



MAGNETISiM

Defining PCB: tools

GUI Overview

TAB MENU

Core Geometry Tab (Ctrl+1)

Winding Geometry Tab (Ctrl+2)

Sim Config Tab (Ctrl+3)

Magnet Tab (Ctrl+4)

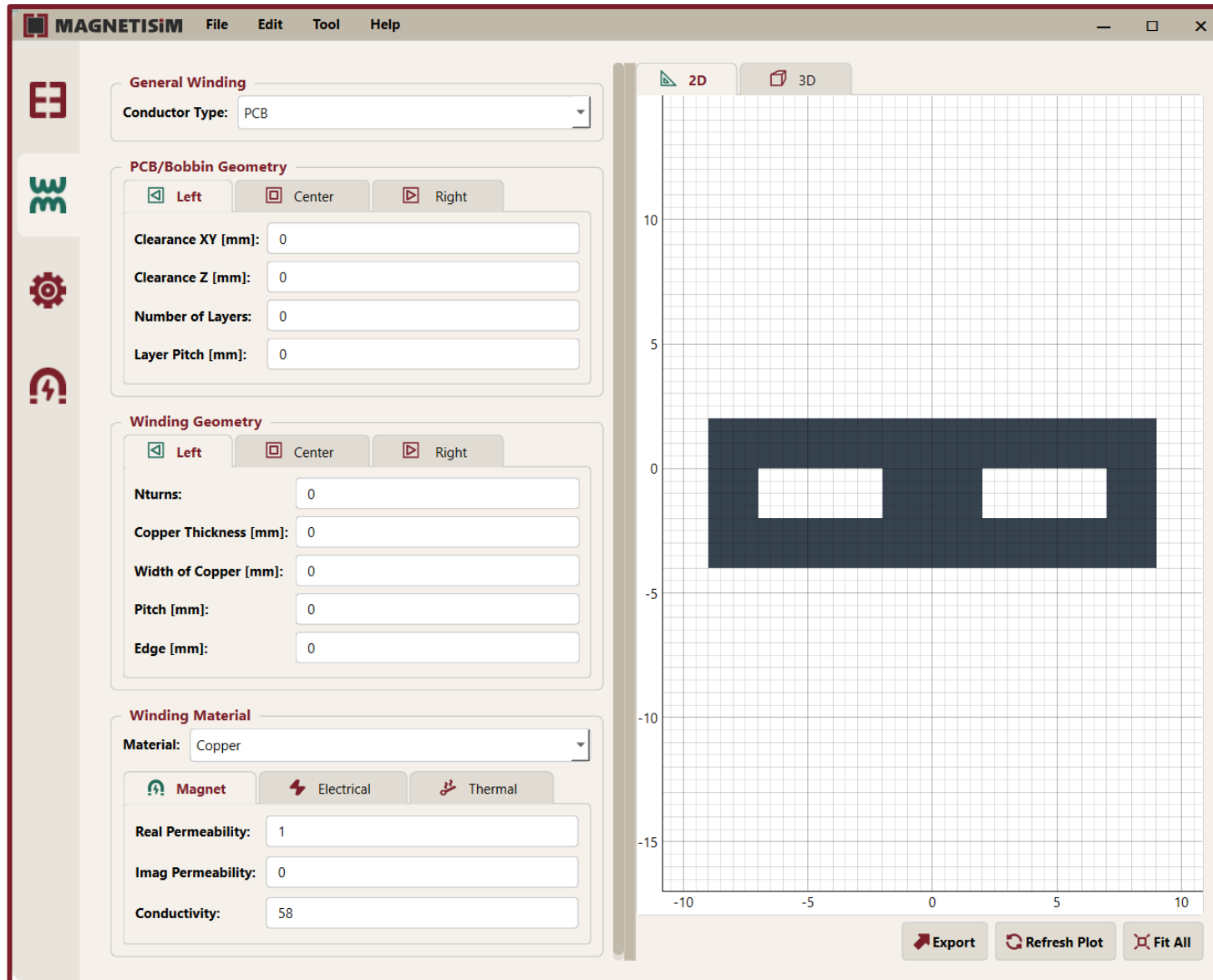
INPUT FRAME

The screenshot displays the MAGNETISiM software interface. At the top, a menu bar includes 'File', 'Edit', 'Tool', and 'Help'. The main window is divided into two primary sections. On the left is the 'INPUT FRAME', which contains two main panels: 'Core Geometry' and 'Core Material'. The 'Core Geometry' panel features dropdown menus for 'Manufacturer', 'Geometry Core', and 'Core Reference', followed by input fields for dimensions A [mm], B [mm], Bp [mm], C [mm], Cp [mm], D [mm], E [mm], F [mm], G [mm], I [mm], and Gap [mm], all currently set to 0. The 'Core Material' panel includes a 'Material' dropdown, four tabs for 'Magnet', 'Power Loss', 'Electrical', and 'Thermal', and input fields for 'Real Permeability', 'Imag Permeability', and 'Conductivity', all set to 0. A button labeled 'Add Permeability(f) Curve' is present, along with a checkbox for 'Permeability(f) from Internal-Database'. On the right is the 'MODELER WINDOW', which shows a 2D coordinate system with a grid. The x-axis ranges from -0.6 to 0.6, and the y-axis ranges from -0.9 to 0.9. At the bottom of the modeler window are buttons for 'Export', 'Refresh Plot', and 'Fit All'. A 'BAR TOOL' is indicated at the top right of the window.

BAR TOOL

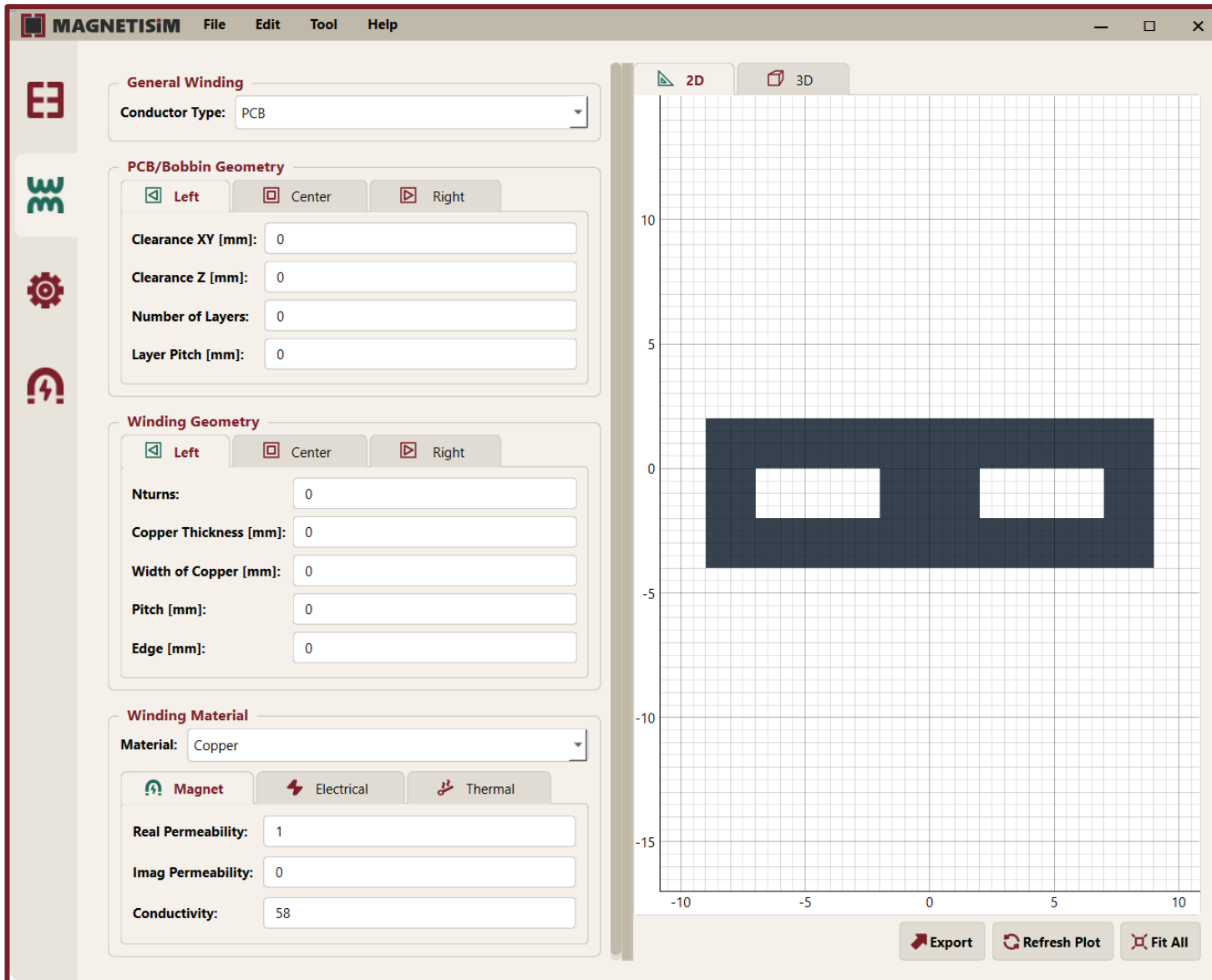
MODELER WINDOW

Winding Geometry Selection – PCB



Before configuring the winding details, make sure your core geometry is properly selected and visualized in the modeler window.

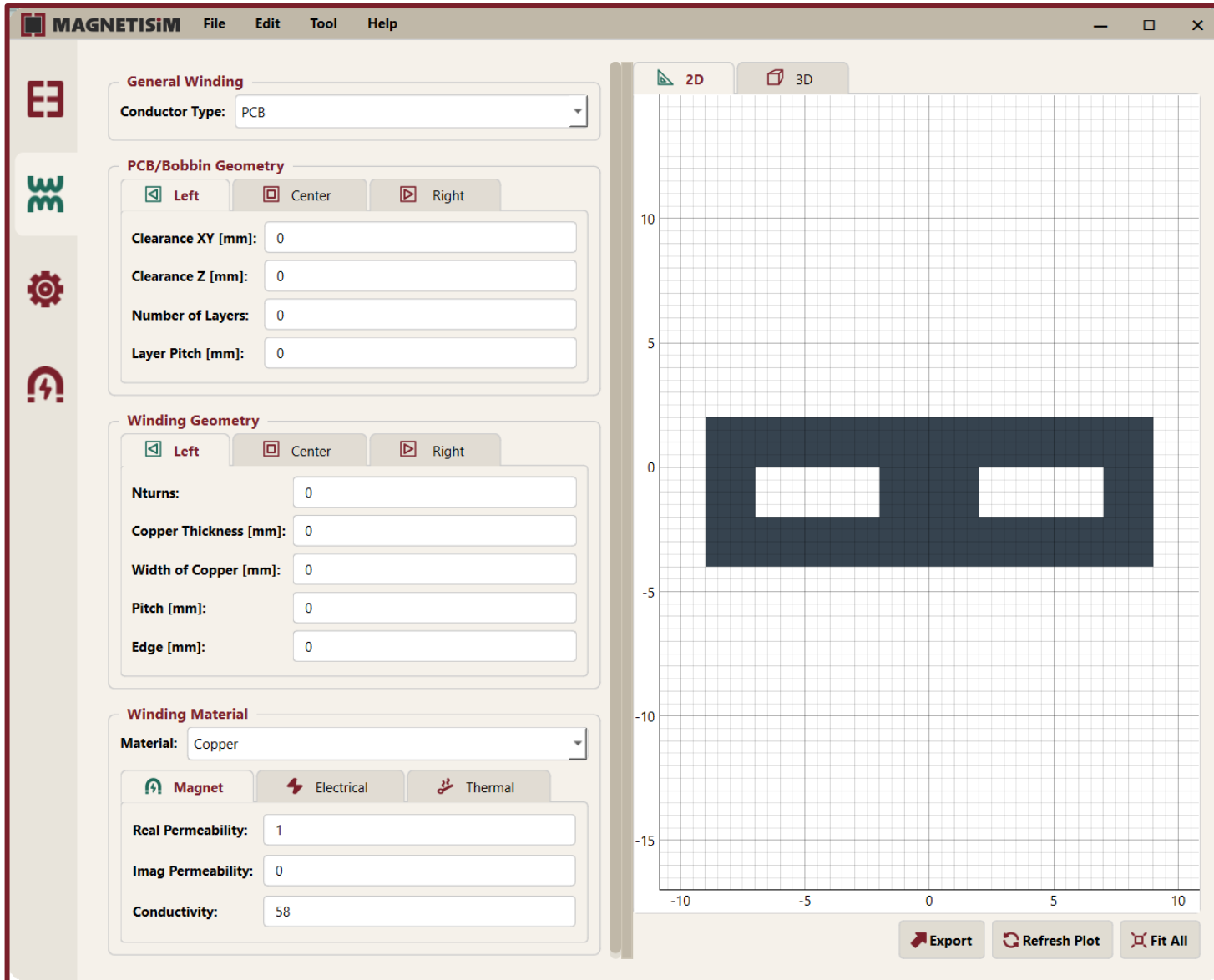
PCB/Bobbin Geometry



Once in the **Winding Geometry tab**, the first step is to configure the overall boundaries of your board in the **PCB/Bobbin Geometry section**.

- **Clearance XY** - This sets the horizontal safety margin or distance between the PCB traces and the core walls.
- **Clearance Z** - This defines the vertical safety distance.
- **Number of Layers** - Here you specify the total number of conductive copper layers your PCB design will have.
- **Layer Pitch** - This dictates the vertical separation between each of those PCB layers.

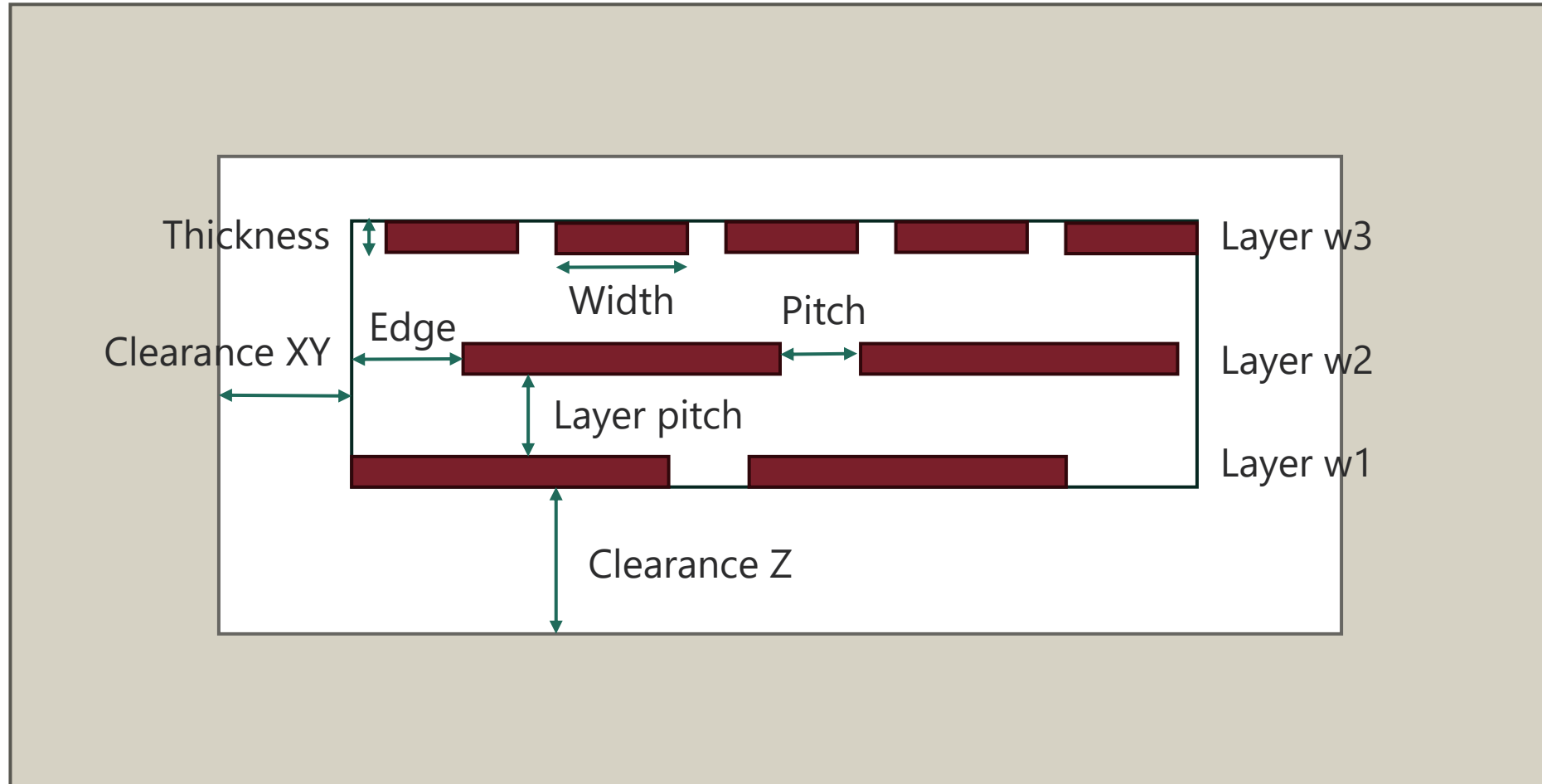
Winding Geometry



After defining the board structure, move to the **Winding Geometry section** to specify the physical dimensions of the tracks.

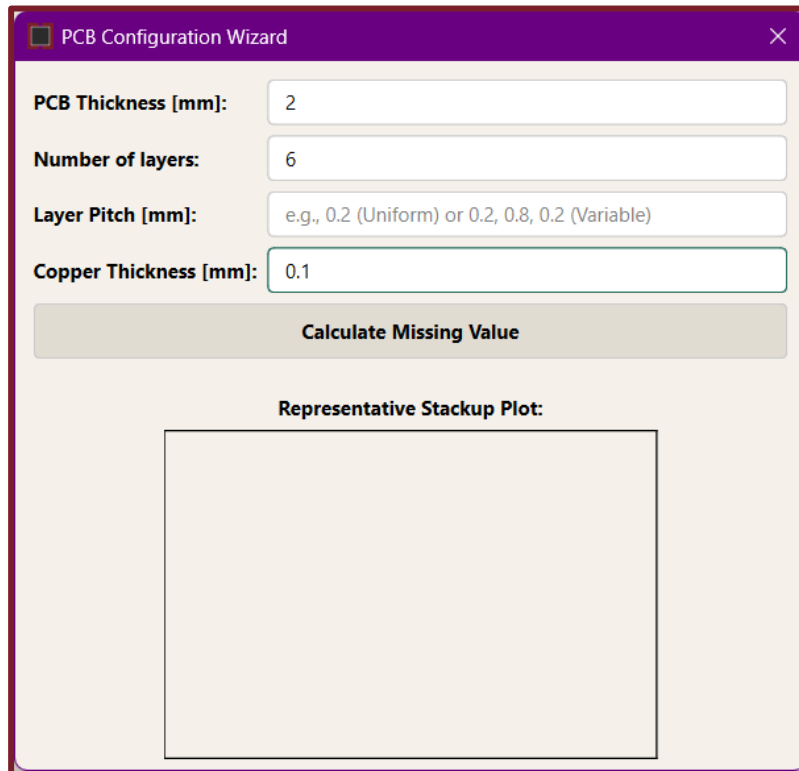
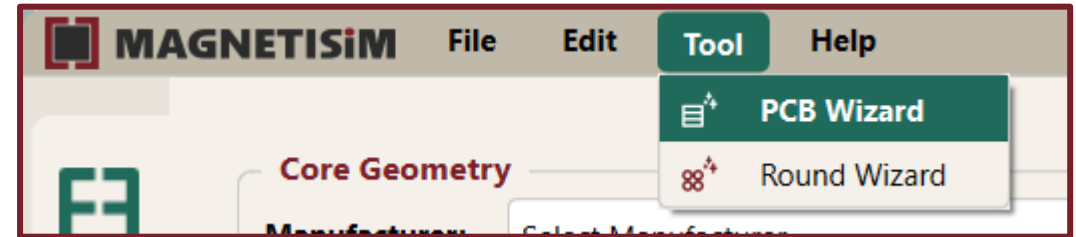
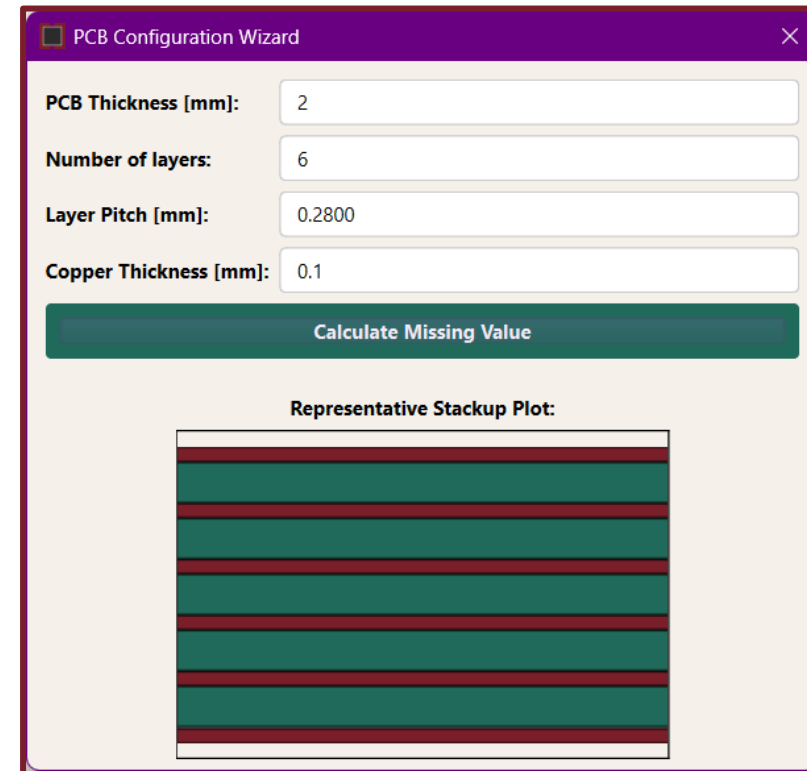
- **Nturns** - The number of turns per layer.
- **Copper Thickness** - The vertical thickness of the PCB copper trace.
- **Width of Copper** - The horizontal width of the copper trace.
- **Pitch** - The spacing between adjacent turns on the exact same layer.
- **Edge** - The physical margin or distance from the edge of the PCB.

PCB Distribution parameters



To greatly simplify this process, MAGNETISiM includes automatic tools. By going to the top **Tool** menu, we can select the **PCB Wizard**.

This wizard allows you to simply input your global constraints, such as the total **PCB Thickness**, the **Number of layers**, and the **Copper Thickness**.

A screenshot of the 'PCB Configuration Wizard' dialog box. It features four input fields: 'PCB Thickness [mm]' with the value '2', 'Number of layers' with the value '6', 'Layer Pitch [mm]' with the text 'e.g., 0.2 (Uniform) or 0.2, 0.8, 0.2 (Variable)', and 'Copper Thickness [mm]' with the value '0.1'. Below these fields is a 'Calculate Missing Value' button. At the bottom, there is a section titled 'Representative Stackup Plot' with an empty rectangular area.A screenshot of the 'PCB Configuration Wizard' dialog box, identical to the previous one but with the 'Calculate Missing Value' button highlighted in green. The 'Representative Stackup Plot' section now displays a visual representation of the PCB stackup, showing alternating layers of dark green and dark red.



Alberto Vital Pagola

alberto.vital@upm.es

CEIMM-UPM

Madrid, Spain



Alberto Delgado

a.delgado@upm.es

CEIMM-UPM

Madrid, Spain



magnetisim.com