Finite Element Model of the Synchronous Machine with FEMM and Matlab – Part 1

Loïc QUEVAL (loic.queval@centralesupelec.fr)
revised Feb 24, 2022

1. Requirements

- FEMM 4.2 (download: www.femm.info)

2. Introduction

In this document, we present a step-by-step tutorial to simulate a synchronous machine (SM) with FEMM. Although the objective of the tutorial is for the reader to build the model on their own, the completed `tutorial_SM_QUEVAL_20220224.fem` can also be downloaded.

3. Model construction

The SM to be analyzed is pictured in Figure 1. It has 3 phases,2 pairs of poles and 24 stator teeth. The rotor has an outer diameter of 138 mm. The airgap width is 1 mm. The stator has an outer diameter of 210 mm. The axial length of the machine is 210 mm. There are 3 armature windings (a, b, c), each wound with 8 turns of copper wire. There is 1 field winding (f) wound with 30 turns of copper wire. In this part, we will consider the case where the field current is 1 A and the armature currents are zero.

In FEMM, one models the problem in 2D, neglecting end effects to reduce the problem complexity. By convention, positive-valued currents flow in the out-of-the-screen direction and positive-valued rotation is in the counterclockwise direction.

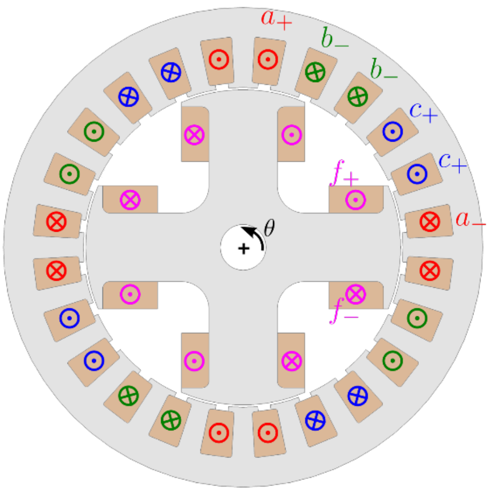

Figure 1 - SM geometry and conventions.

3.1 Create a new model

Run the FEMM application. The default preferences will bring up a blank window with a minimal menu bar.

Select `File > New` from the main menu. A dialog will pop up with a drop list allowing you to select the type of new document to be created. Select `Magnetics Problem` and hit the `OK` button. A new blank magnetics problem will be created, and several new toolbar buttons will appear.

3.2 Set problem definition

The first task is to tell the program what sort of problem is to be solved. To do this, select `Problem` from the main menu. The `Problem Definition` dialog appears. Set `Problem Type` to `Planar`. Make sure that `Length Units` is set to `Millimeters` and that the `Frequency` is set to 0. Set the `Depth` to 210. Set the `Smart Mesh` to `Off`. Hit the `OK` button (Figure 2).

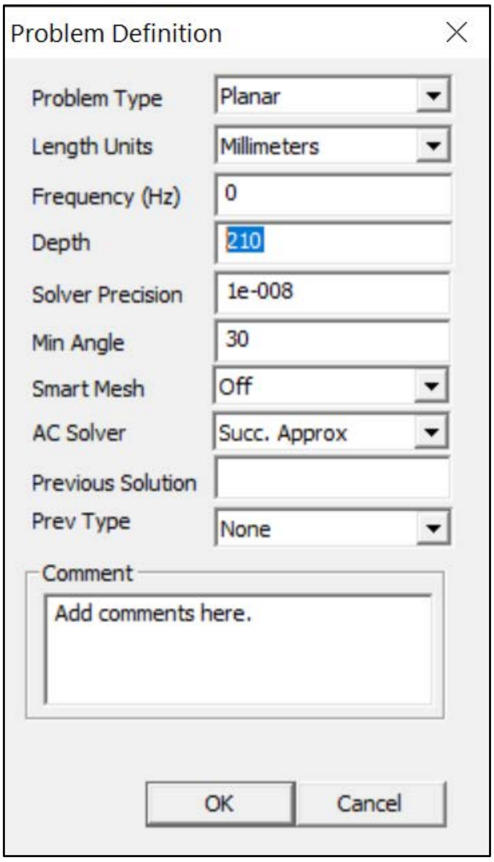

Figure 2 - "Problem definition" dialog.

3.3 Draw the geometry

We will import the geometry from DXF files.

Select `File > Import DXF` from the main menu. Select the rotor DXF file `geom_SM_rotor_QUEVAL_20220224.dxf` from your computer and click on `OK`. The DXF dialog opens. Set the `tolerance for DXF Import` to 0.001. Click on `OK`. Click on `zoom extents` to center the geometry. The result is shown on Figure 3.

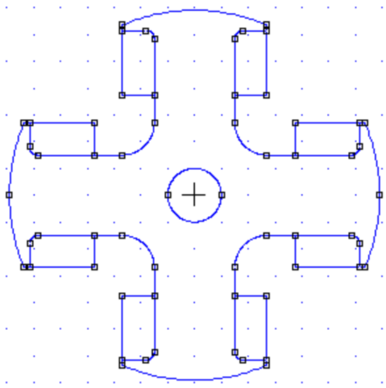

Figure 3 - Rotor geometry.

Let’s assign a group number to the rotor assembly so that we can rotate it easily later on. Select the `Operate on groups of objects` button . Select the `Select a group of entities using the mouse` button . Select all the rotor, it will turn red, denoting that it is selected. Press `<SPACE>` to “open” the `Group Properties` dialog (Instead of pressing the space bar, one can use the `Open up Properties Dialog` toolbar button). The `Group Properties` dialog opens. Set the group number `In Group` to `1` (Figure 4). Click `OK`.

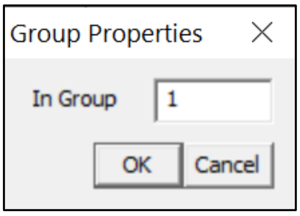

Figure 4 – “Group properties” dialog.

Select `File > Import DXF` from the main menu. Select the stator DXF `geom_SM_stator_QUEVAL_20220224.dxf` file from your computer and click on `OK`. The DXF dialog opens. Set the `tolerance for DXF Import` to 0.001. Click on `OK`. Click on `zoom extents` to center the geometry. The result is shown on Figure 5.

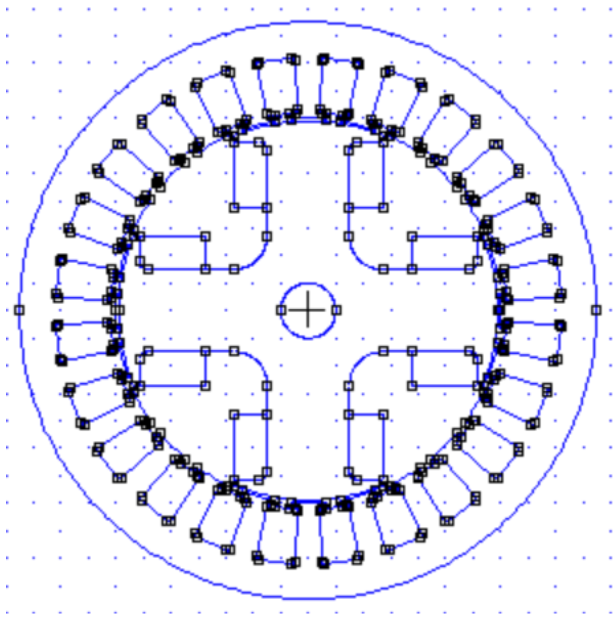

Figure 5 – Rotor + stator geometry.

The drawing of the geometry is finished. Save the model.

3.4 Add materials to the model

Select `Properties > Materials Library` from the main menu. The `Materials Library` dialog appears. Drag-and-drop `Air` from Library Materials to Model Materials to add it to the current model. Go into the `Soft Magnetic Materials > Silicon Iron` folder and drag `M-19 Steel` into Model Materials. Click on `OK`.

3.5 Add circuits to the model

Select `Properties > Circuits` from the main menu. The `Property Definition` dialog opens.

Push the `Add Property` button to create a new circuit property. In the `Circuit Property` dialog, set the `Name` to `a`. Specify that the circuit property is to be applied to a wound region by making sure that the `Series` radio button is selected. Set the `Circuit Current to 0` A. Click on `OK.`

Similarly add the circuit `b`. Set the `Circuit Current to 0` A.

Similarly add the circuit `c`. Set the `Circuit Current to 0` A.

Similarly add the circuit `f`. Set the `Circuit Current to 1` A (Figure 6).

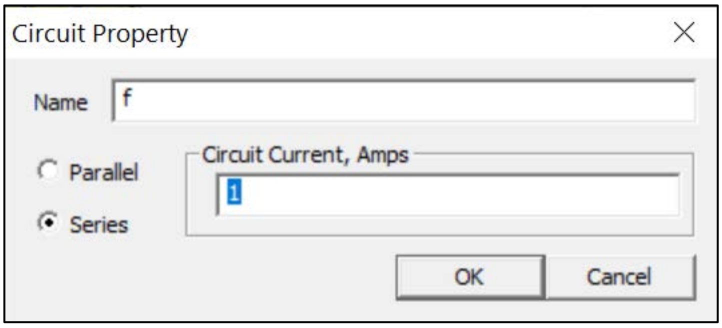

Figure 6 – "Circuit property" dialog for the circuit f.

3.6 Place block labels

FEMM uses block labels to associate materials and other properties with various domains in the problem geometry.

Click on the `Operate on Block Labels` toolbar button denoted by concentric green circles . Place a block label in each one of the 36 domains (10 in the rotor, 1 in the airgap and 25 in the stator). The result is shown on Figure 7.

NOTE: If snap-to-grid is enabled then it may be sometimes difficult to place the block label in the empty space. If this is the case, disable snap-to-grid.

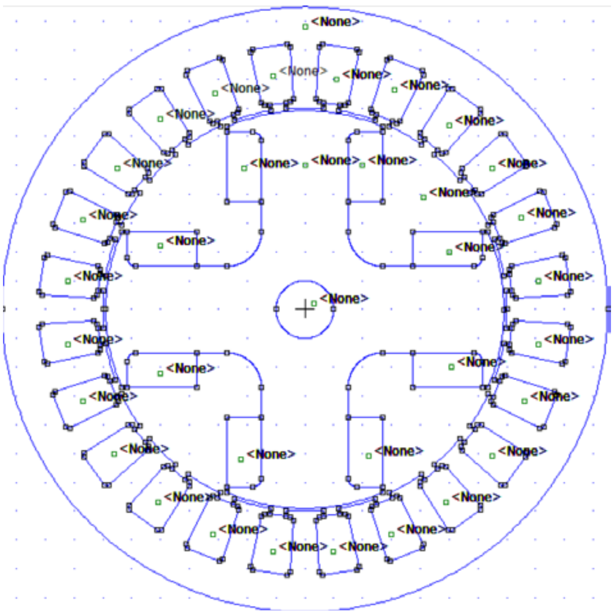

Figure 7 - Geometry after adding the block labels.

3.7 Associate a material and a circuit property to each domain

We want to associate each domain (Figure 8) to its material and circuit properties, as defined in Table 1. This is done using the block label placed in each domain. For the sake of example, let's consider the Coil a+ domains.

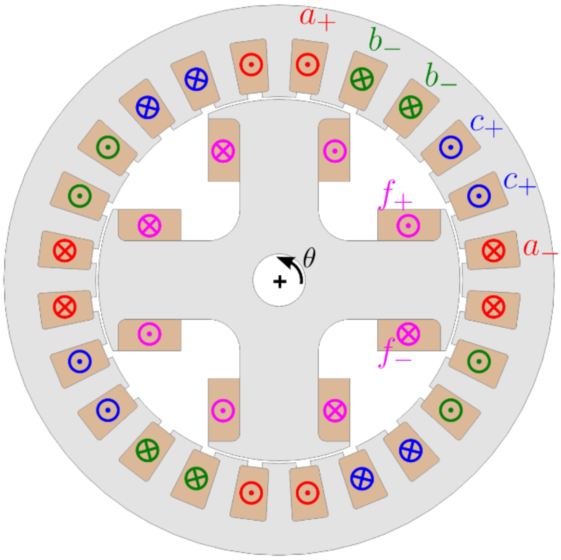

Figure 8 - Geometry with coil domains definitions.

Table 1 - Material and circuit property of each domain

Domain	Block type	In circuit	Turns	In group
Coil a+	Air	a	8	0
Coil a-	Air	a	-8	0
Coil b+	Air	b	8	0
Coil b-	Air	b	-8	0
Coil c+	Air	c	8	0
Coil c-	Air	c	-8	0
Coil f+	Air	f	30	1
Coil f-	Air	f	-30	1
Stator iron	M-19 Steel	<None>	<None>	0
Airgap	Air	<None>	<None>	1
Rotor iron	M-19 Steel	<None>	<None>	1
Rotor shaft	Air	<None>	<None>	1

Right click on all the block labels in the Coil a+ domains. The block labels will turn red, denoting that they are selected. Press `<SPACE>` to "open" the selected block label (Instead of pressing the space bar, one can use the `Open up Properties Dialog` toolbar button). The `Properties for selected block` dialog will pop up, containing the properties assigned to the selected label.

Set the `Block type` to `Air`.

We want to assign currents to flow in these domains, so select `a` from the `In Circuit` drop list. The `Number of turns` edit box will become activated if a series-type circuit is selected for the region (the `Coil` property that was previously defined). Set the `Number of turns` to `8`, denoting that each domain is filled with 8 turns with positive currents flowing in the out-of-the-screen direction.

Click on `OK`. The block labels will then be labeled as `Air[a:8]`.

NOTE: If we wanted to denote that each domain is filled with 8 turns with positive currents flowing in the into-the-screen direction, we could have specified the number of turns to be -8.

Similarly define the other domains according to Table 1. The result is shown on Figure 9.

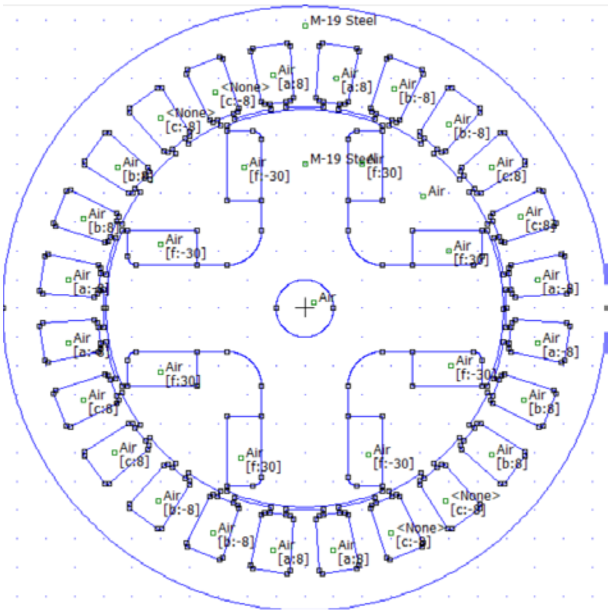

Figure 9 - Geometry after associating material and circuit property to each domain.

At this point, all the domain materials and circuit properties have been defined. Save the model.

3.8 Add boundary conditions to the model

Select `Properties > Boundary` from the main menu.

Push the `Add Property` button. The `Boundary Property` dialog opens. Set the `Name` to `zero`. Specify that the boundary property is a Dirichlet boundary condition by selecting `Prescribed A` in the `BC Type` field. Click on `OK` (Figure 10).

Click on `OK` to close the `Property Definition` dialog.

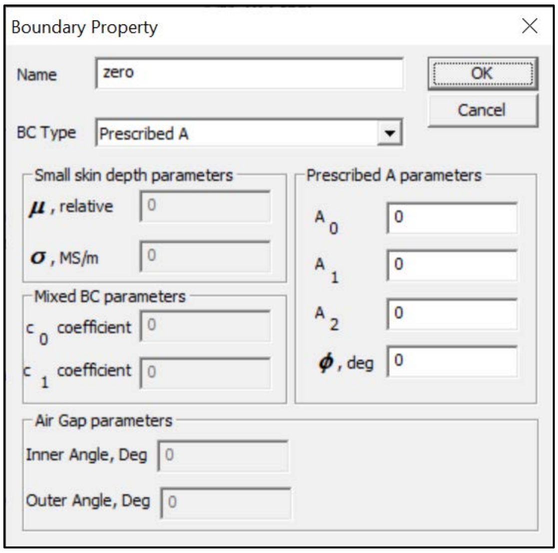

Figure 10 – “zero” boundary condition.

3.9 Associate a boundary condition to each model boundary

We want to associate the outer boundary to its boundary condition.

Switch to Arc segment mode by pressing the `Operate on arc segments` toolbar button.

Right click on the two outer iron boundaries. They will turn red, denoting that they are selected. Press `<SPACE>` to "open" the selected arc (Instead of pressing the space bar, one can use the `Open up Properties Dialog` toolbar button). A dialog will pop up containing the properties assigned to the selected arc. Set the `Max. segment, Degree` to `5`. Set the `Boundary cond.` to `zero`. Click `OK`.

At this point, all the boundary conditions have been defined. Save the model.

4. Generate Mesh

Click on the toolbar `Run mesh generator` button. This action generates a triangular mesh for your problem (Figure 11). You should have approximately 30500 nodes.

NOTE: If the mesh spacing seems to fine or too coarse you can select block labels or line segments and adjust the `Mesh size` defined in the properties of each object. To adjust all the mesh sizes in your model at once, press the `<F3>` button to refine the mesh in all blocks or the `<F4>` button to coarsen the mesh in all blocks.

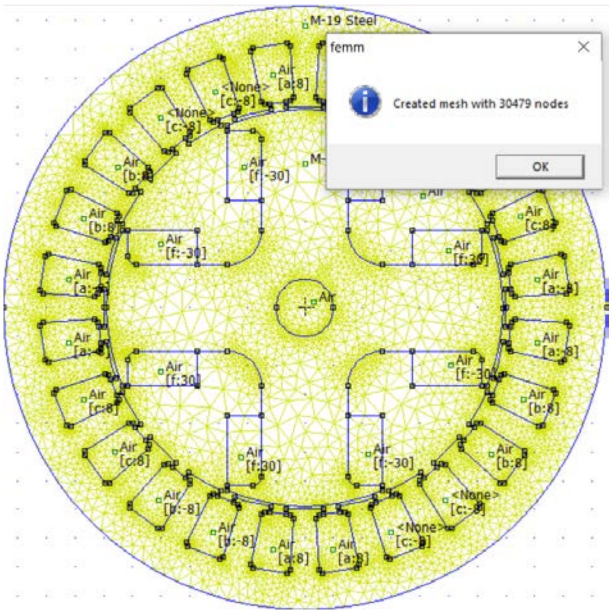

Figure 11 - Mesh.

5. Run the finite element analysis

Click on the `Run analysis` button to analyze your model.

Processing status information will be displayed. If the progress bars do not seem to be moving, then you should probably cancel the calculation. This can occur if insufficient boundary conditions have been specified. For this problem, the calculations should be completed within few seconds. There is no confirmation for when the calculations are complete, the status window just disappears when the processing is finished.

6. Analysis Results

Click on the glasses icon to view the analysis results. A post-processor window will appear. The post-processor window will allow you to extract many different sorts of information from the solution. By default, a black-and-white graph of flux lines is displayed (Figure 12).

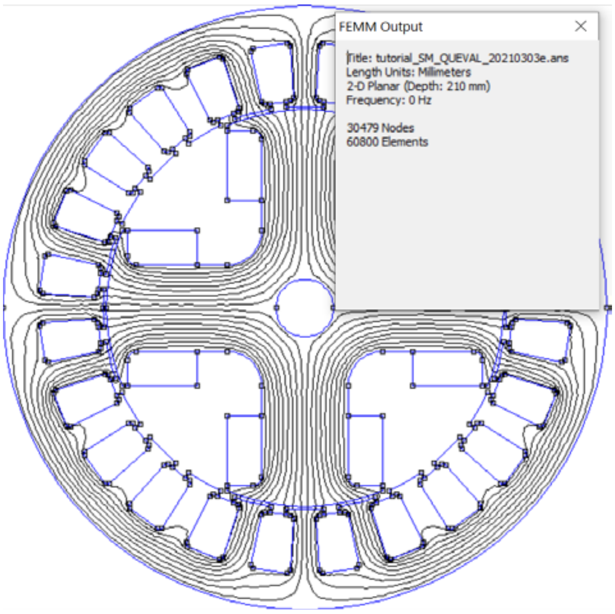

Figure 12 – Default analysis results: flux lines.

6.1 Get field values at a point

Just like the pre-processor, the post-processor window has a set of different editing modes: `Point`, `Contour`, and `Area`. By default, when the program is first installed, the post-processor starts out in `Point` mode. By clicking on any point with the left mouse button, the various field properties associated with that point are displayed in the floating `FEMM Output` window. The location of a point can be precisely specified by pressing the `<TAB>` button and entering the coordinates of the desired point in the dialog that pops up. For example, for the point (0, 45), the `FEMM Output` dialog is shown in Figure 13.

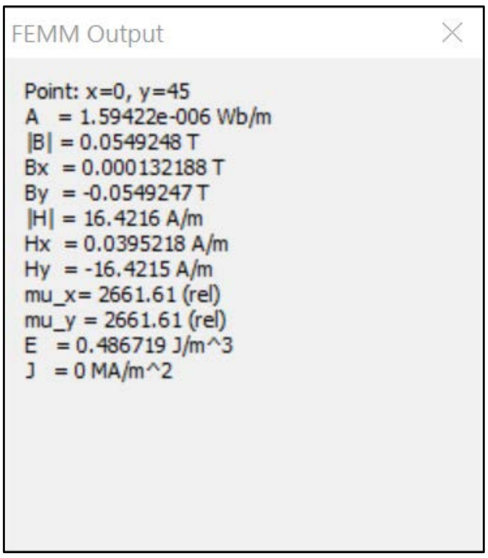

Figure 13 - Point values at the point (0, 45).

6.2 Plot field values along a contour

FEMM can also plot values of the field along a user-defined contour. Here, we will plot the magnetic flux density along the centerline of one field coil. Switch to Contour mode by pressing `Contour`. You can now define a contour along which flux will be plotted. There are three ways to add points to a contour:

1. Left Mouse Button Click adds the nearest input node to the contour;
2. Right Mouse Button Click adds the current mouse pointer position to the contour;
3. <TAB> Key displays a point entry dialog that allows you to enter in the coordinates of a point to be added to the contour.

Here, method 3 can be used. Press <TAB>, add points (-20, 45) and (20, 45) in the above order. Then, press the `Plot` toolbar button. The `X-Y Plot of Field Values` dialog opens (Figure 14). The default selection is `|B| (Magnitude of flux density)`. If desired, different types of plot can be selected from the drop list on this dialog. Hit `OK`. The result is shown in Figure 15.

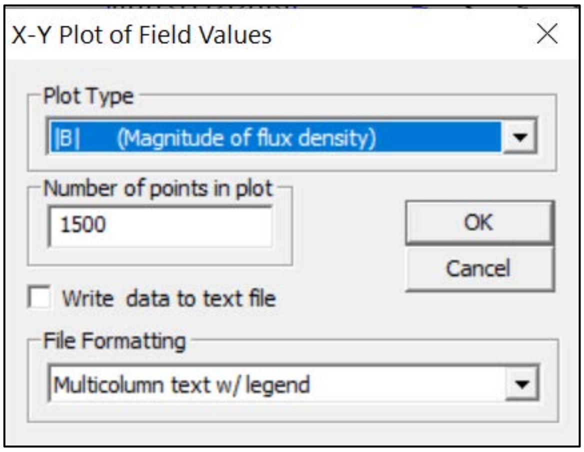

Figure 14 - "X-Y plot field values" dialog.

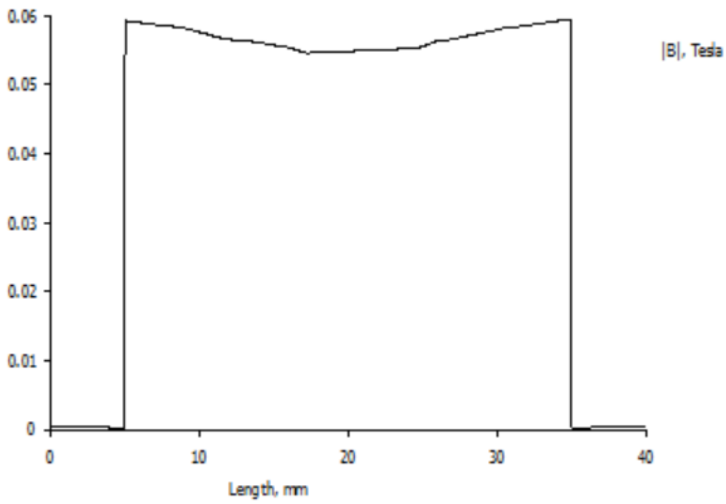

Figure 15 - Plot of the flux density along the centerline of one field coil.

6.3 Plot Flux Density

To make a color density plot of the magnetic flux density, click on the rainbow-shaded toolbar button. When the dialog box comes up, select the `Show Density Plot` check box and accept the other default values. Click on `OK`. The resulting solution is shown in Figure 16.

Note that the magnetic flux density is everywhere below the saturation point of the iron. For this operating point, the circuit is linear.

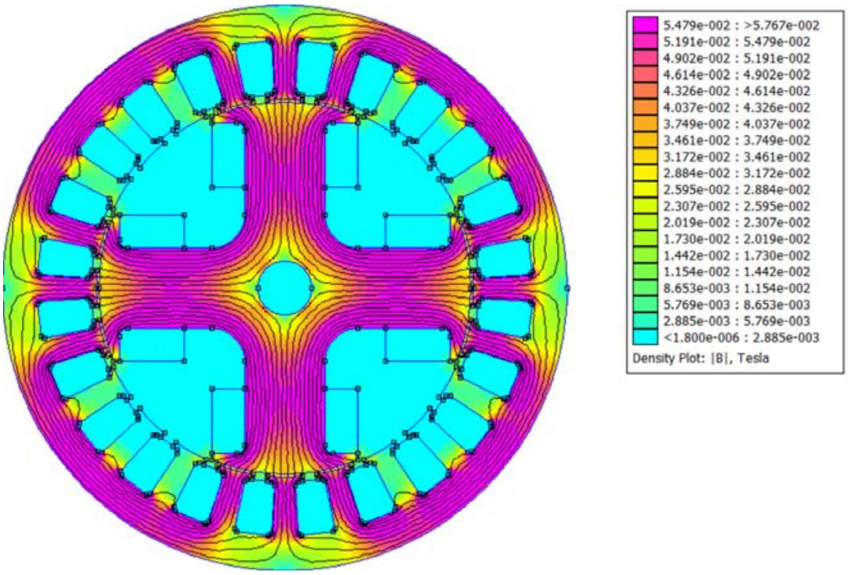

Figure 16 – Plot of the flux density distribution.

6.4 Calculate flux linkages & inductances

Press the button to display the resulting attributes of each `Circuit Property` that has been defined. For each circuit, the calculated flux linkages are shown in Figure 17.

Since there is only one current, and since this current is equal to 1 A, the flux linkages can be unambiguously interpreted as the coil self and mutual inductances. Indeed, in the linear case, by definition,

$$L_f = \frac{\lambda_f}{i_f}, \quad M_{af} = \frac{\lambda_a}{i_f}, \quad M_{bf} = \frac{\lambda_b}{i_f}, \quad M_{cf} = \frac{\lambda_c}{i_f}$$

Observe that for this operating point:

- $L_f > 0$
- $M_{af} \approx 0$
- $M_{bf} < 0$
- $M_{cf} > 0$

Can you calculate all the coefficients of the inductance matrix?

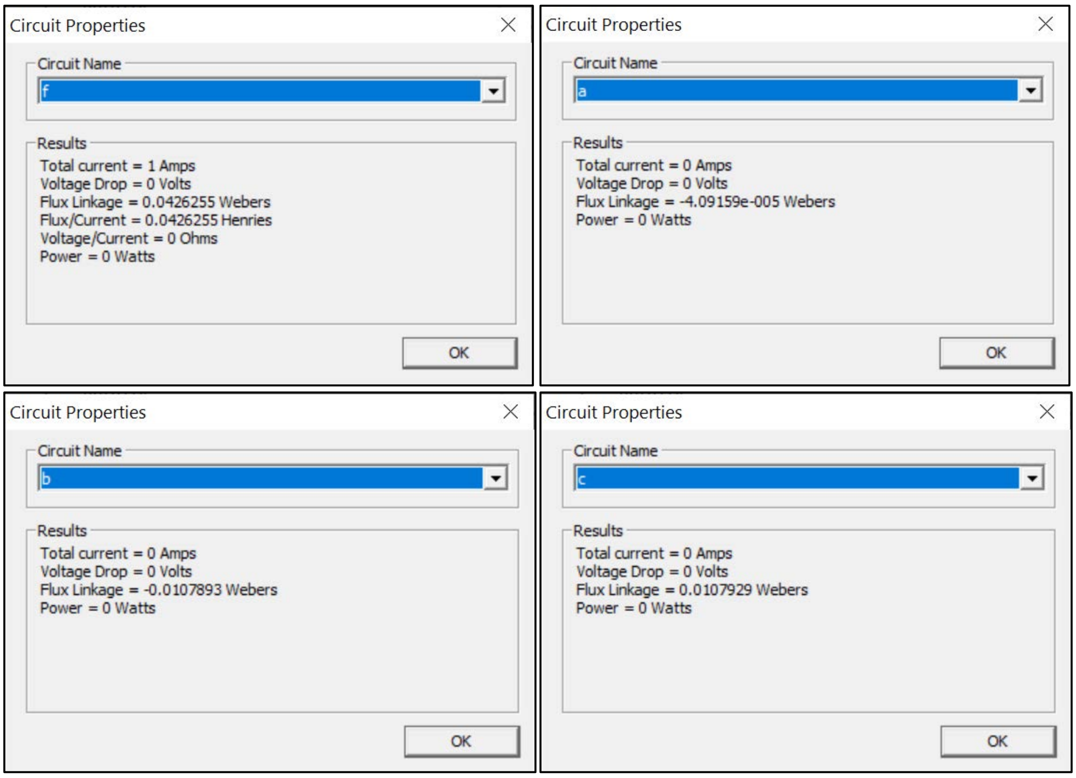

Figure 17 – "Circuit Properties" dialog (θ_m = 0 deg, i_a = 0 A, i_b = 0 A, i_c = 0 A, i_f = 1 A).

7. Conclusions

You have now completed the finite element modeling of a SM with FEMM (part 1). In this tutorial, you learned how to:

- Import a geometry from a DXF file;
- Define the material and circuit properties;
- Define the boundary conditions;
- Generate a mesh;
- Run the analysis;
- Inspect local field values;
- Plot field values along a contour;
- Plot the flux density distribution;
- Calculate flux linkages and inductances for a given operating point.