COBEM-2017-1136
NUMERICAL ANALYSIS OF THE HEAT TRANSFER THROUGH A HEAT TREATMENT FURNACE

Paulo Gabriel Cunha Martins
Kesiany Máxima de Souza
Felipe Brito Gonçalves Barbosa
University of Brasilia
paulomartins92@gmail.com, kesianymaxima@gmail.com, felipebarbosa313@gmail.com

Olexiy Shynkarenko
LAICA, University of Brasilia/FGA
olexiy@aerospace.unb.br

Abstract. The present study is focused on the presentation of numerical analysis that were applied during a project of an electric furnace developed by a student group from the University of Brasília. The heat treatment furnace was developed with the educational objective to assist some student projects during an analysis of some materials treatment that requires a temperature of experimentation lower than 1200°C, for example, aluminum melting or steel tempering. Along the project, the student group were faced with some challenges that aren't easily solved, like on the design and choice of materials for the furnace and the calculation of the temperature changing over time inside the furnace. Like there isn't any simple analytical method that could be used to calculate the temperature distribution inside the electric furnace or its rate change over time operation, it were realized some numerical analysis to determine those properties, in order to save money and input some reliability during the manufacturing process. The first analysis was carried out to validate the materials chosen and the design of the furnace, while on the second analysis was determined the temperature variation over time operation in the furnace during the heating and cooling process.

Keywords: heat treatment, transient heat transfer, electric furnace, finite element method, thermal simulation

1. INTRODUCTION

In heat treatment, heat is added to a material in order to alter its strength, hardness and others properties by using techniques such as annealing, hardening or tempering (Win, 2017). The industrial furnaces main purpose is the heat treatment of materials. It can be used for important applications, such as material phase change or micro-structure change, for example, aluminum melting or steel tempering, which requires environments with temperature lower than 1200°C.

The first objective of current study was the calculation of the temperature behavior inside the heat treatment furnace, which relates with the validation of the insulation materials and the design of the electric furnace, the second objective of this project was the determination of the temperature distribution during the heating and cooling processes of the furnace, because it was a requirement of the project to the installation of the control thermal system unit, since the resistance in continuum work could be destroyed through a high temperature experience over a long time operation.

The first step on this project was the design of the structure and the choose of the refractory material to keep the outer surface of the structure within the limit of 65°C while the inner volume working at 1200°C . Therefore it were realized a stationary simulation to validate the choose of the materials and the design selected.

With the design and materials established, it were realized the transient simulation of the heat furnace by using a finite element analysis (FEA) to calculate the curves of heating and cooling of the oven. Those data are required for the configuration of the thermal control system of the oven. As Knop (2009, pg.1) says, "*This method (FEA) allows companies to foresee how a product will respond in real-time situations before they actually begin the constructions of that product*", and therefore, the validated computational simulation of a product such as a furnace, saves money, time, and offers a wide range of analysis that can be done in order to optimize the product.

The main goal of the present numerical simulations are the validation of the materials and the design of the furnace using numerical methods, without need of prototypes construction and experimental analysis. The basic laws of heat transfer that governs the numerical methods used on ANSYS, which accounts for conduction, convection and radiation modes, are the Fourier's law, Newton's law of cooling and the Stefan-Boltzmann law, respectively (Grebenisan *et al.*, 2014).

P. G. C. Martins, K. M. Souza, F. B. G. Barbosa, O. Shynkarenko
T. A. O. T. H. T. T. A. H. T. F.

2. LITERATURE REVIEW

The heat conduction through the furnace walls is determined based on the energy balance from which Cengel (2012, pg.68-69) and Grishin (2011, pg. 3) derived equation 1 below:

$$\frac{\partial}{\partial x}\left(k_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k_z \frac{\partial T}{\partial z}\right) = \rho C_p \frac{\partial T}{\partial t} - \dot{g} \tag{1}$$

where k_i is the thermal conduction in direction i [W/(m°C)], ρC_p is the heat capacity of the material [J/m^3] which represents the heat storage capability of the material, $\dot{g}$ is the volumetric heat generation [W/m^3] and T is the temperature [°C].

Equation 1 can be applied to a thermally orthotropic material which have different properties on each direction, and it quantifies the conduction heat transfer per unit volume per unit time [W/m^3]. This phenomenon is usually induced by molecular or electronic vibration (Grishin, 2011).

The finite difference equation 1 can be applied for each interior node of the furnace but not to the nodes on the boundaries. In order to obtain the finite difference equations of boundary nodes, accounting for the convection and radiation heat transfer on the furnace surfaces, whether internal or external, a combined convection and radiation boundary condition was assumed, and it can be represented by the following equation 2 as showed by Cengel (2012, pg. 276):

$$hA(T_\infty - T_0) + \epsilon\sigma(T_{sur}{}^4 - T_0{}^4) + kA\frac{T_1 - T_0}{\Delta x} + \dot{g}_0 A \frac{\Delta x}{2} = 0 \tag{2}$$

where h is the convective heat transfer coefficient [W/(m^2°C)], AΔx/2 is the volume of the element [m^3], A is the heat transfer area [m^2], T_∞ is the temperature of the surrounding medium [°C], T_{sur} is the temperature of the surrounding surfaces [°C], ϵ is the dimensionless emissivity of the surface, σ is the Stefan-Boltzman constant [W/m^2 °C^4], 0 is the subscript for the node at the surface, and 1 is the subscript for the node at the Δx distance from node 0.

The convective heat transfer coefficient h_{cv} is a very important parameters since it gives the amount of heat energy that will pass through a unit area of a given substance in a unit time in order to raise the temperature in one degree, and it can characterize either a natural or forced convection (Logan, 2010). On the present study, it was considered the natural convection of air inside and outside the furnace, where h_{cv} was considered to assume a value of 10W/(m^2°C) on the exterior of the furnace, and 5W/(m^2°C) inside the furnace. These h_{cv} values where based on the ones showed on table 1.

Table 1. Typical values for the convective heat transfer coefficient, Lavine (2008)

Process	Physical state	h_{cv} $(W/m^2°C)$
Natural Convection	Gases	2-25
	Liquid	50-1000
Forced Convection	Gases	25-250
	Liquid	100-20000
Convection with phase change	Boiling and condensation	2500-100000

Before the furnace manufacturing, a literature review about the possible materials that could be used on its fabrication, was done. From it, the materials showed on table 2 were chosen according to the requirements and lower cost as possible. The ceramic refractory panel are a good choice for the main furnace structure due to its high capacity to reflect heat, generated from the resistors, back into its interior (DeCrisci, 2015). The resistances Kanthal A1 can be used at temperature up to 1400°C and they have resistivity and very good oxidation resistance, what makes them perfect for heating the furnace for heat treatment presented on this study (Kanthal, 2017). For the furnace bodywork, it was chosen the stainless steel since its properties would offer a good resistance to the furnace from phenomena such as corrosion (ASSDA, 2017).

Table 2. Materials properties involved on the design of the heat furnace.

Materials	Density (Kg/m^3)	Mean conductivity $(W/(m°C))$	Mean specific heat $(J/(Kg°C))$
Ceramic refractory panel	300	0,2	790
Resistances Khantal A1	7100	25	700
Stainless stell	7850	60,5	434

3. COMPUTATIONAL PROCEDURE

In order to validate the furnace structure and to observe its heat transfer, a numerical analysis by Finite Elements Method was made using the Steady-State Thermal and Transient Thermal modules of the ANSYS Workbench 16.2. The analysis and validation of the furnace isolation was done by the steady module, and the cooling and heating procedures were observed by using the transient module.

3.1 Steady State Thermal Simulation

First of all, it was constructed a CAD model of the furnace, then it was realized its simplification in order to make possible its discretization and meshing for the numerical simulations, as shown in figure 1.
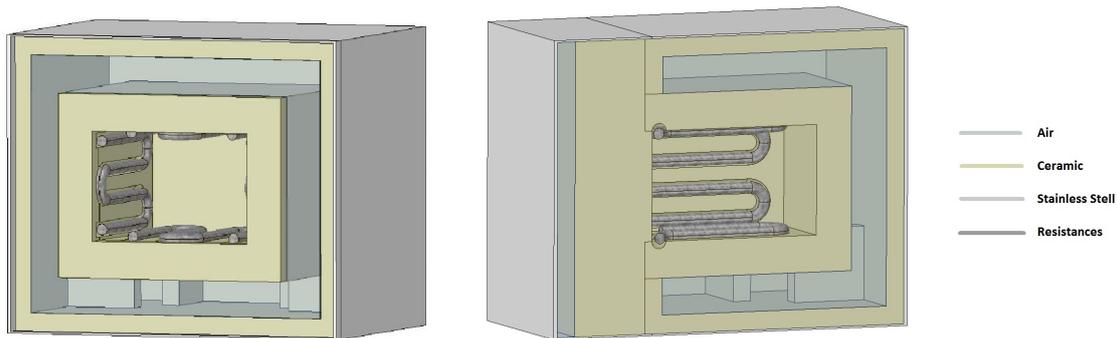
Figure 1. Longitudinal and transversal cut of the electric furnace.

The second step was the discretization of the solid bodies with the creation of a mesh with approximately 115 thousand 3D finite elements, which can be seen in figure 2, and was used for the steady analysis. The mesh mapping allowed to reduce the cell's quantity and to improve the quality of the solution.
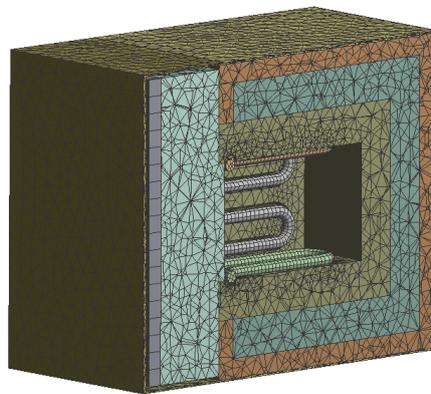
Figure 2. Transversal cut and mesh definition of the electric furnace.

For this analysis the boundary conditions considered were the conduction through the materials, convection and radiation related to the external ambient with ambient temperature (T = 30°C). Beyond those were considered an initial uniform temperature of the resistances at 1200°C and were determined the distribution of the equilibrium temperature over all the structure. Then, the numerical calculation of the solution for the steady-state thermal analysis could be executed by calculating the solutions of the equations 1 and 2 discretized and represented by a general matrix based on Fourier's Law, as shown on equation 3, presented by ANSYS Manual (2005, p. 3).

$$[K(T)]T = Q(T) \tag{3}$$

where [K(T)] is the matrix with the temperature-dependent or constant thermal conductivity, T is the vector which must be solved in order to find the temperatures, and Q(T) is the system where heat flux, heat flow rate, convection and radiation are treated as boundary conditions. This equations represent, therefore, a conduction-based solution with no time-dependent effects (ANSYS, 2005).

3.2 Transient Thermal Simulation

The transient heat transfer analysis was used to simulate the heating and cooling processes of the furnace and to determine the timing necessary to heat the inner air or the material that will suffer the temperature variation. This analysis were one of the most important requirement of the project, since it were required this data to configure the thermal control system unit to act safely without damage the resistance.

Once the materials and design of the heat furnace were validated through the last simulation, the transient thermal simulation were prepared. A transient simulation demands much more computational resources compared with a static simulation, therefore it was realized a simplification of the CAD to get the results, as can be seeing at the figure 3.
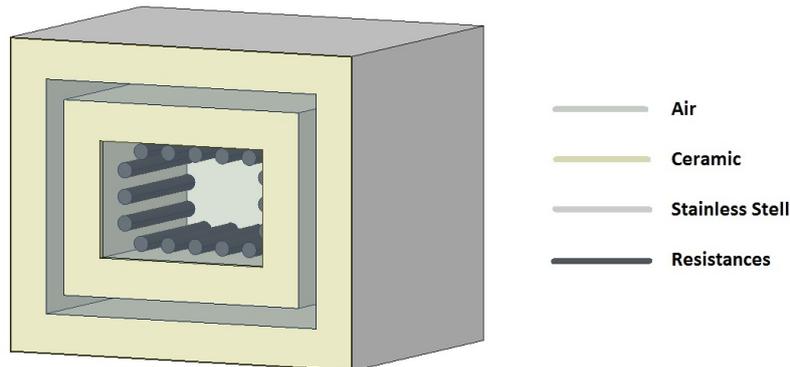

Figure 3. Transversal cut of the electric furnace.

With this simplified CAD and with the air added around the resistances, it were possible to get a mesh with 139 thousand 3D elements with a good distribution of the elements around all over the geometry. For both simulations, the mesh were distributed over the geometry utilizing the methods SOLID87, SOLID90, CONTA174, TARGE170 and SURF152, ANSYS (2017).
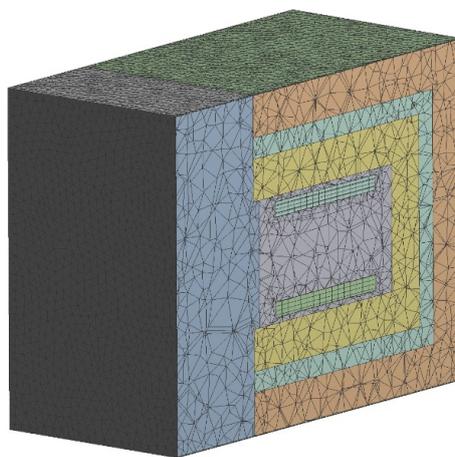

Figure 4. Longitudinal cut and mesh definition of the electric furnace.

On the configuration of the boundary conditions for the heating processes it were considered beyond the convection and radiation presented on the last simulation, constant temperature of 1200°C and an equivalent heat flux over the surfaces at 3 kW. For the cooling simulation, it were considered all the inner air and the resistance with initial temperature of 1200°C. After those configurations, for both cases were determined the distribution over all the structure for the temperature over the time.

In addition, the numerical calculation could be accomplished following basically the same procedures and elements as the steady-state thermal analysis, where the main difference is that the transient thermal analysis have some applied loads varying with time (ANSYS, n.d.).

4. RESULTS AND DISCUSSION

4.1 Validation of the Project

This subsection is devoted to the results of the first simulation. Through the finalization of this simulation the student team were more secure for the mounting of the heat furnace, since it were validated numerically that its design and the materials choose were safe to work and withstand the inner temperature of 1200°C without any damage or overheating of the external surface.
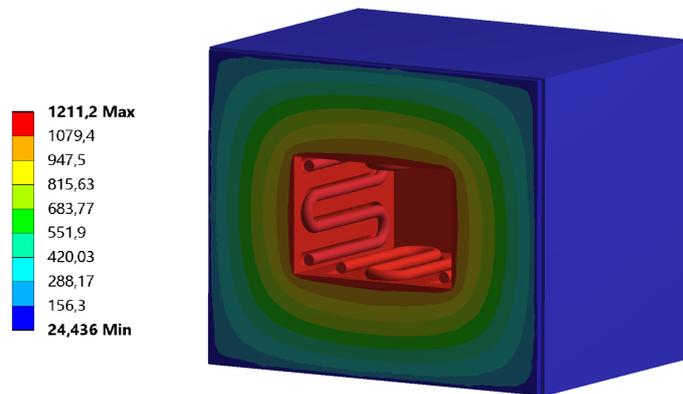

Figure 5. Distribution of the equilibrium temperature over the heat furnace.

As it is shown the external temperature of the oven doesn't exceeds the limit of 60 °C, getting the values between 50 °C and 58 °C while it works on the maximum operational temperature of 1200°C. It were realized during the furnace testing process the measurement of the steady outside wall temperature of 65°C using a pyrometer and it were determined that the simulations results were very accurate. The comparison of the numerical and experimental data allowed to validate the design solutions and the materials choice.

4.2 Cooling of the Furnace

The cooling simulation of the furnace was made using the ANSYS Transient Thermal analysis tool in order to estimate the behavior of the temperature in time. The initial temperature was considered constant and equal to 1200°C over all the structure. The heat dissipation process was realized using the convective and the radiative boundary conditions at the outer surface of the heat furnace.

After the boundary conditions were set and the mesh was established, the simulation runned for about 3600 seconds of real-time, simulating the cooling process the furnace. The distribution of the temperature inside the furnace after 3600 seconds is shown on the figure 6.
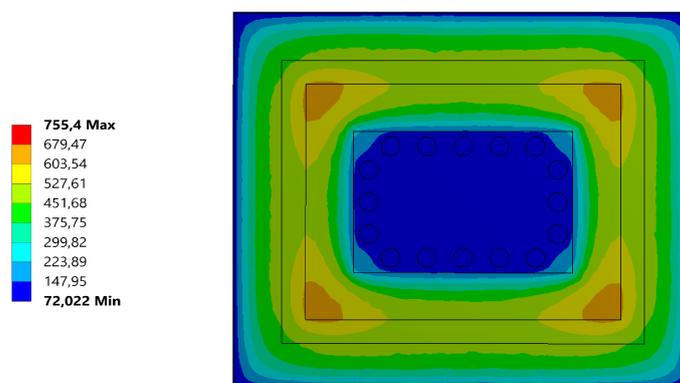

Figure 6. Temperature distribution in the central cross-section after 1 hour of cooling.

Based on the transient solution, it was installed a temperature probe at the center from the useful volume of the heat furnace to calculate the average temperature during the cooling process with the door closed. This result in the following cooling curve presented on the figure 7.
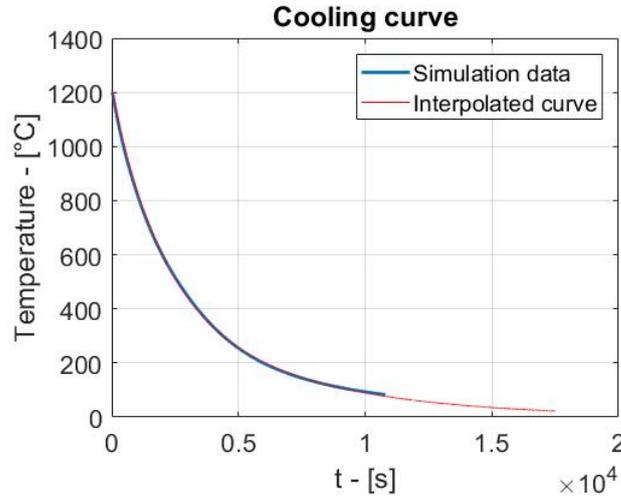

Figure 7. Averaged temperature of the air inside the furnace during the cooling process.

It was done a study of this distribution temperature data over the time and through a interpolation resolution problem were calculated the following cooling function (4) of the heat furnace with door closed.

$$T_{cooling} = 3.31 \cdot 10^{23} e^{-\left(\dfrac{x + 1.831 \cdot 10^5}{2.655 \cdot 10^4}\right)^2} + 1.646 \cdot 10^9 e^{-\left(\dfrac{x + 1.807 \cdot 10^5}{4.657 \cdot 10^4}\right)^2} \tag{4}$$

For data analysis were calculated the standard deviation and the error of the interpolated function over the data determined on the simulations. The standard deviation and the error were calculated respectively through the formulas 5 and 6:

$$\sigma_j = \sqrt{\frac{1}{2} \sum_{i=1}^{2} (x_i - \mu)^2} \tag{5}$$

$$e_j = \frac{|T_{interpolation_j} - T_{simulation_j}|}{T_{simulation_j}} \tag{6}$$

where μ is the mean value between the temperature data determined on the simulation and the interpolated function, while the index j is related to the time interval analyzed. After the calculation for all time interval were determined the maximum value of each parameter and the mean value for all the temperature distribution. The average of those values were calculated simple summing all the values and dividing for the number of steps considered. For the cooling case the parameters calculated are below in the table 3.

Table 3. Data analysis of the cooling function.

$\sigma_{max}\ [^\circ C]$	10.7
$\sigma_{mean}\ [^\circ C]$	3
$e_{max}\ [\%]$	7.4
$e_{mean}\ [\%]$	1.7

4.3 Heating of the Furnace

The simulation of the heating process was performed in order to estimate the temperature evolution in time. The initial temperature of 1200°C and a heat flux of 3kW were considered for the resistors inside the furnace, and the rest of the structure was considered having the ambient temperature of 22°C. The boundary conditions of the convection, conduction and radiation were realized similarly to the previous case. On figure 8 are shown the temperature distributions in the cross sections view.
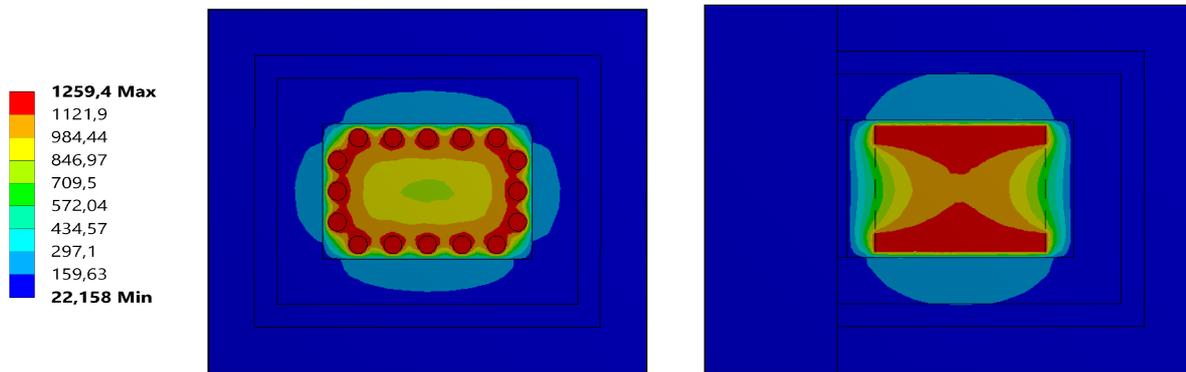

Figure 8. Temperature distributions in the transversal and longitudinal cross-sections of the furnace, respectively.

The furnace was numerically simulated for 3 hour of the real time. Based on this solution, the average temperature of a probe installed on the center point of the useful volume of air inside the heat furnace was calculated and presented on the figure 9.
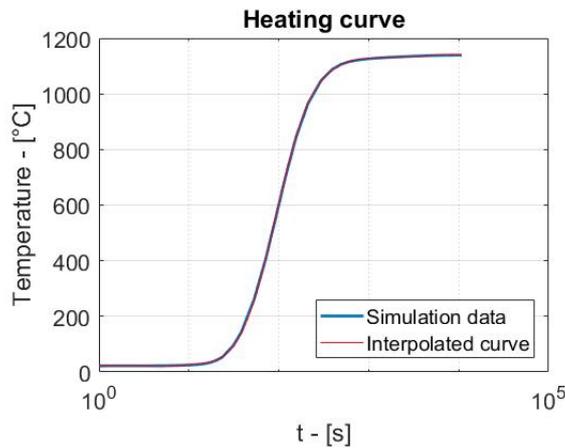

Figure 9. Averaged temperature of the air inside the furnace during the heating process

Like the cooling case, it was determined through an interpolation of the heating curve the average temperature function of a probe installed on the center point of the useful volume inside the electric furnace presented on the equation (7):

$$
\begin{aligned}
T_{heating}(z) = 629.7 \; & - \; 439.8 \cdot cos(z) \; - \; 498.7 \cdot sin(z) \; - \; 14 \cdot cos(2z) \; - \; 85.8 sin(2z) \; + \\
& - \; 114.3 \cdot cos(3z) \; + \; 70.5 \cdot sin(3z) \; - \; 49.2 \cdot cos(4z) \; + \; 6 \cdot sin(4z) \; + \\
& + \; 4.8 \cdot cos(5z) \; + \; 29.7 \cdot sin(5z) \; - \; 1.2 \cdot cos(6z) \; + \; 19.9 \cdot sin(6z) \; + \\
& + \; 4.1 \cdot cos(7z) \; + \; 2.7 \cdot sin(7z) \; + \; 2.1 \cdot cos(8z) \; + \; 1.6 \cdot sin(8z)
\end{aligned}
$$

$$z = 0.5 \cdot ln(t) \tag{7}$$

where t is the time interval considered between 0 and 10800 seconds. The maximum and the average values for standard deviation and error for the heating case were as well determined, they are presented in the table 4

Table 4. Data analysis of the heating function.

σ_{max} $[°C]$	2.5
σ_{mean} $[°C]$	1.5
e_{max} $[\%]$	15.7
e_{mean} $[\%]$	0.2

5. CONCLUSION

The current work allowed to predict the temperature behavior in the prototype furnace using the numerical simulation method. The developed methodology could be used for the selection of the heat insulation material and for the optimization of the geometry in order to attend the requirements to the furnace operation. Furthermore it was determined the cooling and heating curves rate over the operation of the electric furnace to assist the configuration of the thermal control system.

6. REFERENCES

ANSYS, 2005. *Thermal Analysis - Training Manual - Chapter Six.* Ansys Online Manuals - Release 5.5.

ANSYS, 2017. "Ansys online manuals". `http://www.ansys.stuba.sk/html/elem_55/chapter4/`. [Online; accessed 16-Juni-2017].

ANSYS, n.d. *Chapter 3: Transient Thermal Analysis.* Ansys Workbench Simulation. [Online; accessed 24-September-2017].

ASSDA, 2017. "What is stainless steel?" `https://www.assda.asn.au/stainless-steel/what-is-stainless-steel`. [Online; accessed 24-September-2017].

Cengel, Y.A., 2012. *Heat and Mass Transfer: Fundamentals and Applications.* McGraw-Hill Education, 4th edition.

DeCrisci, V., 2015. "Importance of refractory panel replacement". `https://chimneytek.com/importance-of-refractory-panel-replacement/`. [Online; accessed 24-September-2017].

Grebenisan, G., Radu, I.E. and Anton, I., 2014. "Finite elements analysis of thermal steady state, using ansys". *Annals of The University of Oradea - Fascicle of Management and Technological Engineering.*

Grishin, A., 2011. "Mae 323 lecture 8: Heat transfer and multiphysics". `http://www.padtinc.com/mae323/Lecture9_Heat_Transfer_Multiphysics.pdf`. [Online; accessed 24-September-2017].

Kanthal, 2017. *Resistance heating wire and resistance wire.* Sandvik AB. [Online; accessed 24-September-2017].

Knop, N.M., 2009. "Thermal analysis of a fireplace using ansys". *Iowa State University.* [Graduate Theses and Dissertations.10496].

Lavine, I.D.B., 2008. *Fundamentos de Transferência de Calor e Massa [Fundamentals of mass and heat transfer].* LTC.

Logan, D.L., 2010. *A First Course in The Finite Element Method.* Cengage Learning, 5th edition.

Win, P., 2017. "Heat treatment furnace". `http://www.win-therm.com/heat-treatment-furnace/`. [Online; accessed 20-September-2017].

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.