Exercise 16: Magnetostatics

Magnetostatics is part of the huge field of electrodynamics, founding on the well-known Maxwell-equations. Time-dependent terms are completely neglected in the computation of the field values. Therefore, only static magnetic field components arise as a consequence of direct currents or permanent magnets. As an example, the magnetic field generated by a coil can be described by magnetostatics.

Goal

Computation of the magnetic flux density **B** generated by a set of coils. One simple, but powerful configuration is that of a Helmholtz-coil, a serial setup of two coils to generate a homogenous magnetic field in between.

Hint

As this is your first usage of *COMSOL*, this will be a step-by-step manual. Feel free to adjust some points at the end of your work. Saving the project after some modification can prevent you from doing things twice!
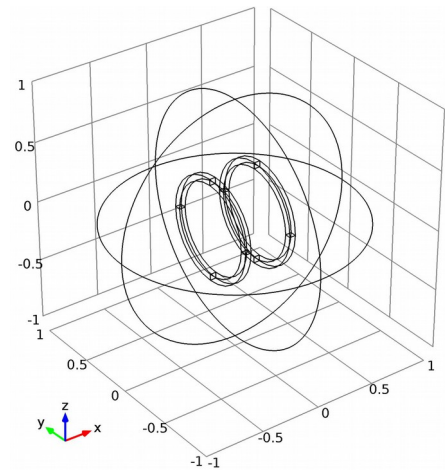
Steps

1. Run COMSOL 4.4 and make sure that the text "Academic Classkit License" appears on the start screen.

2. Creating the project:

    *   Choose *model wizard* to get help for the upcoming choices.

    *   Our problem will be in *3D*. (In some problems, some dimensions do not include new information. For these problems, you can save computational time, reducing the model from 3D to 2D or 1D.)

    *   On the next tab, you have to select the physics you want to calculate. As we like to see the magnetic field, choose the *magnetic fields (mf)* module in the group *AC/DC* and click on *add*.

    *   Now the program asks for a study. Here you have to decide between time-dependent, stationary or frequency domain problems. As we want to solve a static problem, select *stationary* and push *done*.

    *   The main project window appears, which is divided into some subareas. The model builder presents the main steps of any simulation tool. Surely you can easily get the connection to the same points in Ansys Workbench.

3. Global definitions:

    *   To make life easier, use global definitions as much as possible. For example, every coil of our problem should have the same value for the current. If we want to increase the

value without a global definition, every coil would have to be selected and updated manually.

- Right-click on *global definitions* and choose *parameters*.

- Select *parameters* and add a new parameter for the standard current by writing *I0* in the field *name* and *0.5[mA]* in *expression*. The value field should be filled automatically by leaving the row.

- Do the same for coil radius *rC = 0.4[m]* and conductor width *wC = 0.05[m]*

4. Geometry definition:

- As you know from Ansys you can also define volume bodies in *COMSOL* directly. This feature is relatively intuitive and you can use it in the second part of the lab. Now we want to point out the way with sketches as we have done it in Ansys.

- At first, right-click on *geometry* and put in a new *work plane (wp1)*. Work planes are pretty similar to the sketch interface from Ansys.

- Right-click on *plane geometry* and add a *square (sq1)* with the properties:

  ○ *side length = wC*

  ○ *base = center*

  ○ *xw = rC/2*

  ○ *yw = rC*

- Right-click on *sq1* and duplicate the selection to generate a *square (sq2)*.

- Change the property *xw* of *sq2* from *rC/2* to *-rC/2*.

- Select *geometry* to get back to the 3D-view and click on *build all*. Now you see the sketch on the 3D-view.

- Add the object *revolve* from the geometry menu to the problem and select the new item.

- Change the *direction of revolution axis* to the **x** axis direction and select *wp1* for the *input objects*. Find the part *selections of resulting entities* and create a new selection named *coils*.

- *Build all objects* to create the two coils by revolving the two squares.

- As COMSOL cannot do the field calculation for an infinite area, we have to generate a computation area. Therefore add a *sphere (sph1)* with *radius = 1[m]* and *position = (0 / 0 / 0)*. Also create a new selection as before and name it *compArea. Build all objects*.
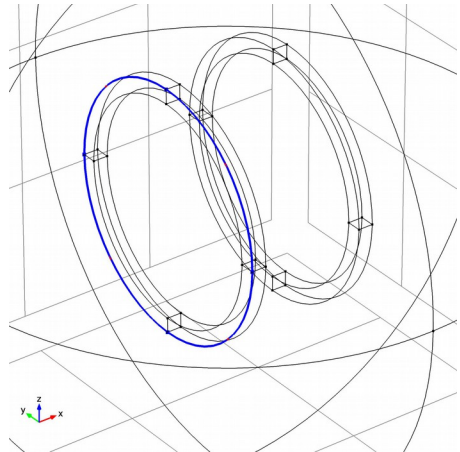
5.   Materials definition:

- Right-click on *materials* and *add material*. A column opens in the window, where you can select different materials.

- Open the node *built-in*, where you find the most typical materials. Add the materials *air* (computation area) and *copper* (coils) to the component.

- Click on *copper* to see the material options.

- In the part of *geometric entity selection*, select the two coils by selecting *coils* in the drop down menu of *selection*.

  HINT: For manual selection like in Ansys, playing with the settings at the graphics area can help.

- For the material *air*, select the big *sphere (sph1)*.

6.   Physics definition / magnetic fields:

- Three predefined items (*ampère's law*, *magnetic insulation* and *initial values*) are included for the magnetic fields computation. For *ampères's law*, one should change the *material type* to the *from material* option.

- Right-click on *magnetic fields*. The pop-up menu presents you volume type selections as well as face type, edge type and point type selections. As the geometry of the coils is defined as the full 3D geometry, insert a *multi-turn coil* from the volume type selections.

- Click on *multi-turn coil* and select one of the two coil geometries. For this coil, set *material type* to *from material* and *coil current = I0*. As the coil is circular, define *coil type = circula*r.

- One might be surprised how *COMSOL* knows the direction of the coil current. That's not the case, so you have to define a *reference edge*. Right-click on *multi-turn coil* and add a *reference edge*. On the *edge selection* frame, select and delete all entries. Select four circular edge lines of the coil as you can see on the picture to the right.

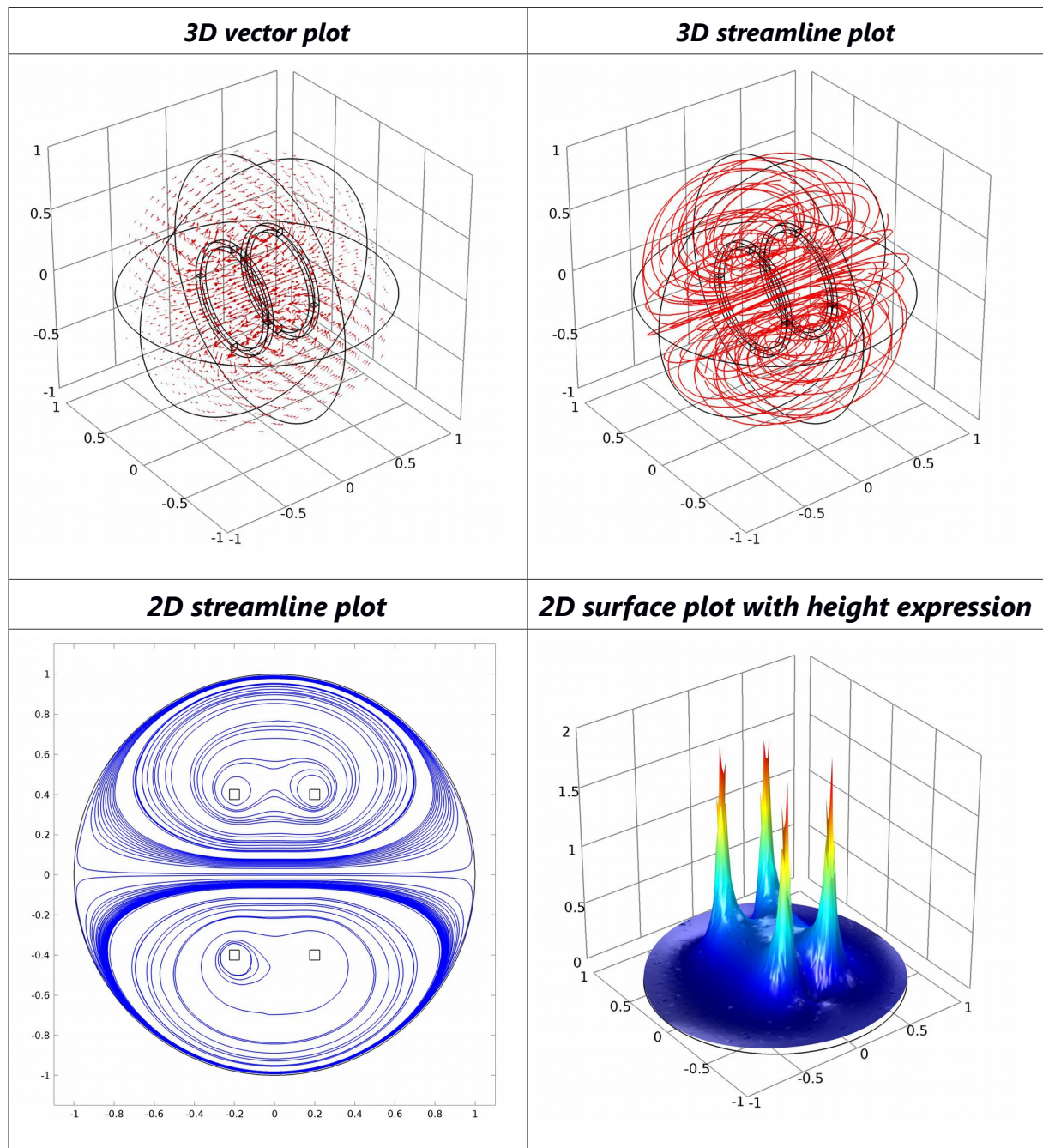- Repeat the last two steps for the second coil.

7.   Mesh:

- The basic ideas of refinement options are more or less the same as you know from Ansys. For your first study, using *sequence type = physics-controlled mesh* and *element size = normal* will already yield good solutions. Click on *build all* to generate the mesh.

- On the messages tab on your window, the following statistic should be displayed:

  - ➢ *Complete mesh consists of 11627 domain elements, 1260 boundary elements, and 336 edge elements.*

- The number of elements is one of the essential information you should check before starting the computation!

8. Study:

- Use the standard entries and right-click on *step 1: stationary*. From the pop-up menu, start the computation by one click on *compute selected step*.

9. Post-processing (results):

- It is often helpful to first visualize the results of a volumetric 3D problem as a 2D intersection through the domain. Right-click on *data sets* and add a *cut plane*. As *data set* define *solution 1*, *plane = xy-planes* and *z-coordinate = 0*.

- Right-click on *results* gives you the opportunity to add a *3D-plot group, 2D-plot group* and a lot of other nice stuff.

- A *2D-plot group* for example, can plot the values on the defined *cut plane*.

- Play around and generate some nice pictures as you can see below...
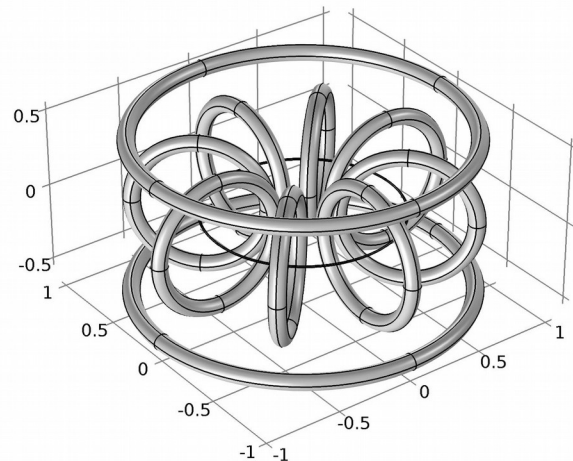

Questions & Jobs

1. Verify your simulation results by checking, whether the magnetic flux density streamlines are actually parallel to each other.

2. Play with 2D and 3D plots to generate pictures, which point out this characteristic.

3. As Helmholtz pointed out, we should set the distance of the two coils equal to the radius of the coils. Check the behavior of the parallelism for other ratios.

4. Check out the influence of changing the extent of the computational area.

Plots of the magnetic flux density calculated for the Helmholtz coil.

3D vector plot	*3D streamline plot*
2D streamline plot	*2D surface plot with height expression*

Tokamak

The Tokamak configuration of a nuclear fusion reactor is one way to achieve the required complex magnetic field structure. As the full geometry is too complex to be created and simulated in the course of a single lab, we will have a look at the simplified geometry you see below to the right. Use your knowledge from the previous steps, to design a model of this problem and have a look at the complex structure of the magnetic field.
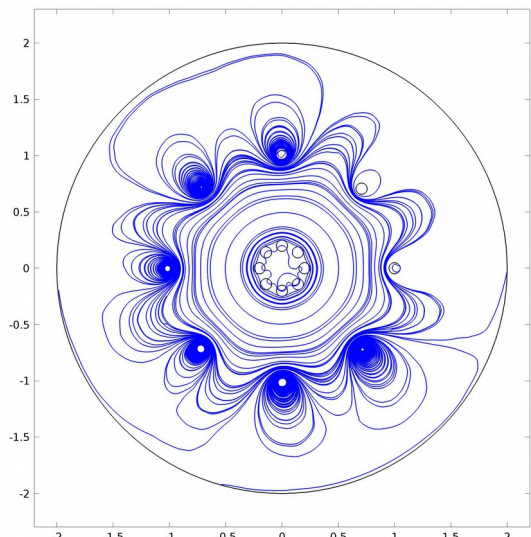
Steps:

1. Add a new parameter for the position of the Coil along the axis. (*pC = 0.6[m]*)

2. Construct eight coils with the center on a circle of the radius *pC*. Use the 3D *torus* geometry element to generate the coils with an inner circle with a radius *wC* and an outer circle with radius *rC*.

3. Add a Helmholtz coil with *major radius = pC+rC* and *minor radius = wC* for the coils. The distance of the coils should be also *pC+rC*.

4. Remember the computation area and generate a sphere. Set up the current definitions as you've done before with an effective coil current of *I0=80[kA]* and build a normal physical mesh.

Hint: Without the Helmholtz coil, the magnetic field of the circular ordered coils, should end up in a ring field.

5. Add a small *torus* of radius *pC* and inner circle *5[mm]* that support the plasma current of fifty times the coil currents. This will bring the typical helix-structure of the magnetic field, that is used to capture the plasma.

6. Simulate at first, only the circular ordered coils, in a second step add the plasma current and at least the full setting with the Helmholtz coil.
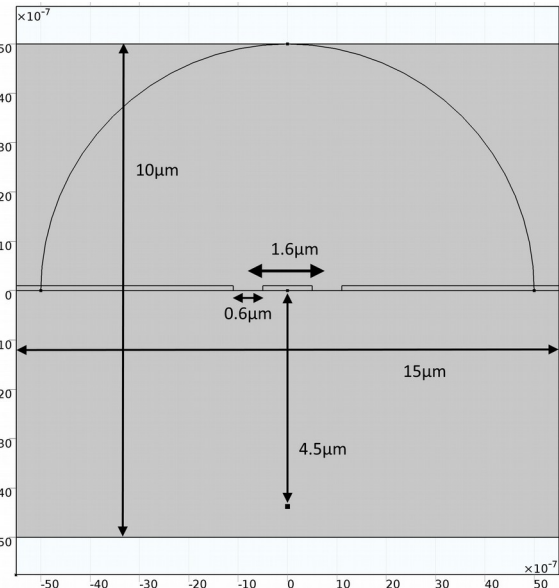
Background:

The circular ordered coils, will build a magnetic field, that looks like the one in the picture over it. To get a helix structured magnetic field, a current will be induced with a transformer technique. The plasma itself can be interpreted as the secondary coil with one turn. The additional Helmholtz coil around the z-axis of the problem, can be used to move the plasma in the z-direction.

Additional exercise: Young's experiment

This experiment is one of the most famous for demonstrating the wave-like behavior of light to students. A wall with two slits of the dimension of the wave length is placed between a point source for electromagnetic waves and a detector. If the slit distance at its second power, divided by the wave length is much smaller as the distance of the detector to the wall, one can observe that the intensity distribution can be described by the mathematical *sinc*-function.

We want to find out how the intensity distribution looks like, if this inequality is not true. Therefore, open a *new project* with the *model wizard* and select the *2D* dimensionality option. The physics will be *electromagnetic waves, frequency domain (emw)* and the study type we want to look at is *frequency domain*.

Generate the *geometry* as displayed in the picture on the right top of this page. As *materials* we use *air* and *aluminum 3003-H18* (for the wall). Add an *electric point dipole* to the *physics* node and choose the point at the bottom of the geometry as it's location. Use the setting *dipole moment* for the *dipole specification* and set it to *0.1[Am]* in *x-direction* and *0[Am]* in *y-direction*. To neglect scattering effects on the computation area boundaries, add a *scattering boundary condition* to the same node. Select the complete boundary lines of the area and use *wave given by E-field* for the property *incident field*.

The *mesh* has to be fine enough in order to resolve the wave length in an adequate way. The wave-length we want to look at is *lambda=500[nm]* as you can define as a *global parameter*. To generate a fine enough mesh change the mesh configuration to *user controlled mesh*. Set the *maximum element size* to *lambda/10* and the *minimum element size* to *lambda/15*.

To compute a study in the *frequency domain*, one has to define the frequencies that have to be computed. Select *step 1: frequency domain* in the study node and type *c_const/lambda* in the *frequencies* text field. The constant *c_const* is a predefined parameter and contains the speed of light in vacuum. Now you can run the simulation to get some nice pictures as the ones below. Try also to produce a function plot of the E-field norm for the top boundary or the half circle we defined in the geometry.

Hints:

- For the halve circle geometry, change the type from solid to curve.

- For the function plot, add edge 2D plot on the data sets node in results, and select the desired edges.

Norm of the electromagnetic field (Intensity is proportional to that)