Model Construction

Start the CST EM STUDIO module without a project template. In the “Home” tab select the “LF Frequency Domain” solver, check your units and change the length units to mm (cf Fig. 9).

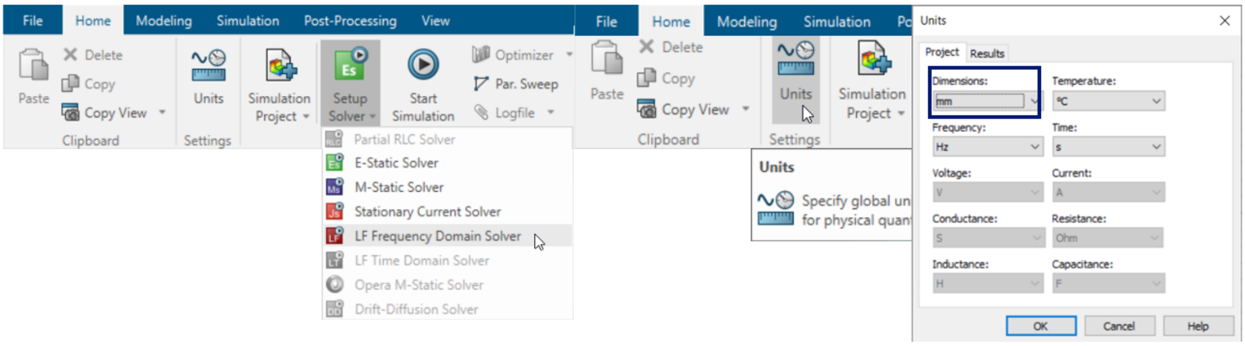

Figure 9: a) Select the “LF Frequency Domain” solver
b) Change units from m to mm.

Next, switch to the “Modeling” tab and construct the copper disc. To do so, simply click on the cylinder symbol in the “Shapes” field (cf. upper part of Fig. 10). You can construct the cylinder graphically by dragging the mouse over the working plane and double clicking, or you can press the Esc key and enter the values by hand (cf. lower part of Fig. 10). Even if you choose the first course of action you will still have an opportunity correct your specifications in the “Cylinder” dialog later on. Since the material “Copper (annealed)” is loaded by default you can directly select it in the material frame at the bottom of the “Cylinder” dialog.

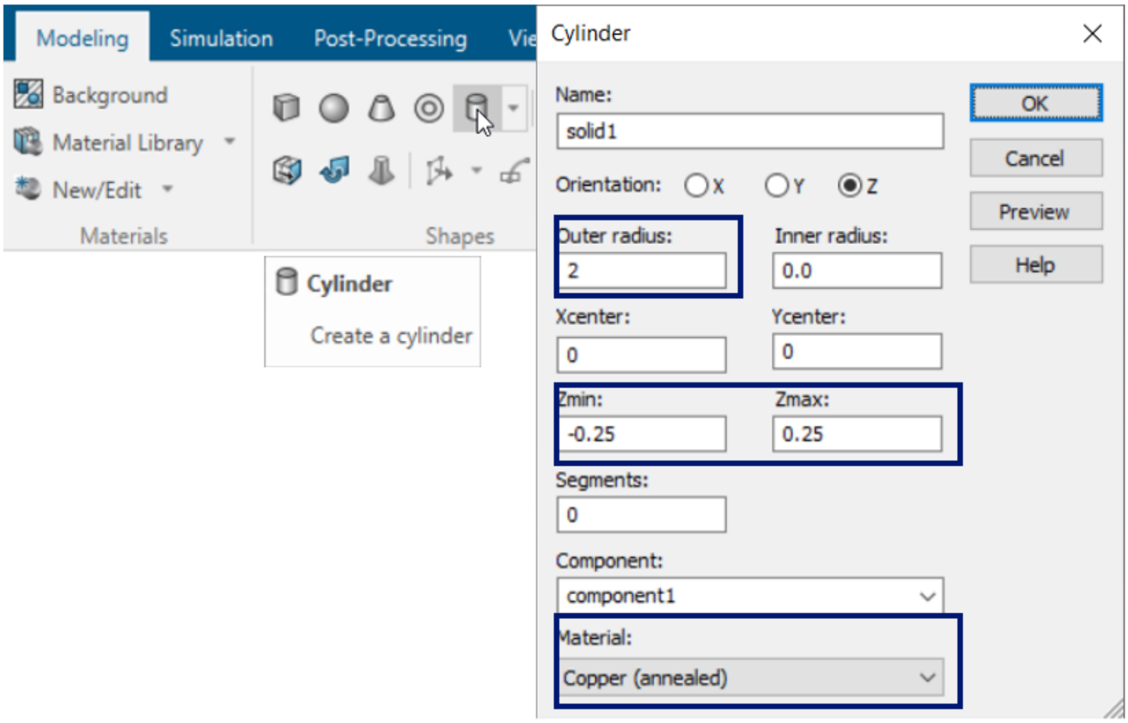

Figure 10: Upper part: Click on the cylinder symbol in order to construct the disc. Lower part: In the “Cylinder” dialog you can set orientation, size, material, etc. of the disc.

Later on we will want to evaluate the conduction current density in the radial direction of the disc. While this can be done without explicitly defining a curve on which the field is then evaluated, it nicely demonstrates the flexibility of this approach.

In order to create a straight curve, go to the “Curves” field, click on “Curves” and select “Line” (cf. Fig. 11 a)). We can use pick operations to select the coordinates of the line. When picking a point on the model the coordinates of the nearest point on the working plane will be selected for the line coordinates. You can either use the tools from the “Pick” field (cf. upper part of Fig. 11 b)) or the handy keyboard shortcuts: “c” will let you pick the point in the center of a circle, while “r” sets a point on the circles boundary. Simply press the key – or select the appropriate pick operation from the “Pick” field – and hover the cursor over the border of the disc. You will notice the disc radius and prospective pick point being highlighted in red (cf. lower part of Fig. 11 b)). Pick the point by double-clicking. After setting the second point, the “Line” dialog will pop up, where you can verify the starting and end coordinates (cf. Fig. 11 c)). When using pick points, you do not see the specific coordinates but rather expressions like “xp(3)”, denoting the x-coordinate of the third pick point.

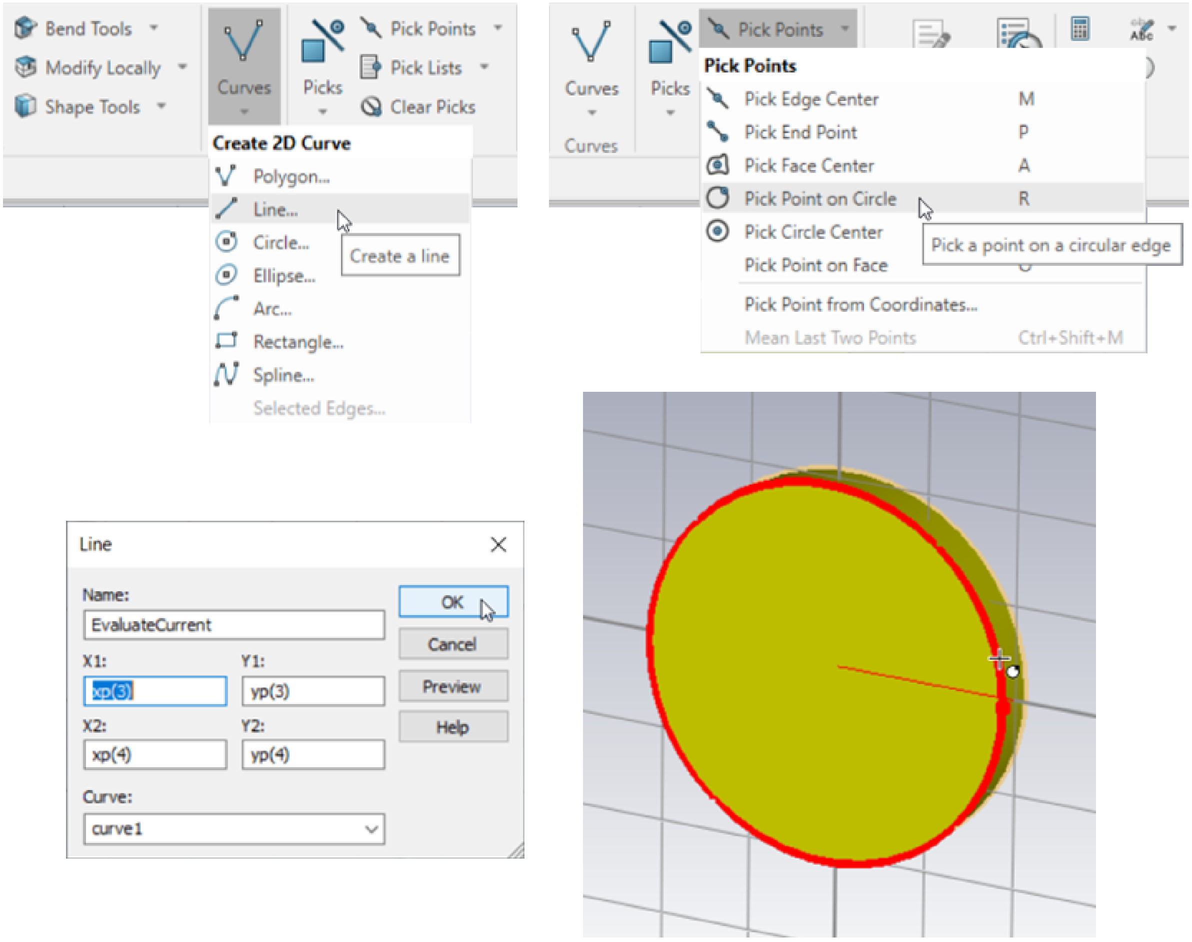

Figure 11:

a) Go to the “Curves” field, click on “Curves” and select “Line”

b) Upper part: Select “Pick Point on Circle”. Alternatively use keyboard shortcut “r”. Lower part: Hover cursor above radius of circular edge. Edge and prospective point are highlighted.

c) Line dialog window, coordinates are already set by the pick points.

Next, switch to the “Simulation” tab. To set the evaluation frequency, go to the “Settings” field and click on “Frequency” (cf. Fig. 12 a)). Instead of a numerical value you can also directly enter a parameter name (cf. Fig. 12 b)). Once the parameter has been set up (cf. Fig. 12 c)) - which happens automatically after the entry - you can find and change it again in the Parameter List. Alternatively you could have defined the parameter beforehand - using the Parameter List - and then directly used it to determine the evaluation frequency. By default the Parameter List can be found at the bottom of CST Studio to the right of the Navigation Tree. You can switch the window between “Parameter List” and “Result Navigator”.

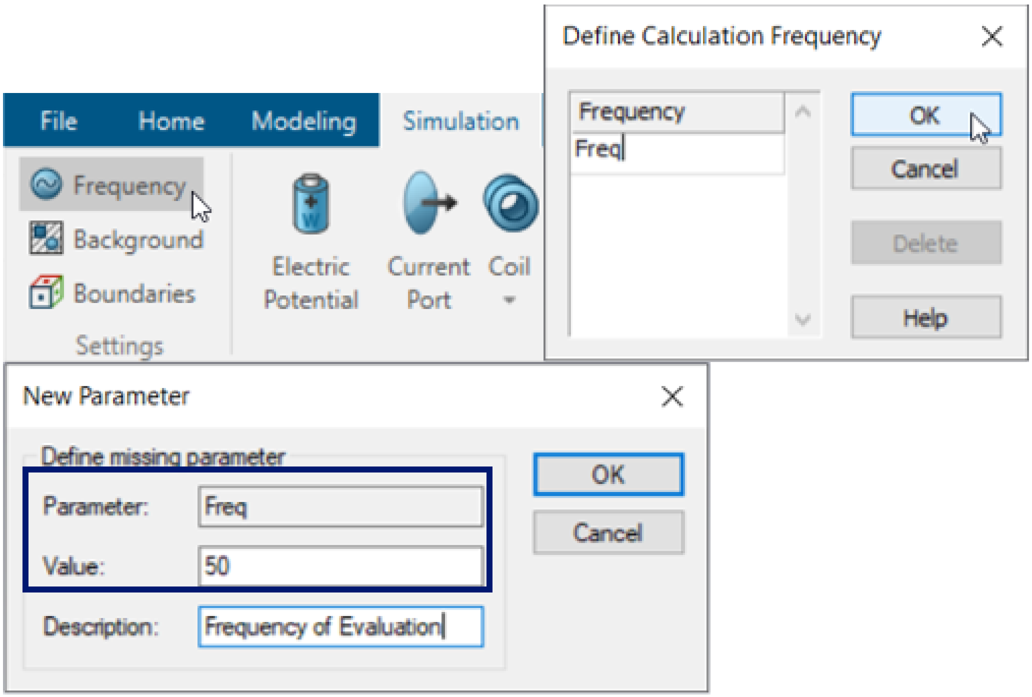

Figure 12

a) In the “Settings” field click on “Frequency” to set the evaluation frequency.
b) Start the set-up of a parameter by entering “Freq” in the frequency field.
c) Finish defining the parameter the adding a value and description.

In the “Settings” field, underneath the “Frequency” button you will find the “Background” button. Here you can set the background material and enlarge the bounding box by assigning extra space in the global axis directions. The background material is already preset to air. Checkmark “Apply in all directions” (cf. Fig. 13 a)) and add a distance of 10 (project units = mm) in all directions.

Next, take care of the boundaries and symmetry planes. To do so, click on “Boundaries” beneath the “Background” button – the “Boundary Conditions” dialog will pop up. In the dialog switch to the “Symmetry Planes” tab first. Each symmetry you exploit reduces the simulation problem by half, so it makes sense to use all possible symmetries. Since we want to have a homogeneous magnetic source field in z-direction, set the XY plane to “magnetic” and the YZ and XZ planes to “electric” (cf. Fig. 13 c)). You will see all changes graphically displayed in the model as well. Then switch back to the “Boundaries” tab. Since we have already set symmetry conditions half of the options are greyed out. In accordance with our source field select boundary type “magnetic” for “Zmax” and set the rest to “open” (cf. Fig. 13 c)).

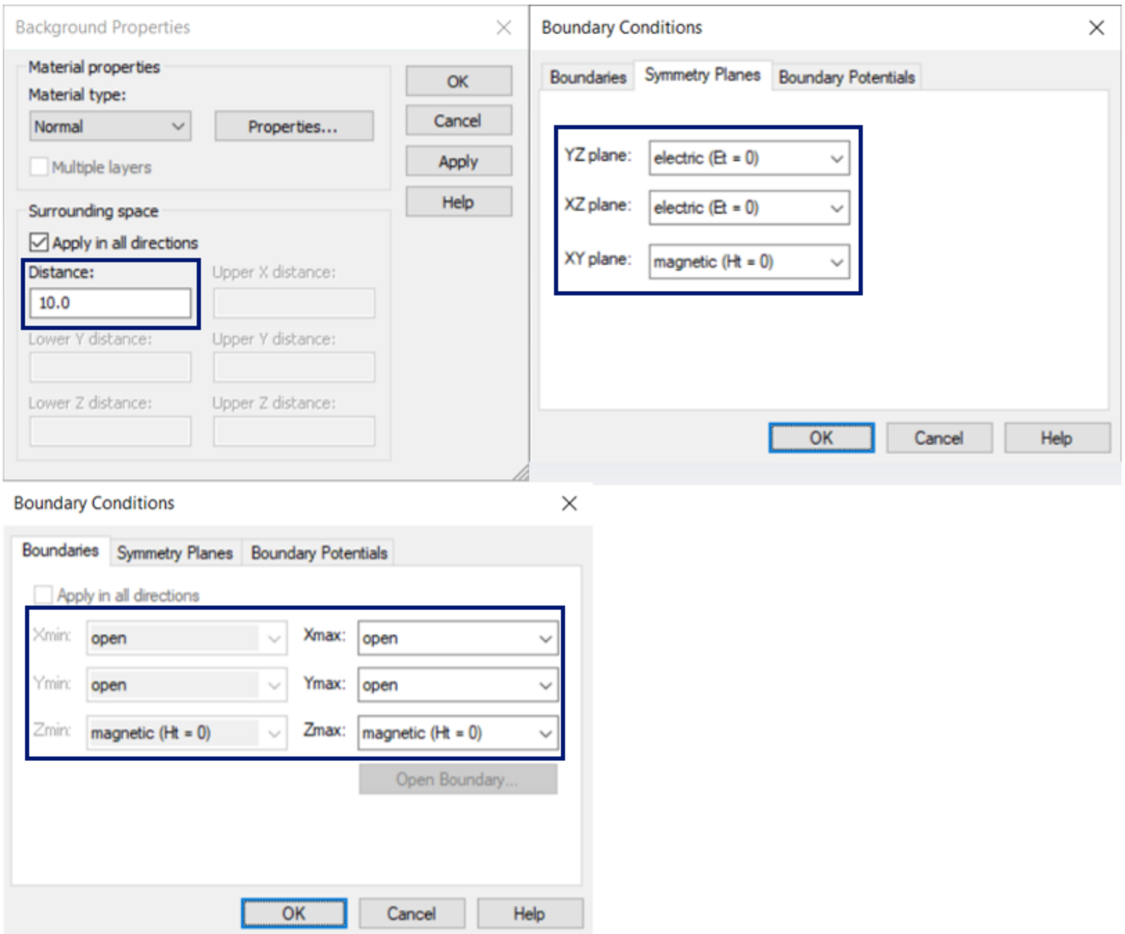

Figure 13

a) Set distance between model and bounding box.
b) Set the symmetry planes.
c) Set the boundary conditions.

Next, set the source field. Excitation sources are also found in the “Simulation” tab, in the “Sources and Loads” field. Click on “Magnetic Source Field” in order to set up the field (cf. Fig. 14 a)). Enter the amplitude as parameter “Hz”. Using parameters not only allows quick and easy manipulation of the setup, but can also ease evaluation, since one only has to type in an expression instead of recalling the exact numerical value. Set “Hz” to a value of 1.0e3 (cf. Fig. 14 b)).

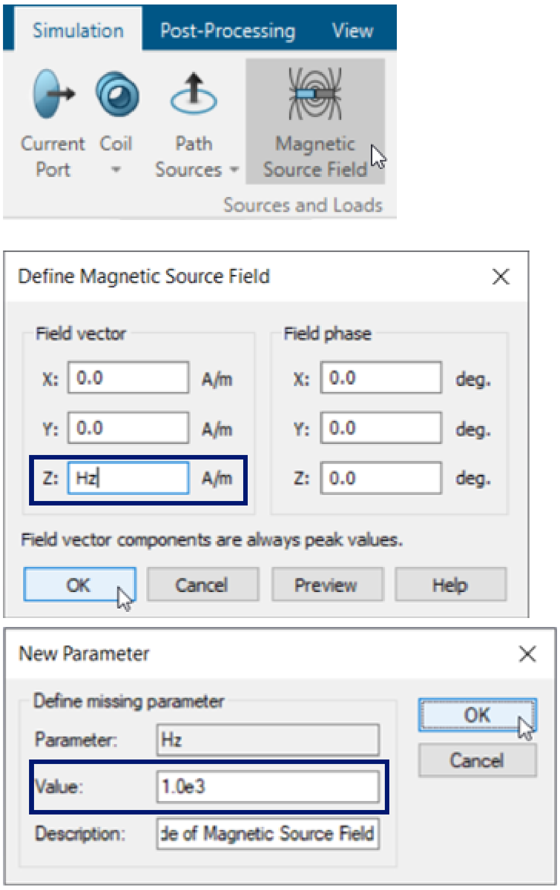

Figure 14

a) Call up the “Define Magnetic Source Field” dialog.
b) Set up Magnetic Source Field.

Next, refine the mesh a bit. Keep in mind that the student edition has a limit of 10 000 mesh cells for the tetrahedral mesh. By using appropriate settings we can generate a very fine mesh and stay below the limit. To access the mesh settings click on the “Global Properties” button in the Mesh-field (cf. Fig. 15 a)). Clicking on the lower part will open a drop down menu where the mesh type itself can be set. The mesh should be set to “Tetrahedral”. Clicking on the upper part opens the “Mesh Properties - Tetrahedral” dialog. In the “Cells per max model box edge” field, enter “20” for “Model” and “3” for “Background”. Then click on the “Specials…” button. In the “Smooth mesh with equilibrate ratio” field enter a value of 1.25 (cf. Fig. 15 b)). This setting controls how quickly neighboring mesh cells can grow in size.

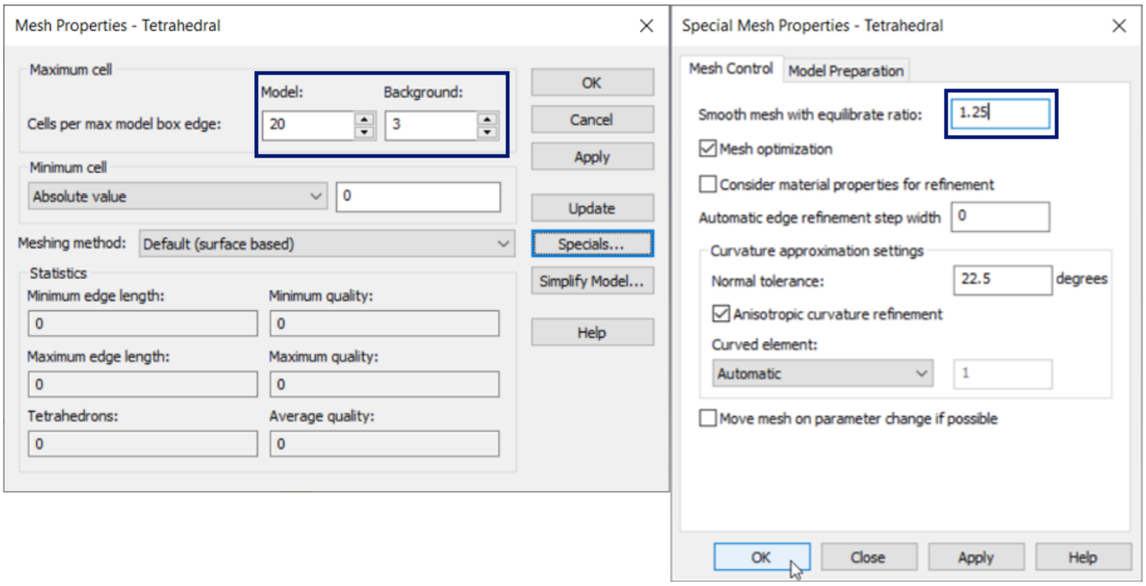

Figure 15

a) Click on the Global Properties button in order to specify the mesh settings.
b) Set the mesh settings.

And with that the modeling part is done. Next, click on the solver icon in either the "Simulation" tab or the "Home" tab (cf. Fig. 16 a)). Take care that the solver equation is set to Magnetoquasistatic and turn off the adaptive mesh refinement (cf. Fig 16 b)). Then click on "Start" in order to run the simulation.

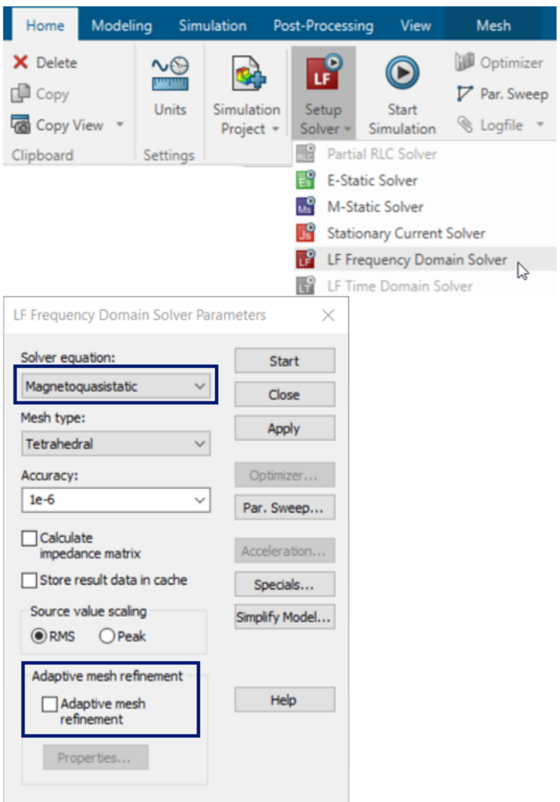

Figure 16

Upper part: LF Frequency Domain Solver icon.

Lower part: Solver settings.

For the evaluation of the results and the comparison to the analytical formula by using Template Based Postprocessing please go back to the main page.