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

Loïc QUEVAL (loic.queval@centralesupelec.fr) & Guillaume KREBS
revised January 04, 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 switched reluctance machine (SRM) with FEMM. Although the objective of the tutorial is for the reader to build the model on their own, the completed `tutorial_SRM_QUEVAL_20210201.FEM` can also be downloaded.

3. Model construction

The SRM to be analyzed is pictured in Figure 1. It has 8 stator teeth and 6 rotor teeth (SRM 8-6). The rotor has an outer diameter of ~68 mm. The airgap width is 0.4 mm. The stator has an outer diameter of 143 mm. The axial length of the machine is 200 mm. There are 4 coils (a, b, c, d), each wound with 18 turns of copper wire. For the purposes of this example, we will consider the case in which a steady current of 50 A is flowing through the coil c.

In FEMM, one models the 2D problem using symmetry conditions to reduce the problem complexity. By convention, positive-valued currents flow in the out-of-the-page direction and positive-valued rotation is in the counterclockwise direction.

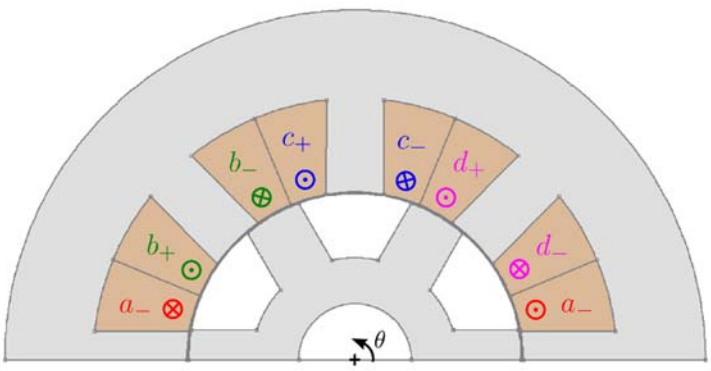


Figure 1 - SRM geometry and conventions.

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 a number of 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 200. Set the `Smart Mesh` to `Off` (Figure 2). Hit the `OK` button.

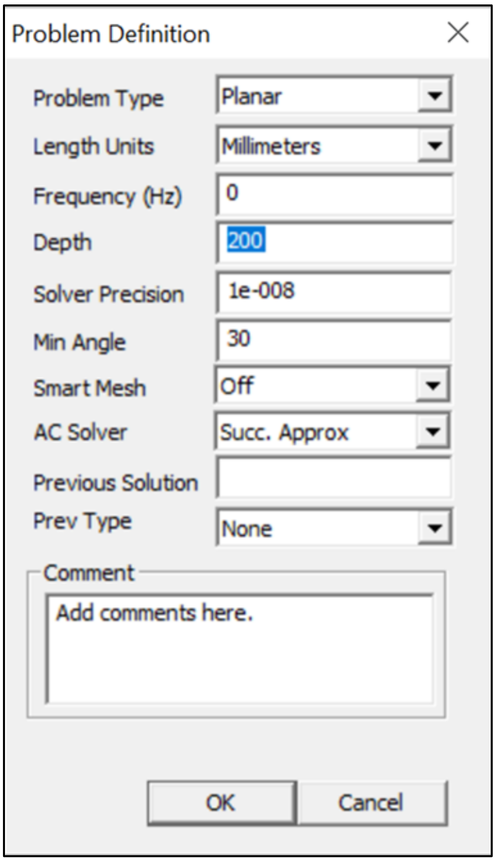


Figure 2 - "Problem definition" dialog.

3.3 Draw the SRM

We will first draw the rotor, then the stator. Then we will cut the full geometry to obtain the half geometry.

3.3.1 Rotor

Select the `Operate on nodes` button, so that nodes can be drawn. One can place nodes either by moving the mouse pointer to the desired location and pressing the left mouse button, or by pressing the <TAB> key and manually entering the point coordinates via a popup dialog. Place nodes at the coordinates given in Table 1. Click on `Zoom extents` to adapt the view to your screen. The result is shown on Figure 3.

Table 1 - Rotor nodes coordinates

Node	x [mm]	y [mm]
A	11.5	0
B	-11.5	0
C	20	6.08
D	20	-6.08
E	33.35	6.08
F	33.35	-6.08

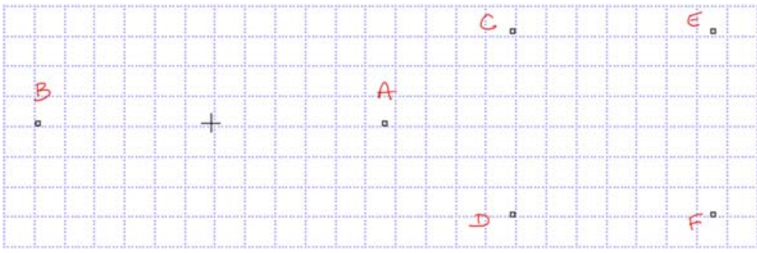


Figure 3 – Rotor geometry after plotting the nodes of Table I.

Draw the circle of diameter [AB]. Do do so, select the `Operate on arc segments` button. By selecting two nodes with left mouse button clicks in sequence, one can obtain arcs between them. Select the node A, then the node B. The `Arc segment properties dialog` appears. Set the `Arc Angle` to `180`, and click `OK`. An arc segment is obtained. Select the node B, then the node A. The `Arc segment properties dialog` appears. Set the `Arc Angle` to `180`, and click `OK`. Another arc segment is obtained. The two arcs form the circle of diameter [AB].

NOTE: An arc segment is always plotted counterclockwise with respect to the origin (0,0) starting from the first point that is added.

Draw the segments [CE] and [DF]. To do so, select the `Operate on segments` button so that lines can be drawn connecting the nodes. By selecting the nodes defining the coil with left mouse button clicks in sequence, one obtains segments between each of the nodes. Select the node C, then the node E. The segment [CE] is obtained. Select the node D, then the node F. The segment [DF] is obtained. The result is shown on Figure 4.

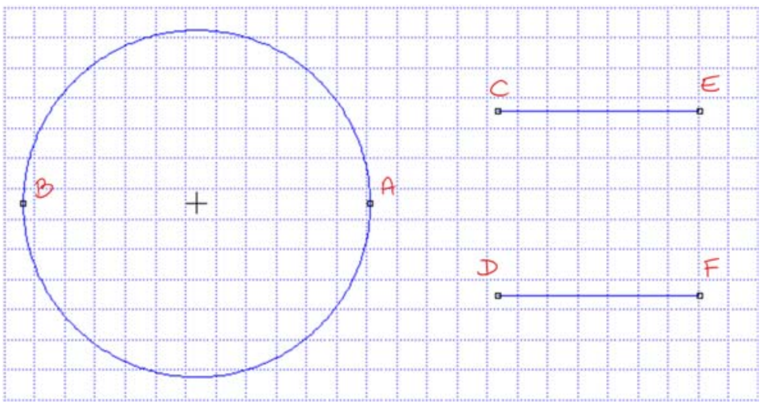


Figure 4 – Rotor geometry after plotting the circle and the segments.

Select the `Operate on segments` button. Right click on the segments [CE] and [DF], they will turn red, denoting that they are selected. Select the `Copy selected objects` button. The `Copy` dialog appears. Select Rotation. Set the `Angular shift` to `60` degrees and the `Number of copies` to `5`. Click `OK`. The result is shown on Figure 5.

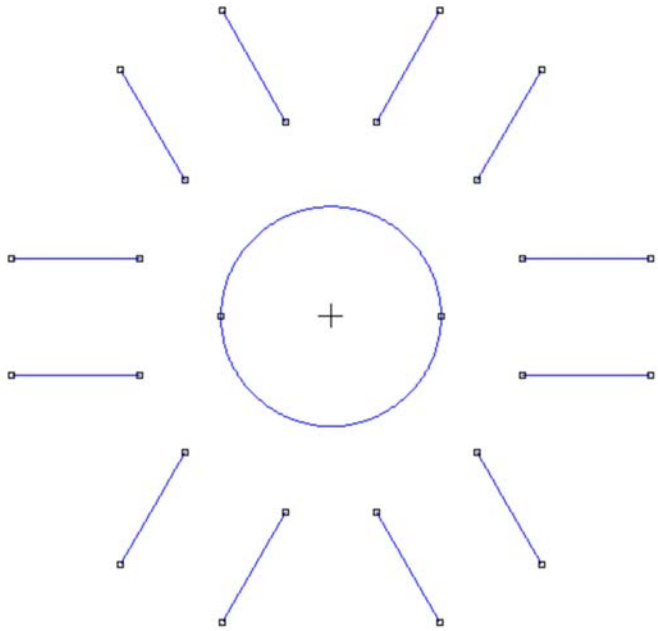


Figure 5 – Rotor geometry after rotating the segments.

Select the `Operate on arc segments` button. Draw the arc segments shown on Figure 6. Set the `Arc Angle` to `26` for the inner arcs. Set the `Arc Angle` to `21` for the outer arcs. The result is shown on Figure 6.

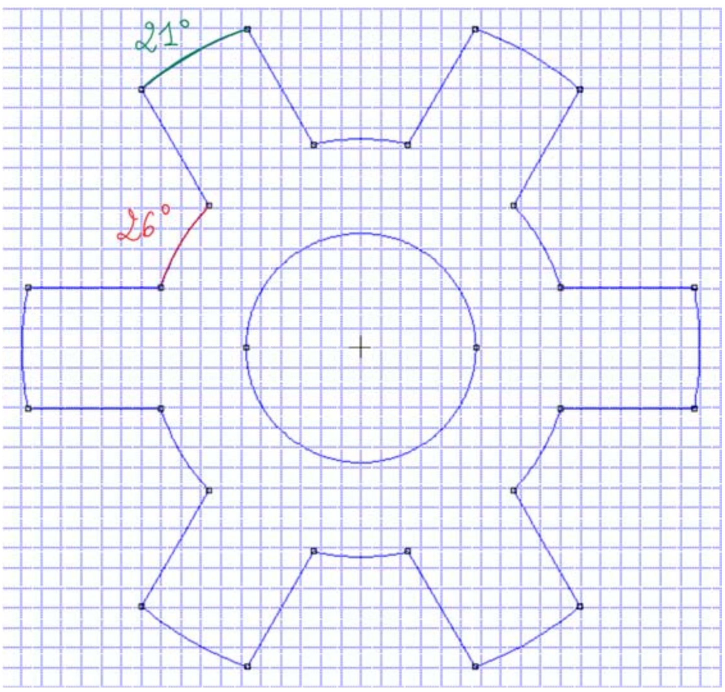


Figure 6 - Rotor geometry after plotting the arcs.

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

3.3.2 Stator

Let’s move the rotor on the side, to have more space to draw the stator. 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. Select the `Move/Rotate selected objects` button . The `Move` dialog appears. Select `Translation`. Set the `horizontal shift` to `200`. Click `OK`. The rotor moves to the right.

Switch to Nodes mode by pressing the `Operate on nodes` button . Place nodes at the coordinates given in Table 2. The result is shown on Figure 7.

Table 2 - Stator nodes coordinates

Node	x [mm]	y [mm]
G	34.3	0
H	53.4	0
I	51.3	14.9
J	51.3	-14.9
K	33.5	7.5
L	33.5	-7.5
M	71.5	0
N	-71.5	0

Figure 7 - Stator geometry after plotting the points of Table II.

Draw the circle of diameter [MN].

Draw the segments [GH], [KI] and [LJ].

Select the `Operate on arc segments` button . Draw the arc segments (LG and (GK with the `Arc Angle` set to `12.6` degrees. Draw the arc segments (JH and (HI with the `Arc Angle` set to `16.2` degrees. The result is shown on Figure 8.

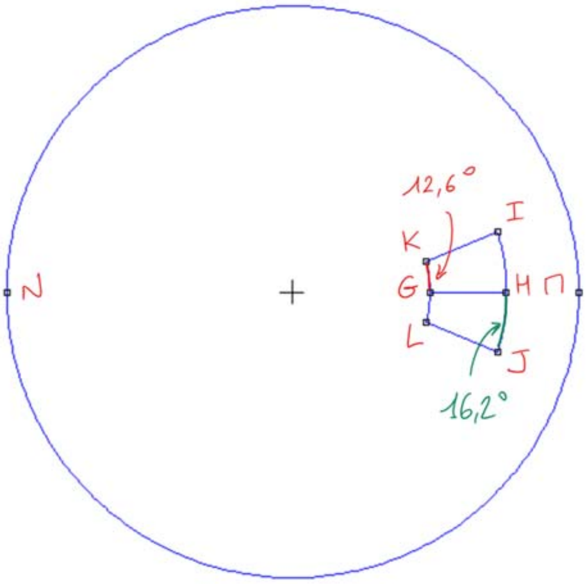


Figure 8 - Stator geometry after plotting the circle, the segments and the arcs.

Select the `Operate on a group of objects` button. Left click on the `Select a group of entities using the mouse` button. Select the segments [GH], [KI] and [LJ] and the arcs (KG, (GL, (IH and (HJ. They will turn red, denoting that they are selected. Select the `Move/Rotate selected objects` button. The `Move` dialog appears. Select Rotation. Set the `Angular shift` to `22.5` degrees. Click `OK`.

Select the `Operate on a group of objects` button. Left click on the `Select a group of entities using the mouse` button. Select the segments [GH], [KI] and [LJ] and the arcs (KG, (GL, (IH and (HJ. They will turn red, denoting that they are selected. Select the `Copy selected objects` button. The `Copy` dialog appears. Select Rotation. Set the `Angular shift` to `45` degrees and the `Number of copies` to `7`. Click `OK`. The result is shown on Figure 9.

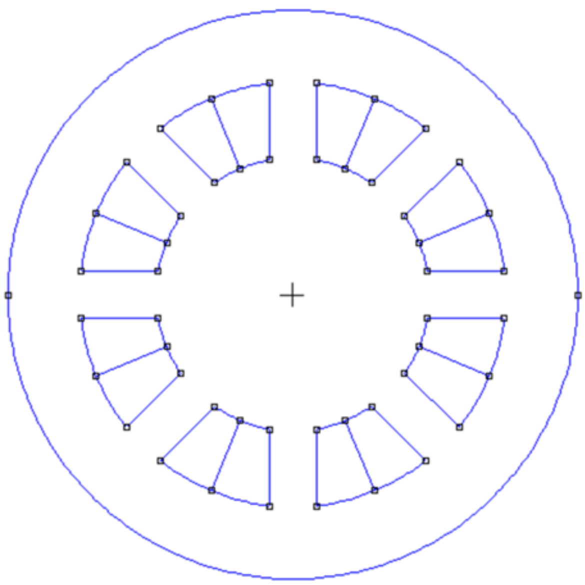


Figure 9 - Stator geometry after moving + rotating the segments and the arcs.

Select the `Operate on arc segments` button. Draw the arc segments shown on Figure *10* with the `Arc Angle` set to `18.8` degree. The result is shown on Figure 10.

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

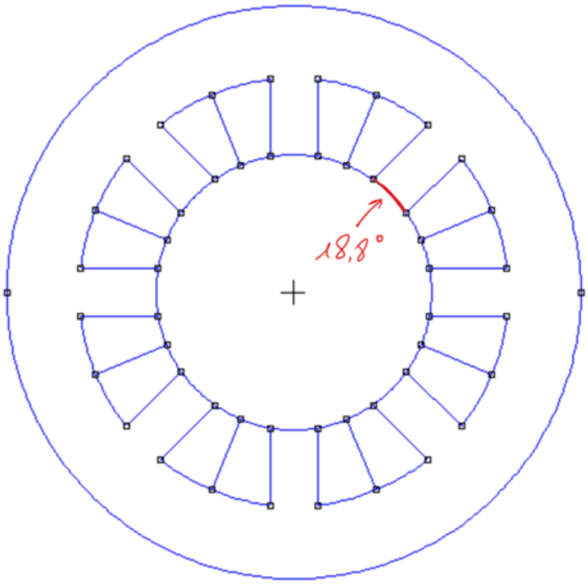


Figure 10 - Stator geometry after plotting the arcs.

3.3.3 Full geometry

Let's move the rotor back to the center. 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. Select the `Move/Rotate selected objects` button. The `Move` dialog appears. Select `Translation`. Set the `horizontal shift` to `-200`. Click `OK`. The result is shown on Figure 11.

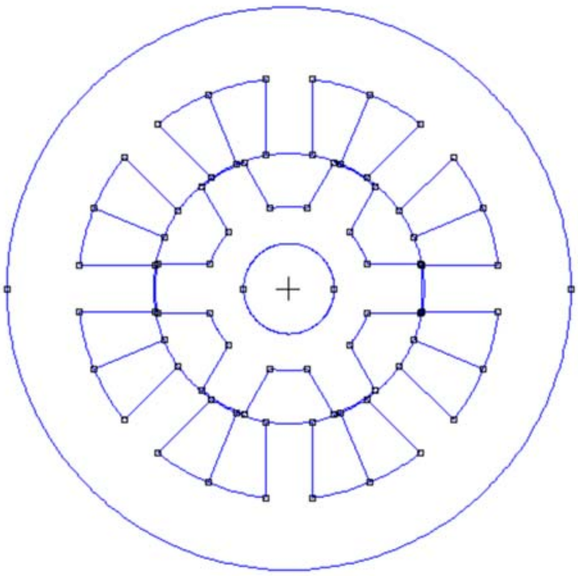


Figure 11 - Full geometry.

Let's add two circles in the airgap that will be used to deal with the rotation. Switch to Nodes mode by pressing the `Operate on nodes` button. Place nodes at the coordinates given in Table 3. Draw the circle of diameter [PQ] and the circle of diameter [RS]. The result is shown on Figure 12.

Table 3 - Airgap nodes coordinates

Node	x [mm]	y [mm]
P	34	0
Q	-34	0
R	34.2	0
S	-34.2	0

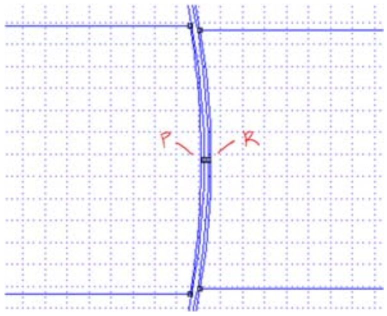


Figure 12 - Zoom on the airgap after plotting the 2 circles in the airgap.

3.3.4 Half geometry

Thanks to the symmetries, we need to model only half the geometry. Select the `Operate on segments` button. Draw the segment [MN]. The result is shown on Figure 13. Observe that new points have been automatically added at the intersections (Figure 14).

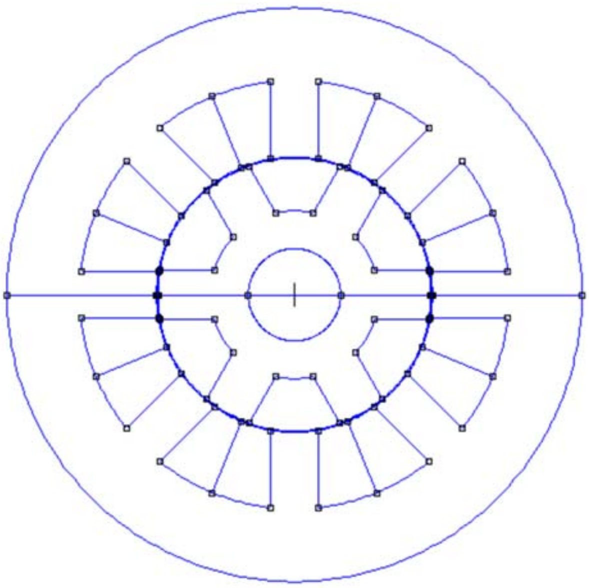


Figure 13 - Full geometry after plotting the segment [MN].

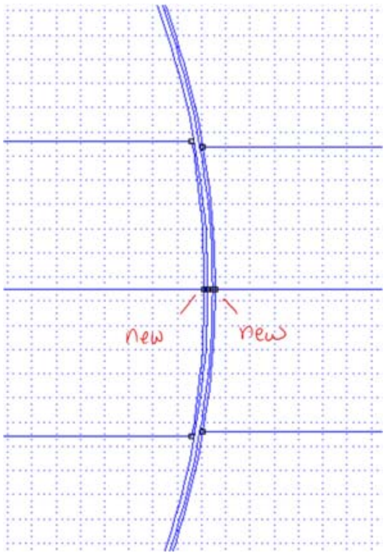


Figure 14 - Zoom on the airgap after plotting the segment [MN]. The new points that are automatically added at the intersections between [MN] and previously existing segments or arcs are denoted “new”.

Select the `Operate on segments` button. Select all the segments of the bottom half of the geometry and click on the `Delete selected objects` button. Select the `Operate on arc segments` button. Select all the segments of the bottom half of the geometry and click on the `Delete selected objects` button. Select the `Operate on nodes` button. Select all the nodes of the bottom half of the geometry and click on the `Delete selected objects` button. The result is shown on Figure 15.

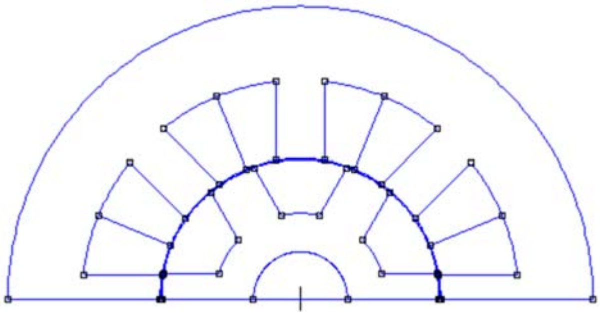


Figure 15 - Half geometry.

Select the `Operate on segments` button. Select the segments [AB], [PR] and [QS] and click on the `Delete selected objects` button. The result is shown on Figure 16 and Figure 17.

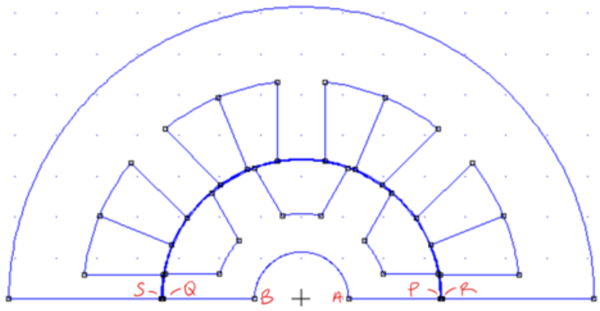


Figure 16 - Half geometry after deleting the segments.

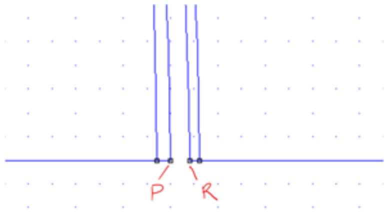


Figure 17 - Zoom on the airgap after deleting the segments.

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`. Click on `OK.`

Push the `Add Property` button to create a new circuit property. In the `Circuit Property` dialog, set the `Name` to `b`. 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`. Click on `OK.`

Push the `Add Property` button to create a new circuit property. In the `Circuit Property` dialog, set the `Name` to `c`. 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 50`. Click on `OK` (Figure 18).

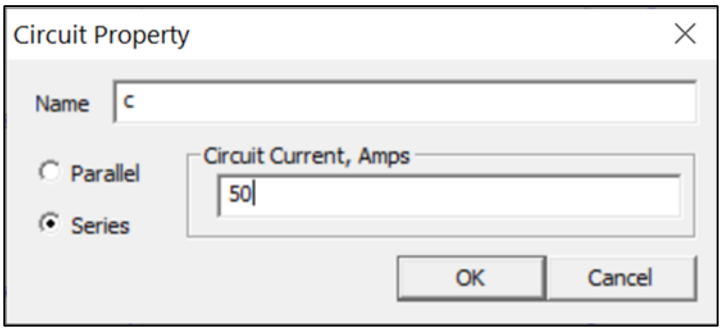


Figure 18 – "Circuit property" dialog for circuit c.

Push the `Add Property` button to create a new circuit property. In the `Circuit Property` dialog, set the `Name` to `d`. 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`. Click on `OK.` Click on `OK` to close the `Property Definition` dialog.

3.6 Place block labels

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

Select the `Operate on Block Labels` button. In the same way as nodes, block labels can be placed either by a click on the left mouse button, or via the `<TAB>` dialog. Place a block label in each one of the 12 domains. Make sure that the labels in the Airgap stator side domain and in the Airgap rotor side domain are correctly located. The result is shown on Figure 19 and Figure 20.

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

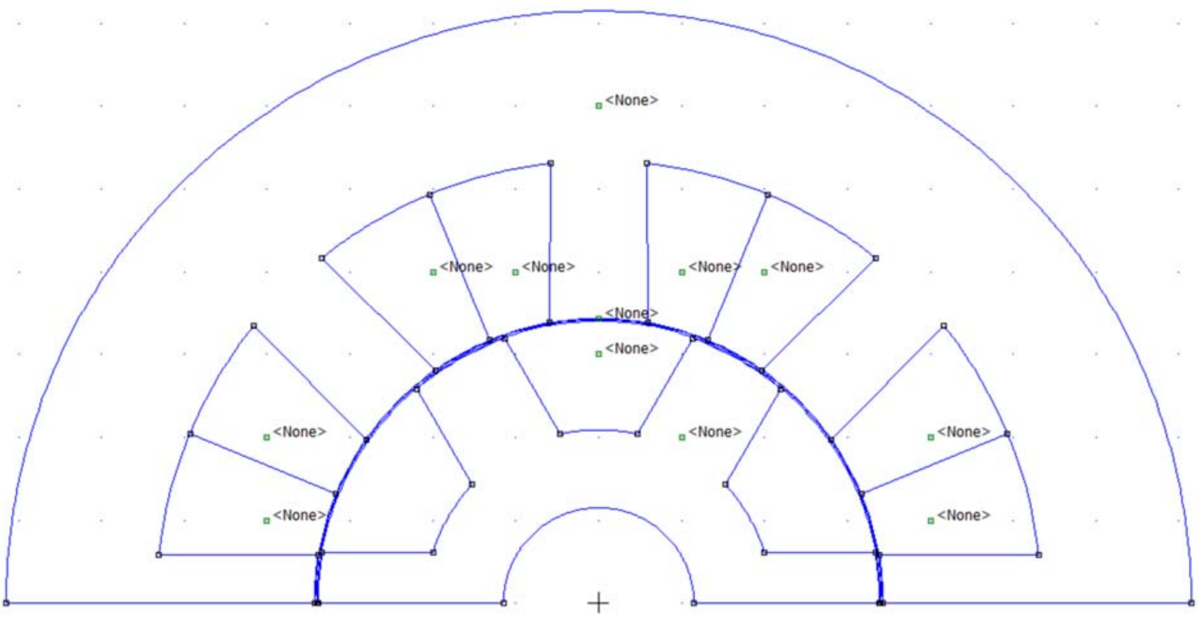


Figure 19 - Half geometry after adding the block labels.

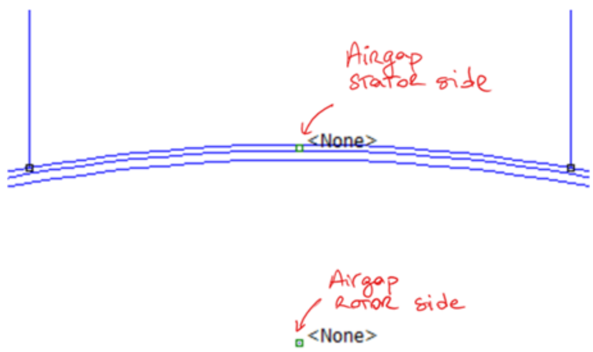


Figure 20 - Zoom on the airgap after adding the block labels. Note that there is one label in the airgap stator side domain and one label in the airgap rotor side domain. There is no label in the domain between them.

3.7 Associate a material and a circuit property to each domain

We want to associate each domain (Figure 21) to its material and circuit properties, as defined in Table 4. This is done using the block label placed in each domain. For the sake of example, let's consider the Coil b+ domain.

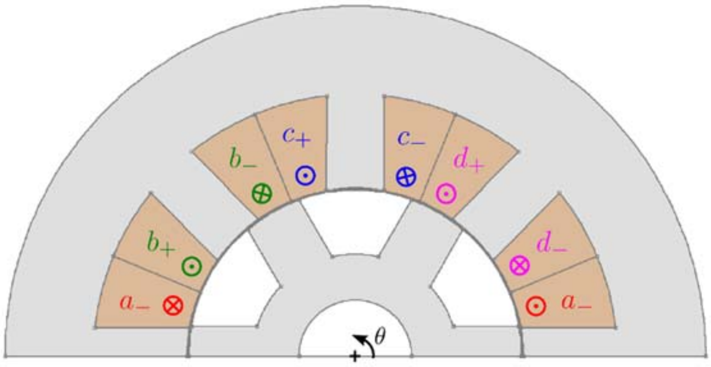


Figure 21 - Half geometry with coil domains definitions.

Table 4 - Material and circuit property of each domain

Domain	Block type	Mesh size	In circuit (turns)	In group
Coil b+	Air	2	b (18)	0
Coil b-	Air	2	b (-18)	0
Coil c+	Air	2	c (18)	0
Coil c-	Air	2	c (-18)	0
Coil d+	Air	2	d (18)	0
Coil d-	Air	2	d (-18)	0
Coil a-	Air	2	a (-18)	0
Stator iron	M-19 Steel	2	<None>	0
Airgap stator side	Air	0.2	<None>	0
Airgap rotor side	Air	1	<None>	0
Rotor iron	M-19 Steel	2	<None>	10

Right click on the block label node in the Coil b+ domain. The block label will turn red, denoting that it is 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`.

Uncheck the `Let Triangle choose Mesh Size` checkbox and set the `Mesh size` to `2`. The mesh size parameter defines a constraint on the largest possible elements size allowed in the associated section. The mesher attempts to fill the region with nearly equilateral triangles in which the sides are approximately the same length as the specified `Mesh size` parameter.

We want to assign currents to flow in this region, so select `b` 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). Enter `18` as the number of turns for this region, denoting that the region if filled with 18 turns with positive-valued currents flowing in the out-of-the-page direction.

Click on `OK`. The block label is then be labeled as `Air[b:18]`.

NOTE: If we wanted to denote that the turns have positive-valued currents flowing in the into-the-page direction instead, we would have specified the number of turns to be –18. The block label would then be `Air[b:-18]`.

Similarly define the other domains according to Table 4. Note that there are two a- domains. The result is shown on Figure 22.

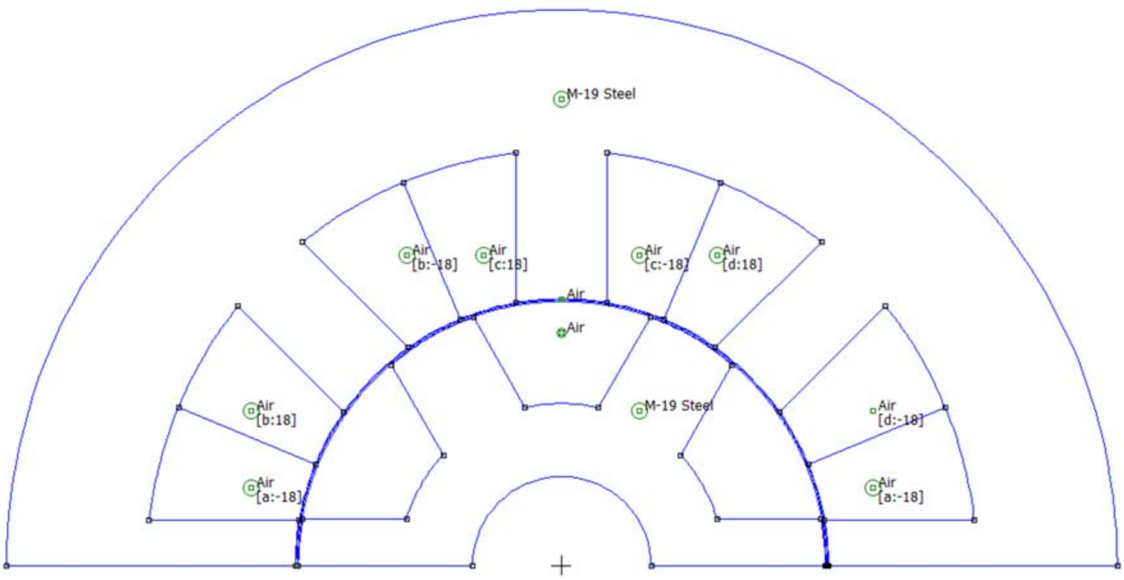


Figure 22 - Half geometry after associating material and circuit property to each domain.

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

3.8 Add boundary conditions to the model

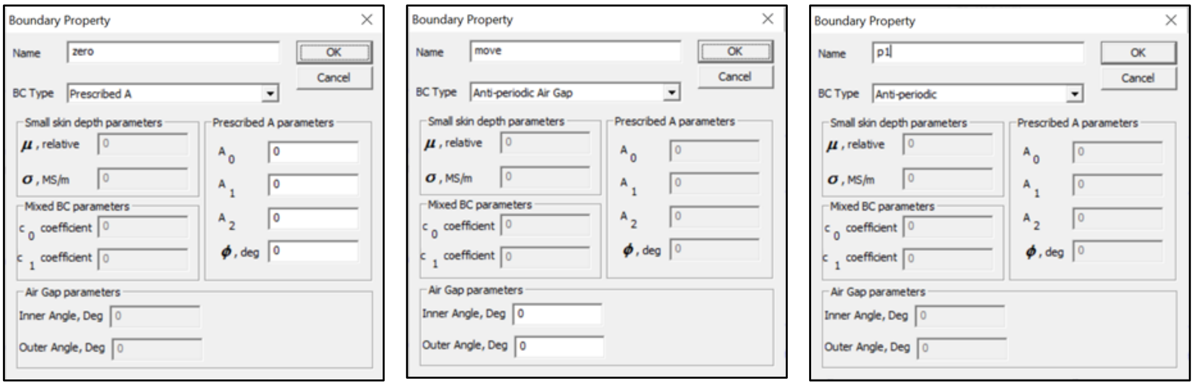
Select `Properties > Boundary` from the main menu.

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 23).

Push the `Add Property` button. The `Boundary Property` dialog opens. Set the `Name` to `move`. Specify that the boundary property is an anti-periodic boundary condition by selecting `Anti-periodic Air Gap` in the `BC Type` field. Click on `OK` (Figure 23).

Push the `Add Property` button. The `Boundary Property` dialog opens. Set the `Name` to `p1`. Specify that the boundary property is an anti-periodic boundary condition by selecting `Anti-periodic` in the `BC Type` field. Click on `OK` (Figure 23). In the same manner, create anti-periodic boundary conditions `p2, p3` and `p4`.

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



"zero" "move" "p1"

Figure 23 – "Boundary Property" dialogs.

3.9 Associate a boundary condition to each model boundary

We want to associate each boundary (Figure 24 and Figure 25) to its boundary condition.

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

Right click on the outer iron boundary. The boundary will turn red, denoting that it is 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`.

Similarly assign the `zero` boundary condition to the inner iron boundary, with the `Max. segment, Degree` set to `5`.

Similarly assign the `move` boundary condition to the inner and outer airgap boundaries, with the `Max. segment, Degree` set to `1`.

Switch to Segment mode by pressing the `Operate on segments` toolbar button .

Right click on the p1 boundary. The boundary will turn red, denoting that it is selected. Press `<SPACE>` to "open" the selected segment (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 segment. Set the property to `p1`. Click `OK`.

Similarly assign the `p1` boundary condition to the p1' boundary.

Similarly assign the `p2` boundary condition to the p2 and p2' boundaries.

Similarly assign the `p3` boundary condition to the p3 and p3' boundaries.

Similarly assign the `p4` boundary condition to the p4 and p4' boundaries.

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

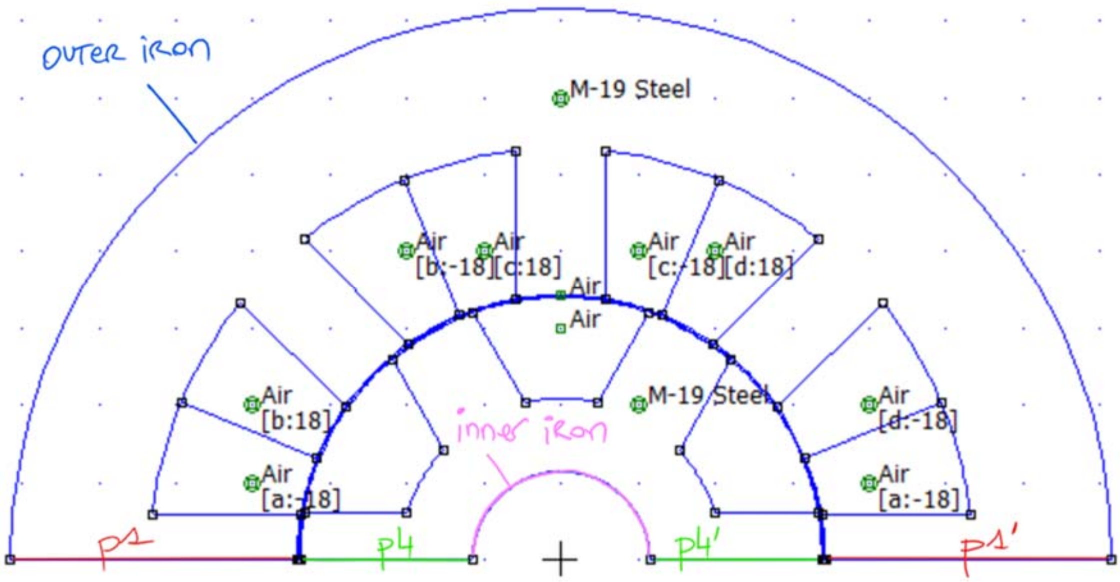


Figure 24 - Half geometry with boundary definitions.

Figure 25 - Zoom on the airgap with boundary definitions.

4. Generate Mesh

Click on the toolbar button with yellow mesh. This action generates a triangular mesh for your problem (Figure 26). You should have approximately 9400 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 of the mesh sizes in your model at once, press* <`F3`> *to refine the mesh in all blocks or* <`F4`> *to coarsen the mesh in all blocks. Then you can regenerate the mesh.*

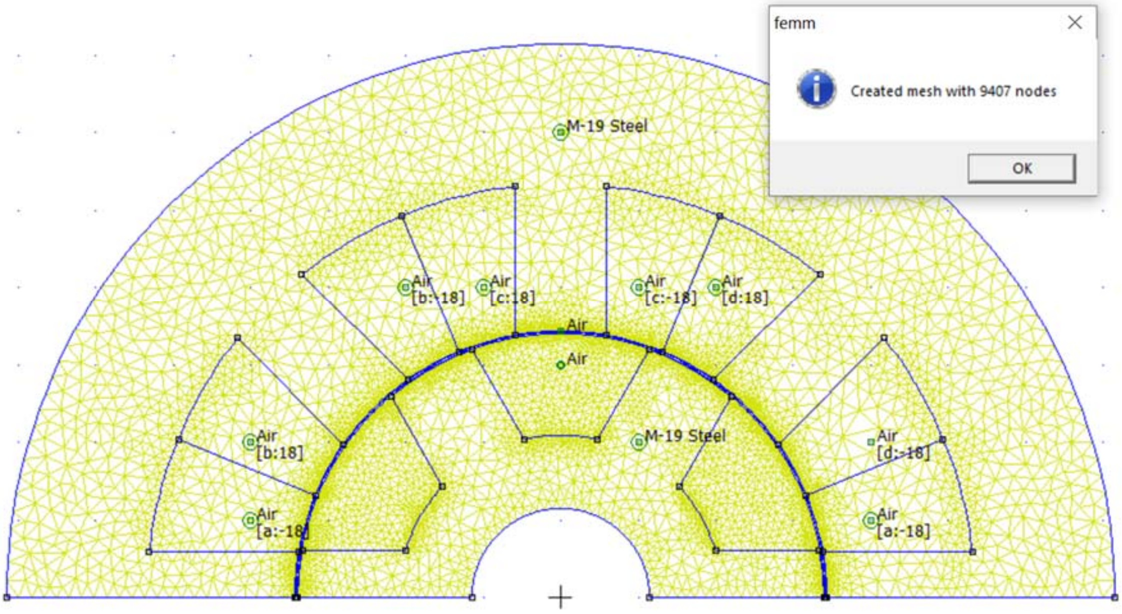


Figure 26 - Half geometry after generating the mesh.

5. Run the finite element analysis

Click on the “turn the crank” 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 particular problem, the calculations should be completed within a second. 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 27).

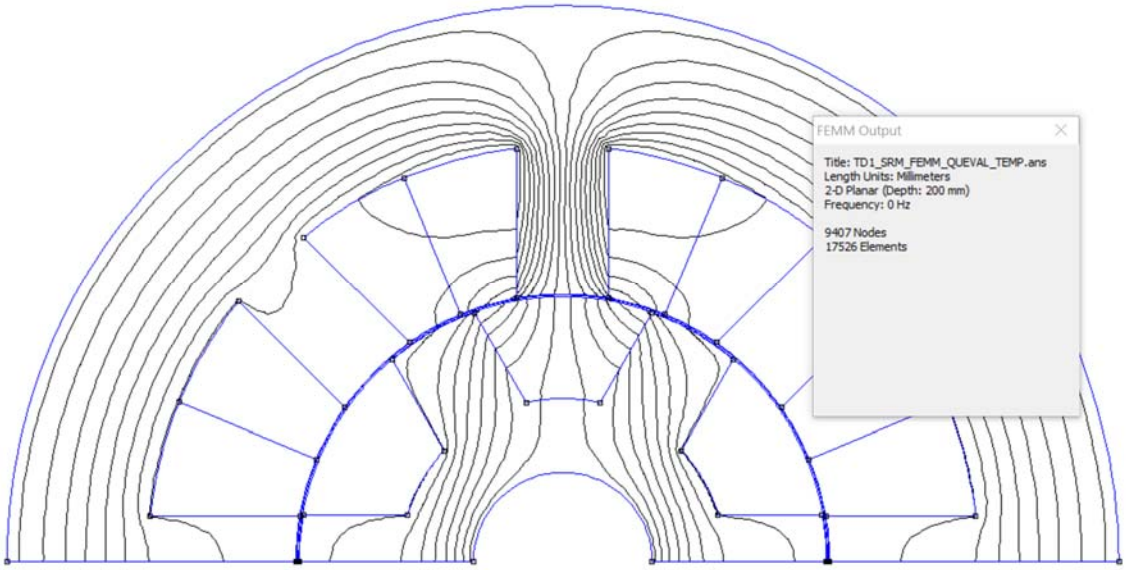


Figure 27 – Default analysis results: flux lines.

6.1 Getting 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`. 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. Similar to drawing points in the pre-processor, 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 28. You should obtain a similar result (but probably not exactly the same values).

Note: If the `FEMM Output` *dialog disappeared, select* `View > Output windows` *from the main menu.*

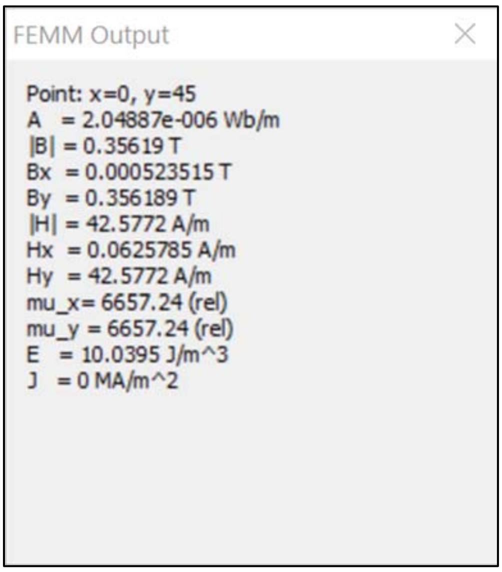


Figure 28 – “FEMM Output” dialog for the point (0, 45).

6.2 Plotting 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 the 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 the point (-12, 45), then add the point (12, 45). Then, press the `Plot` toolbar button . The `X-Y Plot of Field Values` dialog opens (Figure 29). 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 are shown in Figure 30.

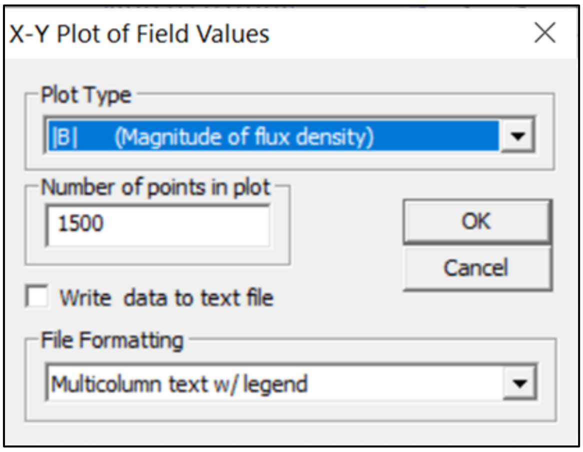


Figure 29 - "X-Y plot field values" dialog.

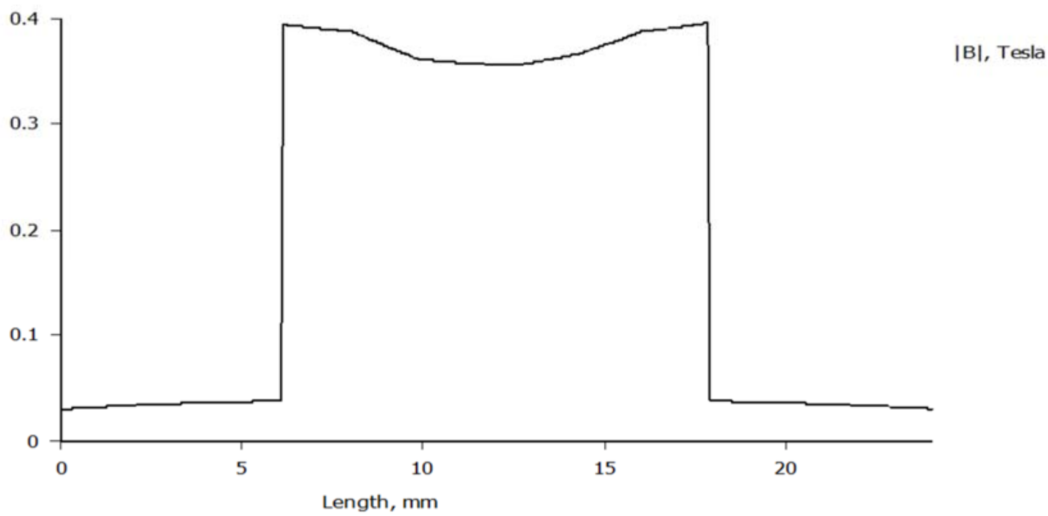


Figure 30 - Plot of the flux density along the axis of the coil c.

Note: It is often the case in the solution to magnetic problems that the field values are discontinuous across a boundary. In this case, FEMM determines which side of the boundary will be plotted based on the order in which points are added. For example, if points are added around a closed contour in a counterclockwise order (with respect to the origin), the plotted points will lie just to the inside of the contour. If the points are added in a clockwise order, the plotted points will lie just to the outside of the contour.

6.3 Plotting 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` radio button and accept the other default values. Click on `OK`. The resulting solution is shown in Figure 31.

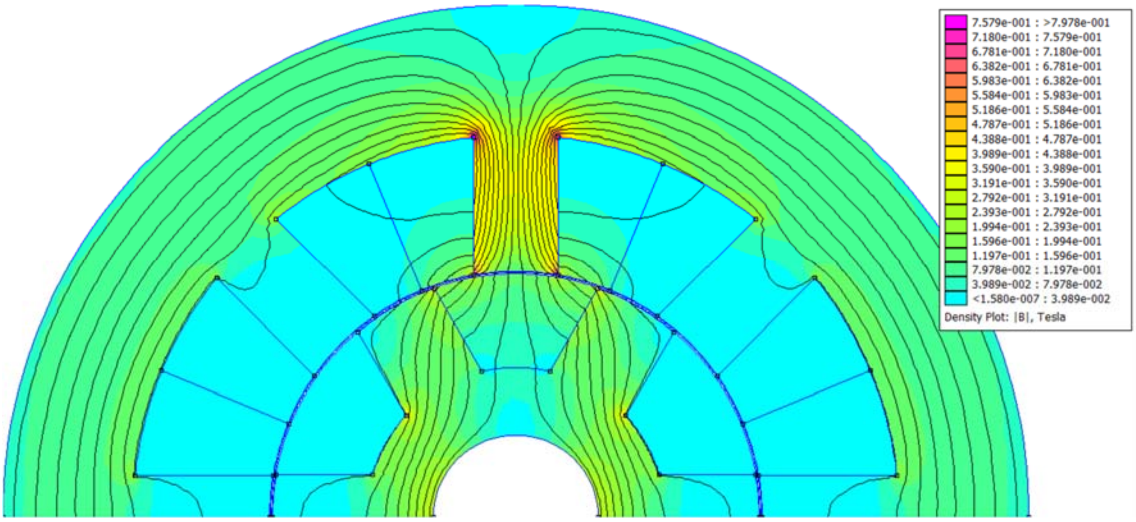


Figure 31 – Plot of the flux density distribution.

7. Conclusions

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

- Draw a geometry using nodes, segments and arcs;
- 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;
- Display color flux density plots.