

Thermal Modeling Using Finite Element Method – A Review

Khushboo Borasi¹, Avin Chandrakar²

¹PG Student, Thermal Engineering, Oriental University Indore, M.P, India

²Associate Professor, Mechanical Engineering, Oriental University Indore, M.P, India

Abstract: *Approximately 75% of energy consumption in petrochemical and refining industries is used by furnaces and heaters. Ambient air conditions (pressure, temperature and relative humidity) and operational conditions such as combustion air preheating and using excess air for combustion, can affect the furnace efficiency. If the furnaces are operated at optimized conditions, the huge amounts of savings in energy consumptions would be achieved. By modeling and optimizing of a furnace the optimal operation conditions can be obtained. The aim of this paper is providing a mathematical model which is able to calculate furnace efficiency with change in operating and combustion air conditions.*

Keywords: finite element method, thermal engineering, blast furnace, optimization, simulation.

1. Introduction

Heat transfer is involved in almost every kind of physical process, and in fact it can be the limiting factor for many processes. Lately, the modeling of heat transfer effects inside industrial furnaces has started drawing attention of many more investigators as a result of the demand for energy conservation through efficiency improvement and for lower pollutant emissions. It also has become ever more important in the design of the products itself in many areas such as the electronics, automotive, machinery and equipment manufacturing industries. Both experimental work and numerical analysis through mathematical models has proven to be effective in accelerating the understanding of complex problems as well as helping decrease the development costs for new processes. In the past, only large companies could afford the cost of sophisticated heat transfer modeling tools, therefore the savings in large production runs justified the costs in specialized engineers and computer software. Nowadays, modeling has become an essential element of research and development for many industrial, and realistic models of complex three dimensional structure of the furnace are feasible on a personal computer.

A heat treatment furnace is a manufacturing process to control the mechanical and physical properties of metallic components. It involves furnace control, turbulent flow, conduction within the load, convection and thermal radiation simultaneously. The thermal history of each part and the temperature distribution in the whole load are critical for the final microstructure and the mechanical properties of workpieces and can directly determined the final quality of parts in terms of hardness, toughness and resistance. To achieve higher treatment efficiency, the major influencing factors such as the design of the furnace, the location of the work pieces, thermal schedule and position of the burners should be understood thoroughly.

The damage to the global environment and the prospective depletion of essential resources caused by growing human activity constitute a dual challenge that calls for coordinated measures by multilateral organizations such as ADEME,

French Environment and Energy Management Agency. This is an industrial and commercial public agency, under the joint supervision of French Ministries for Ecology, Sustainable Development and Spatial Planning (MEDAD) and for Higher Education and Research with a mission to encourage, supervises, coordinate, facilitate and undertake operations aiming in protecting the environment and managing energy.

Since simulation of the heating up process of work pieces in heat treatment furnaces is of great importance for the prediction and control of the ultimate microstructure of the work pieces but specially the reduction of both energy consumption and pollutant emissions, this agency supports our research program and encourages all players and partners in this project to save energy, particularly sectors that consume high quantities of energy on daily basis.

2. Role of Computational Modeling in Heat Furnace Design

As mentioned previously, the major factor to be considered in the working of a furnace is the heat transfer by all the modes, which occur simultaneously. To either study a new furnace or to optimize the heating process in existing ones, the heat transfer in the furnace has to be modeled in the same way of a real situation as closely as possible. Given the geometry of the furnace, different boundary conditions along the furnace length, gas composition and properties and other complexities, an analytical solution is not feasible and computational modeling has to be resorted to. Over the last 20 years, the CFD (Computational Fluid Dynamics) has gained its reputation of being an efficient tool in identifying and solving such problems. Modeling the heating process involves solving coupled heat transfer equations. For solving them computationally, the method should be capable of doing so in an accurate way and within a reasonable time. In the heat transfer of heat treatment furnace, there are conduction, convection, radiation, turbulent flow and furnace control. Conduction mainly occurs in all solids materials. Turbulent convection exists between the atmosphere and solid materials exposed to it, and furnace

walls to the ambient air. Radiation exists between solid materials exposed to each other and to all walls. The tools used in this thesis are the Finite Element Method (FEM) and Computational Fluid Dynamics (CFD). This method is shown as an attractive way to solve the turbulent flow and heat transfer in the furnace chamber and it can be applied for a variety of furnace geometry and boundary conditions.

The entire heat transfer process is a transient one, and iterations are necessary. The main process is detailed in the following flowchart:

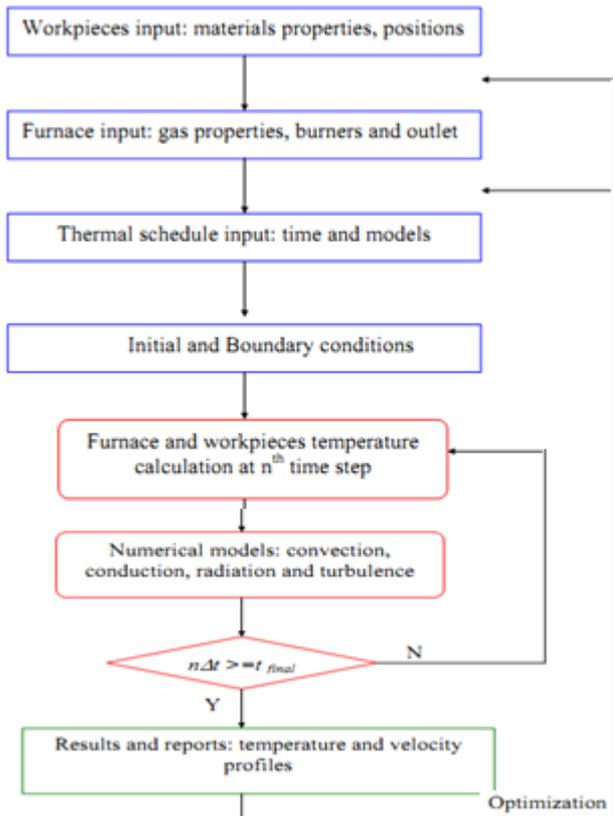


Figure 1: General flowchart for a heat treatment process

3. Finite Element Method

A finite element method (abbreviated as FEM) is a numerical technique to obtain an approximate solution to a class of problems governed by elliptic partial differential equations. Such problems are called as boundary value problems as they consist of a partial differential equation and the boundary conditions. The finite element method converts the elliptic partial differential equation into a set of algebraic equations which are easy to solve. The initial value problems which consist of a parabolic or hyperbolic differential equation and the initial conditions (besides the boundary conditions) can not be completely solved by the finite element method. The parabolic or hyperbolic differential equations contain the time as one of the independent variables. To convert the time or temporal derivatives into algebraic expressions, another numerical technique like the finite difference method (FDM) is required. Thus, to solve an initial value problem, one needs both the finite element method as well as the finite difference method where the spatial derivatives are converted into algebraic expressions by FEM and the

temporal derivatives are converted into algebraic equations by FDM.

The words "finite element method" were first used by Clough in his paper in the Proceedings of 2nd ASCE (American Society of Civil Engineering) conference on Electronic Computation in 1960. Clough extended the matrix method of structural analysis, used essentially for frame-like structures, to two-dimensional continuum domains by dividing the domain into triangular elements and obtaining the stiffness matrices of these elements from the strain energy expressions by assuming a linear variation for the displacements over the element. Clough called this method as the finite element method because the domain was divided into elements of finite size. (An element of infinitesimal size is used when a physical statement of some balance law needs to be converted into a mathematical equation, usually a differential equation).

Argyris, around the same time, developed similar technique in Germany. But, the idea of dividing the domain into a number of finite elements for the purpose of structural analysis is older. It was first used by Courant in 1943 while solving the problem of the torsion of non-circular shafts. Courant used the integral form of the balance law, namely the expression for the total potential energy instead of the differential form (i.e., the equilibrium equation). He divided the shaft cross-section into triangular elements and assumed a linear variation for the primary variable (i.e., the stress function) over the domain. The unknown constants in the linear variation were obtained by minimizing the total potential energy expression. The Courant's technique is called as applied mathematician's version of FEM where as that of Clough and Argyris is called as engineer's version of FEM.

From 1960 to 1975, the FEM was developed in the following directions :

- 1) FEM was extended from a static, small deformation, elastic problems to dynamic (i.e., vibration and transient) problems, small deformation fracture, contact and elastic-plastic problems, non-structural problems like fluid flow and heat transfer problems.
- 2) In structural problems, the integral form of the balance law namely the total potential energy expression is used to develop the finite element equations. For solving non-structural problems like the fluid flow and heat transfer problems, the integral form of the balance law was developed using the weighted residual method.
- 3) FEM packages like NASTRAN, ANSYS, and ABAQUS etc. were developed.

The large deformation (i.e., geometrically non-linear) structural problems, where the domain changes significantly, were solved by FEM only around 1976 using the updated Lagrangian formulation. This technique was soon extended to other problems containing geometric non-linearity :

- dynamic problems,
- fracture problems,
- contact problems,
- elastic-plastic (i.e., materially non-linear) problems.

Some new FEM packages for analyzing large deformation problems like LS-DYNA, DEFORM etc. were developed around this time. Further, the module for analyzing large deformation problems was incorporated in existing FEM packages like NASTRAN, ANSYS, ABAQUS etc.

4. Basic Steps in FEM

The finite element method involves the following steps.

First, the governing differential equation of the problem is converted into an integral form. There are two techniques to achieve this : (i) Variational Technique and (ii) Weighted Residual Technique. In variational technique, the calculus of variation is used to obtain the integral form corresponding to the given differential equation. This integral needs to be minimized to obtain the solution of the problem. For structural mechanics problems, the integral form turns out to be the expression for the total potential energy of the structure. In weighted residual technique, the integral form is constructed as a weighted integral of the governing differential equation where the weight functions are known and arbitrary except that they satisfy certain boundary conditions. To reduce the continuity requirement of the solution, this integral form is often modified using the divergence theorem. This integral form is set to zero to obtain the solution of the problem. For structural mechanics problems, if the weight function is considered as the virtual displacement, then the integral form becomes the expression of the virtual work of the structure.

In the second step, the domain of the problem is divided into a number of parts, called as elements. For one-dimensional (1-D) problems, the elements are nothing but line segments having only length and no shape. For problems of higher dimensions, the elements have both the shape and size. For two-dimensional (2D) or axi-symmetric problems, the elements used are triangles, rectangles and quadrilateral having straight or curved boundaries. Curved sided elements are good choice when the domain boundary is curved. For three-dimensional (3-D) problems, the shapes used are tetrahedron and parallelepiped having straight or curved surfaces. Division of the domain into elements is called a mesh.

In this step, over a typical element, a suitable approximation is chosen for the primary variable of the problem using interpolation functions (also called as shape functions) and the unknown values of the primary variable at some pre-selected points of the element, called as the nodes. Usually polynomials are chosen as the shape functions. For 1-D elements, there are at least 2 nodes placed at the end-points. Additional nodes are placed in the interior of the element. For 2-D and 3-D elements, the nodes are placed at the vertices (minimum 3 nodes for triangles, minimum 4 nodes for rectangles, quadrilaterals and tetrahedral and minimum 8 nodes for parallelepiped shaped elements). Additional nodes are placed either on the boundaries or in the interior. The values of the primary variable at the nodes are called as the degrees of freedom.

To get the exact solution, the expression for the primary variable must contain a complete set of polynomials (i.e.,

infinite terms) or if it contains only the finite number of terms, then the number of elements must be infinite. In either case, it results into an infinite set of algebraic equations. To make the problem tractable, only a finite number of elements and an expression with only finite number of terms are used. Then, we get only an approximate solution. (Therefore, the expression for the primary variable chosen to obtain an approximate solution is called an approximation). The accuracy of the approximate solution, however, can be improved either by increasing the number of terms in the approximation or the number of elements.

In the fourth step, the approximation for the primary variable is substituted into the integral form. If the integral form is of variational type, it is minimized to get the algebraic equations for the unknown nodal values of the primary variable. If the integral form is of the weighted residual type, it is set to zero to obtain the algebraic equations. In each case, the algebraic equations are obtained element wise first (called as the element equations) and then they are assembled over all the elements to obtain the algebraic equations for the whole domain (called as the global equations).

In this step, the algebraic equations are modified to take care of the boundary conditions on the primary variable. The modified algebraic equations are solved to find the nodal values of the primary variable.

In the last step, the post-processing of the solution is done. That is, first the secondary variables of the problem are calculated from the solution. Then, the nodal values of the primary and secondary variables are used to construct their graphical variation over the domain either in the form of graphs (for 1-D problems) or 2-D/3-D contours as the case may be.

5. Advantages of FEM Over Other Numerical Methods

Advantages of the finite element method over other numerical methods are as follows:

- 1) The method can be used for any irregular-shaped domain and all types of boundary conditions.
- 2) Domains consisting of more than one material can be easily analyzed.
- 3) Accuracy of the solution can be improved either by proper refinement of the mesh or by choosing approximation of higher degree polynomials.
- 4) The algebraic equations can be easily generated and solved on a computer. In fact, a general purpose code can be developed for the analysis of a large class of problems.

6. Modeling Boundary Value Problem Using FEM

To illustrate the development of integral formulations, the following model boundary value problem is considered. It represents the axial extension (or compression) of a bar shown in Fig. 2

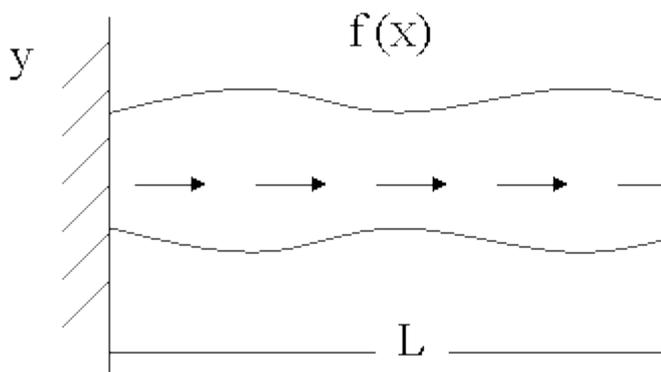


Figure 2: Axial Extension of a bar

The bar has a variable area of cross-section which is denoted by the function $A(x)$. The length of the bar is L . The Young's modulus of the bar material is E . The bar is fixed at the end $x = 0$. The forces acting on the bar are (i) a distributed force $f(x)$, which varies with x and (ii) a point force P at the end $x = L$. The axial displacement of a cross-section at x , denoted by $u(x)$, is governed by the following boundary value problem consisting of a differential equation (DE) and two boundary conditions (BC):

$$DE: -\frac{d}{dx} \left(EA(x) \frac{du}{dx} \right) = f(x) \quad 0 < x < L \quad (3.1 a)$$

$$BC: (i) u = 0 \text{ at } x=0 \quad (3.1 b)$$

$$(ii) EA(x) \frac{du}{dx} \text{ at } x=L \quad (3.1 c)$$

The differential equation represents the equilibrium of a small element of the bar expressed in terms of the displacement using the stress-strain and strain-displacement relations. The boundary condition (2.1b) is a geometric or kinematic boundary condition. Since, it is a condition on the primary variable $u(x)$, it is called as Dirichlet boundary condition. The second boundary condition (condition 1c) is a force boundary condition, or a condition on the secondary variable (i.e., axial force). Since, it is a condition on a derivative of the primary variable; it is called as the Neumann boundary condition.

7. Conclusion

Finite element method is the most widely used method for optimization problems. As seen from the works of [1] that optimization could also lead to efficiency of the overall setup, an effort has been made by us in this research work to implement the concepts of FEM to a resistance and temperature problem of industrial furnace which is the heart of almost any industry. Also the above said method helped us learning the basics of Matlab and would increase our skill set.

References

- [1] Francis Wildy, "Fired Heater Optimization", AMETEK Process Instruments, 2000.
- [2] S. Mussati, Juan I. Manassaldi, "Mixed Integer Non Linear Programming Model For The Optimal Design Of Fired Heaters", Applied Thermal Engineering, Vol. 29, pp.2194-2204, 2009.

- [3] Hassan Al-Haj Ibrahim. "Thermal efficiency of fired heater," 2008.
- [4] Worrell E, Galitsky C. "Energy efficiency improvement and cost saving opportunities for petroleum refineries", Lawrence Berkeley National Laboratory report LBNL-56183. Berkeley, CA.
- [5] Z. Jegla, P. Stehlik, J. Kohoutek, "Plant energy saving through efficient retrofit of furnaces", Applied Thermal Engineering Vol.20, pp. 1545-1560, 2000.
- [6] Z. Jegla, "The Conceptual Design of a Radiant Chamber and Preliminary Optimization of a Process Tubular Furnace", Heat Transfer Engineering, Vol.27, pp.50-57, 2006.
- [7] Jegla, Z., Kohoutek, J., And Stehlik, P., "Global Algorithm For Systematic Retrofit Of Tubular Process Furnaces", Applied Thermal Engineering, Vol. 23, pp.1797-1805, 2003.
- [8] Taal, M., Bulatov, I., Klemes, J. And Stehlik, P., "Cost Estimation And Energy Price Forecasts For Economic Evaluation Of Retrofit Projects", Applied Thermal Engineering", Vol.23, pp. 1819-1835, 2003.
- [9] A. Garg. "Revamp fired heater rating "hydrocarbon processing, pp.67-80, 1998.
- [10] Bussman W, Baukal C. "Ambient condition effects on process heater emissions", Energy, 2008.
- [11] W.R. Bussman, C.E. Baukal, "Ambient condition effects on process heater efficiency", John Zink Co. LLC, 11920 East Apache, Tulsa, Energy, 2009.
- [12] W.E. Lobo, J.E. Evans, "Heat transfer in the radiant section of petroleum heaters", Trans IChE, 743-778, 1939
- [13] Nelson, W.L. "Petroleum Refinery Engineering", 4th ed, McGraw-Hill.