

## **AC 2009-182: APPLICATION OF FEMLAB SOFTWARE FOR SIMULATION OF THE THERMAL METHOD FOR NONDESTRUCTIVE TESTING PROCESSES**

### **Anatoliy Protasov, National Technical University of Ukraine “KPI”**

Dr. Anatoliy Protasov is a Head of Nondestructive Testing Department of National Technical University of Ukraine “KPI”. He graduated from the National Technical University of Ukraine “Kiev Polytechnical Institute”, Electric Engineering Department, MS, Ph.D. He has research experience in Nondestructive Testing Methods.

# Application of FEMLAB Software for Simulation of the Thermal Method for Nondestructive Testing

## Introduction

The thermal nondestructive testing (TNDT) method is widely used for inspection of industrial parts and components. The method involves heating the object and subsequently measuring the temperature of its surface. This change in temperature provides information about the test object's structure. The surface temperature changes if the discontinuity exists inside the object.

The laboratory training allows students to understand the fundamental processes, which take place during the TNDT procedure. Students are able to simulate this procedure use FEMLAB software package. This interactive software package is based on application of partial differential equations for simulation of scientific and technical problems related to TNDT. Applied modes of this software allow the development of models with desired properties of a test object and heat flow. FEMLAB package generates the system of partial differential equations, which represent a complete model of TNDT processes. The finite element analysis method is used for solving partial differential equations.

## I. Creation of the thermal processes model in FEMLAB software

A blast furnace is used as a test object (TO). The cast-iron blocks, which have internal discontinuities, will be tested. A defect (blister or crack) has different heat-transfer properties than metal. Distortion of the thermal field on the surface of a metal block will indicate the presence of the discontinuity.

### Problem statement

The 2D model of a TO will be discussed below in details. A cross-section of the cast-iron 8 x 4 cm rectangular block, which has 2 cm in diameter blister filled with air, will be used as a model. The temperature of one of the block surfaces is 873 K. The other three block surfaces have ambient temperature -293K. Temperature distribution on the block surfaces could be determined according to the heat-transfer properties of cast-iron material and air. These properties are presented in the handbook.

### Simulation technique

There is a need to activate the FEMLAB **Graphical User Interface (GUI)** to start simulation. A figure of the **Model Navigator** panel appears after the **GUI** activation (see Fig.1). It is necessary to mark the button **2D** and choose the model **Physics modes/ Heat transfer/ Time depended** to prepare the **GUI** for creation of the two-dimensional TO model.

### Creation of the TO geometry

A cross-section of the TO can be drawn using the option **Draw** from the main menu. The origin of coordinate is placed in the center of the TO which is represented as a rectangle (8x4 cm) with a circle 2cm in diameter. A rectangle and circle with a necessary size can be drawn in the field of **Axes** using a mouse. The model of the defective TO cross-section is represented on the Fig. 2.

## Boundary conditions specifying

It is necessary to switch the **GUI** application to the boundary conditions mode **Boundary/ Boundary Mode** to specify boundary conditions. This mode is used for displaying the internal and external segments of the TO in the field of axes. The mode **Boundary/ Boundary Setting** allows making selection of the necessary segments (sides) of the TO. The temperature 873K is set for the upper segment in the dialog box. The others segments have temperature 293K.

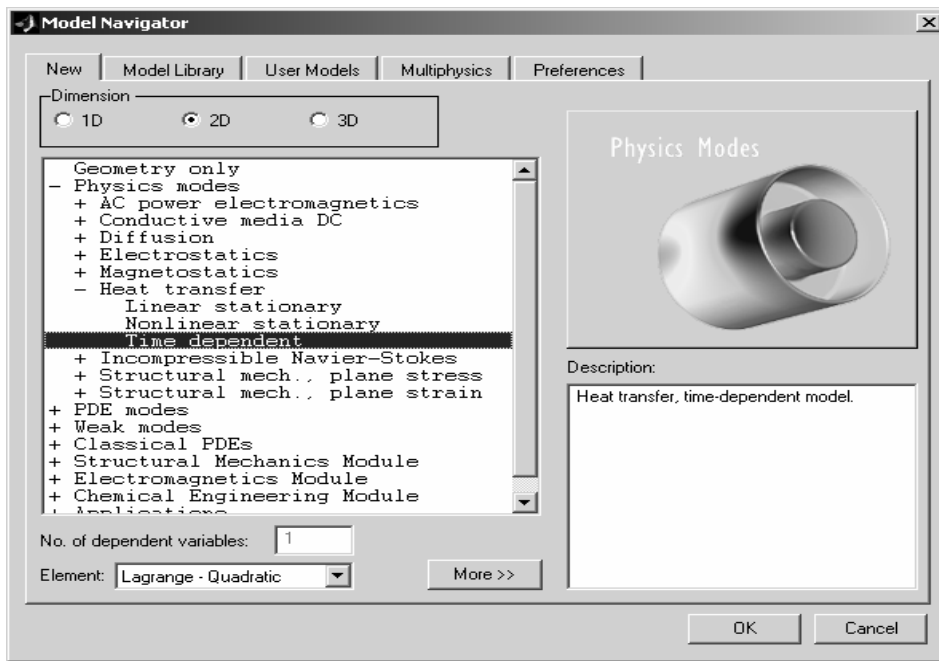


Figure 1: Overview of the Model Navigator panel

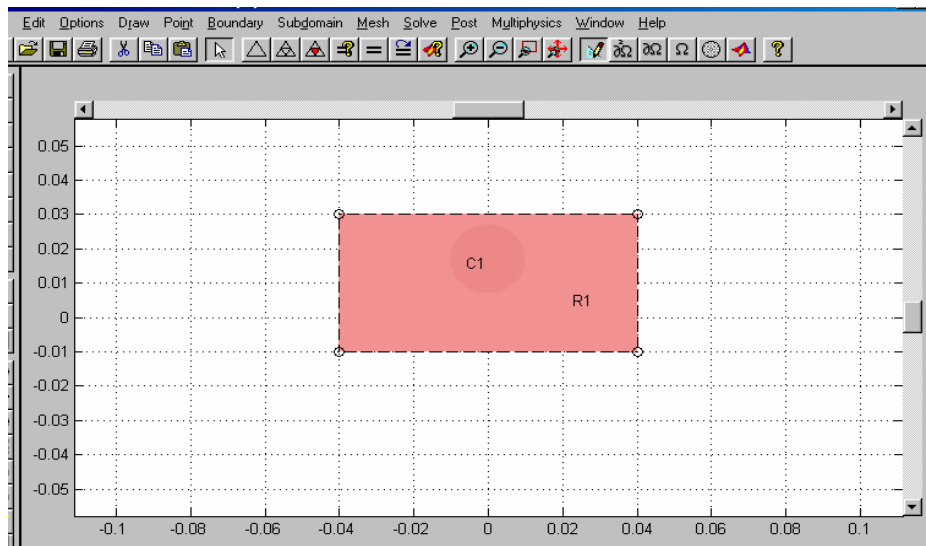


Figure 2: The graphic image of the testing object. R1 - cross-section of the metal block, C1 - discontinuity flaw

## Setting partial differential equations coefficients

The option **Subdomain/ Subdomain Mode** realizes transfer to the mode of setting partial differential equations (PDE) coefficients. This mode displays two zones in the field of **Axes**. The number of zone is indicated by the command **Options/ Labels/ Show Subdomain Labels**. The zone #1 and zone #2 correspond to the TO and a defect accordingly. Heat-transfer coefficients (PDE coefficients) can be set using option **Subdomain/ Subdomain Settings**. The dialog box of this option is shown on Figure 3. The bookmark **Coefficients** allows setting coefficients for zone #1 and zone #2. They are:  $\rho=7850$  (density),  $C=500$  (heat capacity),  $k=46.5$  (thermal conductivity coefficient) and  $\rho=1.29$ ,  $C=1000$ ,  $k=0.029$  for the zone #1 and zone #2 accordingly. The bookmark **Init** is used for setting initial temperature of the TO. This value is equal to 293.

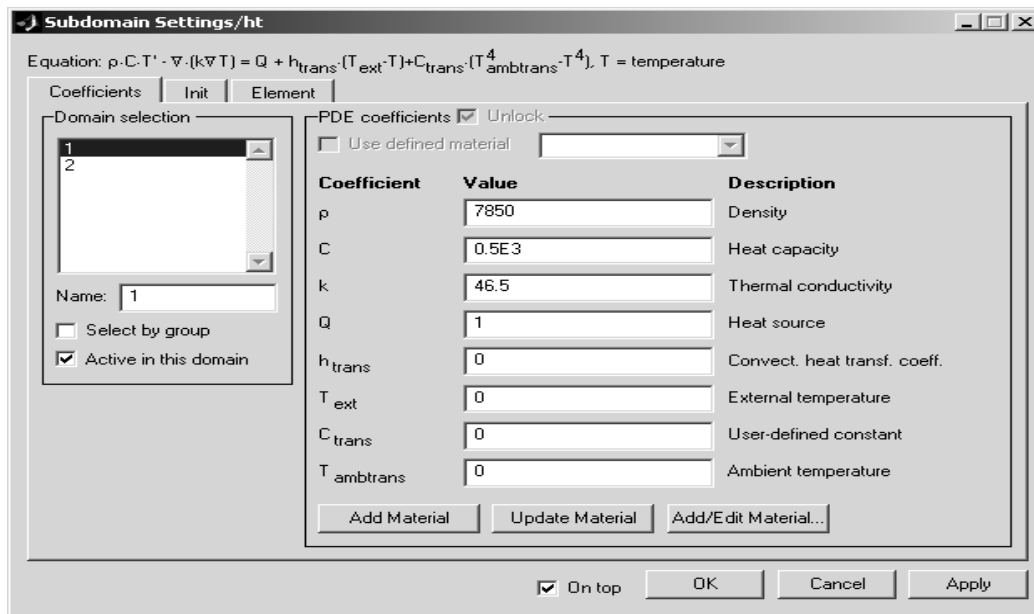


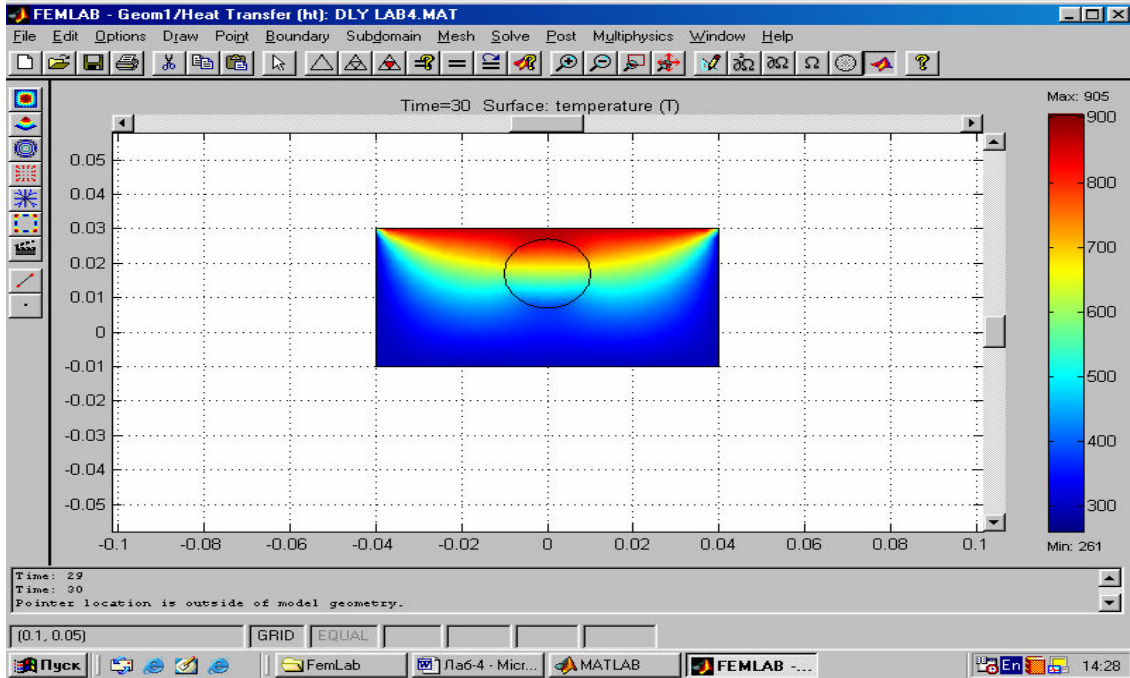
Figure 3: The dialog box for PDE coefficients setting

## Generation of the finite-element mesh and PDE solution

The option **Mesh/ Mesh Mode** realizes generation of the finite-element mesh. The mesh is generated automatically in accordance with current adjustment of a mesh generator. It is possible to increase a mesh using option **Mesh/ Refine Mesh**. The option **Solve/ Parameters** and bookmark **Timestepping** are used for setting the time of simulation and a step of discretization for PDE solution. The simulation time is 30 sec and the discretization step is 1 sec for this case. After pressing the button **Solve**, FEMLAB realizes the mode of the solution visualization (see Fig. 4). It allows to observe temperature distribution down the TO cross section.

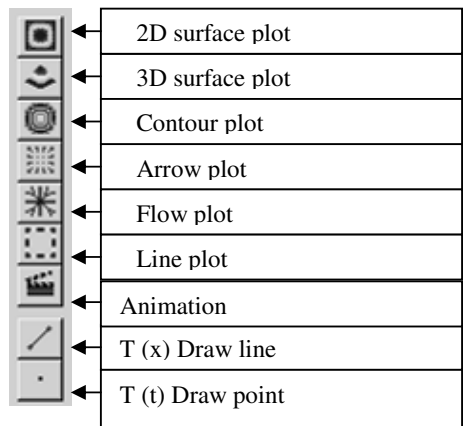
## Visualization modes

Visualization parameters can be changed using the option **Post/ Plot Parameters**. This option changes such parameters as a number of **arrows**, **flow lines**, **contour levels**, and color of the plots. The Figure 5 displays the vertical panel **Plot Type**.



**Figure 4: Visualization of the heat-transfer problem**

There are 9 buttons on this panel. The upper function **2D surface plot** demonstrates a 2D image of a thermal field (in X and Y coordinates). The next function **3D surface plot** makes thermal field visualization in 3 coordinates X, Y, and T, where T is a scale of temperature. The function **Contour plot** shows isotherms placed on the temperature background. The function **Arrow plot** realizes visualization of the vector field of heat flow. The function **Flow plot** shows lines of temperature distribution. The function **Line plot** transforms a plot in the chain line. FEMLAB gives a chance to study and analyze temperature distribution in dynamics. The function **Animation** changes a number of iteration processes and animation speed. It is very important to get temperature distribution along the given line (thermo profile). This function can be realized by the button **Draw line**. The last function **Draw point** shows temperature time change in a given point.



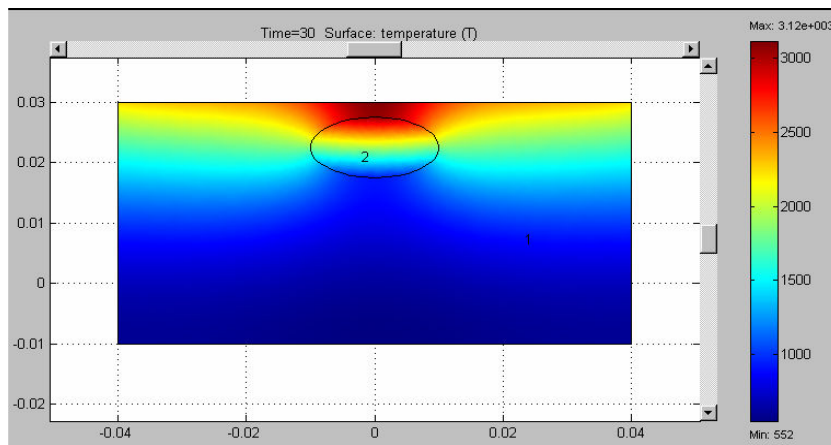
**Figure 5: The image of the vertical panel Plot Type**

## II Definition of the optimal time for testing

It is known that efficiency and reliability of the thermal testing depend on heating time and time of temperature registration. There is a temperature difference between the defective and defect-free zones which reaches maximum only in the definite time. Temperature difference comes down after this time. Determination of this time provides a high reliability of testing. FEMLAB software allows realizing such a procedure.

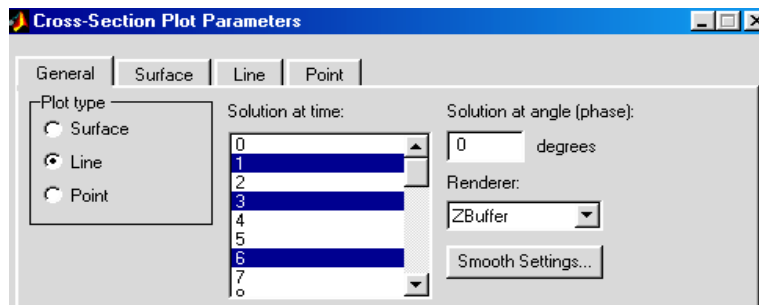
### Definition of the thermal profile

Let's substitute a circle for the ellipse in our model. The ellipse is a more realistic shape for a defect. Figure 6 demonstrates the result of application of the proposed simulation procedure for the ellipse shaped defect.



**Figure 6: Thermal field distribution in the object with the ellipse shaped defect**

It is necessary to open the window **Post/ Cross Section Plot Parameters** to set solution parameters for this task. The box **Solution at time** is used to choose a time domain. The Figure 7 shows that a problem will be solved for the every 3 sec.

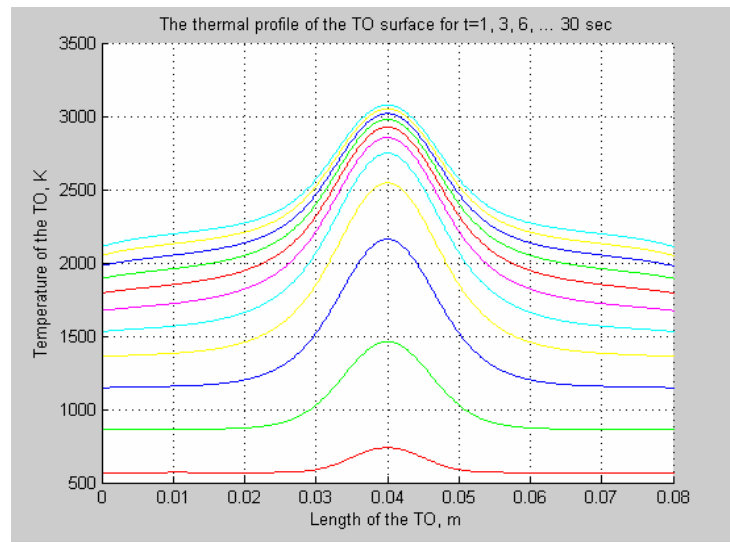


**Figure 7: The dialog box for solution parameters setting**

The bookmark **Line** is used for setting coordinates of the thermal profile line. The box **Select line via** allows to mark the initial and final points of this line. The Figure 8 demonstrates results of the problem solution with mentioned above settings. It is obvious that temperature of the defective zone is higher than defect-free one. It occurs because a defect hinders heat flow propagation. Temperature difference between the defective and defect-free zones

changes with time. This process happens because heat flow goes around the defect and spreads inside of the TO.

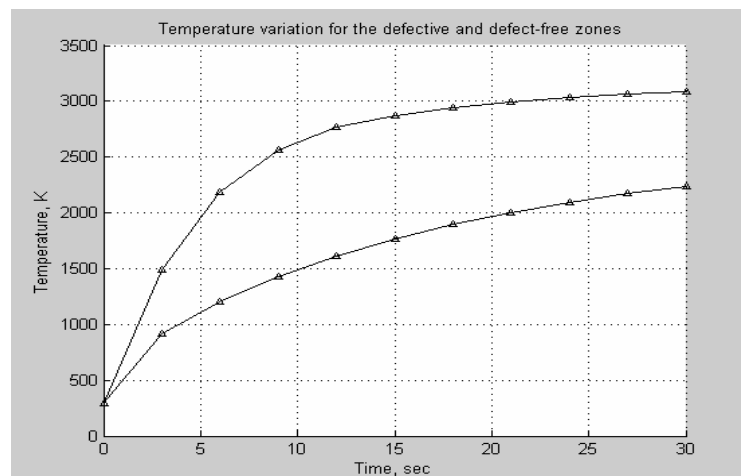
Figure 6 shows that the layer over a defect accumulates thermal energy and the layer beyond is lack of such energy. Thus, a defect leads to local temperature change.



**Figure 8: The thermal profile of the testing object surface in the given moment of time**

#### Definition the local temperature variation in time

There is one more useful option for the TNDT task. It is option **Draw point for cross section point plot**. This option allows finding local temperature variation in time and optimal inspection time. The window **Post/ Cross section plot parameters** and the button **Point plot** in the bookmark **Point** are used to set coordinates for two temperature measuring points. The first point is placed over the defect and the second is over the defect-free surface. The bookmark **General** allows setting time steps for this task (for instance 0, 3, 7, 10, 18, 22, and 30 sec). The result is shown on the Figure 9.



**Figure 9: Temperature variation in time for the defective (upper line) and defect-free (lower line) zones**

These plots demonstrate that the optimal time for inspection is about the 12-13 sec. The temperature difference has unsteady nature and tendency to decrease after the 14 sec. The proposed technique can be used for inspection time estimation.

### **Application for the educational process**

The proposed simulation procedure has been applied in the educational process as a laboratory work used in the context of the subject “Thermal Method of Nondestructive Testing”. The purpose of this work is to teach the 4-th year students to apply FEMLAB software for simulation of the TNDT process. Using instructions (see Appendix A) students learn how to create a complete model of TNDT procedure. They research influence of such factors as properties, size, and depth of a defect on the inspection result. The final result of simulation is visualization of temperature field distribution in the object at arbitrarily selected time.

After application mentioned laboratory work, the students’ progress in studies has been investigated. Students were divided into two groups. The first group (controllable group) didn’t use this laboratory training during the term. The second one (experimental group) applied this training. The results of investigation are adduced in the **Appendix B**.

### **Conclusion**

FEMLAB software package can be used for simulation of the TNDT process. It gives opportunity to investigate a potential of this testing method. The proposed laboratory training helps students to understand physical processes occurred in a solid body under the action of temperature. They can observe processes which are invisible for a human eye and better understand thermal wave propagation in the testing object. It improves the students’ progress in studies.

### **Appendix A: Instructions for laboratory work accomplishment**

**Instructor** specifies materials of a testing object (conductor) and a defect (dielectric), and boundary conditions (temperature **T**, heat flow **Q**, convection heat transfer coefficient **h**, ambient temperature **T<sub>amb</sub>**).

#### **Tasks for students:**

1. Use a hand-book to find necessary material parameters for model creation (density **p**, heat capacity **C**, thermal conductivity **k**).
2. Use the option **Draw** to create a geometric object; set boundary conditions and PDE coefficients; generate the finite-element mesh, set parameters for a PDE solution (bookmark **Timestepping**), and get visualization of a solution.
3. Get thermal distribution into a TO for two arbitrarily selected time points. The plots must be placed in separate windows and have names. These results should be saved in the folder named **Lab Model**.
4. Get the plots of following types: **Contour**, **Arrow** (match optimal number of vectors), **Flow** (match optimal number of lines), **2D surface**, and **3D surface**.



5. Realize the animation process of heat propagation into a TO with different time steps. The optimal result must be saved as the \*.mpeg file.
6. Get the heat process model for the following cases:
  - a test object and a defect are conductors,
  - a test object is dielectric and a defect is conductor,
  - a test object and a defect are dielectrics.
 Compare received results, provide an explanation of these results, and draw conclusions.
7. Estimate the optimal time for a TO heating by heat flow (using the option **Draw line for cross section line plot**).
8. Define the optimal time domain for testing (using the option **Draw point for cross section point plot**).
9. Find the functional relation between the time of testing and defect depth for the given object parameters and temperature conditions.

Finally students prepare a report with analysis of received results.

**Appendix B: The results of students' progress testing**

Student groups	Percentage of good marks			
	quiz 1	midterms	quiz 2	final test
Controllable	78	74	76	72
Experimental	86	80	85	79
Difference	8	6	9	7