



# MAGNETISiM

Configure Magnet Simulation

# GUI Overview

## TAB MENU

Core Geometry Tab (Ctrl+1)

Winding Geometry Tab (Ctrl+2)

Sim Config Tab (Ctrl+3)

Magnet Tab (Ctrl+4)

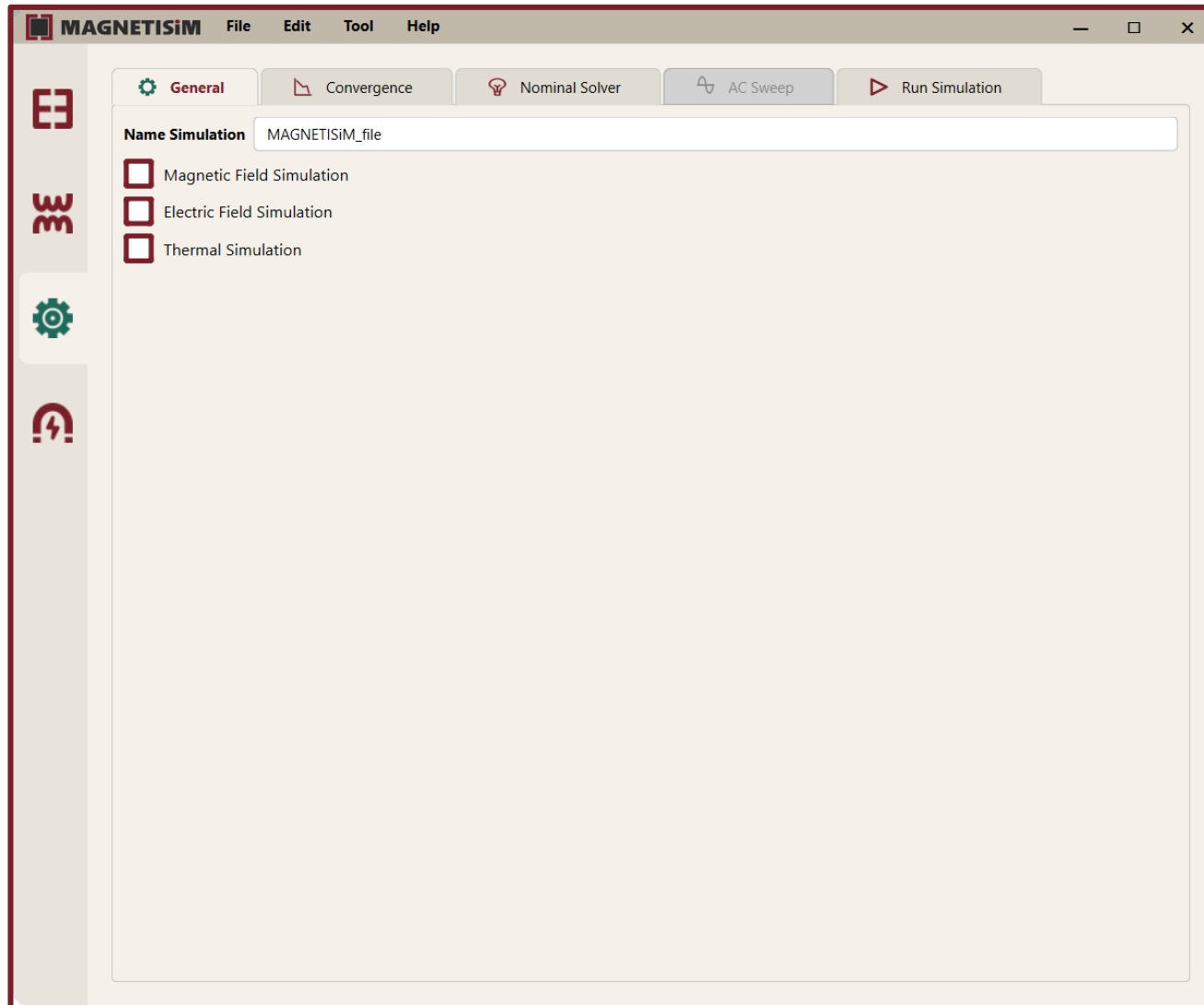
## INPUT FRAME

The screenshot displays the MAGNETISiM software interface. At the top, a menu bar includes 'File', 'Edit', 'Tool', and 'Help'. On the left, a vertical 'TAB MENU' contains four icons: a core symbol (Ctrl+1), a winding symbol (Ctrl+2), a gear (Ctrl+3), and a magnet symbol (Ctrl+4). The main area is divided into two input frames. The 'Core Geometry' frame includes dropdown menus for 'Manufacturer', 'Geometry Core', and 'Core Reference', followed by input fields for dimensions A [mm] through I [mm] and 'Gap [mm]', all currently set to 0. The 'Core Material' frame includes a 'Material' dropdown, tabs for 'Magnet', 'Power Loss', 'Electrical', and 'Thermal', and input fields for 'Real Permeability', 'Imag Permeability', and 'Conductivity', all set to 0. A button 'Add Permeability(f) Curve' and a checkbox 'Permeability(f) from Internal-Database' are also present. On the right, the 'MODELER WINDOW' shows a 2D plot with axes ranging from -0.6 to 0.6 on both the x and y axes. The plot area is currently empty. At the bottom of the plot window, there are buttons for 'Export', 'Refresh Plot', and 'Fit All'. A 'BAR TOOL' is indicated at the top right of the window.

## BAR TOOL

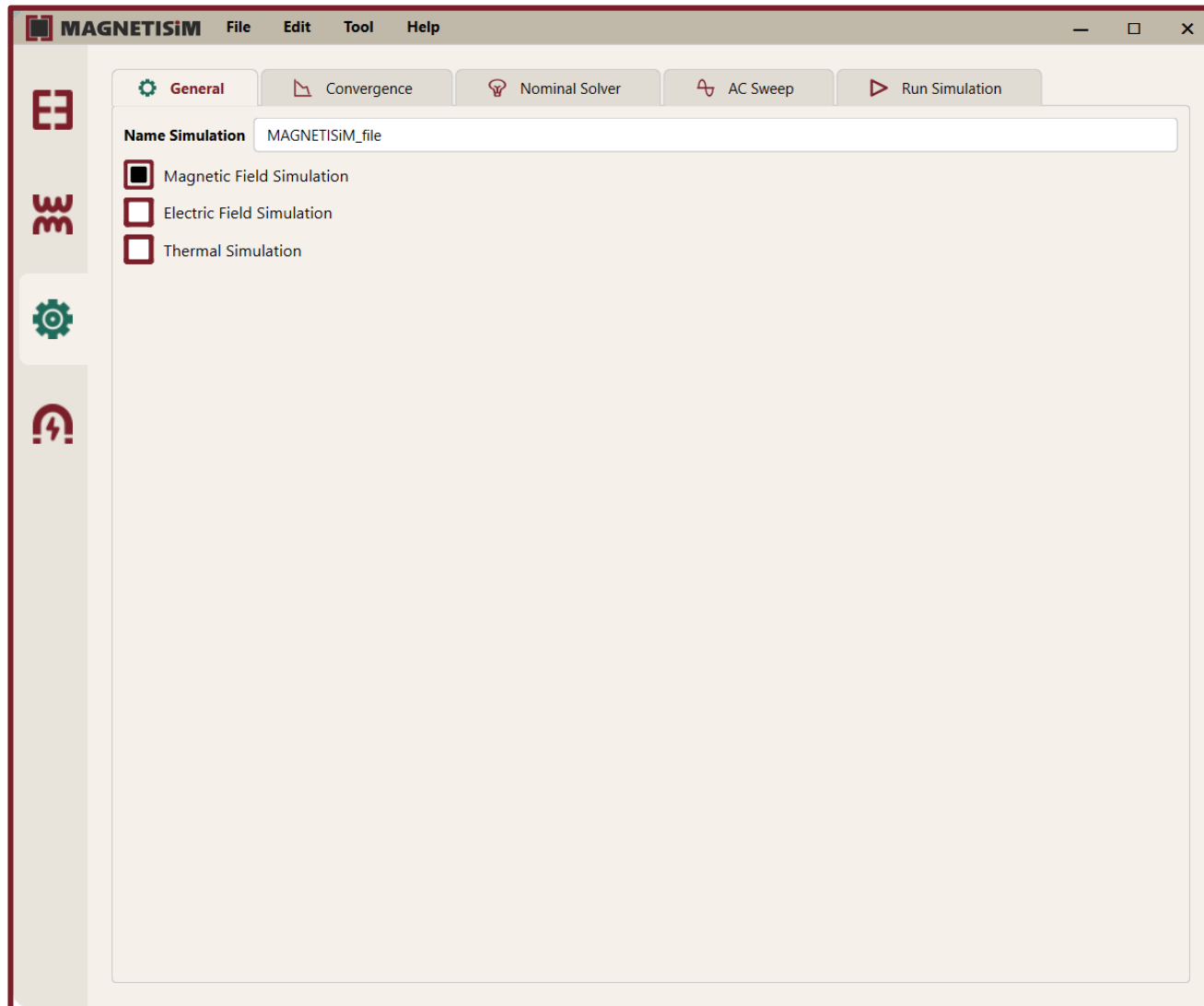
## MODELER WINDOW

# Defining the simulation



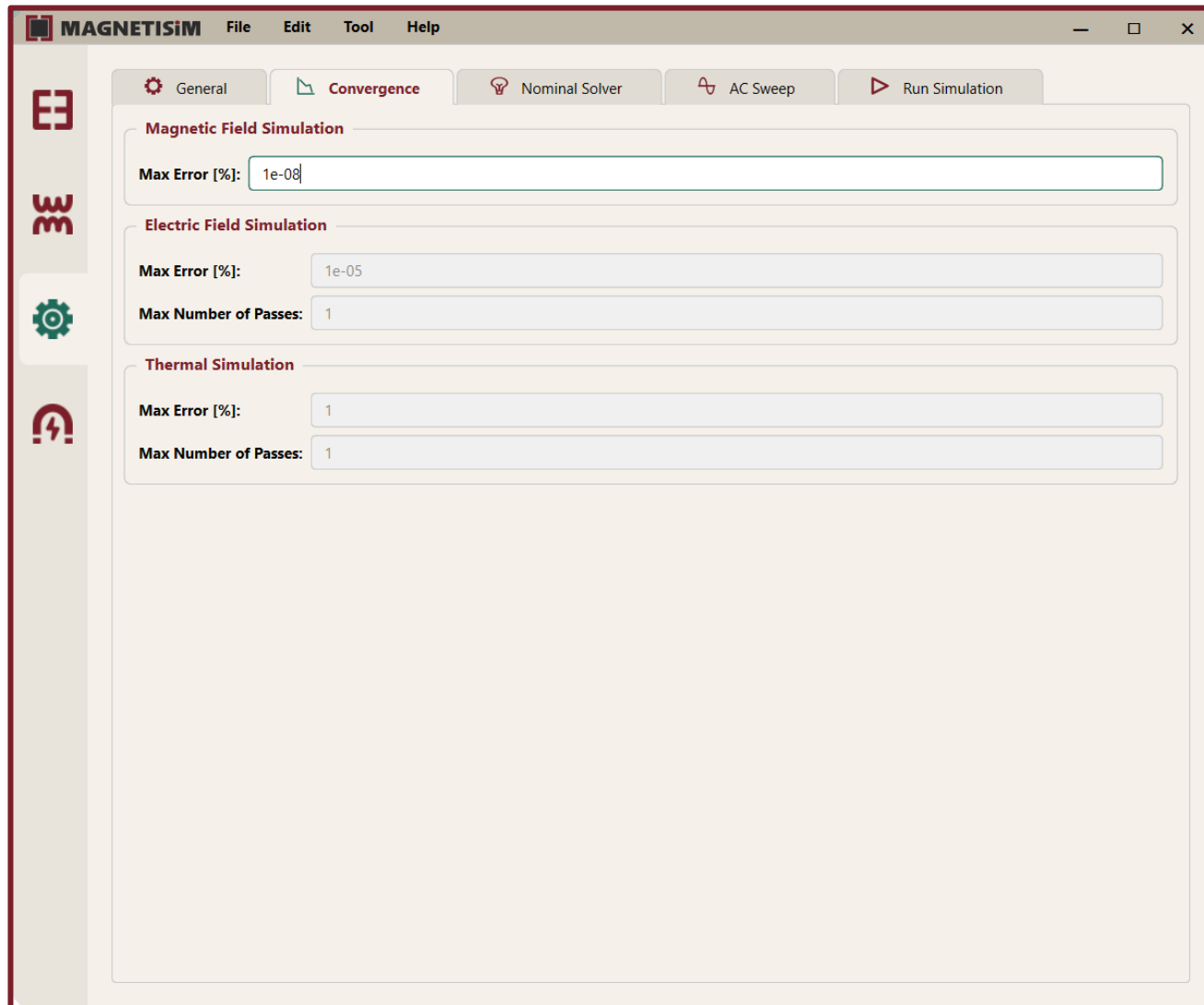
In the **General tab** under the Simulation Configuration menu, you must first assign a name to your file in the "**Name Simulation**" field.

# Magnet Simulation



Then, check the box next to "**Magnetic Field Simulation**" to activate the magnetic calculations for your project.

# Convergence

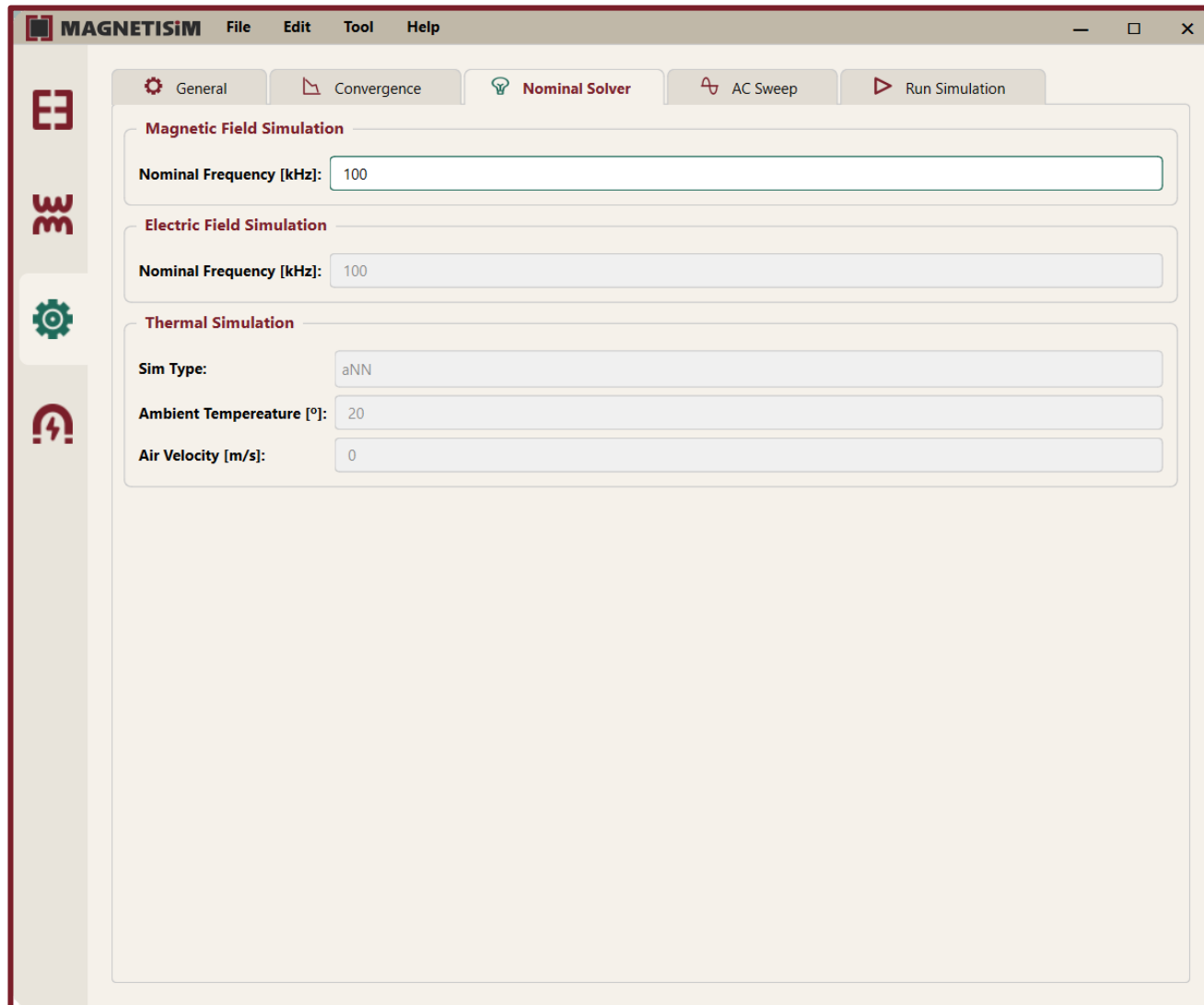


Next, move to the **Convergence tab**, where we define the mathematical precision criteria for the simulation solver.

By setting the "**Max Error [%]**" to a low value, such as 1e-08, you ensure high accuracy for your magnetic field results before the solver considers the simulation complete.

Due to the simulator requirements, it needs to be equal or lower than 1e-8.

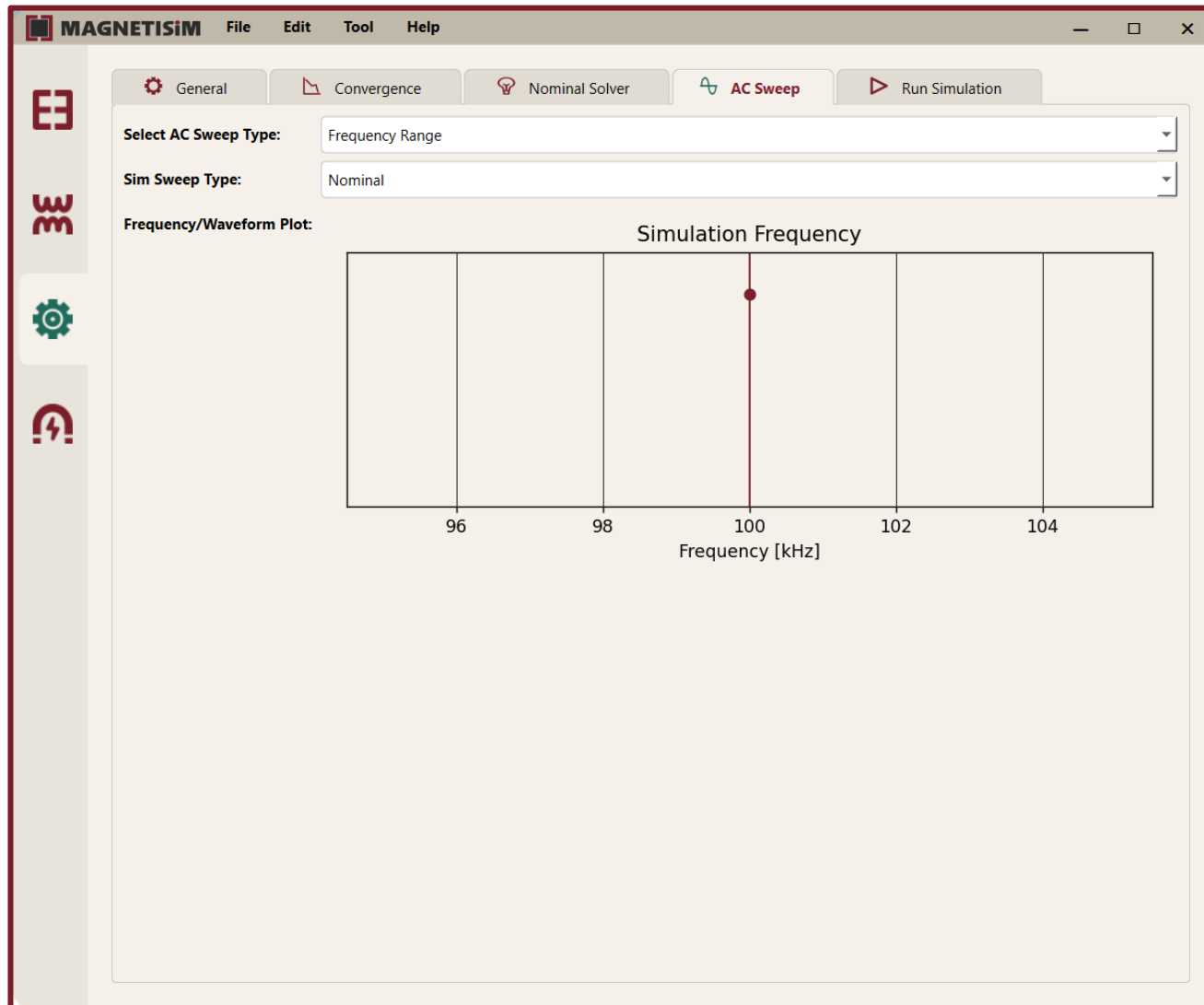
# Nominal Solver



In the **Nominal Solver tab**, you establish the main operating frequency of your component.

Simply input the value into the "**Nominal Frequency [kHz]**" field (for example, setting it to 100 kHz).

# AC Sweep



The **AC Sweep tab** allows you to analyze how your design responds across different frequencies. First, set the "**Select AC Sweep Type**" to "**Frequency Range**".

**Nominal** - If you select "**Nominal**" as your **Sim Sweep Type**, the system will only perform calculations at the base frequency you defined earlier. The **Frequency plot** will show a single point (e.g., at 100 kHz).

# AC Sweep

**MAGNETISiM** File Edit Tool Help

General Convergence Nominal Solver **AC Sweep** Run Simulation

Select AC Sweep Type: Frequency Range

Sim Sweep Type: Logarithmic

Min Frequency [kHz]: 1

Max Frequency [kHz]: 200

Number of Points: 3

Include Nominal Frequency

Frequency/Waveform Plot: Simulation Frequency

10<sup>0</sup> 10<sup>1</sup> 10<sup>2</sup>

Frequency [kHz]

**Logarithmic / Linear** - You can sweep across a broad spectrum by choosing a Logarithmic or Linear sweep type. You define the scope by entering a "**Min Frequency**" (e.g., 1 kHz), a "**Max Frequency**" (e.g., 200 kHz), and the "**Number of Points**" you want to calculate.

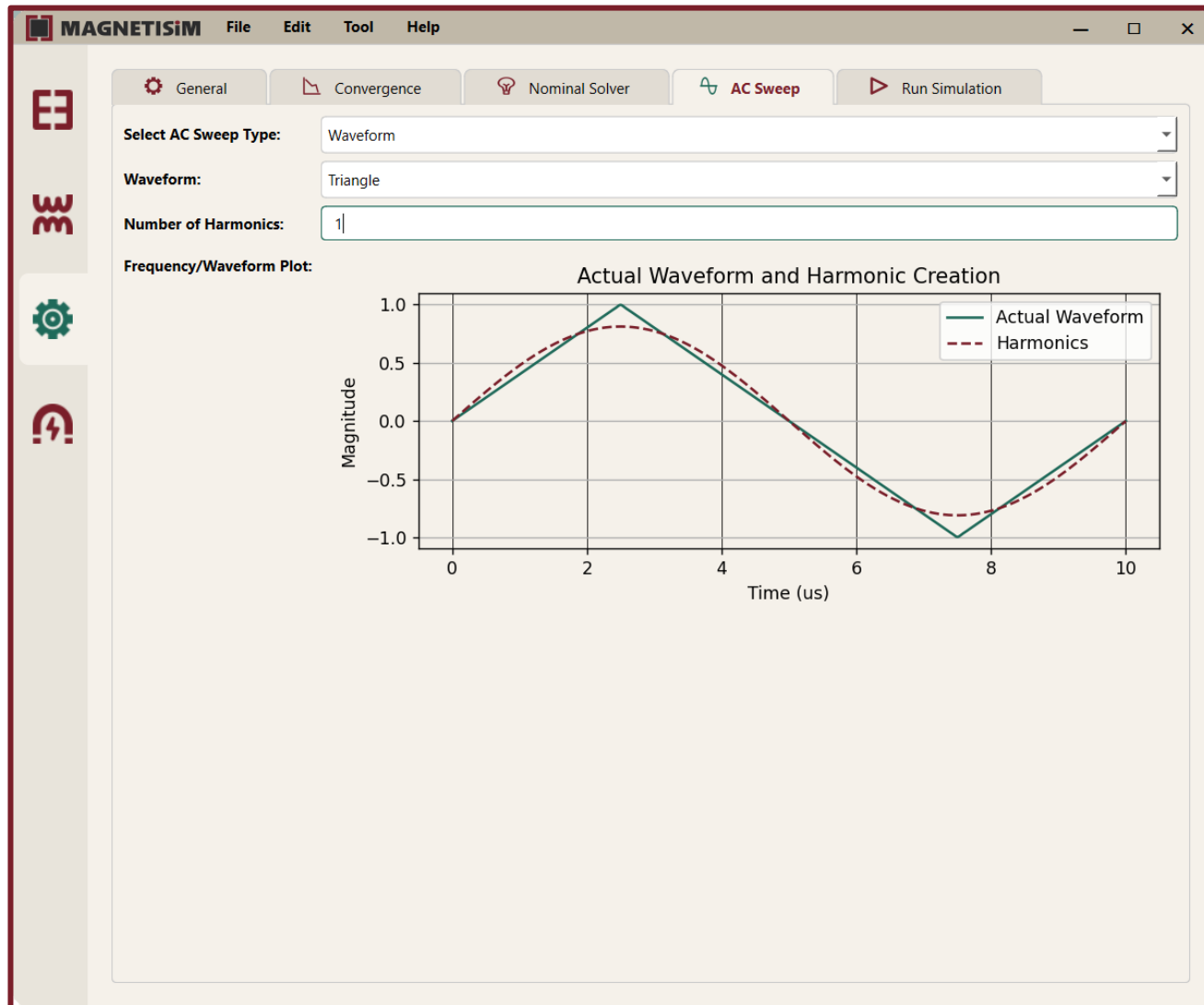
# AC Sweep

The screenshot shows the MAGNETISiM software interface with the 'AC Sweep' configuration panel. The 'Include Nominal Frequency' checkbox is checked. Below the configuration is a 'Simulation Frequency' plot showing four discrete points at 0, 66.67, 100, and 133.33 kHz.

Simulation Point	Frequency [kHz]
1	0
2	66.67
3	100
4	133.33

Checking the "**Include Nominal Frequency**" box ensures that your main operating point is evaluated within this sweep. The **visual Frequency Plot** below will update to show you exactly where the simulation points will occur along the axis.

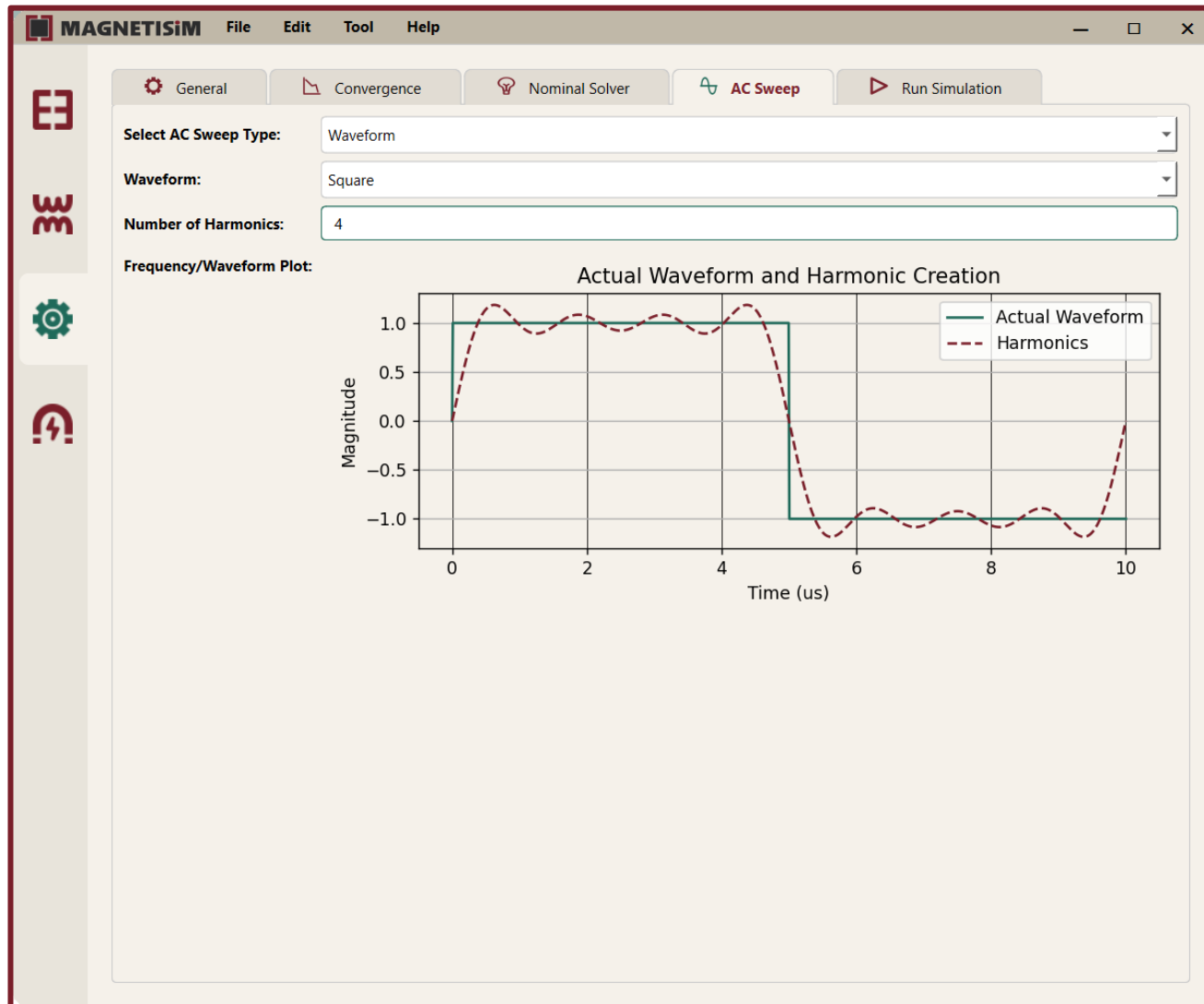
# AC Sweep



If you change the **Sweep Type** to **Waveform**, you can simulate non-sinusoidal excitations.

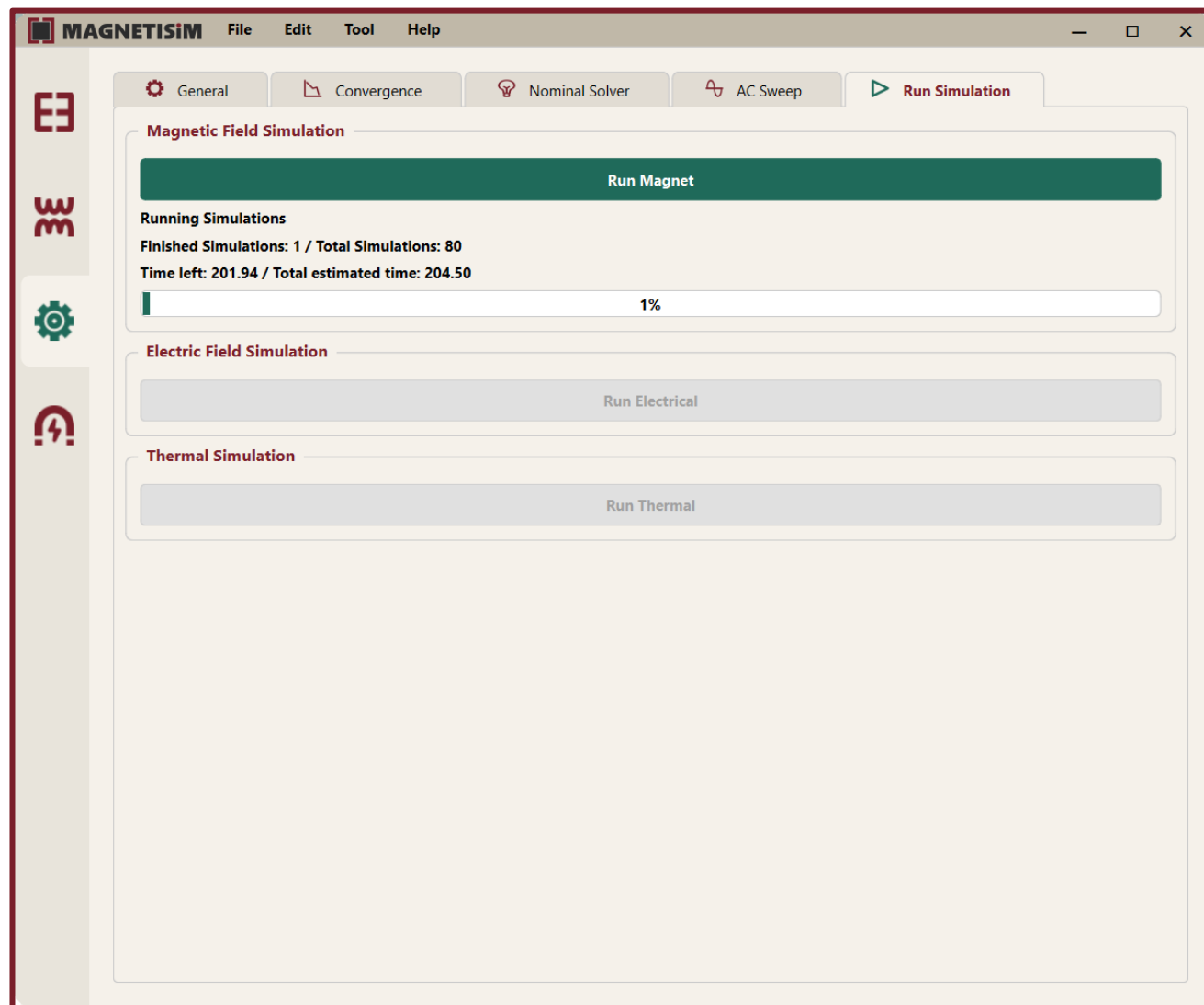
You can select specific wave shapes like "**Triangle**" or "**Square**" from the drop-down menu.

# AC Sweep



A critical parameter here is the "**Number of Harmonics**". As shown in the "**Actual Waveform and Harmonic Creation**" plots, a low number (like 1 harmonic for a triangle wave) provides a basic approximation.

However, increasing the harmonics (like using 4 harmonics for a square wave) makes the simulated dashed line closely match the ideal solid line, which is essential for accurate power loss calculations.



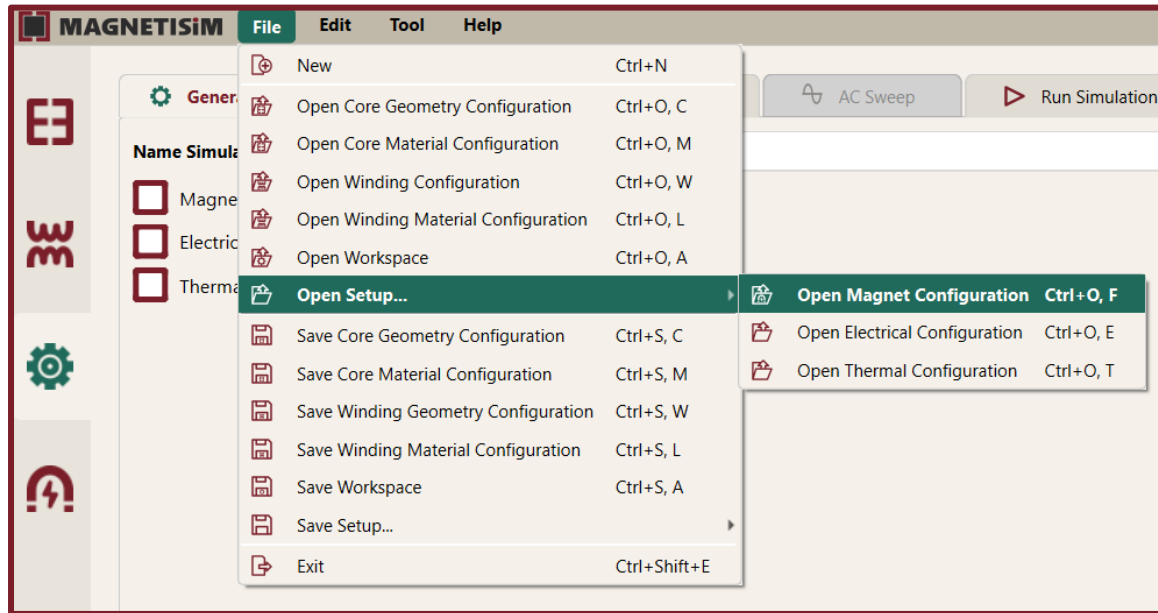
Once all parameters are set, go to the **Run Simulation tab** and click the large green "**Run Magnet**" button.

You can monitor the progress through the status bar, which displays the percentage completed, the number of finished simulations out of the total (e.g., 1 / 80), and the estimated time remaining.

# Magnet Simulation – .json

All of the **simulation parameters** you just configured (including maxError, NominalFreq, ACType, SweepType, and waveform harmonics) are structured and saved automatically as a .json file. This makes it easy to load previous setups.

You can import configurations by navigating to the top menu: **File -> Open Setup... -> Open Magnet Configuration**, or by using the keyboard shortcut **Ctrl+O, F**.



```
{  
  "SimName": "MagnetSim_File",  
  "MagnetCheck": true,  
  "maxError": 1e-08,  
  "NominalFreq": 25.0,  
  "ACType": "Frequency Range",  
  "SweepType": "Logarithmic",  
  "f_min": 10.0,  
  "f_max": 50.0,  
  "n_points": 2,  
  "NomIncluded": true,  
  "WaveformAC": "Triangle",  
  "nHarm": 1  
}
```



**Alberto Vital Pagola**

[alberto.vital@upm.es](mailto:alberto.vital@upm.es)

CEIMM-UPM

Madrid, Spain



**Alberto Delgado**

[a.delgado@upm.es](mailto:a.delgado@upm.es)

CEIMM-UPM

Madrid, Spain



[magnetisim.com](http://magnetisim.com)