SIMULATION OF HEAT TRANSFER AND ELECTROMAGNETIC FIELDS OF PROTECTED MICROCOMPUTERS

J. Lakatoš, J. Čuntala, A. Kondelová

Department of mechatronics and electronics, Faculty of Electrical Engineering,
University of Žilina, Univerzitná 1, 010 26, Žilina, tel.: +41/ 513 16 01,
e-mail : lakatos@fel.utc.sk, cuntala@fel.utc.sk, kondelova@fel.utc.sk

Summary   The paper presents results of collaboration between Department of mechatronics and electronics at University of Žilina and VÚVT Engineering a.s. Žilina in area of heat transfer simulations and disturbing electromagnetic radiation simulations in computer construction. The simulations results were used in development of protected microcomputer prototypes in frame of applied research at both of workplaces.

1. INTRODUCTION

Protected computers prevent electromagnetic radiation from computer case inside and through supply and interface wiring in the area. Shielding technology or optical fibber connection is used to suppress radiation. Electromagnetic shielding technology, which is applied at the computer system, provides two practical aspects:

- Protection of a user against unfavorable effects of electromagnetic field radiated from a computer
- Reduction in risk of misuse of information leaking from computer.

2. DESCRIPTION OF SIMULATING SYSTEM FEMLAB 3.1

FEMLAB 3.1 [1] is engineering tool that performs partial differential equation-based modelling and simulating of physics phenomena. FEMLAB can solve custom partial differential equations (PDEs) or perform specialized physics application modes. The modes execute predefined PDEs for specific areas of physics. By combining the application modes into a single task, it is possible to simulate a multiphysics problem. FEMLAB contains several application modes which consist of graphical user interfaces, equations, parameters and variables.

The application models in FEMLAB are divided into the following categories:

- Acoustics
- Diffusion
- Electromagnetic
- Fluid Dynamics
- Heat Transfer
- Structural Mechanics

FEMLAB 3.1 allows choosing two application modes in the area of transfer:

- Heat transfer by conduction
- Heat transfer by convection and conduction

3. DESCRIPTION OF GEOMETRY MODEL

The real size of a protected computer prototype was used at geometry model generation. Dimensions of the monitor case are: width 36cm, height 36cm and depth 11cm. The opening in front part of monitor steel case for LCD display has width 32cm, height 26cm. Dimensions of two openings in back cover of monitor are 7,5 x 7,5cm. Dimensions of PCB board of display are 30 x 22cm. An active element is aluminium cooler with dimensions 10 x10, height 2cm. Fluorescent tubes for LCD monitor lighting have length 26cm and diameter 10mm. Fig.1 shows 3D monitor model in FEMLAB.
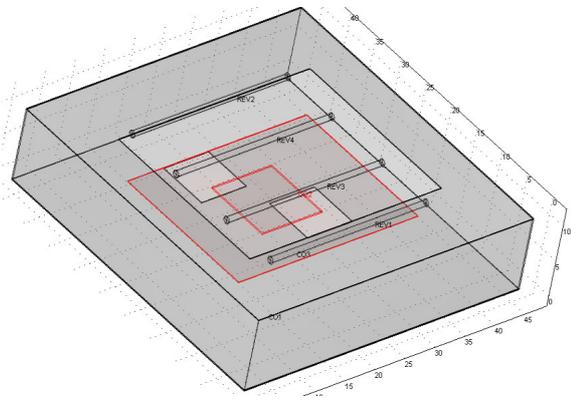
Fig.1   FEMLAB geometry model of the protected monitor

a.   HEAT TRANSFER MODEL

The simulation was performed in application mode Convection and Conduction from Heat Transfer category of FEMLAB modes [2]. The task uses explicitly defaulted Lagrange - quadratic mathematical solver with conservative form of heat equation. FEMLAB represents this formulation in

the Convection and Conduction application mode as:

$$Q = \delta_{ts}\rho C_p \frac{\partial T}{\partial t} + \nabla.(-k\nabla T + \rho C_p u.\nabla T), (1)$$

where: Q is the heat source, δ_{ts} is time scaling coefficient, ρ is the density, C_p is the heat capacity T is the temperature, u is the velocity in the x_1 –direction, k is the thermal conductivity

The used materials are selected from FEMLAB material library. The monitor case is made from steel plate, the cooler of power semiconductor elements is from aluminium and printed circuits are from copper. The mentioned materials have got following material constants:

Material:	Density ρ	Heat capacity C_P	Thermal conductivity λ
Steel	7,8kg/dm³	475J/kgºC	44,5W/mºC
Aluminium	2,7kg/dm³	900J/kgºC	201 W/mºC
Copper	8,7kg/dm³	385J/kgºC	400 W/mºC

At the simulation the surroundings temperature Ta = 20 ºC is specified and a maximal temperature of the cooler is 100 ºC.

FEMLAB 3.1 has generated for 3D model a mesh of 31 194 elements. A predefined mesh size has generated an extra coarse mesh.

4. POSTPROCESSING RESULTS AND GRAPHICAL INTERPRETATION OF HEAT TRANSFER

At heat transfer simulation in the monitor in dependence on a time, a temperature vas computed at 10 seconds intervals in frame of 5 minutes in various parts of the monitor. The total computing time was 267 seconds. The simulating system allows determining exact temperatures in various places of simulated object at various time of simulation. It is possible to find out the time of temperature consolidation and the maximal temperature. These values help to detect heat-overburdened areas of simulated object. Consequently it is possible to change the design and favourably influence lifetime and reliability of the designed device. The temperature value is presented by colour scale or by graphs which relate a temperature to given simulation place and time. Fig.2 and Fig.3 present 2D results of heat simulation of the monitor including influence of the fluorescent tubes in LCD monitor. Fig.4, Fig.5, and Fig.6 present 3D results of heat transfer simulation of the monitor.
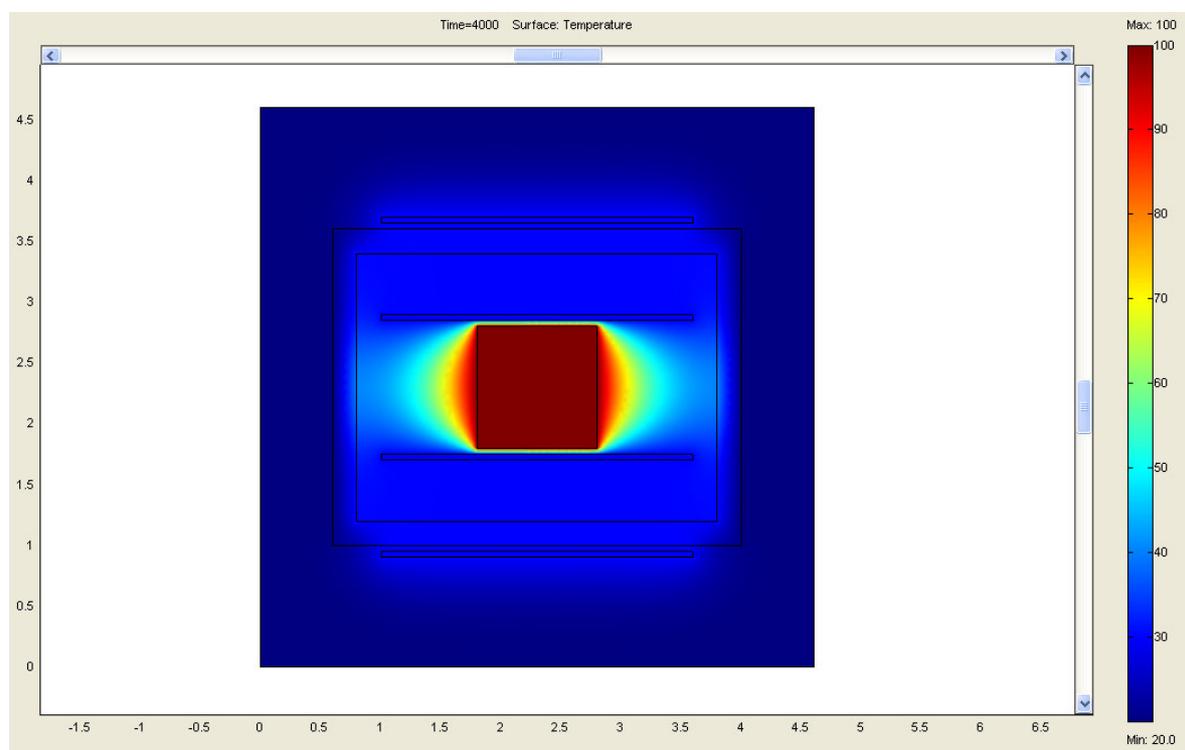

Fig.2 The colour shows interaction of heat from supply and fluorescent tubes in LCD
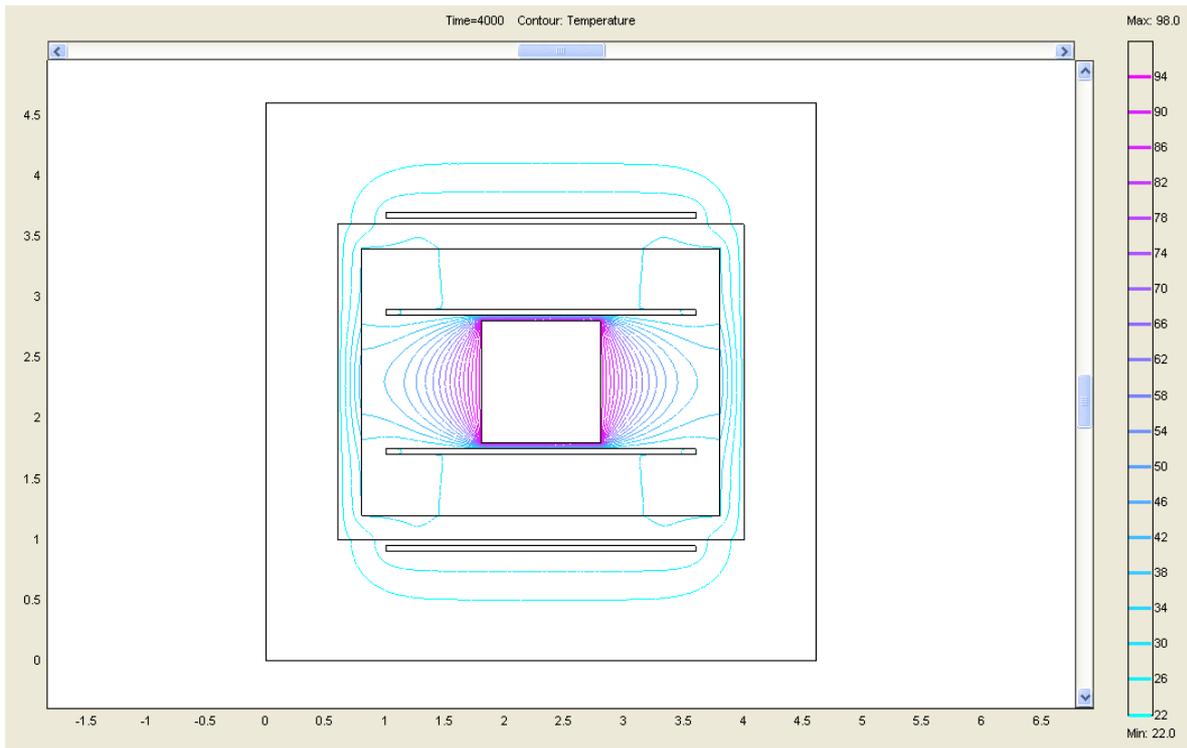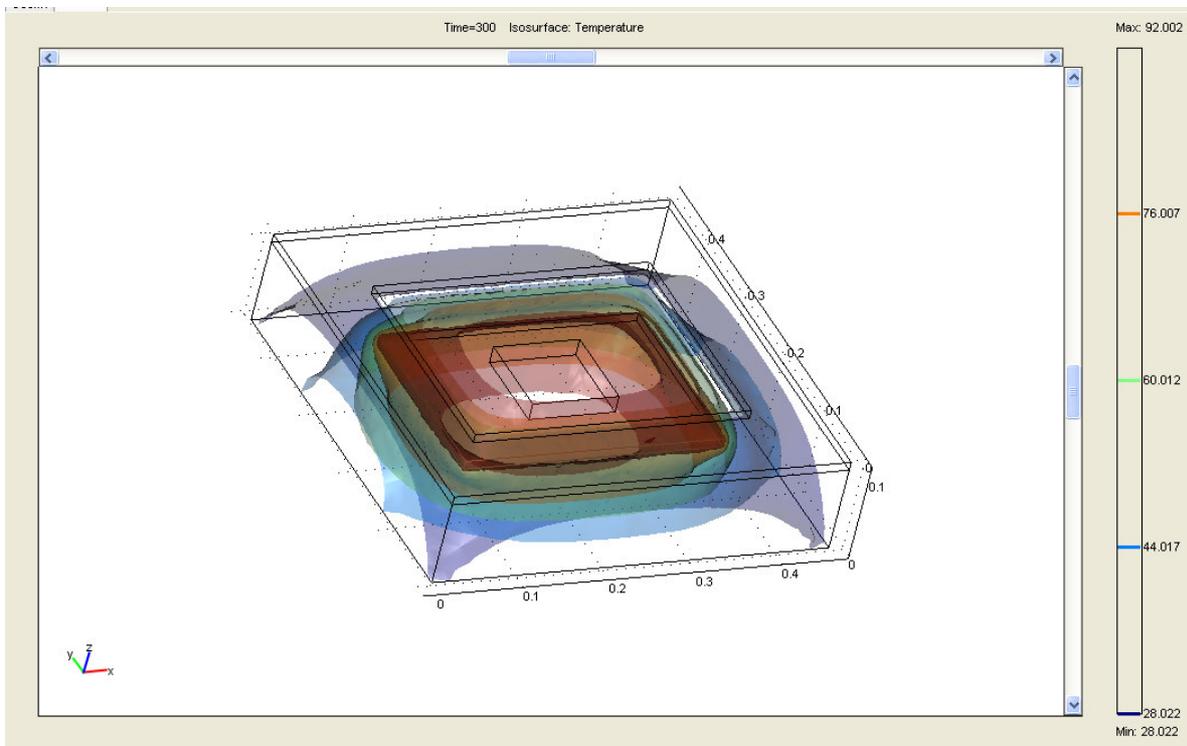
Fig.3 The contour plot of the solution in 2D

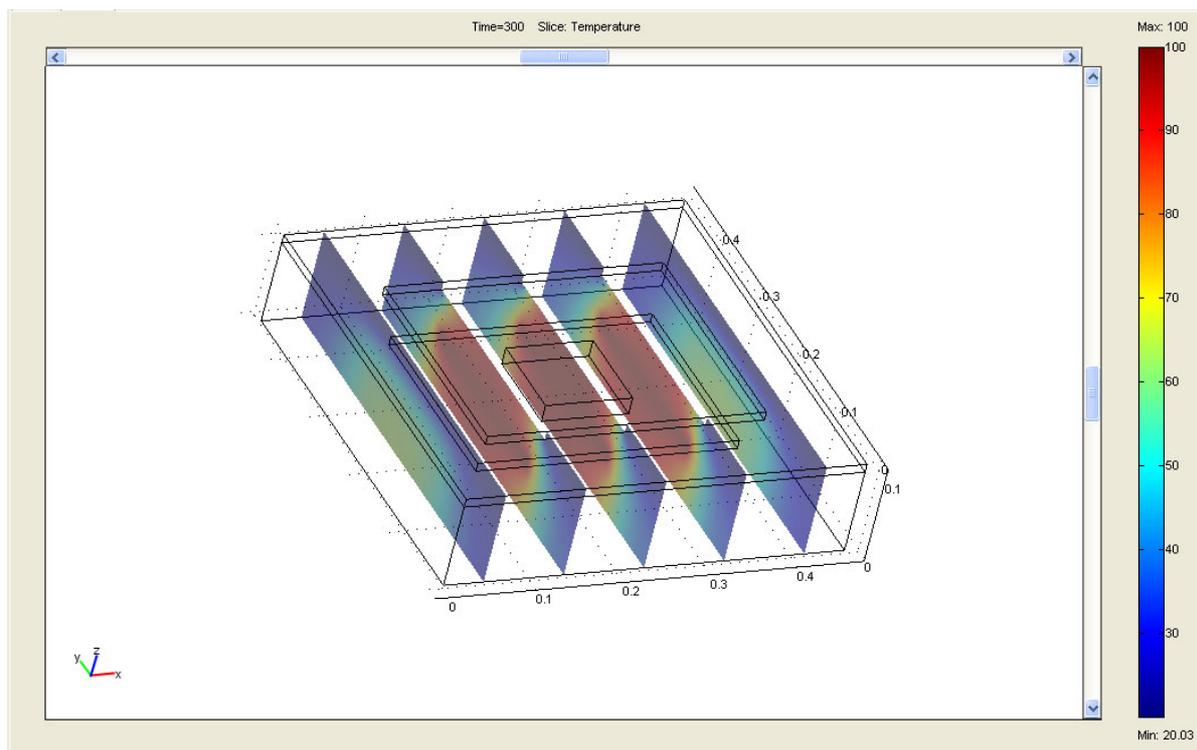*Fig.4 The isosurface plot of the solution in 3D*

Fig.5 A slice plot of the solution to a 3D Poisson's equation
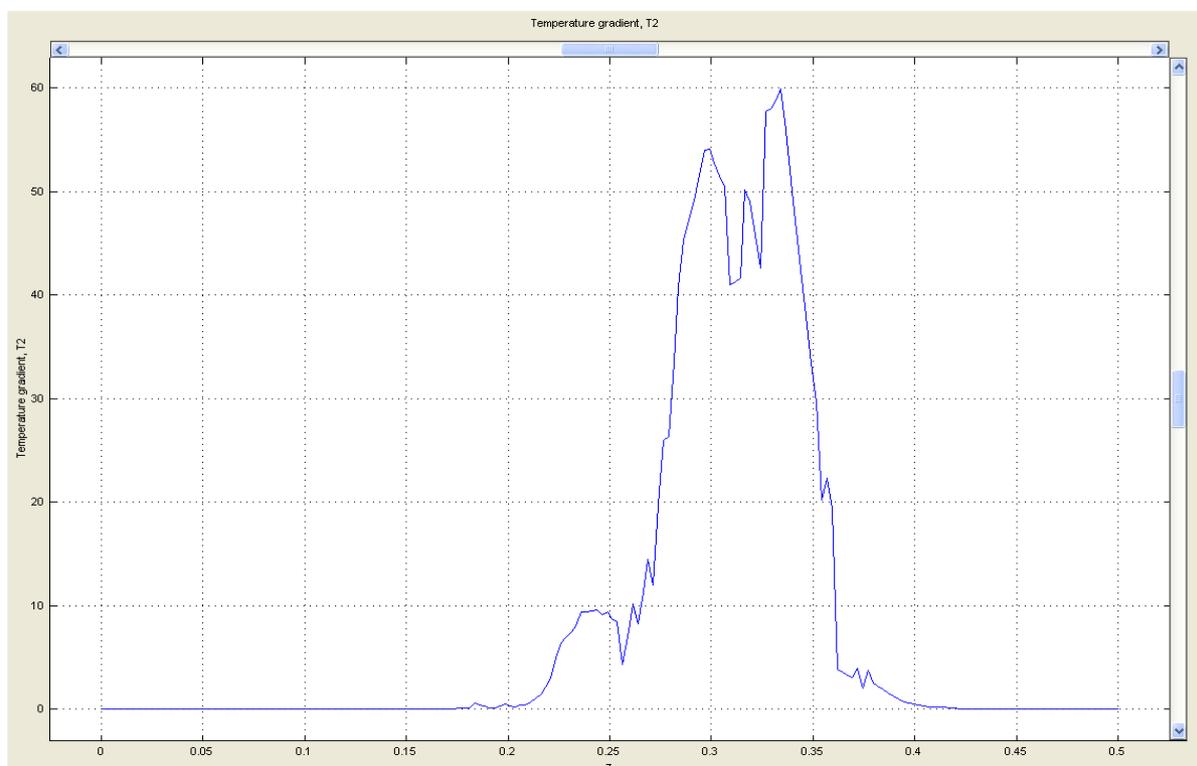

Fig.6 The temperature gradient in the chosen space slice and at chosen time

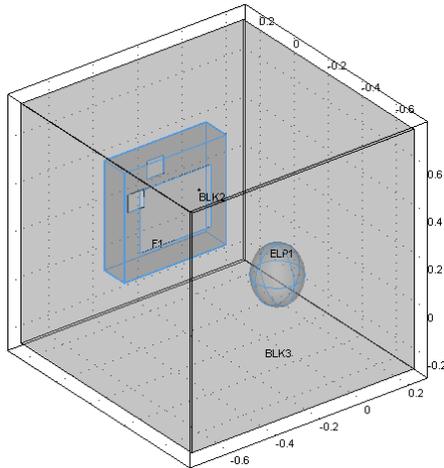## 5. RADIATION MODEL OF ELECTROMAGNETIC FIELD

Fig.7 The geometry model of simulation space of electromagnetic fields

The intention of simulation was to find out the radiation of electric field through the opening in a case of protected computer. 3D geometry model consists of steel display case (component F1) with opening for LCD display in front part and with two openings for supply and periphery connection. The element BLK2 represents a source of electromagnetic radiation. The component BLK3 represents a free space for spread of electric signal waves. The object ELP1 models an observer.

A radiated field from the source forms transverse waving (TE) which is described in 3D by equation [3]:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \varepsilon_{rc} \mathbf{E} = \mathbf{0} \; , \qquad (2)$$

The wave number k0 of free space is defined:

$$k_0 = \omega \sqrt{\varepsilon_0 \mu_0} = \frac{\omega}{c_0} \qquad (3)$$

where c0 is the velocity of a light in a vacuum and εrc is the complex materials permittivity:

$$\varepsilon_{rc} = \varepsilon - j \frac{\sigma}{\omega} \qquad (4)$$

The radiation source is represented by a magnetic dipole supplied by a current I0 = 1A. The frequency is given by the parameter a = (4, 5, 6, 7, 8, 9). The parametric solver computes with frequency value:

$$f = 10^a \qquad (5)$$

TE wave has got only one component of the electric field along a z-axis and a magnetic field is situated in the model plane. The harmonic electric and magnetic fields can be described in the shape:

$$\mathbf{E}(x, y, t) = E_z(x, y)\mathbf{e}_z e^{j\omega t} \; , \qquad (6)$$

$$\mathbf{H}(x, y, t) = (H_x(x, y)\mathbf{e}_x + H_y(x, y)\mathbf{e}_y)e^{j\omega t} \; , \qquad (7)$$

The geometry model has assigned boundary conditions in mentioned application mode. The geometry boundaries like contours, surfaces, planes or chosen points have got concrete physical values.

6. RESULTS AND GRAFICAL INTERPRETATION OF ELECTROMAGNETIC FIELD DISTRIBUTION

The mesh for finite element method is in the Fig.8. It contains 29 620 elements, 2776 elements on boundaries and 256 elements on the edges of geometrical model. The linear parametric solver vas
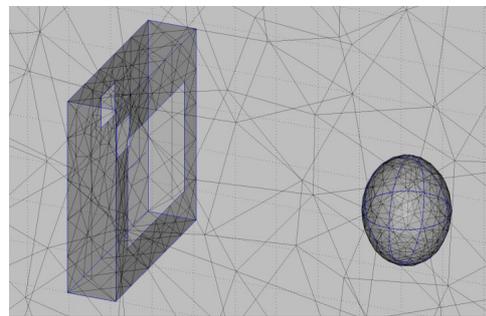

Fig.8 The mesh for PDE processing

used.

Fig.9 shows visualization of tangential component of electric field intensity on a surface of protected computer case (an ideal conductor) and on a surface of an observer head.

Electromagnetic waving is radiated from dipole supplied by a current 1A along a z-axis. It means in direction forward to observer's forehead.

The radiated electric power is simulated in various distances of observer along a z-axis by frequency 1MHz, 10MHz and 100MHz and result is in Fig.10.

7. CONCLUSION

The simulation works were divided into two independent parts. The first part has dealt with simulation of heat field extension from heat sources inside a monitor case. The critical domains of device were followed. The temperature dependence on time was interpreted and value of steady temperature too. The purpose of simulation is to ensure a reliability of developed device.

The second part has dealt with electromagnetic fields in electronic systems. The examples of simulation and visualization presented in this paper cover only small part of possibilities of multiphysical simulating program FEMLAB.
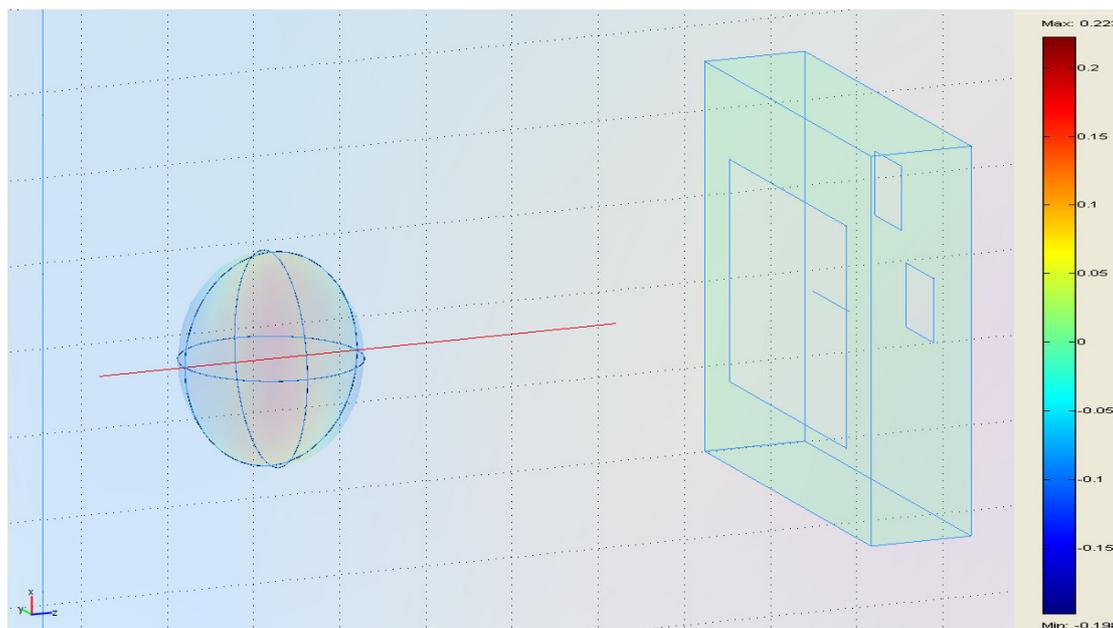
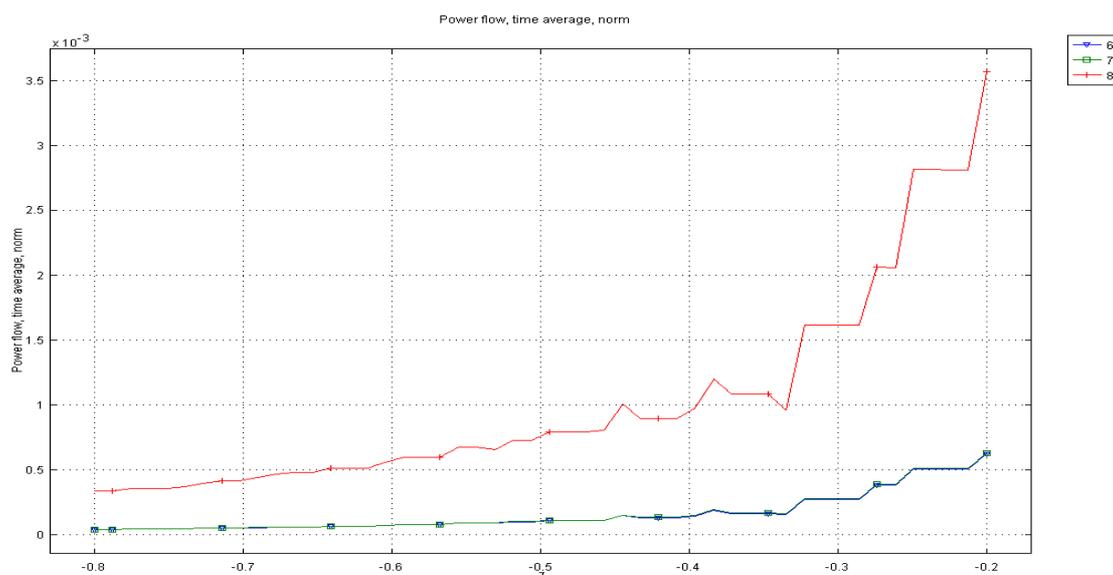*Fig.9 The tangential electric field on surface of a case and an observer*

Fig.10 The normal component of power flux from the case forward to observer's forehead along a z-axis. Graph of radiated signal frequency1 MHz (line 6), 10 MHz (line 7) and 100 MHz (line 8).

REFERENCES

[1] *http://www.comsol.com*

[2] FEMLAB 3 – User's Guide, *COMSOL AB, January 2004*

[3] ELECTROMAGNETICS MODULE FEMLAB 3 – User's Guide, *COMSOL AB, January 2004*