7. THERMAL COMPUTATION OF ELECTRICAL MACHINES

K. Hameyer, U. Pahner and R. Belmans
Katholieke Universiteit Leuven, Belgium

H. Hedia
Université de Liège, Belgium

Abstract

Performance and rated data of electrical machines are strongly related to the heat transfer and the temperature distribution inside the machine caused by losses. To estimate the heat flux accurately, numerical design tools are required. The paper focuses on the application of the implementation of a thermal finite element (FEM) solver into a commercial FEM package for electromagnetics. Suitable data structures and the general program coding methodology of the implementation are discussed. Using an electrical machine as an example, the thermal field solution obtained by the thermal FEM module demonstrates the suitability of the method.

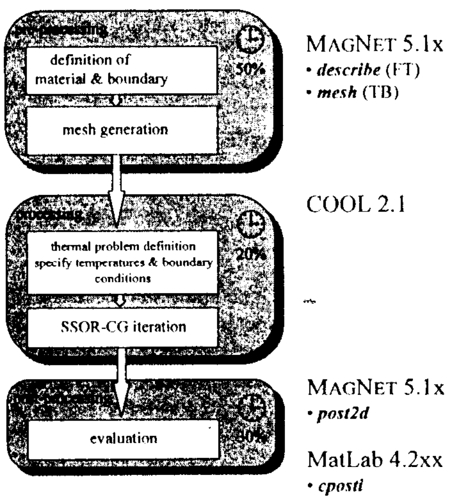


Fig. 1. FEM scheme for a thermal computation.

INTRODUCTION

Numerical techniques to compute the magnetic field are in common use. The finite element method is the most frequent and flexible tool to calculate the electromagnetic field in electrical machines. The general scheme of the FEM procedure can be divided up into three parts, the

- pre-processing,
- processing,
- post-processing.

Starting from a sketch of the geometry, closed regions have to be identified. A unique label can be assigned to all of these regions. The definition of material properties and of the driving sources is based on the assigned labels. Finally, boundary conditions are defined. This has to be done by the design engineer. In the last step of the pre-processing the whole problem has to be discretised by the finite elements. This is mostly automated and in a two-dimensional model the mesh consists of non-overlapping triangles or rectangles. The processing step only needs quantities concerning the algebraic solution of the system of equations, such as the accuracy of the solution and other more or less artificial quantities. After computing the solution of the defined problem, the post-process evaluates the desired field quantities and generates graphics or diagrams of the solution. This straight forward procedure of solving a finite element problem offers the possibility to add self developed software modules in all process steps. The given scheme then slightly changes. As a consequence, the solution of the thermal field can be obtained by using data from a problem definition concerning the geometry and labelled material domains of a previously FEM electromagnetic field computation. Using such data reduces the efforts of pre-processing the thermal problem. Thus, the parameters concerning the thermal problem, thermal conductivity, convection and radiation coefficients and the boundary conditions have to be supplied in the pre-process for the thermal problem only. The new FEM scheme including a thermal problem definition is illustrated in Fig. 1. The additional thermal pre-process is put to the processing block of the electromagnetic FEM procedure. Here, a commercial FEM package [1,2] suited for electromagnetic problems is extended by this thermal solver. Any open FEM package, which allows the formatted import and output of all key data, is suited.

Taking advantage of the equivalence between electrostatic and thermal fields, the necessary evaluations of the thermal solution can be performed in the post-processor of the electromagnetic FEM solver. This combined approach is also a step towards an overall solution of the behaviour of an electrical machine. As the losses are linked to the magnetic field distribution and the currents, the calculation of thermal and magnetic quantities has to be linked. Therefore, using the same pre- and post-processor is very advantageous.

HEAT TRANSFER

The knowledge of the temperature distribution within a body is important in many engineering problems. In particular in the design and optimisation of electric motors, the temperature distribution plays an essential role. The limits of overload capabilities are set by maximum temperatures in specific regions of the machine. The values of rated current and voltage of the machine are specified by the steady state temperature distribution.
For the two-dimensional, steady state problem, the governing partial differential equation is:

$$\frac{\partial}{\partial x}\left(k_x \cdot \frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_y \cdot \frac{\partial T}{\partial y}\right)+\dot{q}=0 \qquad (1)$$

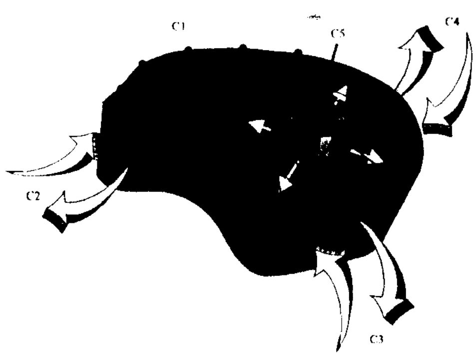


Fig. 2. Two-dimensional heat transfer problem with the boundary conditions.
C1 - prescribed nodal temperature,
C2 - heat flow across the boundary due to convection,
C3 - heat flow across the boundary due to radiation,
C4 - prescribed heat flux across the boundary,
C5 - internal heat source.

where k_x, k_y are the thermal conductivity coefficients, T the temperature and $\dot{q}$ the strength of the heat source per unit volume. Compared to the magnetostatic or electrostatic case, a more versatile set of boundary conditions has to be taken care of (figure 2.):

- C1: prescribed temperature T_0 at a node (Dirichlet condition)

$$T_0 = T(x,y) \qquad (2)$$

- C2: surface heat flow $h(T-T_\infty)$ across the boundary due to convection (Neumann condition)

$$k_x \frac{\partial T}{\partial x}\cdot l_x + k_y \frac{\partial T}{\partial y}\cdot l_y + h(T-T_\infty)=0 \qquad (3)$$

with T_∞ the ambient temperature and h the convection heat coefficient, l_x and l_y the direction cosines of the outward normal to the boundary surface.

- C3: surface heat flow $h_r(T-T_\infty)$ across the boundary due to radiation (Neumann condition)

$$k_x \frac{\partial T}{\partial x} \cdot l_x + k_y \frac{\partial T}{\partial y} \cdot l_y + h_r(T - T_\infty) = 0 \tag{4}$$

with T_∞ the ambient temperature. The radiation heat coefficient h_r is defined by

$$h_r = \sigma F(T^2 + T_\infty^2)(T + T_\infty) \tag{5}$$

where σ is the Stefan-Boltzmann constant and F an emissivity function.

- C4: prescribed heat flux q across the boundary (Neumann condition)

$$k_x \frac{\partial T}{\partial x} \cdot l_x + k_y \frac{\partial T}{\partial y} \cdot l_y + q = 0 \tag{6}$$

- C5: internal heat source $\dot{q}_0$

The steady state problem formulation described by eq. (1) and considering all boundary conditions (3-6) is equivalent to the problem of finding the temperature $T(x, y)$ which minimises the functional

$$\begin{aligned} f = {} & \frac{1}{2} \iint_A \left\{ k_x \left(\frac{\partial T}{\partial x}\right)^2 + k_y \left(\frac{\partial T}{\partial y}\right)^2 - 2\dot{q}T \right\} dA \\ & + \int_{C2} h(T^2 - 2TT_\infty)\,dC_2 \\ & + \int_{C3} h_r(T^2 - 2TT_\infty)\,dC_3 \\ & + \frac{1}{2} \int_{C4} qT\,dC_4 \end{aligned} \tag{7}$$

and simultaneously satisfies the first given boundary condition, the prescribed node temperatures at the contour C1, according to eq. (2) [3, 4, 7]. To obtain an approximated solution of (7) it is necessary to discretise the problem. Here, standard triangular elements with linear shape functions over each finite element are used. The derivation of the element matrices can be taken from literature [3, 4, 5, 7]. Due to the choice of elements a sparse system of equations has to be solved to compute the temperature at each node. The system of equations turns into a non-linear one, if heat flow due to radiation is considered as h_r from eq. (5) is a temperature dependent heat transfer coefficient. In this case an iterative solution procedure must be used.

IMPLEMENTATION OF THE THERMAL MODULE

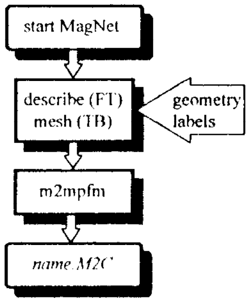


Fig. 3. Mesh generation using MAGNET.

In this section the implementation of the developed thermal module is described. To generate a mesh for the thermal module a commercial finite element package is used. The following methodology for the implementation of the thermal solver can be applied to any open FEM package available. Figure 3 illustrates the procedure. The geometry is prepared either with the utility programs *describe* or *draw2d*, whereas the program *mesh* is used for the discretisation of the regions and the definition of the constrained boundaries. Using the MAGNET filemanager *m2mpfm*, the mesh is transferred into an ASCII-format file. This is the input data file for the thermal module.
The steps in the pre-process (meshing and labelling) are:

1. generation of the geometry using the programs *describe* or *draw2d*
2. mesh generation using the program *mesh* to define the material labels. Materials, sources and constraints are defined in the thermal solver by using the labels given here.
3. definition of constraints (nodes are constrained by labels)
4. output of the mesh data into an ASCII-file using the program *m2mpfm*

Particular attention must be paid to the difference of boundary conditions between the electromagnetic and thermal field types while generating an admissible discretisation. Convection and radiation boundaries can only be applied to boundaries of the whole domain and not to contours inside this area (Fig. 2). In an electrical machine one pole pitch is assumed to be form a periodical symmetry.
After the meshing and labelling step and according to fig. 1 the thermal problem definition follows. The thermal material properties describing quantities have to be given by the user. The thermal module is started and inquires for the values. This part forms the most problematic step preparing the problem definition. The thermal material parameters depend on the temperature itself, on conditions concerning the air flow such as the velocity and turbulence caused by cooling fans, the end-winding geometry and the motor speed. A serious problem represents the choice of the radiation heat coefficient. F in eq. (5) accounts for the geometry of the radiating surfaces and their emissivities. The emissivity is defined as the ratio of total emitted power to that of a black body at the same temperature. Its value depends on the roughness, on the degree of oxidation if metal is considered and the temperature of the surface. Practical values for this parameter are known for parallel surfaces [3]. On the other hand, practical surfaces may not be parallel nor flat. The calculation of F is sufficiently complicated that it should be computed by external programs. Design experience is necessary to chose the right values. Some publications are dealing with the calculation of the parameters to be used particularly in electrical machines [10-12].
The straight forward internal structure of the developed thermal FEM module is given in fig. 4. The structure of the matrices inside the element loop can be taken from the literature [3, 4, 5, 7]. Extensions in the source code to obtain the thermal transients can be implemented without enormous efforts.

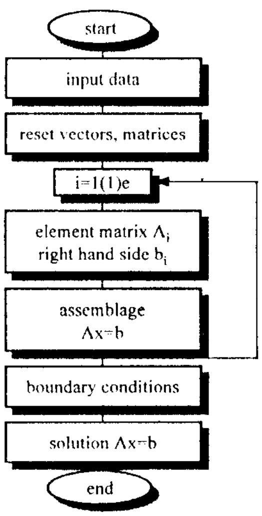


Fig. 4. Structure of the thermal FEM module.

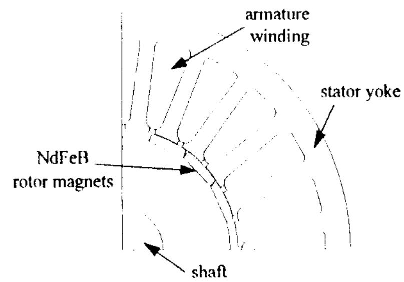
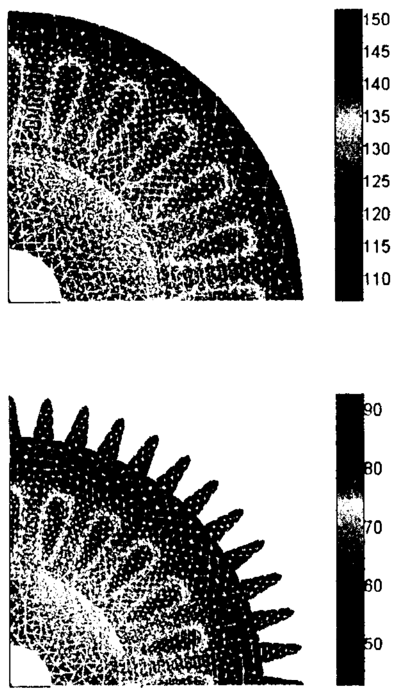


Fig. 5. Four-pole servo motor geometry.



a)

b)

Fig. 6. a) Steady state temperature distribution and b) temperature distribution with heat dissipater.

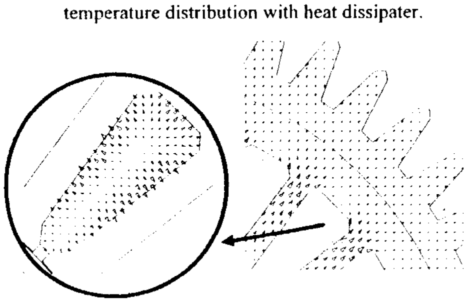


Fig. 7. Heat flux inside a slot and the heat dissipator.

SIMULATION EXAMPLE

The magnetic properties of the high-energy permanent magnet material NdFeB is highly temperature dependent. With increasing temperature the magnetic characteristics changes. A rapid decrease in the magnetic characteristics above 100° can be observed. Therefore, permanent magnet excited servo motors constructed with NdFeB must be carefully designed from the magnetic as well as from the thermal point of view. Here, a 4-pole servo motor is considered. Three NdFeB permanent magnet blocks per pole are fixed to the rotor and the armature winding is arranged in 24 stator slots (Fig. 5). In this example, the FEM was used to complete the electromagnetic design. Thus, a finite element mesh consisting of 2500 triangular elements already exists and can be used in the thermal analysis. A convection boundary condition is applied at the outer diameter of the motor. For the simulation an ambient temperature of 50° C is assumed. The ohmic losses of the armature winding are calculated for the rated winding current and serve as the heat sources in the thermal simulation. Due to the electrical insulation of the single copper conductors, the thermal conductivity of the slot regions is set to 0.5 W/mK. A diagram with values for the thermal conductivity considering the copper fill-factor of a slot can be found in [10]. Figure 6a) shows the computed temperature distribution inside the motor. The slot areas show the highest temperatures. Caused by the high ambient temperature the stator lamination at the outer diameter is in the range of 100°C. The permanent magnets are heated up to 125°C. A second simulation of the servo motor with the same input data shows a possible solution to protect the magnet material against over-heating. Here, a heat dissipator is arranged to the stator. The temperature of the magnet material from this simulation lies now in the range of 68°C. A closer view to the heat flux out of a stator slot to the thermally good conducting iron lamination can be taken from fig. 7. The main part of the slot heat flux through the stator teeth.

CONCLUSIONS

The paper discusses possibilities off a software module which extents a commercial FEM package suited for electromagnetics to treat thermal problems. Using the pre- and post-process utilities from the FEM package for magnetics has the advantage that the user has not to learn the handling of an additional software. The internal structure of a thermal FEM module is introduced. A permanent magnet excited servo motor selected as an example demonstrates the suitability of the methods used. The implementation of thermal transients is in progress.

ACKNOWLEDGEMENTS

The authors are indebted to the Belgian Nationaal Fonds voor Wetenschaptelijk Onderzoek for its financial support for this work and the Belgian Ministry of Scientific Research for granting the IUAP No. 51 on Magnetic Fields.

REFERENCES

[1] Edwards, J.D. and Freeman, E.M.: *MagNet 5.1 User Guide - Using the MagNet Version 5.1 Package from Infolytica*, Infolytica, London and Montreal, 1995.

[2] *Magnet 5, 2D Toolbox Reference Manual*, Infolytica Corporation, 1995.

[3] Akin, J.E.: *Finite elements for analysis and design*, Academic Press, London, San Diego, 1994.

[4] Cook, R.D.: *Finite element modelling for stress analysis*, John Wiley & Sons, Inc., New York, 1994.

[5] Zienkiewicz, O.C. and Taylor, R.L.: *The finite element method*, fourth edition, Vol. 1, Basic formulation and linear problems, McGraw-Hill Book Company, 1994.

[6] Zienkiewicz, O.C. and Taylor, R.L.: *The finite element method*, fourth edition, Vol. 2, Solid and fluid mechanics dynamic and non-linearity, McGraw-Hill Book Company, 1994.

[7] Rao, S.S.: *The finite element method in engineering*, Pergamon Press, 1989.

[8] Goering, H., Roos, H.-G. and Tobiska, L.: *Finite-Element-Methode*, Akademie Verlag, 1993.

[9] Schwarz, H.R.: *Methode der finiten Elemente*, Teubner Studienbücher, Mathematik, 1984.

[10] Plejic, M., Gorican, V. and Hribernik, B.: "FEM thermal modeling of an induction motor", ed. in Nicolet, A. & Belmans, R., *Electric and Magnetic Fields*, Plenum Press, 1995, pp. 155-158.

[11] Pickering, S.J., Lampard, D., Hay, N. and Roylance, T.F.: "Heat transfer from the stator end-windings of a low voltage concentric-wound induction motor", *Proc. Conf. Electrical Machines and Drives, IEE Conf. Pub. No. 412*, September 11-13, 1995, pp. 477-481.

[12] Henneberger, G., Ben Yahia, K. and Schmitz, M.: "Calculation and identification of a thermal equivalent circuit of a water cooled induction motor for electric vehicle application", *Proc. Conf. Electrical Machines and Drives, IEE Conf. Pub. No. 412*, September 11-13, 1995, pp. 6-10.