Acta
Technica
Jaurinensis

Vol. 7., No. 4., pp. 368-377, 2014
DOI: 10.14513/actatechjaur.v7.n4.339
Available online at acta.sze.hu

Finite Element Simulation by Free Software Tools

M. Kuczmann, T. Budai

Széchenyi István University, Department of Automation
Egyetem tér 1, H-9026 Győr, Hungary
E-mail: kuczmann@sze.hu

Abstract: This paper presents a finite element based environment for the simulation of two dimensional electromagnetic field problems, especially electric motors. The graphical user interface, the finite element mesh generator and the computational functions are free software tools that have been developed by scientific teams from all over the world. The aim of this work is to realize a new software environment that can be used in the simulation of two dimensional problems as well as in education.

Keywords: electromagnetic field, finite element method, parallel computation

1. Introduction

A new software tool has been developed for the simulation of two dimensional electromagnetic field problems; the focus of our research is especially on electric motors. The finite element method [1-5] (FEM) has been applied in the numerical field analysis. Simple one dimensional problems can also be solved by the developed tool.

The main aims of the developing team can be summarized as follows. The software tool must be based on free environments and functions realized by other scientific teams from all over the world, i.e. this FEM code is also available to download and to use. The environment of GMSH [6] is a three-dimensional finite element mesh generator with built-in preprocessing and postprocessing facilities, which is a user-friendly interface for FEM applications. GMSH contains a built-in CAD (Computer Aided Design) interface to build up the geometry of the application to be simulated. The postprocessing scheme can also be realized in the frame of GMSH. The geometry of the problem can be imported from many file formats, i.e. from many other CAD applications. The bridge between the geometry with physics and the postprocessing task (i.e. the numerical field analysis) can be realized by the functions of PETSc [7], which is a portable, extensible toolkit for scientific computation. It consists of many functions, e.g. to assemble and to solve large system of equations, and it is written in the C programming language. The FEM simulation is a time consuming and computer consuming task, realization of parallel algorithms is necessary to speed up the analysis. Our goal is to study the possibilities of parallel algorithms [8-10,29] and domain decomposition techniques [11-15]. The different models of the different materials are very important to include in a new environment, here the modelling of permanent magnets [16] and ferromagnetic hysteresis [17] are the most important goals.

2. The Finite Element Method, a short introduction

The main steps of FEM simulations are shown in fig. 1 [1]. The first step is the model specification, i.e. to build up the models of the real life problems which simulation require electromagnetic field calculations (the partial differential equations have to be found, which must be solved with prescribed boundary and continuity conditions; it has to be found out, whether it is a linear or a nonlinear problem and how the characteristics look like). After selecting potentials, the weak formulation of these partial differential equations must be worked out. As it is presented in the introduction, the geometry of the problem must be defined by a CAD software tool. The chosen free environment is GMSH [6].

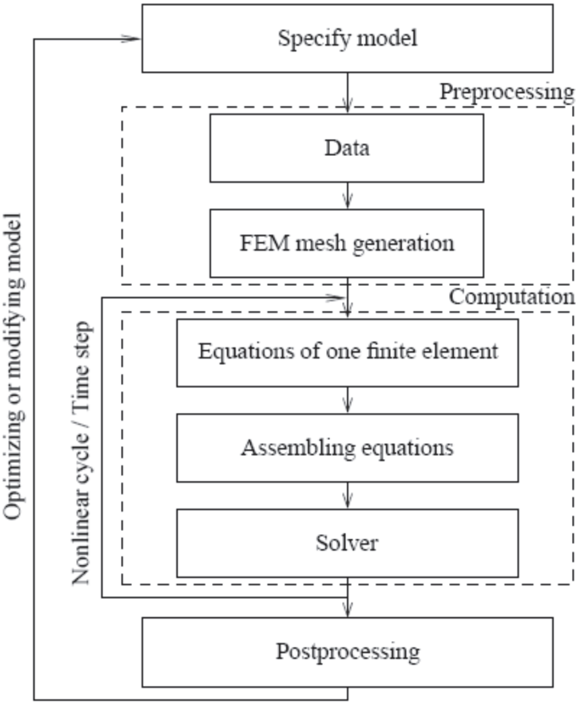

Figure 1. The main steps of the finite element simulations

The next step is the preprocessing task. Here the values of different parameters are given, e.g. the material properties, the excitation signal and so on. The geometry can be simplified according to symmetries. The geometry of the problem must be discretized by a FEM mesh [1-5]. The fundamental idea of FEM is to divide the problem region to be analyzed into smaller finite elements with given shape, as triangles or quadrangles in 2D, or tetrahedra, hexahedra or prisms in 3D.

The next step in FEM simulations is solving the problem by the help of PETSc functions [7]. The FEM equations, based on the Galerkin weak formulations, must be set up in the level of one finite element, and then these equations must be assembled through the FEM mesh [1-5]. Then this global system of equations must be solved [7]. The computation may contain iteration if the constitutive equations are nonlinear. This is the situation when simulating ferromagnetic materials with nonlinear characteristics. Iteration means that the system of equations must be set up and must be solved step by step until convergence is reached. If the problem is time dependent, then the solution must be worked out at every discrete time instant.

The result of computations is the approximated potential value above the mesh. Any electromagnetic field quantity can be calculated by using the potentials at the postprocessing stage. Inductance, energy, force, torque and any other quantities can also be calculated [1-5]. The postprocessing step gives a chance to modify the geometry, the material parameters or the FEM mesh to get more accurate result. The above listed quantities are calculated by functions developed in the frame of PETSc.

3. The developed environment

A test environment has been built at the Széchenyi István University to realize the above mentioned technique, it is shown in fig. 2, and it is presented in [18] in more detail. Here a brief presentation is given.

The test-bed system consists of four IBM HS21-8853 blades (nodes) housed in a BladeCenter E Chassis [19]. The free and open-source Linux distribution, Debian Linux 7.4 has been used as operating system on all nodes. To handle the communication between the nodes an open-source implementation of the MPI [20] standard, OpenMPI 1.6.5 [20] has been installed.

The above mentioned PETSc has been used to numerically solve equation-systems according to FEM.

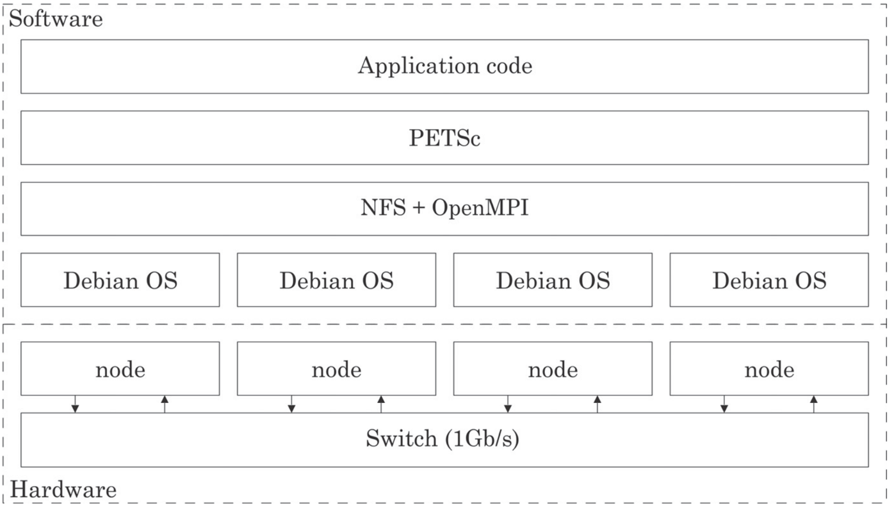

Figure 2. The completed cluster

Compilation and execution of parallel C programs using PETSc requires the software packages mentioned above.

In the time of writing the Debian repository [22] contains outdated versions of OpenMPI and PETSc. Another way of installation is to compile these packages from source on the target machine. This consists of installing compiler, setting configuration parameters and starting the compilation process. Since this task is fairly complex and has to be repeated on each node of the cluster and on machines used for development, automation via shell-scripting is strongly advised. To ease the installation on the cluster and to help other members in the research team who wants to develop such software in the future, an automated installer has been developed [18].

The first node has been chosen as master; the other nodes are slaves and have been controlled by the master. All application sources were compiled on and all measurements were controlled by the master machine.

4. Illustrative examples

4.1. Laminations with ferromagnetic hysteresis

A simple one dimensional problem has been solved by the above mentioned software tool. The schematic view of the problem can be seen in fig. 3, which is a lamination placed into a magnetic field defined by the time varying $\boldsymbol{B}_0$ of one single frequency [23,24]. The thickness d of the lamination is much smaller than the other two dimensions, i.e. a one dimensional model can be set up. The orthogonal components of the electric and the magnetic field are depending only on x as it is shown in the figure.

The following Maxwell's equations must be solved to simulate the lamination core taking eddy currents into account [23,24]: $\nabla \times \boldsymbol{H} = \sigma \boldsymbol{E}$, $\nabla \times \boldsymbol{E} = -\dot{\boldsymbol{B}}$, where $\boldsymbol{H}$, $\boldsymbol{B}$, $\boldsymbol{E}$ and σ are the magnetic field intensity, the magnetic flux density, the electric field intensity, and the conductivity, respectively.

The nonlinear constitutive relationship with static hysteresis is decomposed into a linear term, and a nonlinear residual term, as follows: $\boldsymbol{B} = \mu \boldsymbol{H}_{\mathrm{st}} + \boldsymbol{R}$. Here, μ is the optimal value of permeability selected as [25,26] $\mu = \frac{2}{\nu_{\max} + \nu_{\min}}$, where $\nu_{\max}$ and $\nu_{\min}$ are the maximum and the minimum slope of the inverse static hysteresis characteristics [27], i.e. the maximum and the minimum reluctivity. The index st in the polarization formulation is for the word static, because the static hysteresis model represents the relationship between $\boldsymbol{B}$ and $\boldsymbol{H}_{\mathrm{st}}$, i.e. $\boldsymbol{H}_{\mathrm{st}} = \mathcal{H}\{\boldsymbol{B}\}$. Excess loss term is ignored by this representation, only the nonlinearity, i.e. hysteresis losses, and eddy current losses are present, $\boldsymbol{H}_{\mathrm{st}} = \boldsymbol{H}_{\mathrm{h}} + \boldsymbol{H}_{\mathrm{eddy}}$. These terms are calculated by the FEM procedure. Decreasing the frequency of excitation, the term $\boldsymbol{H}_{\mathrm{eddy}}$ is decreasing automatically, $\boldsymbol{H}_{\mathrm{h}}$ is the frequency independent term of the magnetic field intensity [24].

The above equations are nonexpansive, and the fixed point iteration scheme through the polarization formulation results in a contraction mapping, meaning convergent iterative process [25,26].

However, excess losses are not present. Excess losses can be represented by the following scheme.

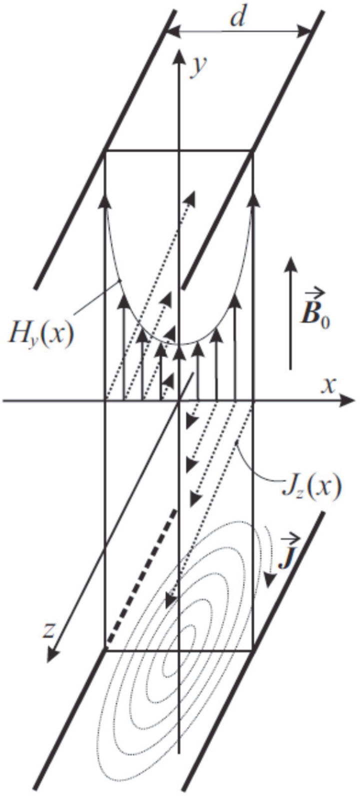

Figure 3. Electromagnetic field inside laminations excited by external field

The magnetic field intensity is decomposed into three parts in the more complex model, $\boldsymbol{H} = \boldsymbol{H}_{\mathrm{h}} + \boldsymbol{H}_{\mathrm{eddy}} + \boldsymbol{H}_{\mathrm{exc}}$. The last term is responsible for the anomalous or excess losses [23,24,28]. The advantages and convergence properties of the fixed point technique can be hold by introducing the excess field term $\boldsymbol{H}_{\mathrm{exc}}$ in the Maxwell's equations as follows [24]: $\nabla \times \boldsymbol{H} = \sigma \boldsymbol{E}$, $\nabla \times \boldsymbol{E} = -\dot{\boldsymbol{B}}$, $\boldsymbol{B} = \mu(\boldsymbol{H} - \boldsymbol{H}_{\mathrm{exc}}) + \boldsymbol{R}$, i.e. the decomposition of the nonlinear constitutive relationship must be rewritten. The following nonlinear partial differential equation can be obtained:

$$\nabla \times \nabla \times \boldsymbol{H} + \mu\sigma\dot{\boldsymbol{H}} = \mu\sigma\dot{\boldsymbol{H}}_{\mathrm{exc}} - \sigma\dot{\boldsymbol{R}}. \tag{1}$$

This problem can be solved numerically by the finite element method. It is noted that the excess field term is present on the right hand side of (1), and the solution of the problem is the total magnetic field intensity, containing the excess field term, too.

The three terms can be seen separately in fig. 4 supplying sinusoidal external magnetic field with three different amplitude of the magnetic flux. A comparison between measured and simulated higher order dynamic minor loops can be seen in fig. 5.

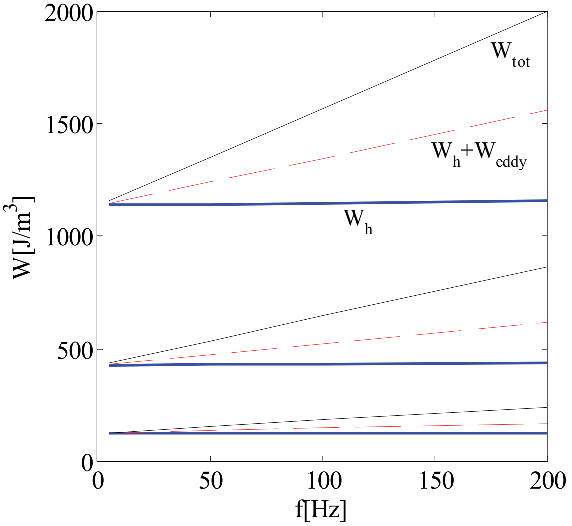

Figure 4. Losses inside the lamination

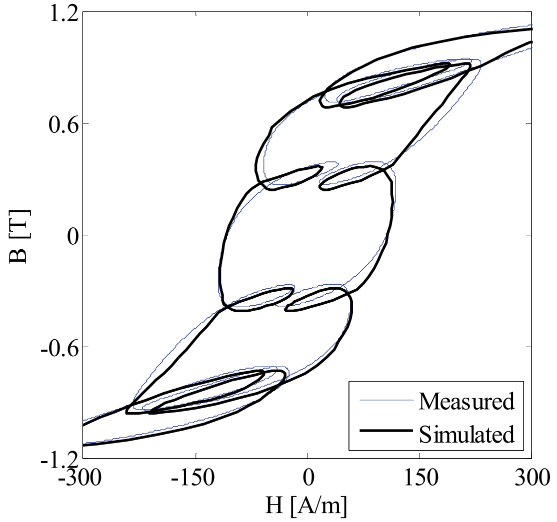

Figure 5. Comparison between measured and simulated higher order dynamic minor loops

4.2. Three phase transformer core

The model of a three phase test transformer has been built to simulate the complex behaviour of the transformer core. The geometry of the transformer can be seen in fig. 6, and the simulated locus of the magnetic field intensity vector around the two signed areas has been plotted in fig. 7.

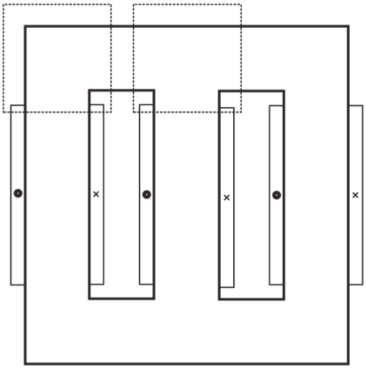

Figure 6. Three phase transformer core

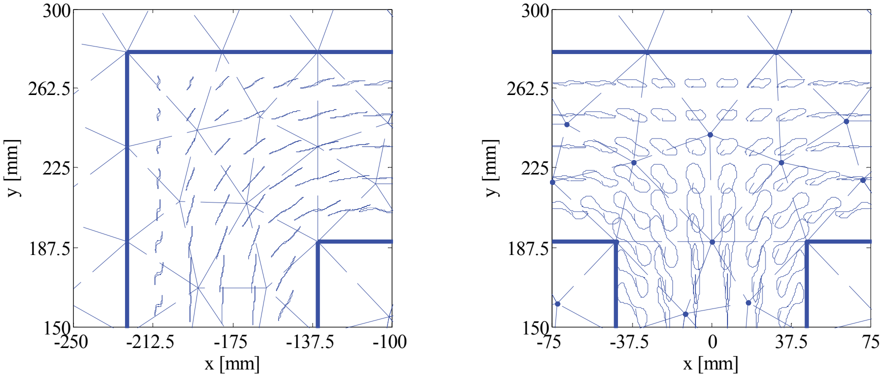

Figure 7. Locus of magnetic field simulated by the vector Preisach model of hysteresis

Here, the so-called vector Preisach model of hysteresis [17,24,27] has been implemented in the FEM code combining it with the fixed-point scheme [25,26].

4.3. Capacitor

The electric field of a capacitor has been simulated by the above mentioned environment [18]. The linear problem, as a simple and first case study, has been solved as a simple one dimensional, a more difficult two dimensional, and finally, as a three dimensional arrangement to increase the number of unknowns to test the speedup of the tool.

Looking at the achieved speedup for each solver, it is clear that problems which require complex computation steps in the assembly phase can benefit the most from parallel processing.

The deviation of the individual results shows that system-level functions, like caching has no significant impact on the results. Fig. 8. shows the execution times of the sequential and best parallel solvers compared [18].

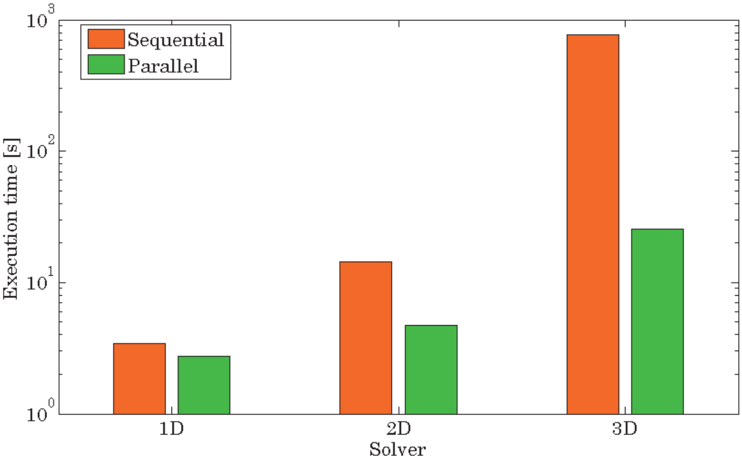

Figure 8. Execution times of the different solvers

5. Conclusion

The developed code has just started to simulate permanent magnet synchronous machines applied in hybrid vehicles.

The next step of the work is to implement the model of permanent magnets in the code, and to give a graphical user interface to handle hysteresis. Modelling of motor geometry is also under construction, and the insertion of the code into control algorithms and identification tasks is also an open question.

Acknowledgement

This paper is sponsored by ”TÁMOP-4.2.2.A-11/1/KONV-2012-0012: Basic research for the development of hybrid and electric vehicles -The Project is supported by the Hungarian Government and co-financed by the European Social Fund”.

References

[1] Kuczmann, M., Iványi, A.: *The Finite Element Method in Magnetics*, Akadémiai Kiadó, Budapest, 2008
DOI: 10.13140/2.1.3104.1927

[2] Bíró, O.: *CAD in Electromagnetism*, Advances in Electronics and Electron Physics, Vol. 82, pp. 1–96, 1991
DOI: 10.1016/S0065-2539(08)60911-7

[3] Bastos, J.P.A., Sadowski, N.: *Electromagnetic Modeling by Finite Element Methods*, Marcel Dekker Inc., New York, 2003

[4] Jin, J.: *The Finite Element Method in Electromagnetics*, John Wiley and Sons, New York, 2002

[5] Meunier, G.: *The Finite Element Method for Electromagnetic Modeling*, John Wiley and Sons, New York, 2008

[6] Geuzaine, C., Remacle, J.F.: *Gmsh: a Three Dimensional Finite Element Mesh Generator with Built-in Pre and Postprocessing Facilities*, International Journal for Numerical Methods in Engineering, Vol. 79, No. 11, pp. 1309–1331, 2009 DOI: 10.1002/nme.2579

[7] PETSc: http://www.mcs.anl.gov/petsc/ (last visited: 2014. 09. 14.)

[8] METIS, http://glaros.dtc.umn.edu/gkhome/views/metis (last visited: 2014. 09. 14)

[9] Molnárka, G., Varjasi, N.: *A Simultaneous Solution for General Linear Equations on a Ring or Hierarchical Cluster*, Acta Technica Jaurinensis, Vol. 3, No. 1, pp. 65-73, 2010

[10] Toselli, A., Vasseur, X.: *Robust and Efficient FETI Domain Decomposition Algorithms for Edge Element Approximations*, COMPEL: The International Journal for Computation and Mathematics in Electrical and Electronic Engineering, Vol. 24, No. 2, pp. 396-407, 2005
DOI: 10.1108/03321640510586033

[11] Kruis, J.: *Domain Decomposition Methods for Distributed Computing*, Saxe-Coburg Publications, Kippen, Stirling, 2006

[12] Nikishkov, G.P.: *Basics of the Domain Decomposition Method for Finite Element Analysis*, in Mesh Partitioning Techniques and Domain Decomposition Methods, Editor: Magoulés, F., Saxe-Coburg Publications, Kippen, Stirling, pp. 119-142, 2007

[13] Farhat, C., Pierson, K., Lesoinne, M.: *The Second Generation FETI Methods and Their Application to the Parallel Solution of Large Scale Linear and Geometrically Nonlinear Structural Analysis Problems*, Computer Methods in Applied Mechanics and Engineering, Vol. 184, No. 2-4, pp. 333–374, 2000
DOI: 10.1016/S0045-7825(99)00234-0

[14] Marcsa, D.: *Domain Decomposition Algorithms for Edge Element Based Parabolic Type Problems*, Acta Technica Jaurinensis, Vol. 7, No. 2, pp. 193-206, 2014
DOI: 10.14513/actatechjaur.v7.n2.278

[15] Farhat, C., Roux, F.X.: *Method of Finite Element Tearing and Interconnecting and its Parallel Solution Algorithm*, International Journal for Numerical Methods in Engineering, Vol. 32, No. 6, pp. 1205-1227, 1991
DOI: 10.1002/nme.1620320604

[16] Kovács, G.: *Up-to-Date Finite Element Based Simulation for Permanent Magnet*, Acta Technica Jaurinensis, Vol. 7, No. 2, pp. 172-182, 2014
DOI: 10.14513/actatechjaur.v7.n2.281

[17] Bertotti, G., Mayergoyz, I.D.: *The Science of Hysteresis*, Academic Press, New York, 2006

[18] Budai, T., Kuczmann, M.: *Parallel Implementation of Finite Element Solvers by the help of MPI*, MSc Conference at the Budapest Univeristy of Technology and Economics, Budapest, pp. 39-42, 2014

[19] IBM BladeCenter E Chassis, www.ibm.com/systems/bladecenter/hardware/chassis/bladee/ (last visited: 2014. 09.14.)

[20] Message Passing Interface Forum, http://www.mpi-forum.org/ (last visited: 2014. 09. 14.)

[21] Open MPI, http://www.open-mpi.org/ (last visited: 2014. 09. 14.)

[22] Debian Packages Repository, https://packages.debian.org/stable/ (last visited: 2014. 09. 14.)

[23] Dlala, E., Belahcen, A., Fonteyn, K.A., Belkasim, M.: *Improving Loss Properties of the Mayergoyz Vector Hysteresis Model*, IEEE Transactions on Magnetism, Vol. 46, No. 3, pp. 918-924, 2010
DOI: 10.1109/TMAG.2009.2034846

[24] Kuczmann, M., Kovács, G.: *Improvement and Application of the Viscous-Type Frequency-Dependent Preisach Model*, IEEE Transactions on Magnetism, Vol. 50, No. 2, Paper No. 7009404, 2014
DOI: 10.1109/TMAG.2013.2283398

[25] Bottauscio, O., Canova, A., Chiampi, M., Repetto, M.: *Iron Losses in Electrical Machines: Influence of Different Material Models*, IEEE Transactions on Magnetism, Vol. 38, No. 2, pp. 805-808, 2002
DOI: 10.1109/20.996208

[26] Hantila, F.I., Preda, G., Vasiliu, M.: *Polarization Method for Static Fields*, IEEE Transactions on Magnetism, Vol. 36, No. 4, pp. 672-675, 2000
DOI: 10.1109/20.877538

[27] Iványi, A.: *Hysteresis Models in Electromagnetic Computation*, Akadémiai Kiadó, Budapest, 1997

[28] Bertotti, G.: *General Properties of Power Losses in Soft Ferromagnetic Materials*, IEEE Transactions on Magnetism, Vol. 24, No. 1, pp. 621-630, 1988
DOI: 10.1109/20.43994

[29] Iványi, P., Radó J.: *Preprocessing to Parallel Computation*, Budapest: Typotex Kiadó, Budapest, 2014
ISBN: 978-963-279-381-8