3D Magnetostatics With ElmerFEM

Richard Crozier

July 20, 2016

Contents

1 Introduction **1**

2 Cylindrical Magnet in Air **2**
- 2.1 Creating the Geometry in FreeCAD 2
- 2.2 Meshing the Geometry in Salome 5
- 2.3 Converting Mesh for Elmer 6
- 2.4 Creating the Case File for Elmer and Solving It 6
- 2.5 Viewing the Results in Paraview 6

3 A Current Carrying Coil in Air **7**

4 Flux Linkage and Mutual Inductance In a Pair of Iron Cored Coils **7**

5 A Permanent Magnet Machine With Symmetry **7**

6 Permanent Magnet Machine With Sliding Air Gap Mesh **7**

1 Introduction

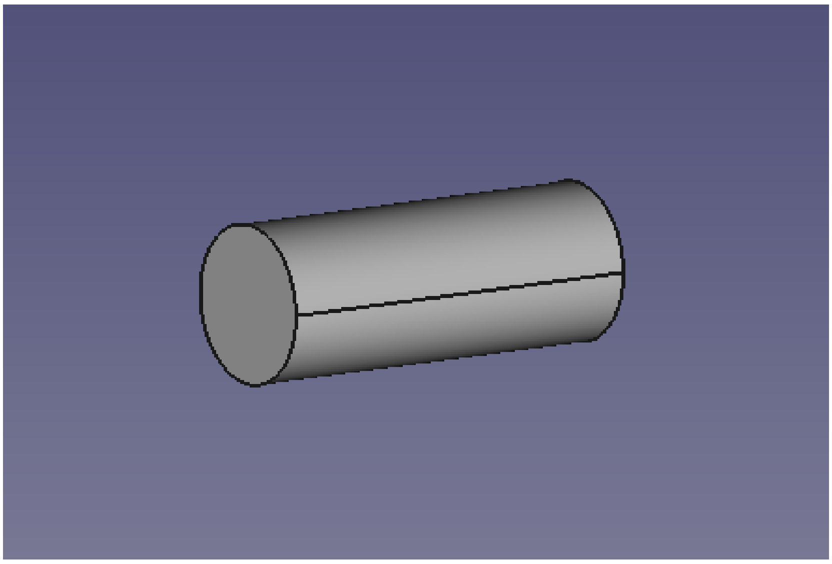
This tutorial describes the full steps to go from nothing to a 3D magnetic simulation using Elmer FEM. This is done in the the following steps:

1. Geometry Creation in FreeCAD
2. Meshing in Salome (or Salome Meca)
3. Converting mesh for Elmer
4. Creating the case file for Elmer and solving it
5. Viewing the results in Paraview

As can be seen, unfortunately quite a few different programs are involved, at least to do the various steps manually. Once you get a grip of each step, however, many tasks can implemented in python or scripted to automate them, making the process a bit smoother.

2 Cylindrical Magnet in Air

The first problem we are going to solve is that of a simple cylindrical bar magnet in free space that looks like this:

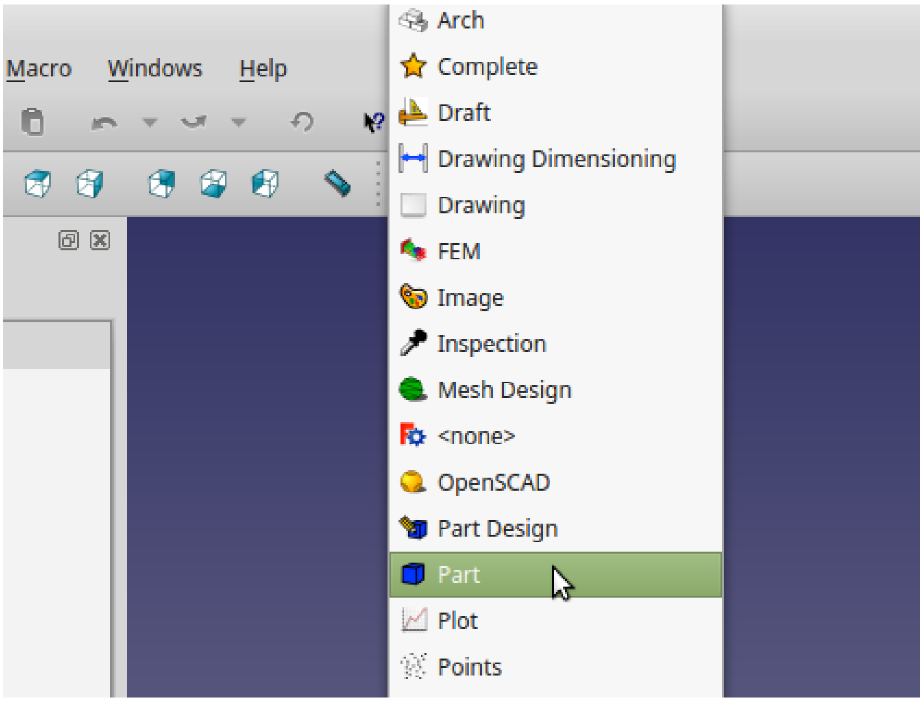


The magnet will be 20mm in diameter, 50mm long and will be magnetised along it's axis. To model this, we will place it in a box of air, to approximate the infinite space region around the magnet.

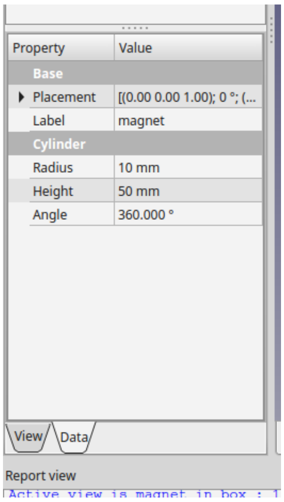
2.1 Creating the Geometry in FreeCAD

[url=http://www.freecadweb.org/]FreeCAD[/url] is a powerful free and open source parametric modelling program based on the industrial grade OpenCASCADE geometry modelling kernel. We could create the geometry diectly in Salome if we wanted, but for anything other than very simple object FreeCAD makes life much easier. Our geometry in this case is very simple, a small cylinder inside a box. Start FreeCAD, and create a new model with File=¿New.

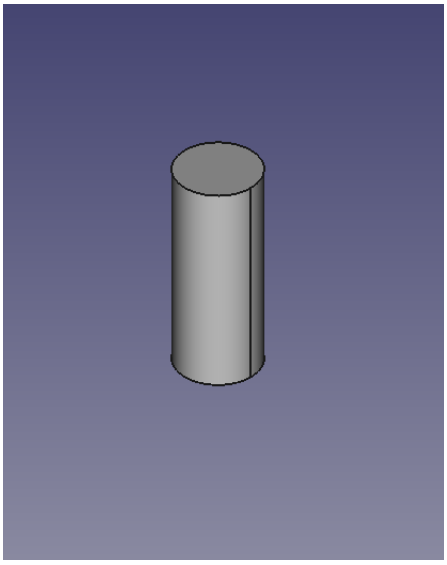
Activate the 'Part' module by choosing from the drop-down menu.



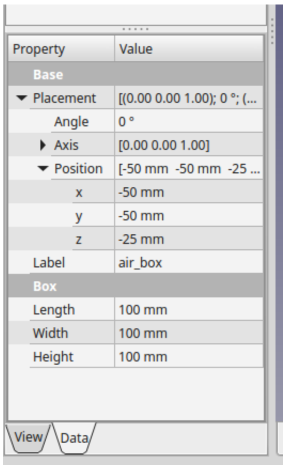
This adds some tools to the toolbar. We can create a cylinder by clicking on the little yellow cylinder icon. This creates a cylinder with default sizes. We will change the size by selecting the cylinder and going modifying properties on it's 'Data' tab:



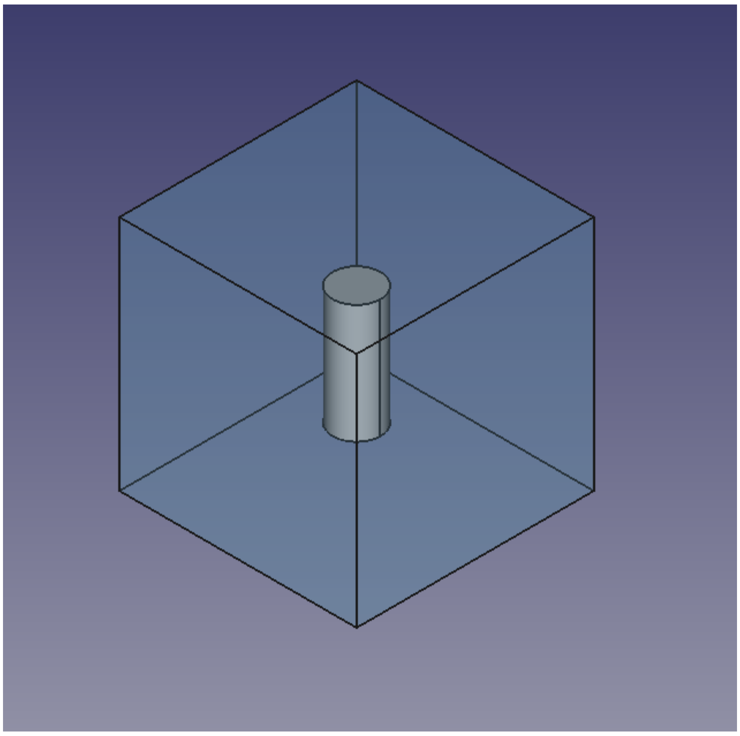
This should leave us with something like the following, a cylider with one face on the XY plane and it's centre on the origin.



Now we need to make a box of air to put the cylinder into, do this by selecting the ’Create a cube solid’ tool. This creates a new cuboid with one face on the XY plane, and one corner on the origin, with default dimensions. Set the Length, Width, and Height to all be 100mm using the Data tab for the box in the same way as was done for the cylinder. Then, expand the ’Placement’ property, also in the Data tab, and then expand the ’Postiion’ property. Change the ’x’ and ’y’ position to both be -50mm, and the ’z’ position to be -25mm. This makes the box fully overlap our magnet.



Now we can’t see the cylinder any more as it overlaps the box. To make everything easier to see, make the box transparent by going to it’s ’View’ tab. Scroll down to the ’Transparency’ property and change to 75. You will now be able to see the magnet inside the box.



This is our geometry complete. With our geometry created, we now have to mesh it. This will be done in another program, Salome, and to get it into this we must first export the geometry to a format Salome can import, in this case STEP. To export the objects to a STEP file, simply select both object in the model browser and then choose File=¿Export and choose the 'STEP with colors' file type. The advantage of STEP is that the model units of FreeCAD, which are mm by default are saved in the file, and can be correctly imported by Salome when we load it again.

2.2 Meshing the Geometry in Salome

1) load the objects from file

2) select both solids and execute partition

3) select the Partition, explode to solid

4) go to mesh module, build a mesh on the partition

then apply different rules on the solid-meshes: 5) select the partitions mesh, create subMesh, Geometry: select an exploded solid of the partition Do this for all solids special rules are wished for 6) change submesh priority, sort in the desired sequence (e.g. the finer mesh above the coarser ones, this enforces the finer rules to be applied on the interface)

create mesh groups and export

7) select the new mesh ¿ create Groups from geometry and select the from the exploded solids of the partition -¿ in the mesh there now exists a sibling groups of volumes 8) rename the groups to something sensible we can use later in Elmer 9) select the mesh, rightclick, export mesh to e.g. UNV file for Elmer FEM

2.3 Converting Mesh for Elmer

Elmer doesn't natively support the unv mesh format, but we can use `ElmerGrid` to convert the unv file to Elmer's mesh format. Elmer's mesh format is not a single file, but rather a directory containing a collection of files. The command to convert a unv file to Elmer's format is:

```
ElmerGrid 8 2 <file name>.unv
```

The first number is the input file format, the second is the output file format, and these are followed by the name of the file to be converted. So in our case we run

```
ElmerGrid 8 2 elmer_3d_magnet_mesh.unv
```

There should now be a new directory created called `elmer_3d_magnet_mesh` containing the files `mesh.boundary`, `mesh.elements` `mesh.header`, `mesh.names` and `mesh.nodes`. The file `mesh.names` is useful to us, as it contains the names we assigned earlier to the groups of volumes and faces, the contents of the file in our case looks like this:

```
! ----- names for bodies -----
$ magnet = 1
$ air_box = 3
! ----- names for boundaries -----
$ air_box_surface = 4
$ bnry5 = 5
```

By setting the appropriate variables in our Elmer case file, we can use these names instead of having to figure out what numbers have been assigned to the mesh groups.

2.4 Creating the Case File for Elmer and Solving It

2.5 Viewing the Results in Paraview

1) Open the output (vtu) file

2) Apply Glyph filter choose magnetic flux density turn off vector scaling color by flux density

Viewing mesh cross-section

Clipping

3 A Current Carrying Coil in Air

4 Flux Linkage and Mutual Inductance In a Pair of Iron Cored Coils

5 A Permanent Magnet Machine With Symmetry

6 Permanent Magnet Machine With Sliding Air Gap Mesh