L1

Axially symmetrical coil model

Instructions for the laboratory exercise

ver. 09/29/2016

Arkadiusz Lewicki, Jarosław Guziński

1. Aim of the exercise

The purpose of the exercise is to prepare the model and perform calculations and their coil analysis axially symmetrical using CAE (*Computer Aided Engineering*) tools .
The exercise is carried out in the FEMM 4.2 (*Finite Element Method Magnetics*) program - http://www.femm.info/wiki/HomePage.

2. Assumptions

The analysis concerns the distribution of the magnetic field created by the air coil. located in an open space. The analyzed element is shown in Fig. 1.

axis of symmetry

air

winding

Fig. 1. Air coil

Coil dimensions assumed:
- inner diameter: 20mm
- outer diameter: 60mm
- height: 50mm

Winding: 100 turns of 18AWG copper wire ie 1 mm diameter (see attachment).
Current value 1.5 A.

3. Construction of the coil model

The coil will be modeled as axially symmetrical. The symmetry axis is marked in Fig. 1.

This will draw the geometry for only half the cross section.

Problem definition

After starting the FEMM program, select from the menu: *FILE-New* or the icon . IN
in the dialog box choose *Magnetics Problem* and confirm with *OK* . will
then the window in which the model in question should be drawn.

Page 2

Page 2

The model can be prepared earlier eg in AutoCAD - FEMM program
allows you to import DXF files.

Since the analysis will be carried out for the axisymmetric model, it should be
select from the menu: *Problem.* A dialog box with the settings and will be displayed
parameters:

Set *Problem Type* as *Axisymetric* - the coil geometry will be represented
as axially symmetrical. The axis of symmetry is a line with the coordinate r = 0.

Set Lenght units as *Millimeters* - coil dimensions are given in mm,
it is convenient to switch units to mm (*Lenght units - Millimeters*). Further
frequency is defined

Frequency - assume 0 in the task, because the task is solving by accident
magnetostatic.

Depth - model depth, ie the third, invisible dimension of the drawing, which at
the axisymmetric model is not set, the default value is *1* .

Precision Solver - precision of calculations, default value is 10^{-8}.

Min Angle - a parameter used to construct a finite element mesh [1].
the default is *30* .

AC Solver - selection of numerical method for solving problem [2] , default
Succ. Approx. denoting the iterative method, ie subsequent approximations (
Successive Approximation).

Comment - - a comment describing the implemented task.

Adopting an axially symmetrical approach requires that the axis of symmetry of the modeled coil
was the vertical axis of the *xy* rectangular system .

Definition of the analysis area

The finite element method used by the program requires that
the area concerned was limited. The first task is therefore limitation
of the analyzed area by drawing its borders. The area is also to include
projected magnetic field lines outside the coil itself.

The area concerned should fit in the program window, so it should
set the displayed area in the menu: *View / Keyboard accordingly* . For the considered
assume problem:

1 The parameter defines the minimum angle of the triangles forming a mesh of finite elements. The smaller the value the calculations are more accurate. In most cases, correct results are obtained for angle 33.8. More complex problems may require reducing the parameter below 20 to ensure sufficient accuracy of floating point calculations. Values from 1 to 33.8 are accepted.

2 The method chosen in *AC Solver* is only relevant for troubleshooting circuits with alternating currents. In this exercise, the coil is powered by direct current, so this parameter does not matter.

Page 3

The program window will be scaled accordingly. After scaling one of the value will automatically change depending on the resolution and size of the monitor computer.

Then define the area analyzed by the FEMM package. Area is defined by defining its border points with the buttons: *Operate on Nodes* :

Operate on Nodes

Place border points with coordinates: (0, -50), (0.50), (0.0). Can use *Tab* key and enter the coordinates of the points, e.g.:

The symmetry axis is further defined. Select operation on lines: *Operate on segments,* and indicate previously drawn points with coordinates (0, -50) and (0.50).

Operate on segments

Then select the work on arches: *Operate on arc* and indicate points (0, -50) and

(0.50). When working on arches, the order in which points are selected is important.

Operate on arc

A dialog box will appear that allows you to specify the arc parameters. This draw *Arc Angle* semicircle should be set to 180. Set the *Max Segment* parameter to 2.5. The arc is approximated by joining straight lines together. *Max Segment* parameter determines what the next straight line is drawn by. The analysis area is as follows area:

Page 4

Page 4

Area analyzed
inside the semicircle

Drawing windings

The program should be switched to node mode. You must define nodes in such way that the distance between them corresponds to the dimensions of the cross-section of the coil. Warning - only the part of the coil marked in fig. 2 is drawn.

established
coil cross section

Fig. 2. A fragment of the coil with nodes entered in the program

Coil nodal coordinates are as follows: (10, 25), (30, 25), (10, -25) (30, -25), because:

50 mm is the height of the coil,

20 mm is the inside diameter,

60 mm is the outer diameter.

The first coordinate is the radius, the second is the coil height.

Then switch the program to simple drawing mode and connect prepared points so that a rectangle is formed. In this way, a cross-section is obtained coil windings.

Adding materials and assigning them to areas

The next operation is to define the material for the areas. In consideration there are two materials - air and copper. Select the *Operate on* icon *labels* :

Operate on labels

Page 5

Then indicate two areas - inside the rectangular cross-section of the windings and outside coils but inside the semi-circle of the analysis area.

The program uses area names to assign material parameters. In the *Properties* menu select *Materials Library* and drag Air and 18AWG from the *Library Materials* to *Model Material* - Fig. 3 . You can edit the parameters of the materials in *Materials Library* after double-clicking on the indicated material.

Fig. 3. Material library window

Select menu *: Properties / Circuit* . In the dialog box press the *ADD* button *Property* to create a circuit. Name the *winding* . Set *Series* which means that all wires in the cross-sectional area of the coil are connected in series. Enter accepted in 1.5 A coil current

The next step is to assign drawing areas to materials and perimeter supply. Click with the right mouse button near the area name (color

point label of the area changes to red) and press *Space* or the icon .

For air, set *Block type* to *Air.* Uncheck *Let Triangle choose Mesh Size* and enter 1 as *Mesh size* . This parameter defines the largest possible size of the grid element. The mesh is made of equilateral triangles with no side greater than the defined value. When *Let Triangle choose Mesh Size* is selected, the program chooses the size of the mesh with as few elements as possible.

For winding cross-section - *Block type* must be set : 18AWG *, In circuit* - *Winding* and number of turns per *Number of turns* : 100.

If the number of turns is positive then the winding direction is opposite to the movement clockwise, while the negative number of turns indicates the winding direction according to clockwise [3] .

[3] Clockwise looking at the coil from above.

Table 1 gives the keyboard shortcuts available in the FEMM program

Tab. 1. Selected keyboard shortcuts available in the FEMM program

KEY	**FUNCTION**
Space	Editing parameters of the indicated object
Tab	Displays a dialog box in which the new coordinates are defined point or block
Escape	Deselects selected objects
Delete	Deletes selected objects
Arrows	They move the screen view
Page Up	magnification
Page Down	Reduction
home	Sets the window view so that all entered objects are visible

4. Calculations

Boundary conditions

From the *Properties | Boundary* menu , choose *Add Property* . Then give a name boundary conditions (ABC) and change *BC Type* to *Mixed* . The name ABC is an abbreviation Eng. *"Asymptotic Boundary Condition* " meaning asymptotic method of solving edge creams. This condition allows the reproduction of an unlimited impact

external area (despite the fact that the analysis itself is carried out in a limited area).
In the case of the *Mixed* boundary condition [4], parameters c_0 and c_1 should be determined in accordance with

$$c_0 = \frac{1}{\mu_0 \mu_r R} \qquad c_1 = 0 \tag{1}$$

where μ_r - magnetic permeability at the edge of the area,
μ_0 - magnetic permeability of the vacuum [5].
R - radius semicircle Analyzed area (for Considered in the case of R = 50 mm, ie in the SI unit system radius R = 0.05 m).

To assign a boundary condition, switch to arc drawing, then right-click the mouse near the arc bounding the area under consideration press *Spacebar* (or the *Properties* button). From *Boundary Cond* , select *ABC* .

Mesh generation

The file with the prepared models should be saved to the computer disk and press the button mesh generator:

Run mesh generator

[4] Description in [1] - annex A.3.2.

[5] $\mu_0 = 4\pi \cdot 10^{-7}$ H / m

Page 7

If the mesh is too thin / too dense, you can edit it again parameters of areas, lines and arcs.

Start calculations

Calculations can be started by pressing the button:

Run analysis

If the progress of the calculations are not satisfactory, they should be stopped (calculations for correctly defined and set problem last a few seconds). Most likely boundary conditions are incorrectly defined.

5. Analysis of results

The analysis of results is started by the button:

View results

It is possible to carry out several types of analysis of results:

Spot analysis - button pressed:

By indicating the appropriate points it is possible to determine for them parameters, including flux, energy loss, induction. You can enter coordinates after pressing the *Tab* key .

Coil parameters (including inductance, resistance):

Field distribution along the contour:

Page 8

Page 8

Select eg points along the coil's symmetry axis in the order: upper, middle, bottom resulting in a line and then press the button:

The induction distribution in the color version

6. Exercise program

1. For the adopted air coil, prepare its model, perform calculations and analyze the results.

2. Place the coil on the iron core. Check how the field layout will change magnetic coil and inductance.

3. Place the iron ball in the coil's axis of symmetry and determine the force with which it is attracted ball through the coil. Determine the attraction force characteristics as a function of coil current. Determine the attraction force characteristics as a function of the ball's distance from the coil.

   Determining the force with which the ball is attracted requires:

   Marking the ball cross-sectional area by selecting *Operation Area* and an indication of any point inside the ball that will be marked in green color

   select *Integrate Force via Weighted Stress Tensor* .

4. Place the permanent magnet in the magnetic field of the coil. Check how the magnetic field distribution will change. Examine the magnetic field distribution for different magnetization directions.

5. Model the actual cylindrical coil in the laboratory and determine number of turns.

   The coil is wound on a steel core. Fig. 4 shows photos of the coil while

   drawing with marked dimensions in fig. 5. The coil is wound with copper wire with a diameter of 0.8 mm. Winding inductance L = 9.5 mH was measured in the laboratory. Take current density J = 2.5 A / mm ^{2} . Assume that the steel core does not protrude beyond the outline coil.

   For the selected number of turns, determine the winding resistance.
   Calculate the wire length in meters to wind the coil.

Fig. 4. Views of the coil.

Fig. 5. Coil dimensions.

7. Literature

1. David Meeker: *FiniteElement Method Magnetics. User's Manual* .
2. David Meeker: *FEMM 4.0 Magnetostatic Tutorial* .
3. Konopiński T., Pac R.: *Transformers and chokes for electronic power devices* . WNT, Warsaw 1979.

Attachment 1
Designation of wire diameters

AWG (*American Wire Gauge*) is a designation of the American standardized series wire diameters, marked with conventional numbers, used for non-ferrous wires. AWG to mm converter are given in the table below.

If the construction of the wire is SOLID, it means that it is wire, whereas if in the *Construction* section *the cable* is a number / number type code, it is a multi-core cable - a strand. For example, for 32AWG 7/40 means that the cable consists of 7 lines 40AWG diameter, ie

$$7 \text{ x } 0.005 \text{ mm}^2 = 0.035 \text{ mm}^2$$

which roughly corresponds to a 32AWG wire cross-sectional area.