

Models of Simple Iron Cored Electromagnets

Author J.Mammadov. Flat 40, 208 Plymouth Grove, Manchester, UK. email: jahangirmcr@gmail.com

Abstract: This report mainly discusses the implementation and results of a project proposal, “Modelling using Finite Element Methods”. The report is devoted to implementation, which is a model of an electromagnet. The software tool that is used to model the electromagnet is COMSOL Multiphysics, a commercial FEA package provided by the University of Manchester, Computer Science School. Additionally, the report includes other electromagnet models and their comparison with the original model.

Keywords: Electromagnet, Multi-Turn Coil, Magnetic Field, AC/DC

1. Introduction

Electric currents flowing through a wire generates magnetic field. A solenoid is a cylindrical wire that generates magnetic field B when it carries electric currents. A ferromagnetic material iron core multiplies magnetic field ten and even thousand times when it is added to a solenoid [1]. All electromagnets work with the same principle of iron core solenoid. The laboratory electromagnet has also the same working principle. There were many electromagnet designs in the laboratory differing in their geometry, power and materials. In this project the requirement was to model a simple iron core electromagnet, shown in **Figure 1**. The electromagnet has a simple geometry, which is made of two elements. It is composed of an iron core and a multi-turn coil. The electromagnet is used as part of a larger experimental apparatus, where there is a need to create a magnetic field, which can be controlled by changing the current in coils. A typical use would be to measure the magnet-optical response of novel data storage-media such as bit pattern media (BPM).

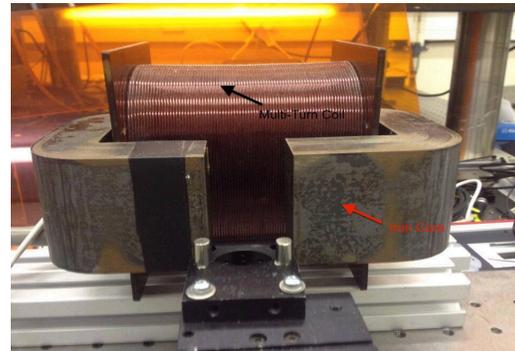


Figure 1. The laboratory electromagnet.

The electromagnet is experimented by measuring the dimensions of the coil and the iron core using a vernier calliper. Looking from top, the iron core is 165mm in length and 94mm in width. Its height and thickness are measured to be 51.5mm and 25mm. The gap in the core is 32.2mm wide. The multi-turn coil is 107mm in length and 80mm in width and its height is 124mm.

The core is made of laminated iron, a special kind of iron used in electromagnets production, which has relative permeability of $\mu_r = 200$ [1]. The coil is made of insulated copper wire. The wire has 1.18mm cross-section area and it is turned 1614 times. The electromagnet generated nearly 0.056 T of magnetic flux density in the middle of the gap applying 2A current to the coil.

2. Geometry and Materials

The geometry of the model is composed of two main elements, a multi-turn coil and an iron core. For building each part of the model a **Work Plane** is added to **Geometry** node to convert a 2D geometry drawn in the plane to a 3D object in the space. 2D geometry objects and features are added to the plane to create a 2D object sequence. All the length units used in **Geometry** are in mm, while angular units are in deg. Two rectangles and one Bezier polygon are used for building the geometry of the multi-turn coil. Firstly, a rectangle (Rectangle 1) with 80mm width and 124mm height is drawn and it is centered about the position (0,0). A smaller rectangle (Rectangle 2) with 34mm width and 58mm height is added and centered about the same position with the first rectangle. So, the two rectangles coincide. Next, by using

Difference Boolean operation smaller rectangle is subtracted from the bigger to draw a final 2D object sequence. Inner and outer corners of the object are rounded in the next step by adding two **Fillet** (Fillet 1 and Fillet 2) operations. The inner and outer corners are filleted with 4mm and 25mm circular fillet arches, respectively. Then, a line between the triangles is drawn using **Bezier Polygon** (Bezier Polygon 1).

This line is an internal boundary, which will represent an input for coil excitation during physics definition.

After two-dimensional object sequence created in **Work Plane 1**, it is converted to a three-dimensional object using **Extrude** (Extrude 1) operation. Object is extruded 107mm. It means its distance from the plane is 107mm. The second **Work Plane** (Work Plane 2) is added to **Geometry** node for drawing an iron core object sequence using three different-sized rectangles. Two Difference operations are used to subtract smaller two rectangles (Rectangle 4 and Rectangle 5) from the biggest one (Rectangle 3). Rectangle 4 is 114.2mm in length and 44mm in width. Rectangle 5 is 32.2mm in length and 40mm in width while Rectangle 3 is 164.2mm in length and 94mm in width.

After subtraction operations, inner and outer corners of the object except the corners surrounding the gap are rounded. The inner corners are filleted 10mm, while the outer corners filleted 25mm each. Finally, 2D core is extruded 51.5mm. After creating the coil and the core on different planes, they are repositioned in space to form a proper electromagnet figure. The only way to reconfigure the object sequence is to relocate the objects on different planes and move them to different directions using XY coordinates. The **Work Plane** of the multi-turn coil (Work Plane 1) is located on YZ plane and has X and Y displacements of 132mm, 25.5mm, respectively. The plane (Work Plane 2), where the core was drawn is relocated on XY plane and has -47mm and -1mm of X and Y displacements.

It is recommended to add a sphere (Sphere 1) to the geometry and put the electromagnet inside. The added sphere will be filled with air during material allocation in order to simulate room environment. This would be important when considering thermal effects such as thermal heating of the electromagnet. However, in the

work these effects are not taken into account and it remains as a future goal.

Although, these 3 objects are positioned properly to model the laboratory electromagnet, COMSOL considers them as three unrelated objects. We need to specifically define those objects as a single object by calling **Form a union** built-in operation under **Geometry** node. The software then forms a union from all geometry objects. The union is divided into domains, separated by boundaries according to the participating geometry objects [2]. It is also possible, but often not necessary to specify boundary conditions on interior boundaries among domains in the geometry [2]. COMSOL ensures continuity in the physics fields across interior boundaries by default. Uniting the objects is the last step in forming the geometry, which results in **Figure 2**.

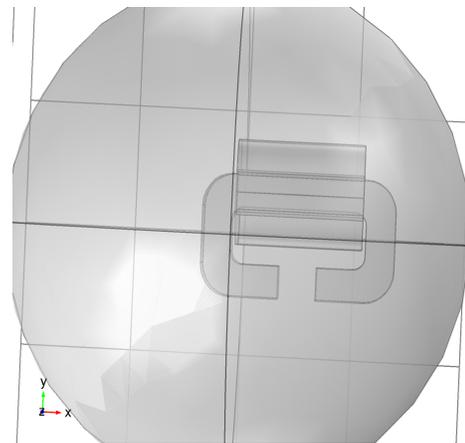


Figure 2. Complete geometry of the model, an electromagnet in a sphere.

According to COMSOL's geometry statistics the model contains 3 domains, which are built from 51 boundaries. Domain 1 is sphere, while Domain 2 and Domain 3 are the iron core and the multi-turn coil, respectively.

After creating the geometry of the model, the second step in the process is to assign materials to each object. The sub-nodes under **Materials** are used to add predefined or user-defined materials, to specify specific material properties using model inputs, functions, values, and expressions or to create a custom material library

[2]. In the model, domains are used to assign materials to the objects.

Materials are grouped according to physics interfaces in COMSOL. For assigning materials to the multi-turn coil, the iron core and the sphere, three materials are chosen from the built-in materials group. COMSOL even provides the functionality to change, remove or add properties to materials and this functionality is used to make the materials more similar to the material properties of the laboratory electromagnet. Copper with relative permeability of $\mu_r = 1$ and relative permittivity of $\epsilon_r = 1$ are assigned to the multi-turn coil, while soft iron (with losses) is assigned to the core. Soft iron is used for different purposes in industry, that's why some properties, especially, relative permeability of the material is not given initially by COMSOL and requires the user to define them. Relative permeability of soft iron can vary depending on the application for which it was produced. Soft iron with relative permeability of $\mu_r = 200$ is the material mainly used in electromagnets [1]. The user must define it manually in COMSOL. Finally, the sphere object is filled with air to imitate the laboratory environment during simulation.

3. Physics Interface-Magnetic Field

The *AC/DC module* in COMSOL is widely used by engineers and scientists to understand, predict and design electric and magnetic fields in statics and low-frequency applications [3]. The *AC/DC module* includes stationary and dynamic electric and magnetic fields in two-dimensional and three-dimensional spaces along with traditional circuit-based modelling of passive and active devices [3]. All modelling formulations are based on Maxwell's equations [3]. The *AC/DC module* supports modelling with its various physics interfaces [3]. *Magnetic field* is one of the physics interfaces under the *AC/DC module*, which allows users to compute magnetic field and induced current distributions in and around coils, conductors and magnets [3]. When *Magnetic Field (mf)* physics interface is added to the model, three nodes, *Ampère's Law*, *Magnetic Insulation* and *Initial Values* nodes are automatically added under the interface to

define the basic principles and equations to compute the magnetic field.

The *Ampère's Law* node adds Ampère's law for the magnetic field and provides an interface for defining the constitutive relation and its associated properties as well as electric properties [3].

Domain selection for *Ampère's law* node is predefined by the parent node (Magnetic Field interface) and cannot be changed. As all the domains were chosen in the physics interface for the electromagnet model, equations will be calculated for each domain. Users are required to choose and define some fields in the node to customize Ampère's law properties to their needs. These fields are "Model Inputs", "Material Types", "Coordinate System Selection", "Conduction Current", "Electric Field" and "Magnetic Field" with their subfields. In the current model (a simple iron core electromagnet) "Temperature", "Absolute pressure" and "Magnetic Flux Density (B)" are variables, which were set as model inputs for these simulations. "Temperature" and "Absolute pressure" are included as 293.15K and 1atm respectively. "Magnetic Flux Density" subfield is entered as an initial guess of simulation results or a good start point for solvers. No value is entered to this subfield. Coordinate system is selected as "Global Coordinate System", while "Material Type" selected as 'From material'. In this context, "Material Type" decides how materials behave and how material properties are interpreted when the mesh is deformed. "From material" is chosen to get the corresponding properties from the domain materials. "Conduction Current" defines "Electrical Conductivity σ (SI unit: S/m)" for the model, and chosen to be picked up from the material properties. "Electric Field" gets "Relative Permittivity" from material properties as well. "Magnetic Field" specifies constitutive relation that describes the macroscopic properties of the medium (relating the magnetic flux density \mathbf{B} and the magnetic field \mathbf{H}) and the applicable material properties, such as relative permeability [3]. "Constitutive relation" is specified as "Relative permeability", which is obtained from the material properties. "Magnetic Insulation" is another component under the physics interface, which is added automatically according to the default settings. It sets magnetic vector potential

to zero at the selected boundaries. As other default nodes it inherits its selection from “Magnetic Field (mf)” parent node. Thus, all boundaries are selected, but insulation is not applicable to the boundaries constituting the coil and the core. Only the sphere is insulated and magnetic potential at its boundaries vanishes. Magnetic vector potential of the electromagnet (coil and core) cannot be zeroed, because by default the interface calculates magnetic field for those domains when the user assigns proper materials (iron and copper). “Initial Values” node is provided by the “Magnetic Field” interface to add initial values for “Magnetic Vector Potential A (Wb/m)” that can serve as an initial value for the simulation results or a good guess for the non-linear solver [3]. Default XYZ components of the vector are 0 Wb/m and they are unchanged for this model, too.

3.1 Multi-turn Coil

A **Multi-Turn Coil** represents the current carrying coil (Domain 2) and as the name suggests it consists of a strand of Copper wire coated with an insulator. Shorting does not occur between conductors due to insulation [3]. Current flows along the wire and is negligible in other directions [3]. The interface requires the selected domain to have magnetic and electric properties in order to be treated as a coil. This node also has fields and properties that are specified by the user. **Multi-Turn Coil** node is the most critical part in defining the physics interface, because the accuracy of the results is highly dependent on this. The most important part in the **Multi-Turn Coil** node under the interface is to specify the type of the coil. COMSOL provides three coil types: Linear, Circular and Numeric. Users are allowed to define the direction of the wire as a vector field and the length of the coil, if they select “User Defined” option under “Coil Type” field. Users need to choose a proper coil type. Otherwise, COMSOL can fail to solve the equations for the simulation and may produce erroneous results. Coil current direction is the only reason that, the node offers three coil types. So, current can flow straightly, circularly or the direction can be calculated in the Study step. It is suggested to select “Numeric coil” type while assigning physics to the coil, because it is the general form of all coil models in COMSOL.

Linear and Circular coils are the special cases where the coil is straight and circular. After examining the geometry of the coil, Numeric coil type is selected to define the multi-turn coil domain in the model.

After coil type selection, values are included to “Number of Turns”, “Coil Conductivity”, “Coil cross-section area” and “Coil Excitation” fields in order to specify the parameters of the coil that will be used for calculations. “Number of Turns” are 1614 in the model as it is in the real electromagnet. “Coil Conductivity” for wire is entered as 6×10^7 [S/m], which is the conductivity of copper. The cross-section area of the wires is defined as “User Defined” and entered to be 1.18mm. COMSOL uses “Coil Conductivity” and “Cross-section area” to compute coil resistance. The current density flowing in the coil domain is computed from a lumped quantity that constitutes the coil excitation. [3] The coil can be excited either by current excitation or voltage excitation. In this case, current excitation is selected and “Coil Current” is entered as 2A for the very first simulation results.

4. Meshing and Study

After defining the physics interface for the model, the next step in the process is mesh creation. Meshing geometry is an essential part of the simulation process, and can be crucial for obtaining the best results in the fastest manner [4]. The geometric model is divided into thousands of tiny finite elements, which can be in different shapes. The elements constituting the model mesh are mostly in tetrahedral shape, pyramid like figure. COMSOL offers two mesh sequence types: “Physics-controlled mesh”, “User defined” meshed. “Physics-controlled” mesh is preferred for the model due to the simplicity of the geometry. “User Defined” mesh sequence types is usually preferred when a model has a complex geometry. By selecting physics-controlled mesh as the mesh sequence type, the mesh is adapted to the current physics settings in the model. The software automatically selected “Normal” element size for the mesh. Using normal-sized elements the mesh is built from 233592 elements.

Study step is the sixth step in COMSOL after mesh creation. Equations and data specified in the previous steps are solved in this step to give the simulation results. In electromagnetics, **Stationary** step is used for calculating static electric and magnetic fields, as well as direct currents. In the model study **Stationary** step is used for calculating magnetic field. It is highly recommended not to forget **Coil Current Calculation** step while solving the equations, because it computes the current of a **Multi-Turn Coil** domain and produces a current density corresponding to a strand of wire [3]. **Coil Current Calculation** study step is only available for 3D models using **Magnetic Field** interfaces and **Multi-Turn Coil** domain nodes. Added **Automatic Current Calculation** sub-node to the **Multi-Turn Coil** domain sets automatic calculation of the current flow in the coil domain. The boundary conditions of **Electric Insulation** and **Input** provide the needed data for **Coil Current Calculation** to solve the equations. After **Study** steps (Coil Current Calculation, Stationary) added, the computation can begin.

5. Results

Results branch contains two **Solution** data sets for the model, because two **Study** steps, **Coil Current Calculation** (Eigenvalue Solver) and **Stationary** studies computed the model. 2A of current applied to the coil for the first simulation. **Figure 3** shows the simulation results for the modeled iron core electromagnet.

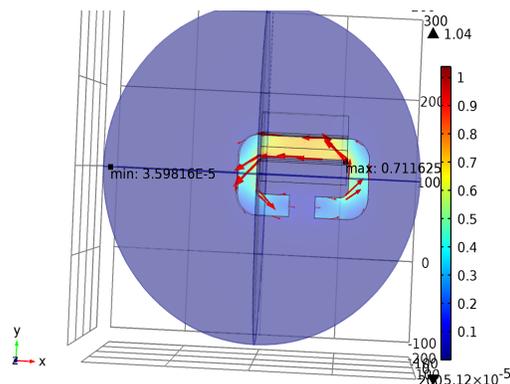


Figure 3. The simulation result of the model electromagnet. Image shows magnetic flux (T) density the model generates

Magnetic flux density (T) is low in blue areas, outside the immediate vicinity of the iron core and coil. A residual of 2mT of magnetic flux density can be measured in those areas, as expected from a simple interpretation of Maxwell's equations. The magnetic flux density is quite high at meeting points of the coil and iron core where it ranges between 0.6 T and 0.7 T. The magnetic flux density is diminishing in the arms of the core, because those parts are far from the coil.

The main concern for the study is the amount of magnetic flux density (T) the electromagnet generates in the middle of the gap. The model electromagnet generates (0.056 ± 2) Tesla magnetic flux density at the midpoints of the gap. The maximum amount of magnetic flux density is nearly 0.7 T, which is in the inner right corner of the iron core, where it meets with the coil. This is also in agreement with Maxwell's laws where a geometric discontinuity leads to magnetic flux concentration. Using this property we sharpened the gap field in other models, discussed in the next chapter, to get more magnetic flux density than the base model. It is not a surprise that, the minimum magnetic flux density is in the far corner of the sphere as it is consistent with Maxwell's equations.

Moreover, other properties of the electromagnet and the coil itself can be found using COMSOL's useful simulation features. According to the simulation results, the coil has 5.54Ω of resistance, when 2A of current applied to it. In addition to resistance, the simulation shows that current density in the coil is $9.14 \times 10^5 \text{ A/m}^2$. Nearly 1300 J/m^3 of energy density is generated in the gap field of the iron core according to the results.

Current (A)	Magnetic flux density (T)	
	Laboratory electromagnet	Model electromagnet
2A	0.056 ± 0.002	0.057 ± 0.002
3A	0.083 ± 0.002	0.084 ± 0.002
5A	0.139 ± 0.002	0.140 ± 0.002
10A	0.275 ± 0.002	0.276 ± 0.002

Table 1. Magnetic Flux Density generated by the laboratory and model electromagnets while 2A, 3A, 5A and 10A of current applied.

The model electromagnet generated the same amount of magnetic flux density in lower currents as the laboratory electromagnet. Although, it behaves like the real electromagnet in lower currents, the accuracy of the model cannot be verified with only this. In order to verify that the model behaves as the real electromagnet it is simulated three times by conducting different amounts of electric current to the coil. The coil is conducted 3A in the first, 5A and 10A in the second and the third simulations respectively. Possible magnetic flux density the model and laboratory electromagnets can generate by applying 3A, 5A and 10A current is given in **Table 1**. The table verifies that the model electromagnet is designed properly, because both electromagnets give the same results under the same conditions.

6. New electromagnet Designs

Possible electromagnet models were evaluated and as a result four electromagnets with different geometries are proposed to test their suitability as laboratory electromagnets. Three of them are quite similar to the original model and have some changes on the dimensions and the geometry surrounding their gap fields. But, one of them is quite different in the shape. It has two separate coils on the iron core. This model required making some alterations on the physics interface applied and the geometry drawn. In the last section newly designed models are compared with the base model and the laboratory electromagnet. The main purpose is to find a model that generates more magnetic flux density than the real electromagnet. Of course, the newly designed models are not expected to

have big differences in magnetic flux generation. However, small differences will certainly occur.

To design the electromagnet with two coils, the coil of the original model is divided into two pieces. It means all the properties of the original coil are divided into two, to create two coils out of one. As it is seen in **Figure 4**, there are two coils on an iron core. The coils are created in the same way with the coil of the original model, but different numbers are used for parameters.

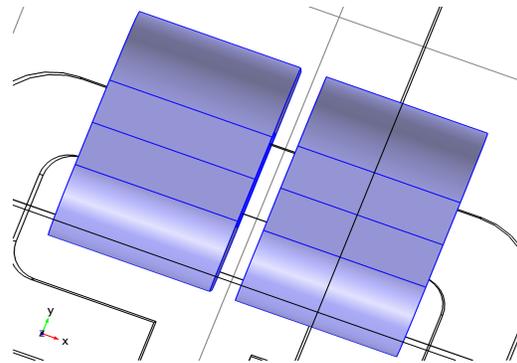


Figure 4. Proposed geometry of the electromagnet with two coils.

Each coil is created using the same techniques and the same simple objects. **Magnetic Field (mf)** interface is added to define physics for the final geometry as it was in the original model. However, two multi-turn coil domains are used each with 807 turns. In total, 1614 turns. Both coils are excited with 2A current for the first results. In **Study** phase, two **Coil Current Calculation** study steps are added to a **Stationary** step, because the current model has two separate coils and each requires calculating their equations separately. The model generates about $53 \pm 2\text{mT}$ of magnetic flux density at the midpoints of the gap field when each coil is excited with 2A current. Moreover, the coils have 2.77Ω of resistance each and $9.14 \times 10^5 \text{ A/m}^2$ of current density together.

Remaining electromagnet models are designed by making alterations on the geometry of the original model, mainly on the gap field. Firstly, the gap field of the electromagnet is rounded to examine whether a change happens in magnetic flux density generated there. The gap is rounded. However, rounding the gap does not make a big difference in the amount of magnetic flux density generated in the gap. The model

produces $55 \pm 2\text{mT}$ of magnetic flux density in the middle of the gap. According to the results it is easily seen that, if we design an electromagnet with rounded gap, we will not be successful in our aim of producing more powerful electromagnet than the laboratory electromagnet.

In an alternative model the gap is chamfered 8mm by using *Chamfer* built-in operation provided by the software. Chamfered electromagnet produced nearly 0.052 T of magnetic flux density in the gap. Moreover, the gap field without any geometry change reduced 5mm. It is expected to produce more magnetic flux density than the original model, because the gap is smaller. 5mm reduction in the distance resulted in 0.019 T of increase in magnetic flux density produced in the gap. Now the gap is producing approximately 0.075 T of magnetic flux density.

Simply reducing the gap can remarkably strengthen the electromagnet making 0.019T of change when 2A of current is applied. The difference is larger in higher currents as it is interpreted in **Table 2**.

However, it is known that geometric discontinuity leads to magnetic flux concentration according to Maxwell's laws. It means that the electromagnet with a chamfered gap should produce the highest magnetic flux density among the models. In order to prove that idea, the chamfered gap was reduced along with the electromagnet with the smallest gap.

Chamfered electromagnet began to generate more magnetic flux density than the simply reduced one when the gap was 18mm and below. For 10A current applied, chamfered electromagnet gives 0.53T of B field, while the simply reduced one gives 0.505T.

We can draw a conclusion from the experiments driven that, chamfered electromagnet with $\leq 18\text{mm}$ gap generates more magnetic flux density than other electromagnets. Although, there is no big difference in the amount of B field produced by the electromagnets in low currents, conducting 10A of current clears all the doubts and makes it easy to identify the strongest electromagnet.

Current	Magnetic Flux Density norm (Tesla)				
	Laboratory	Model	Model with 2 coils	Gap chamfered	Gap reduced
2A	0.056 ± 0.002	0.057 ± 0.002	0.053 ± 0.002	0.052 ± 0.002	0.075 ± 0.002
3A	0.083 ± 0.002	0.084 ± 0.002	0.080 ± 0.002	0.079 ± 0.002	0.111 ± 0.002
5A	0.139 ± 0.002	0.140 ± 0.002	0.134 ± 0.002	0.133 ± 0.002	0.185 ± 0.002
6A	0.275 ± 0.002	0.276 ± 0.002	0.266 ± 0.002	0.260 ± 0.002	0.365 ± 0.002

Table 2. Magnetic flux density norm. produced in the gap fields of the models

References:

1. I.S. Grant, W.R. Phillips. Electromagnetism. University of Manchester; 1976
2. COMSOL Multiphysics Reference Guide. Version: COMSOL 4.3a Date: November 2012
3. COMSOL Multiphysics The AC/DC Module. Version: COMSOL 4.3 Date: May 2012
4. Andrew Griesmer, Size parameters of Free Tetrahedral Meshing in Comsol Multiphysics. 30 Jan 14 [cited 25 Apr 14] COMSOL BLOG [Internet]