

## Low-frequency Magnetic Field Modeling

P. Tarvydas, E. Pačėsaitė, A. Noreika

Department of Electronics Engineering, Kaunas University of Technology,

Studentų str. 50, LT-51368 Kaunas, Lithuania; phone: +370 37 300524; e-mail: ali.us.noreika@stud.ktu.lt

### Introduction

Finite Element Analysis (FEA) provides a reliable numerical technique for analyzing engineering design [1,2]. The process starts with the creation of a geometrical model. Then, the model is subdivided (meshed) into small pieces (elements) of simple shapes connected at specific points (nodes). Thus, a network of discrete interconnected elements (mesh) represents the continuous model. The process assumes that the behavior of each element varies in a particular known fashion for various conditions. The finite element method (FEM) predicts the behavior of the model by manipulating the information obtained from all the elements making up the model.

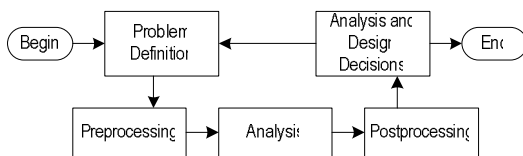


Fig. 1. The basic steps in a finite element analysis

The success of a finite element method for modeling and analyzing a design is based largely on the procedures used. Regardless of the type of application, numerical simulation by the FEM requires complete information about the domain under consideration. Whatever the case might be, the problem's model must contain all the necessary data for each of the different steps in the numerical computation (geometry, elements, loads, boundary conditions, solution of the system of equations, visualization of results, etc.). Thus, the basic steps in a finite element analysis, as schematically shown in Fig. 1. [3,4].

### Modeling in COSMOSM environment

COSMOSM (Structural Research and Analysis Corporation) is a complete, modular, self-contained finite element system [5]. The program includes modules to solve linear and nonlinear static and dynamic structural problems, in addition to problems of heat transfer, fluid mechanics, electromagnetics and optimization. ESTAR module is intended for modeling of low frequency electromagnetic fields [6].

The COSMOSM system consists of a preprocessor and postprocessor, various analysis modules, interfaces,

translators and utilities. The program is completely modular allowing acquiring and loading only the modules that are needed. GEOSTAR is the basic preprocessor and postprocessor of the COSMOSM finite element system [5,6]. It is an interactive full three-dimensional CAD-like graphic geometric modeler, mesh generator and FEA preprocessor and postprocessor. It is possible to create the geometric model, mesh it, provide all analysis related information, perform the desired type of analysis, review, plot and print the results, without having to leave the GEOSTAR screen.

The ultimate goal of the preprocessing stage is a finite element model that is fully described for the desired analysis module in terms of nodes and elements. Loading, boundary conditions, analysis flags, and options, are all specified for an analysis module in terms of nodes and elements. It is always possible to define the FEA model without any geometry, but this task becomes increasingly difficult as the complexity of the model increases. It is possible to import the geometry from most of CAD systems, or use GEOSTAR to create it.

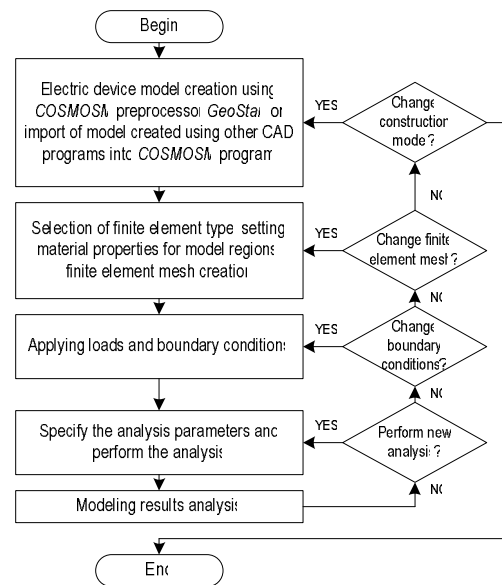


Fig. 2. Modeling algorithm

Users have a wide range of meshing options in COSMOSM. Selecting the most appropriate method for the application at hand can save time without sacrificing

accuracy. Finite element mesh can be formed using following techniques: direct mesh generation, parametric meshing, automatic meshing, carrying the mesh with geometry and creating 2D and 3D meshes from 1D and 2D meshes respectively, local mesh control, adaptive meshing, mesh refinement, bonding incompatible meshes, automatic generation of contact lines and surfaces, mesh information.

Loads in *GEOSTAR* can be applied to nodes or elements directly or through the association with geometric entities, which accelerates significantly the preprocessing stage. Loads and boundary conditions must be applied after meshing. If a the mesh is deleted, the specified loads and boundary conditions are deleted as well. It is also possible to couple degrees of freedom, define point-to-point, point-to-curve, and point-to surface constraints as well as introduce constraint equations for degrees of freedom. The bonding feature permits connecting of non-compatible separately meshed components of the same model. Model's loads and boundary conditions can be associated with time/temperature curves.

The finite element method leads to a system of equations that must be solved simultaneously. A complex model can generate a very large system of equations. Each equation represents an unknown quantity that we seek to solve for. Each unknown quantity is also referred to as a degree of freedom. Traditionally, solving a large system of simultaneous equations requires a long time and large computer resources. Solvers can be broadly classified into two main categories: direct and iterative. Direct solvers are exact but are generally slower than iterative solvers and require more memory. Their efficiency increases in comparison with iterative solvers when stress analysis is solved with more load cases. Iterative solvers are based on trial and error. They produce a solution when errors get smaller than a specified tolerance. Following solvers are used: direct skyline, direct sparse, iterative.

The results obtained by the numerical solution are used to test whether the finite element mesh is acceptable or changes are necessary. Postprocessing refers to the graphical manipulation and display of results after successfully completing the analysis. This step of the finite element application makes the analysis, interpretation and evaluation of the analysis results extremely simple. It should be noted, however, that the postprocessing phase is only a tool to review the analysis output; it is engineering sense and experience that will help interpret the results and evaluate how appropriate the mesh is. The adaptive techniques of *COSMOSM* provide an automatic procedure for such evaluation during the engineering decision-making process.

### Numerical Computation of Magnetic Field

The theory used in *ESTAR* program is based on the application of the potential function theory to Maxwell's equations [6]. Magnetic vector and scalar potentials are used for two and three dimensional analysis, respectively.

For the two dimensional models in the  $x$ - $y$  plane, the only non zero components of  $A$  and  $\nabla\phi$  are the  $z$  components which are functions of  $x$  and  $y$  only and do not vary in the  $z$  direction. Therefore the above equation takes the following scalar form:

$$\begin{aligned} \partial/\partial x(v\partial A/\partial x) + \partial/\partial y(v\partial A/\partial y) - \sigma\partial A/\partial t = \\ = -J_s - (\partial/\partial x)H_{cy} + (\partial/\partial y)H_{cx}, \end{aligned} \quad (1)$$

where  $A$  – magnetic vector potential;  $J_s$  – source current;  $H_c$  – coercitivity of permanent magnets;  $v$  – reluctivity;  $\sigma$  – electric charge density.

For the axisymmetric case, taking  $z$  in the azimuthal direction the only non zero component of  $A$  is the azimuthal component. For this case the following equation is used:

$$\begin{aligned} \partial/\partial x(v\partial A/\partial x + vA/x) + \partial/\partial y(v\partial A/\partial y) - \sigma\partial A/\partial t = \\ = -J_s - (\partial/\partial x)H_{cy} + (\partial/\partial y)H_{cx}. \end{aligned} \quad (2)$$

For the axisymmetric case,  $x$  and  $y$  correspond to radial and axial components of cylindrical coordinates. Two dimensional and axisymmetric analysis in *ESTAR* are based on equations (1) and (2), respectively.

When current sources are not present, the three dimensional magnetostatic analysis in *ESTAR* is based on the following equation:

$$\nabla \cdot \mu \nabla \psi - \nabla \cdot \mu H_c = 0. \quad (3)$$

Three dimensional magnetostatic analysis with current sources is also available in *ESTAR*. A reduced potential method is used where the total magnetic field intensity  $H$  is divided into two parts:

$$\vec{H} = \vec{H}_s + \vec{H}_m, \quad (4)$$

where  $\vec{H}_m$  – is the induced magnetization field,  $\vec{H}_s$  – is the current source field in the free space and it can be calculated from Biot-Savart equation.

When current sources are present, three dimensional magnetostatic analysis in *ESTAR* is based on the equation:

$$\nabla \cdot \mu \nabla A - \nabla \cdot \mu \vec{H}_s - \nabla \cdot \mu \vec{H}_c = 0, \quad (5)$$

where  $A$  – magnetic scalar potential,  $\vec{H}_c$  – is the coercivity of permanent magnet,  $\mu$  – the magnetic permeability.

The finite element method used in *ESTAR* is based on stationarity of suitable energy functionals for equations (1), (2), (4) and (5). For nonlinear problems, a Newton-Raphson method is used which can be expressed in the following matrix form:

$$\left( {}^{t+\Delta t}[K]_L + {}^{t+\Delta t}[K]_{NL} \right) \Delta \psi^i = {}^{t+\Delta t}[R] - {}^{t+\Delta t}[K]_L \psi^{i-1}, \quad (6)$$

where  ${}^{t+\Delta t}[K]_L$  – linear stiffness matrix at time  $t + \Delta t$ ,  ${}^{t+\Delta t}[K]_{NL}$  – nonlinear stiffness matrix at time  $t + \Delta t$ ,  ${}^{t+\Delta t}[R]$  – loading vector at time  $t + \Delta t$ ,  $\Delta \psi^i$  – magnetic potential increment vector at iteration ( $i$ ),  $\psi^{i-1}$  – magnetic potential vector at iteration ( $i-1$ ).

### Finite Elements used in FEA

Elements are the fundamental building blocks of finite element analysis. The elements' shape approximates the geometry of the structure, while their mathematical model simulates the physical behavior. In *COSMOSM*, both H-version and P-version of the finite element method are available in addition to the HP-approach. The following elements are available for linear and nonlinear electromagnetic analysis [5,6]: *MAG2D* – 2D 4-Node Magnetic Element, *MAG3D* – 3D 8-Node Magnetic Element, *TETRA4* or *TETRA10* – 3D 4-Node or 10-Node Tetrahedron Element.

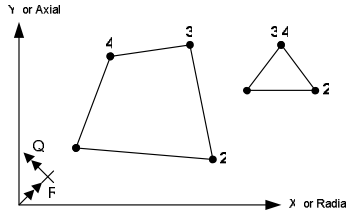


Fig. 3. Magnetic 2D 4-Node Element *MAG2D*

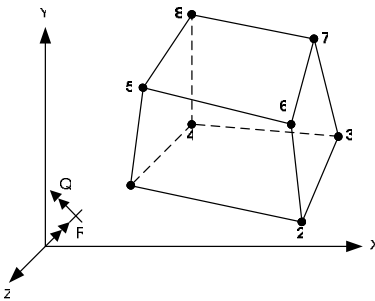


Fig. 4. Magnetic 3D 8-Node Element *MAG3D*

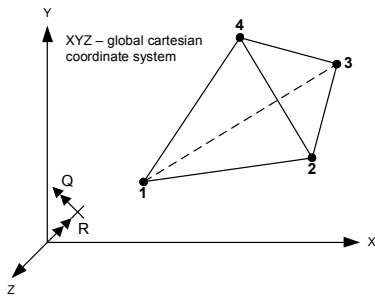


Fig. 5. Magnetic 3D 4-Node Tetrahedron Solid Element *TETRA4*

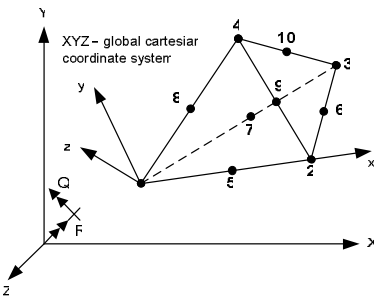


Fig. 6. Magnetic 3D 10-Node Tetrahedron Solid Element *TETRA10*

Magnetic 2D 4-Node Element *MAG2D* is a 4-node quadrilateral element. It can be used to model both two dimensional and axisymmetric electromagnetic analysis. All the elements must be defined in the *X-Y* plane as shown in Figure 2. Axisymmetric models have to be defined in the positive *X* half-plane in which *X* represents the radial direction and *Y* refers to the axis of symmetry. Each node has one degree of freedom representing the magnetic potential. The element node numbering is shown in Figure 2. For this element, a 2x2 Gauss point integration scheme is used. In case of a collapsed-to-triangle element, a one-point integration is performed.

Magnetic 3D 8-Node Element *MAG3D* is an 8-node element for three-dimensional electromagnetic analysis. Magnetic potential is the only degree of freedom considered at each node. The element node numbering is shown in Figure 3. For this element a 2x2x2 Gauss point integration scheme is used.

Magnetic 3D 4-Node or 10-Node tetrahedron solid element *TETRA4* or *TETRA10* is a 4-node or 10-node three dimensional solid element of the analysis of electromagnetic problems. Magnetic potential is the only degree of freedom considered at each node. The element node numbering is shown in Figure 4 for the 4-node element and Figure 5 for the 10-node element. Both clockwise and counter-clockwise numbering are allowed.

### Computation of Magnetic Field in a Solenoid Actuator

A problem of nonlinear magnetostatic axisymmetric analysis is solved using *MAG2D* finite elements. The task is to obtain the magnetic field for a solenoid actuator consisting of a coil enclosed in a ferromagnetic core with a plunger [7] as shown in Fig. 7. For axisymmetric models, a fine mesh should be used in the radial direction near the center axis to avoid inaccuracy caused by  $1/x$  term in the formulation. In the following case relative permeability of air and coil  $\mu = 1$ ; current density in coil  $J = 1 \cdot 10^6 \text{ A/m}^2$ .

Table 1. B-H curve data for core and plunger

B (T)	0.80	0.95	1.00	1.10	1.15	1.25	1.40	1.55	1.65
H (A/m)	460	640	720	890	1020	1280	1900	3400	6000

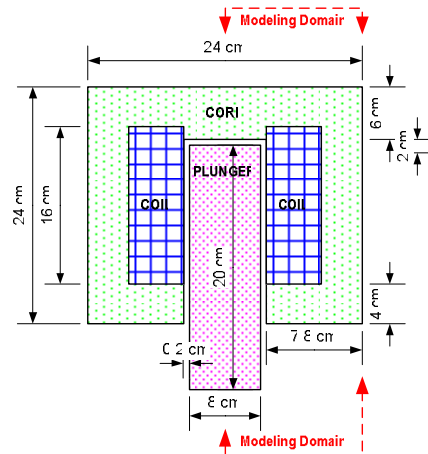
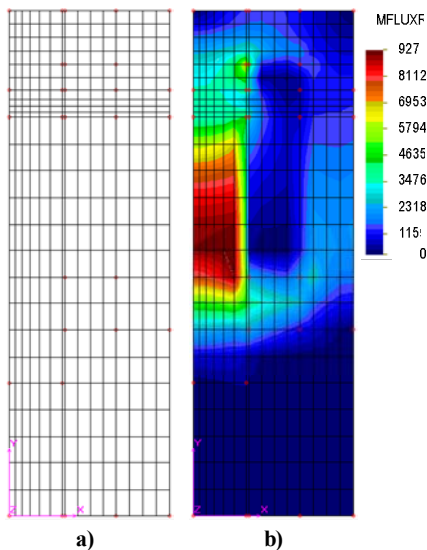


Fig. 7. Solenoid actuator



**Fig. 8.** Finite element mesh (a) and magnetic field strength distribution (b)

Created finite element mesh is presented in Fig. 8a. Finite element mesh consists of 375 finite elements. Calculated magnetic field distribution is presented in Fig. 8b. It is possible to save values of magnetic field strength components to text files using postprocessor *GEOSTAR*.

### Conclusions

Preprocessor and postprocessor *GEOSTAR* are used in *COSMOSM* program. It is an interactive full three-dimensional CAD-like graphic geometric modeler, mesh generator and FEA preprocessor and postprocessor.

So-called "from the bottom up" model building technique was used to design the construction model. In this case keypoints, the points that define the vertices of the model, are the "lowest-order" solid model entities.

**P. Tarvydas, E. Pačėsaitė, A. Noreika. Žemojo dažnio magnetinių laukų modeliavimas // Elektronika ir elektrotechnika. – Kaunas: Technologija, 2005. – Nr. 8(64). – P. 18–21.**

Nagrinėjami žemojo dažnio magnetinių laukų modeliavimo baigtinių elementų metodu klausimai. Analizuojama programa *COSMOSM* ir jos grafinio redaktoriaus *GEOSTAR* galimybės, aptariamos pagrindinės magnetiniams laukams modeliuoti naudojamų baigtinių elementų *MAG2D*, *MAG3D*, *TETRA4* ir *TETRA10* charakteristikos. Pateikiamas sudarytas apibendrintas modeliavimo algoritmas, pasiūlyta žemojo dažnio magnetinių laukų modeliavimo metodika. Aptariami baigtinių elementų tinklelio sudarymo klausimai. Pateikiamas solenoidinio elektromagneto sukuriamo magnetinio lauko modeliavimo pavyzdys. Il.8, bibl.7 (anglų kalba; santraukos lietuvių, anglų ir rusų k.).

**P. Tarvydas, E. Pačėsaitė, A. Noreika. Low-frequency Magnetic Field Modeling // Electronics and Electrical Engineering. – Kaunas: Technologija, 2005. – No. 8(64). – P. 18–21.**

Investigated low-frequency magnetic field modeling using finite element method. Analyzed programs *COSMOSM* and its preprocessor and postprocessor *GEOSTAR*, their possibilities, discussed main characteristics of finite elements *MAG2D*, *MAG3D*, *TETRA4* and *TETRA10* used for magnetic field modeling. Created overall modeling algorithm presented and low-frequency magnetic field modeling method proposed. Problems of creation of finite element mesh are discussed. Example of modeling of magnetic field created by solenoid actuator is presented. Ill.8, bibl.7 (in English; summaries in Lithuanian, English and Russian).

**П. Тарвидас, Э. Пачесайте, А. Норейка. Моделирование магнитного поля низкой частоты // Электроника и электротехника. – Каунас: Технология, 2005. – № 8(64). – С. 18–21.**

Изучаются вопросы моделирования магнитных полей методом конечных элементов. Анализируются возможности программы *COSMOSM* и ее графического редактора *GEOSTAR*, обсуждаются основные характеристики конечных элементов *MAG2D*, *MAG3D*, *TETRA4* и *TETRA10*, используемых для моделирования магнитных полей. Предоставлен составленный обобщенный алгоритм моделирования и предложена методика моделирования магнитных полей низкой частоты. Обсуждаются вопросы составления сети конечных элементов. Предоставлен пример моделирования магнитного поля, создаваемого катушкой с сердечником. Ил.8, библи.7 (на английском языке; рефераты на литовском, английском и русском яз.).

When creating construction solid model, keypoints are created first, and then those keypoints are used to define the "higher-order" solid model entities (that is, lines, areas, and volumes). In other FEA programs, for example *ANSYS*, so-called "from the top down" model building technique is used additionally. The *ANSYS* program gives the ability to assemble the model using geometric primitives, which are fully-defined lines, areas, and volumes. As a primitive structure is created, the program automatically creates all the "lower" entities associated with it. In this case the speed of model design is increased.

It is possible to use *MAG2D*, *MAG3D*, *TETRA4* and *TETRA10* finite elements for linear and nonlinear electromagnetic analysis. Wide-functionality direct and iterative solvers are used to solve the system of linear equations.

### References

1. **Monk P.** Finite Element Methods for Maxwell's Equations. – Oxford: Oxford University Press, 2003. – 336 p.
2. **Jin J.** The Finite Element Method in Electromagnetics. – New York: J. Wiley & Sons, 2002. – 753 p.
3. **Tarvydas P., Markevičius V.** Computation of magnetic fields using a finite element method // Elektronika ir elektrotechnika. – Kaunas: Technologija, 2000. – Nr.1(24). – P.33–37.
4. **Tarvydas P., Pačėsaitė E., Noreika A.** Coil magnetic field modeling in 3-D space // Electronics and Electrical Engineering. – Kaunas: Technologija, 2004. – No. 6(55). – P.17–20.
5. **COSMOSM Users Guide:** <http://www.cosmosm.com/>
6. **COSMOSM Electromagnetic Module ESTAR:** <http://www.cosmosm.com/>
7. **Ostergaard D. F.** Magnetics for Static Fields. – ANSYS Revision 4.3, Tutorials, 1987.

Pateikta spaudai 2005 04 26