COMPUTATIONAL WORKSHOP

CDT Fusion Power: Icy Durham Practical Course

(Version 2)

Dr. M. J. Raine (m.j.raine@durham.ac.uk)

1 Introduction

In this workshop you will use QuickField, a finite element analysis (FEA) software package for simulating electromagnetic, thermal and stress problems. You will use it to create a model of a simple multi-layered copper solenoid, analyse its field profile, replace the copper turns with high temperature superconducting (HTS) turns, and investigate the effect on the magnetic field from changing the solenoid's geometry.

2 Installation

You can download the installer file (student license) from https://quickfield.com/free_soft.htm. This requires you to create an account. Once downloaded, double-click the file and follow the instructions.

3 Modelling a copper solenoid coil

3.1 Create a new project:

1. Open QuickField, if it's not already open on your desktop.
2. Click on **File -> New Problem…**
3. Give the file a name that makes sense to you, "Copper coil" for instance, and save it to your preferred location using the **Browse** button. When done, click **Next** at the bottom of the dialog and the next screen will appear (Figure 1 (LEFT)),

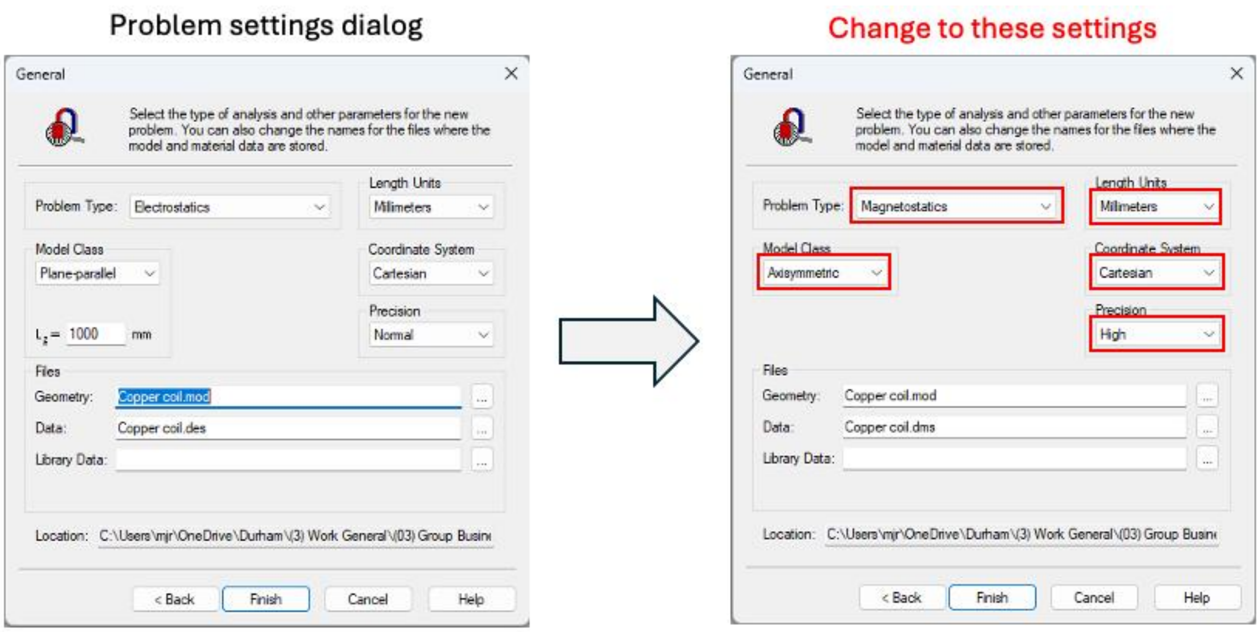

Figure 1: Setting up the project.

4. Change the settings to those shown in the red boxes in Figure 1 (RIGHT) (if they are not already the same) i.e. "Magnetostatics", "Axisymmetric", "Millimeters", "Cartesian", and "High". Click **Finish**.
5. The model plane will now appear in the application window (see Figure 2). By setting "Axisymmetric" in the previous step you told QuickField you are going to draw a 2D cross-section of a 3D object that is symmetrical about a central axis of rotation. Using the appropriate symmetry of the object reduces model complexity and computational resource requirements – it simplifies the model.

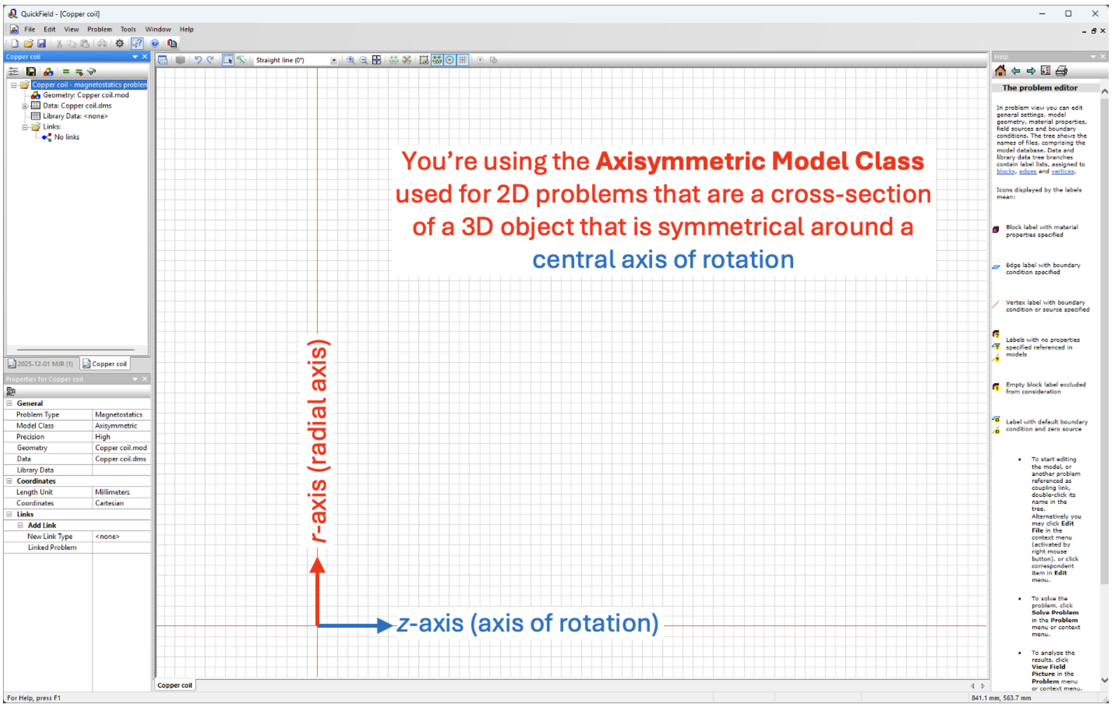

Figure 2: A newly opened project in QuickField showing the orientation of the axes.

3.2 Set up the model

Let's model a solenoid with

a) an *inside* radius, r_{in}= 20 mm (40 mm bore diameter)
b) an *outside* radius, r_{out}= 30 mm (10 mm coil thickness)
c) and *length*, L = 80 mm

Due to the symmetry of a solenoid, we need only deal with a cross-section through one side of the multi-layered coil (see Figure 3). There are four main model definitions we must create by either drawing or labelling and then setting their conditions. These are,

1) the **computational domain** (the area we want to limit the computation to) – *by drawing*,
2) the **solenoid's cross-section** and properties – *by drawing and labelling*.
3) the **environmental conditions** within the domain – *by labelling*,
4) the **boundary conditions** of the domain – *by labelling*,

We'll first deal with 1) and 2) above, which simply requires two rectangles to be drawn.

3.2.1 Rectangle one - The computational domain:

This needs to be somewhat larger than the coil's cross-section. Let's make it 260 x 200 mm by drawing a rectangle of that size.

1. Click on the "Insert Vertices/Edges" toolbar button (see (3) in Figure 4).
2. Click on the "Insert Shape" toolbar button (see (8) in Figure 4).
3. Enter the values shown in Figure 5 (LEFT) into the dialog box that appears.
4. With your mouse, click on the bottom-left point on the diagram of a rectangle at the top of the dialog box. This will set that point as the origin of the rectangle and an "(x,y)" should move from wherever it was to the point you clicked.
5. Click **Insert** and then **Close**.
6. Click the 'zoom to fit' button (see (5) in Figure 4).

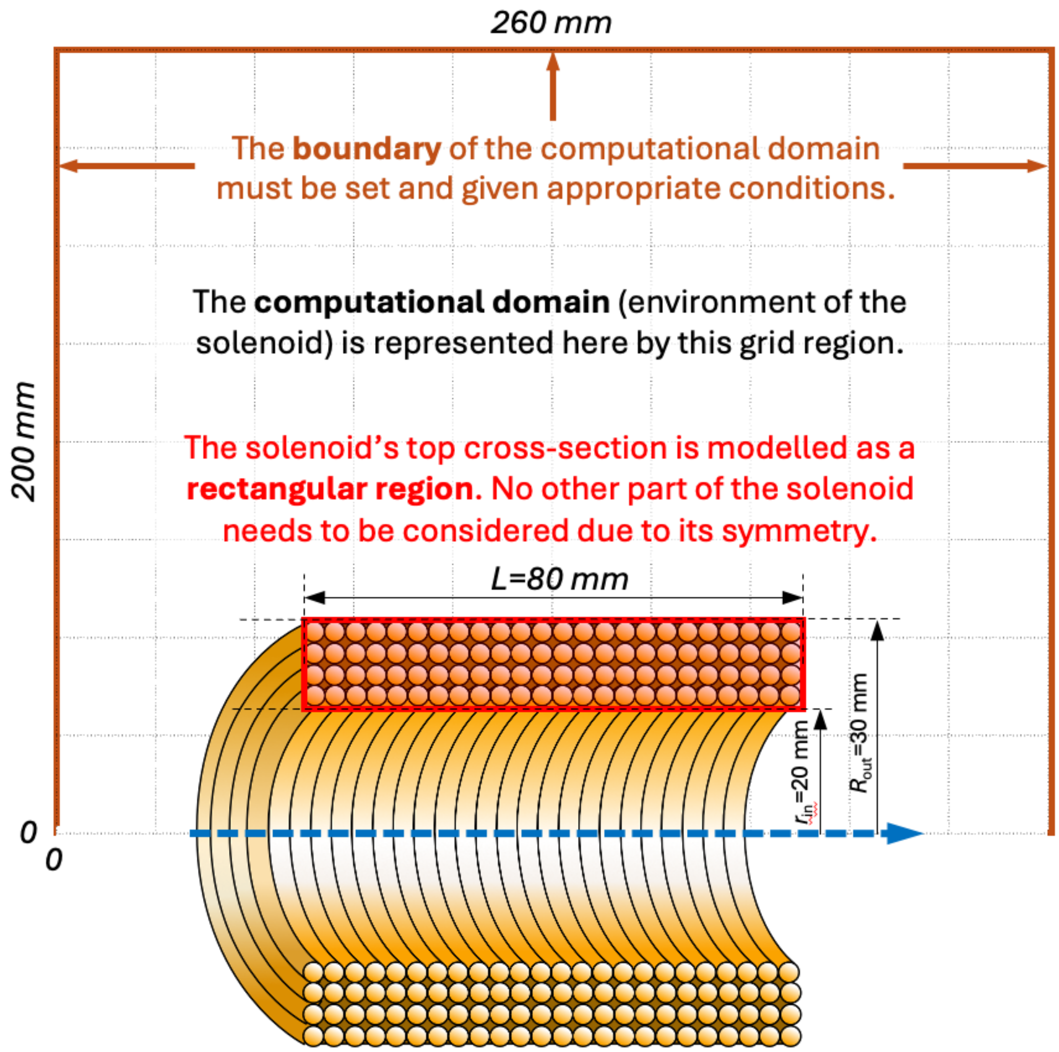

Figure 3: The main definitions of the model; the domain (environment), the *boundary* and the *solenoid's cross-section*.

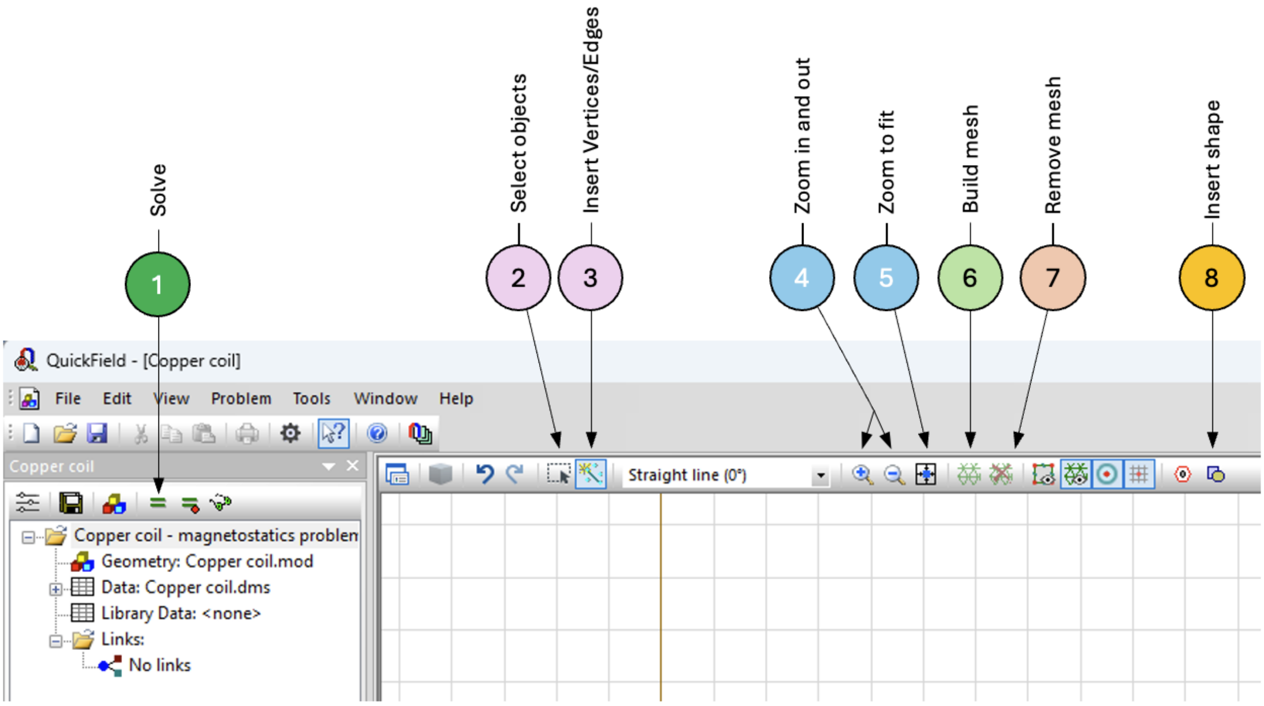

Figure 4: The location of the tools required to set up and solve the model.

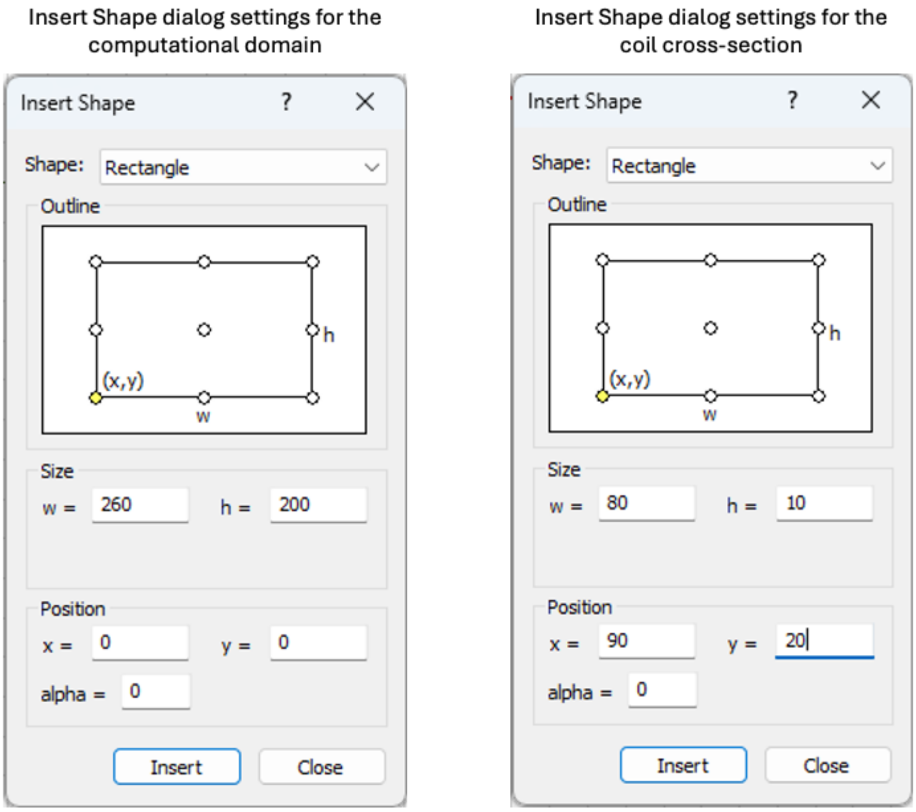

Figure 5: Insert Shape dialog box settings for, LEFT: the computational domain and, RIGHT: the solenoid's cross-section.

3.2.2 Rectangle 2 - The solenoid's cross-section

We now need to draw the coil's cross-section. This needs to be the same overall size as all the individual turns that constitute the solenoid.

We will,

a) make the solenoid cross-section 80 mm by 10 mm.
b) use copper wire with a cross-section of 1.0 mm^2, which at room temperature can continuously carry ~11 A, and even though wire is usually round, for simplicity, we'll assume it has a square cross-section, meaning in an 80 mm x 10 mm coil cross-section we have 80 x 10 = 800 turns. This also means we have an artificial packing density of 100 % but this should not affect the results too much.

We can now draw the solenoid's cross-section.

1. Click the "Insert shape" button again (see (8) in Figure 4).
2. Enter the values shown in Figure 5 (RIGHT) into the dialog box that appears… check that the bottom-left point is selected to be the origin, "(x,y)".
3. Click **Insert** and then **Close**.

Your screen should now look like Figure 6.

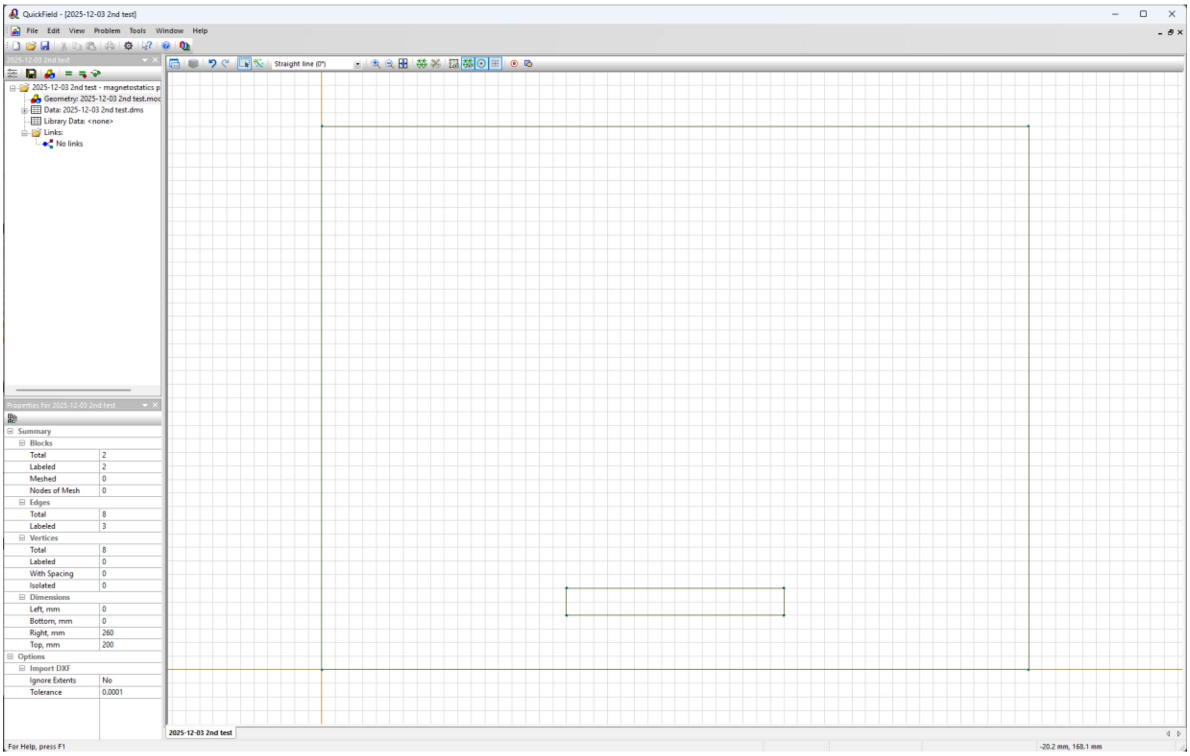

Figure 6: The QuickField window should look like this once you've got to the end of section 3.2.2.

3.2.3 Labelling the different regions

We now need to label the regions within the model so QuickField knows what they are and their properties can be set.

Label the boundary of the computational domain (**BOUNDARY**):

1. Click the "Select Objects" button (see (2) in Figure 4).
2. Double click on one of the boundary lines, let's do the left-hand one first.
3. The Selection Properties dialog box will appear (see Figure 7).

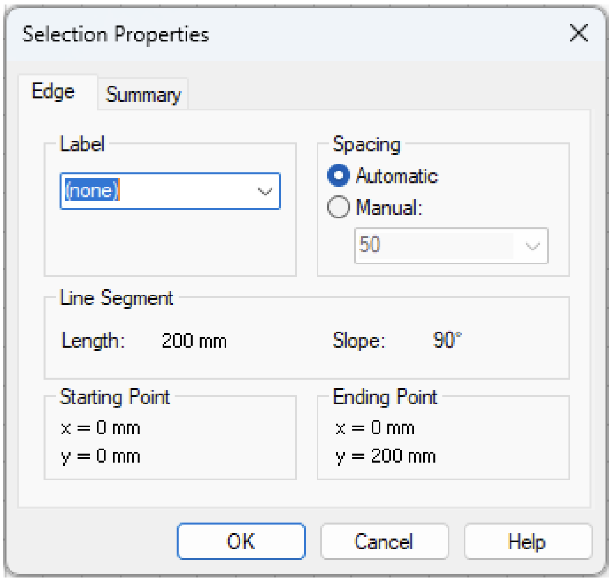

Figure 7: The Selection Properties dialog box used to name model elements.

4. In the “Label” dropdown box (that currently says “(none)”) type “BOUNDARY” and click OK.
5. Repeat for the top and right-hand boundary lines (Note: because you’ve already created a label called “BOUNDARY” you just need to select it using the Label dropdown in the dialog box.) ***Do not label the bottom boundary line***. *We don’t require a bottom boundary because of the symmetry of the model. The boundaries you’ve drawn are rotated about the z-axis, creating an enclosed region (see Figure 3 that only shows three sides to the boundary)*.

Now label the environment (**AIR**):

6. Hover your mouse over the computational domain, outside of the coil cross-section rectangle, it will change colour to show it’s selectable. Double click this region and name it “AIR”.

Label the solenoid’s cross-section (**COIL**):

7. Do the same for the solenoid region and label it “COIL”.

In QuickField’s left-hand panel (see Figure 8) you should now see two Block Labels, **AIR** and **COIL** and one Edge Label called, **BOUNDARY**. They have question marks on their icons to indicate that their properties need to be set.

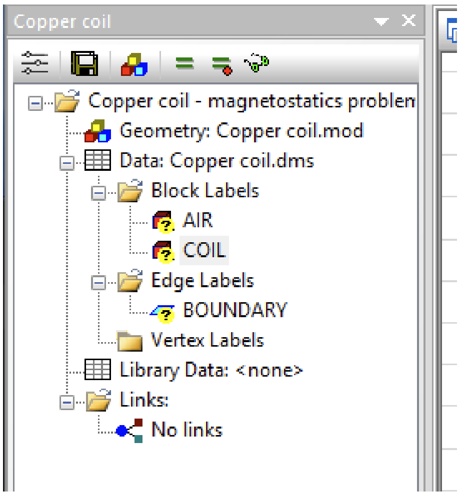

Figure 8: QuickField’s left-hand panel where the Block Labels and Boundary Label can be found.

3.2.4 Set region properties:

1. In the left-hand panel (see Figure 8) double click on the AIR label. Set the relative permeability to 1, as is shown in Figure 9 (LEFT). Then click **OK**.
2. Now double click on the COIL label (see Figure 9 (RIGHT) for values to be entered). Set the relative permeability to 1. In the Field Source section at the bottom, in the current, “I =” entry box, we need to enter the total Ampere-turns of the coil. We are using square cross-section wire 1 mm by 1 mm. This wire is capable of continuously carrying, at room temperature, 11 A. So, in this box we enter “800*11” i.e. the number of turns multiplied by the current in each turn. Make sure the “Total Ampere-Turns” radio button is selected (this can only be done after you’ve entered the “800*11”) and that the conductors are connected in series. When finished, click **OK**.

Label Properties Dialog for **AIR** Label Properties Dialog for **COIL**

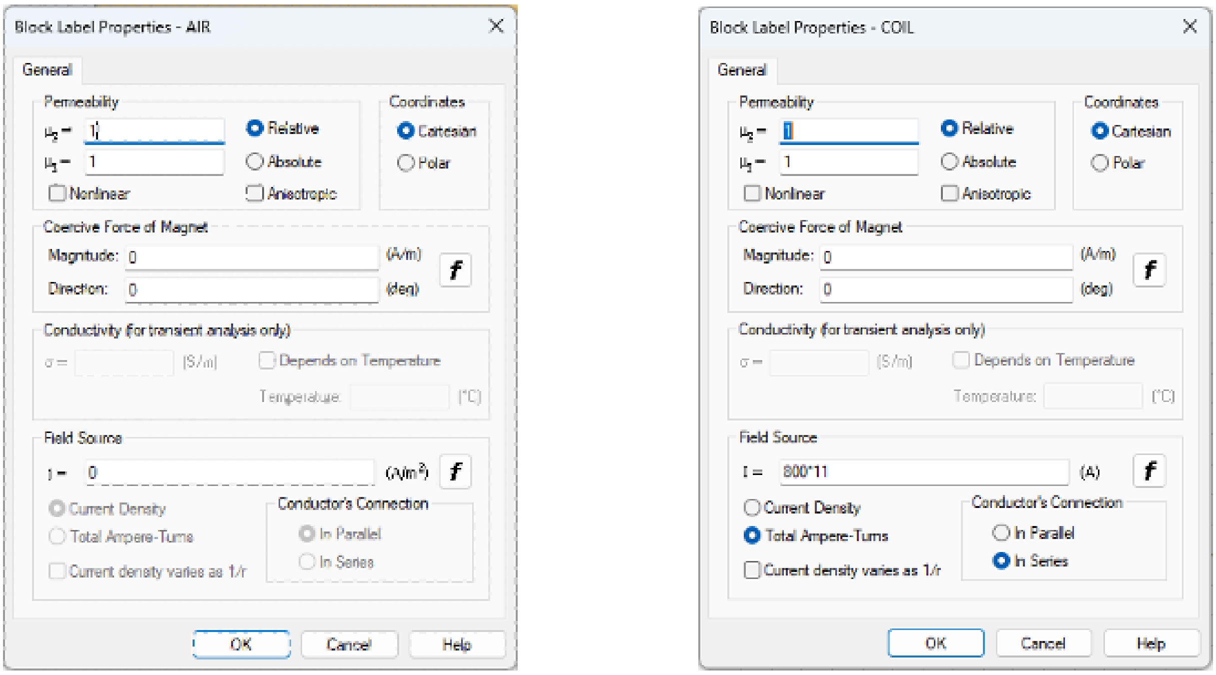

Figure 9: The Label Properties dialog box settings for, LEFT: AIR and, RIGHT: the COIL.

3. Now double-click the BOUNDARY label in the left-hand panel. In the dialog that appears click the box at the top labelled "Magnetic Potential: A=A_0" and click **OK**. This sets the magnetic vector potential at all boundaries to a constant value. In this case, *zero*. This anchors the solution, which ensures uniqueness. If this condition was not set then QuickField would not be able to find a unique solution since the vector potential can be shifted by a constant value without changing the magnetic field, **B**.

3.3 Solve the model:

We are now ready to solve the model.

1. Click the "Build Mesh" button (see (6) in Figure 4)
2. Click the "Solve" button (see (1) in Figure 4).
3. Click away any dialog boxes that appear.

QuickField will now create a new tab (see bottom of QuickField window) that says, "Field Picture Copper Coil" ("Copper Coil" will be replaced with whatever you called your model). A colour field profile map will now be visible on your screen, and a new "Settings" panel will be visible on the left. Play with these settings to see what difference they make to the visualisation of the field map. For instance, open "Vectors" in the Settings panel and change "Show" from "No" to "Yes".

3.4 Check the model's maximum field:

Check how realistic the model's maximum field result is by using the equation (3.1) below for the maximum field generated at the centre of a finite solenoid.

$$B_{max} = \frac{\mu_0 N I}{2(r_{out} - r_{in})} \ln\left[\frac{r_{out} + \sqrt{r_{out}^2 + \frac{L^2}{4}}}{r_{in} + \sqrt{r_{in}^2 + \frac{L^2}{4}}}\right] \qquad (3.1)$$

4 Main tasks

Choose one of the variables that determines the size of the solenoid from the RHS of equation (3.1) and use QuickField to,

1. calculate six different magnetic fields for six different values of the variable you have chosen. *To change the dimensions of the solenoid geometry (the rectangle that represents the solenoid) you can drag and drop the required corner vertices to new locations. To do this, first switch the mesh off by clicking on the "Remove Mesh" button (see (7) in Figure 4), then simply click on the vertex you want to move and let go – this will change its colour. Then click and hold it while dragging it to its new location, before releasing the click. Tip: You don't want to make the solenoid too large within the computational domain so restrict each of the six dimension changes to nothing greater than 5 or 10 mm. After each dimension change, switch the mesh back on before re-solving the model.*
2. Plot and fit your data with equation (3.1) (use Excel or any software package you are comfortable with).
3. Can you explain to what degree your computational data agree with the equation?
4. Call a demonstrator over and show them your graph with datapoints and an analytic fitted line.
5. Would equation (3.1) be useful to approximate the field of a tokamak's
    a. central solenoid?
    b. toroidal field coil?
6. Now let's try to think creatively and join up what you knew before the course about magnetic fields, and connect it with the new material from this course:
    a. Look up and write down Ampere's law – it can be used to describe the spatial dependence of the magnetic field produced by an infinitely long straight wire carrying a current.
    b. Each toroidal field (TF) coil in ITER has 134 turns, each carrying 67 kA. Each turn is a cable with a diameter of about 5 cm (see the wall-display outside the experimental labs). The inner radius of the TF coil is ~ 1 m and the outer radius is ~ 8 m. Using Ampere's law, calculate (i) a very rough value for the maximum magnetic field near the turns and (ii) the field at the centre of the plasma (NB: a very rough and simple calculation is preferred).
    c. How would you improve the accuracy of your calculations (i) using the tools from the course and (ii) more generally. Feel free to discuss this with other students (There's not one unique answer). Estimate the error bar on your calculated magnetic field values. When a small group of you have settled on your answers, ask a demonstrator for their opinion on the answers you have.

5 Additional tasks

5.1 Replace the copper turns with a high temperature superconducting tape

HTS tape comes in all shapes and sizes, but a widely used width is 4 mm with a thickness of ~ 0.1 mm. This thickness is mostly occupied by "normal" material i.e. copper and a Hastelloy substrate, for instance, and the superconducting material (REBCO) only occupies about 1/100th of the tape's thickness. However, we will ignore these complications and use a tape cross-section of 4 mm x 0.1 mm i.e. again we assume a 100 % packing density and that the current is carried by the whole cross-section of the tape.

5.1.1 HTS coil at 77 K:

At 77 K the critical current of the HTS, in *zero* applied field, is around 140 A. However, every part of the HTS used to wind the coil will experience the local field generated by every other part of the coil, and so, since the tape's critical current is field (and temperature) dependent, the solenoid's field will reduce the critical current of the tape.

So, with the dimensions of our (original) solenoid and the HTS tape we are using, there's a maximum field that can be generated by the coil where the HTS reaches its maximum current. The maximum current, given our solenoid characteristics and temperature, is ~ 43 A and this should generate a maximum field of ~ 1.14 T within our solenoid bore.

1. Using this information change the solenoid's properties and re-solve the model.
2. Does your maximum field agree with ~ 1.14 T? If not, why not?

5.1.2 HTS coil at 20 K:

For magnetically confined fusion, it is expected that the HTS magnet system will be at 20 K. At this temperature, the optimum current for our solenoid is ~ 292 A.

1. Adjust the solenoid's properties.
2. What is the maximum field the solenoid can generate at 20 K?
3. At this current and magnetic field, what are the three or four other most important issues you should be concerned about for this superconducting magnet?
4. Can you estimate the forces in your magnet and decide whether the magnet will stay in one piece or not?

5.2 Explore the field profile of a copper solenoid:

1. Select **Contour -> Add Lines...** in the main menu.
2. In the dialog box that appears click **Set Start Point**, which selects the origin (0,0). Then type in any arbitrary lengths in the "z=" and "r=" text boxes that are within the computational domain, say 30 mm for each, and click **Add Line** followed by **Close**.
3. A red vector will now be visible starting at the origin.
4. You can now drag the end points of the red vector anywhere you want. Start by placing the start point at the centre of the coil's bore, in the highest field region. Move the end point through the coil into the low field region, so the vector is pointing along the radial axis, through the centre of the coil.
5. Right click the red vector and select "XY-Plot". This shows the field profile along the vector you drew. You can move the vector anywhere and in any direction in the computational domain to see any field profile you might be interested in viewing.