AN-SOF User Guide › AN-SOF Antenna Simulation Software

AN-SOF Antenna Simulation Software

Accurate, Fast, and Easy-to-Use Tool for Antenna Modeling, Analysis, and Design

Welcome to AN-SOF!

Congratulations on choosing **AN-SOF**, the best combination of **ease of use** and **accuracy** you can find in an electromagnetic simulator for the modeling and design of antennas and wire structures in general. This **User Guide** describes AN-SOF and its many features in detail. Here, you will also find step-by-step examples and tips to help you quickly progress with your antenna modeling projects.

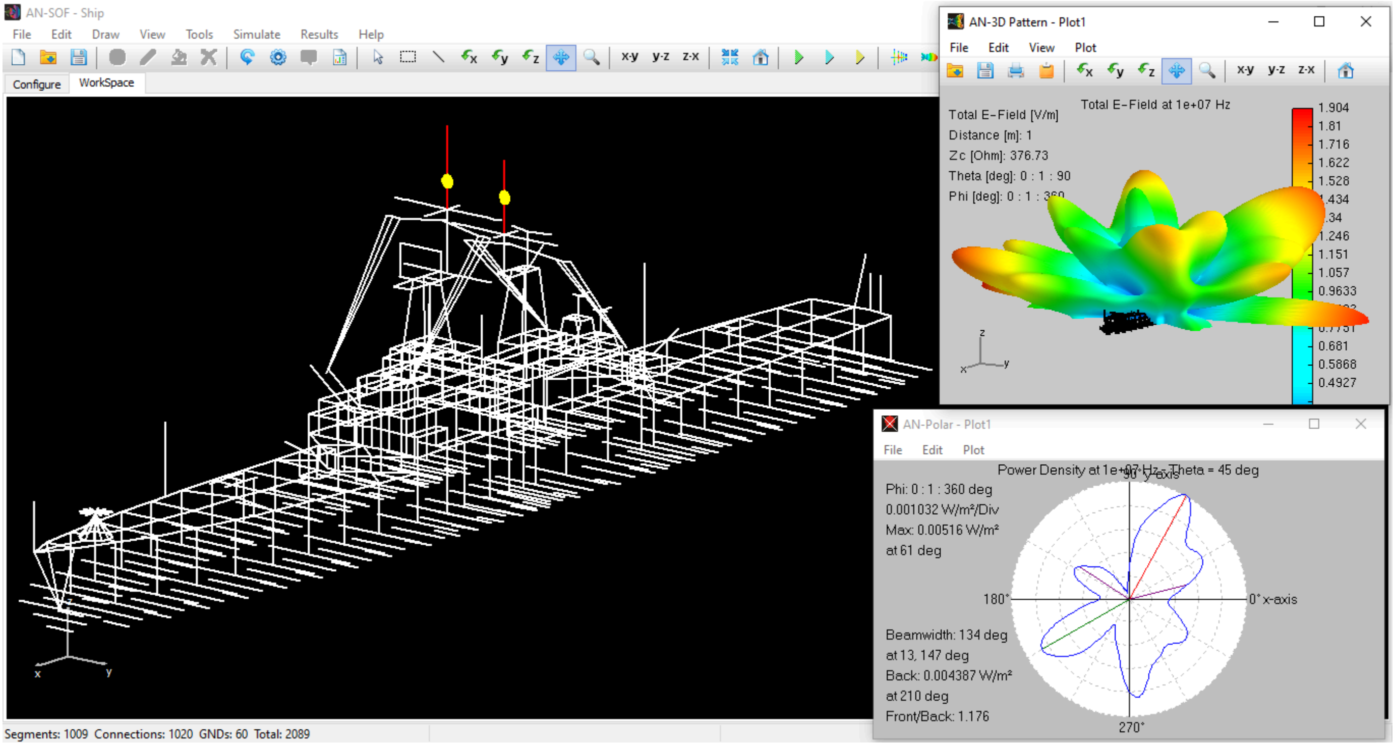
Table of Contents

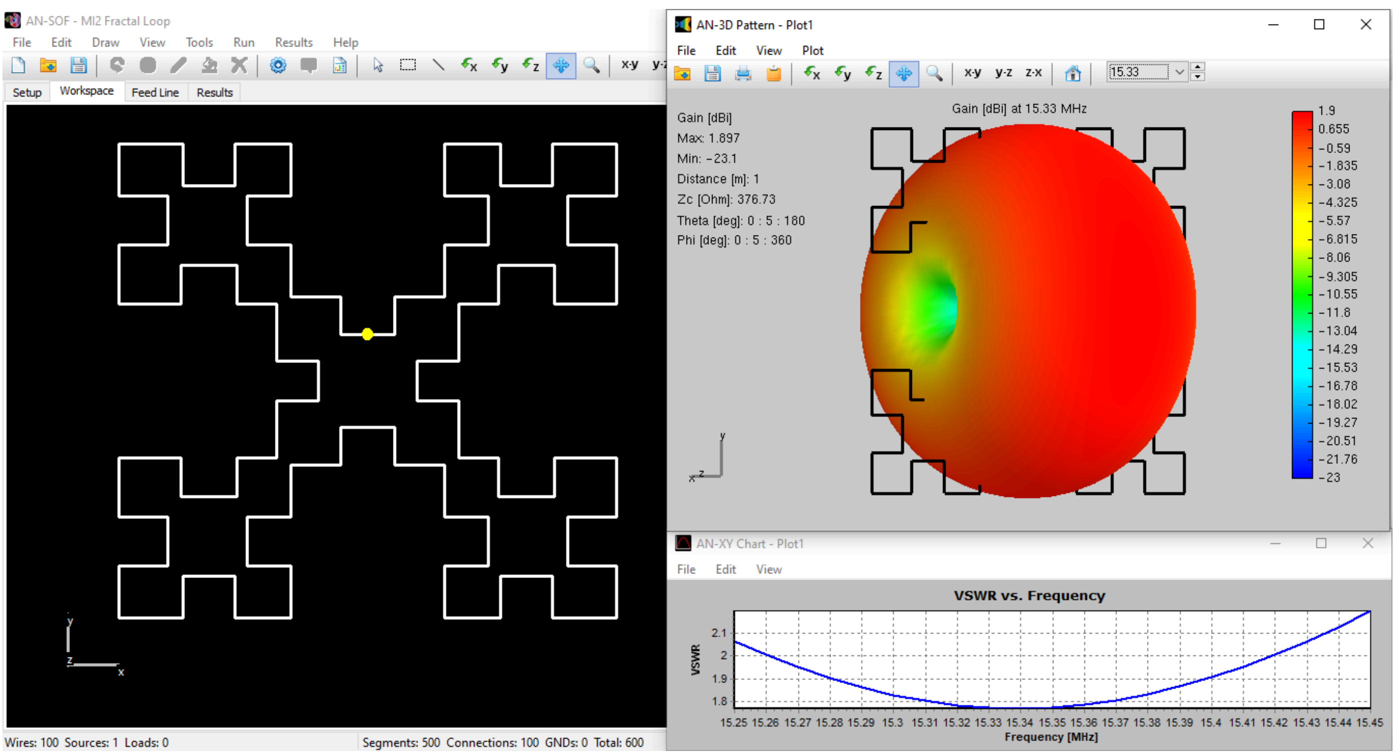
Getting Started

Quick Start Guide

Enhancing Antenna Design Through Simulation Software

An antenna model represents a real-world antenna within a computer program. It is important not to confuse this type of model with a scale model, which is sometimes constructed to measure the radiation characteristics of a larger, physically identical antenna. Due to the **mathematical complexity** involved in modeling, specialized software is often used to **predict and analyze antenna performance**.





Computer simulation plays a critical role in overcoming challenges and driving innovation throughout the product development lifecycle. A computer model offers significant advantages, including the ability to **modify, redesign, break, destroy, and rebuild designs multiple times** without wasting physical materials. By leveraging simulation software, engineers can **significantly reduce the costs** associated with building successive physical prototypes, streamlining the design process.

AN-SOF is a comprehensive simulation software suite designed for **antenna modeling and optimization**. It supports the design of a wide range of wire antennas, including dipoles, monopoles, Yagis, log-periodic arrays, helices, spirals, loops, horns, fractals, phased arrays, and many other types. Additionally, AN-SOF enables **detailed modeling of feeding systems using transmission lines**, allowing users to analyze complex antenna configurations with precision. The software can simulate antennas positioned above **lossy ground planes** or broadcast antennas above radial wire ground screens, providing **realistic and accurate results**.

Furthermore, **AN-SOF's calculation capabilities** have been extended to include **single-layer microstrip patch antennas** and the computation of radiated emissions from **Printed Circuit Boards (PCBs)**. As a result, AN-SOF is a powerful tool for **Electromagnetic Compatibility (EMC) applications**. The software also supports passive circuits with **lumped impedances** and non-radiating networks, enabling **comprehensive analysis of antenna systems**.

> **Note**
>
> In the realm of antenna applications, AN-SOF proves invaluable as it empowers users to achieve the following:
>
> - Design superior antennas.
> - Predict and optimize antenna performance.
> - Fine-tune antenna parameters for optimal results.
> - Account for environmental effects on antenna performance.
> - Employ script-based optimization to refine designs.
> - Gain valuable insights into antenna behavior.
> - Experiment multiple times prior to physically building the antenna model.
> - Deepen understanding of antennas and their properties.
> - Facilitate knowledge sharing and collaboration with colleagues.

With **AN-SOF at your disposal**, you can explore the exciting possibilities of **antenna analysis and optimization**. The software provides an **extensive toolkit** for designing, evaluating, and enhancing antenna performance, empowering engineers and enthusiasts alike to push the boundaries of innovation.

> **Note**
>
> AN-SOF enables us to perform a wide range of tasks, including:
>
> - Describing the antenna's geometry accurately.
> - Selecting appropriate construction materials.
> - Specifying the environmental and ground conditions.
> - Determining the antenna's height above the ground.
> - Analyzing the radiation pattern and front-to-back ratio.
> - Plotting directivity and gain.
> - Evaluating impedance and SWR (Standing Wave Ratio).
> - Predicting bandwidth.
> - Obtaining numerous additional parameters and plots.

Drawing the geometry of structures in AN-SOF is intuitive and user-friendly. Wires can be created in a 3D space using the mouse, menus, and easy-to-navigate dialog windows. Tools are available to **zoom, move, and rotate the structure**, ensuring precise control over the design process.

To visualize simulation results, AN-SOF integrates seamlessly with a suite of specialized applications: **AN-XY Chart**, **AN-Smith**, **AN-Polar**, and **AN-3D Pattern**. These tools allow users to display graphs and analyze data effectively. They can also be executed independently for further graphic processing, offering **flexibility and convenience** for advanced users.

With AN-SOF and its software suite for displaying graphics, we have all the necessary tools to guide us through the stages of an antenna design process.

Learn more

Introduction to AN-SOF: Antenna Simulation Essentials

AN-SOF performs computations of **electric currents flowing on metallic structures**, including antennas in transmitting and receiving modes, as well as **scatterers**. A scatterer refers to any object capable of reflecting and/or diffracting radiofrequency waves. For instance, wave scattering analysis can be conducted on the surface of an aircraft to determine **optimal antenna placement**, on a parabolic reflector to examine gain in relation to the reflector shape, or on a car's chassis to predict **interference effects**.

The **Method of Moments (MoM)** stands as one of the most widely validated techniques for antenna simulation. AN-SOF incorporates an enhanced and advanced version of this method called the **Conformal Method of Moments (CMoM) with Exact Kernel**, which addresses various challenges associated with traditional MoM approaches and achieves **unparalleled accuracy**.

Interested in learning more about the CMoM implementation in AN-SOF? Read this article:

Overcoming 7 Limitations in Antenna Design: Introducing AN-SOF's Conformal Method of Moments

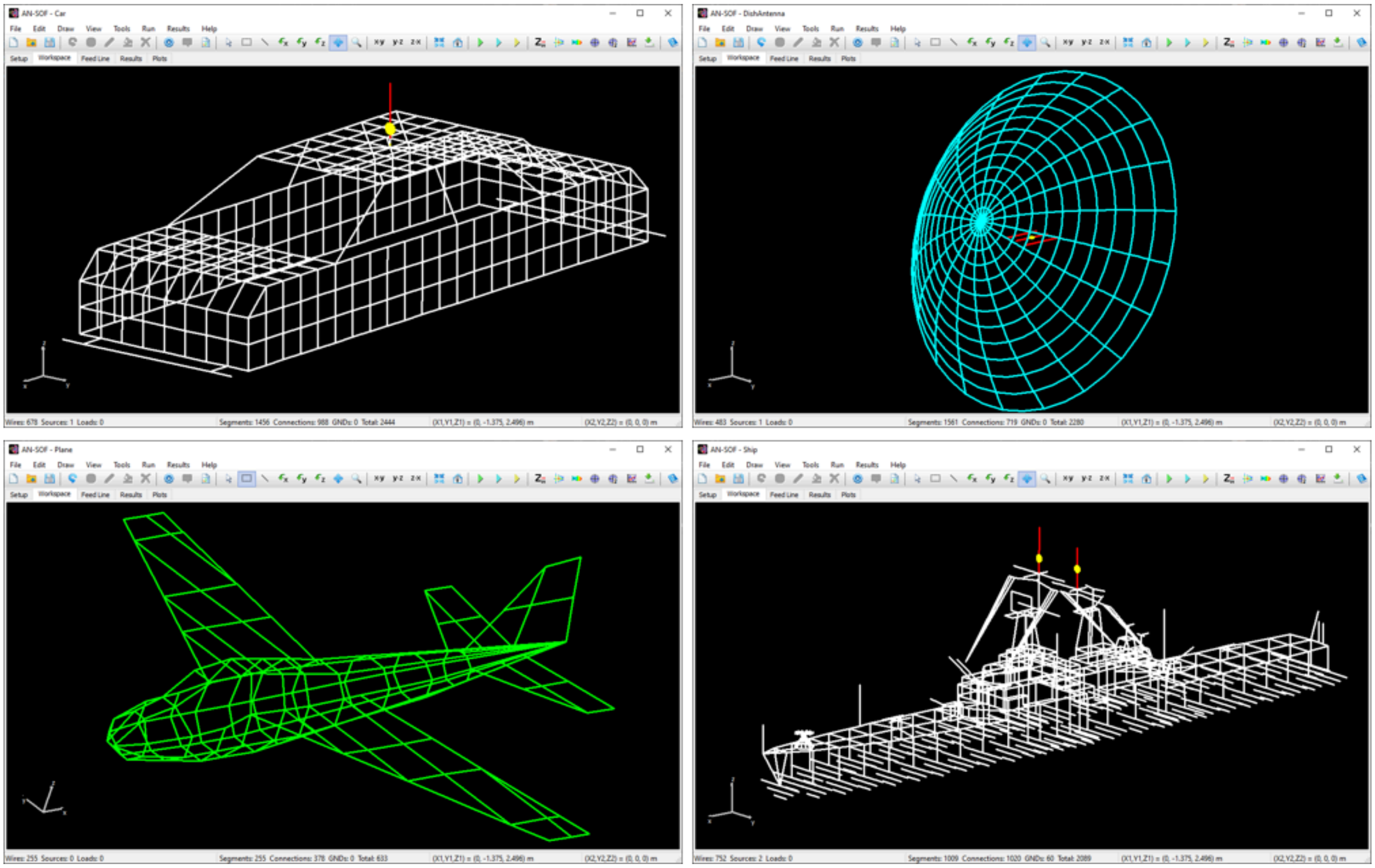


Fig. 1: Computer models of a car, a parabolic reflector, an airplane, and a ship using wire grids.

According to the MoM, any metallic structure can be represented using **conductive wires**, as illustrated in Fig. 1. These wires are subdivided into **small segments**, which assume the shape of cylindrical tubes. To obtain **accurate results**, the length of each wire segment should be comparatively **short compared to the wavelength**, as depicted in Fig. 2. However, this concern can be alleviated during the initial simulation since **AN-SOF automatically handles the segmentation of wires**.

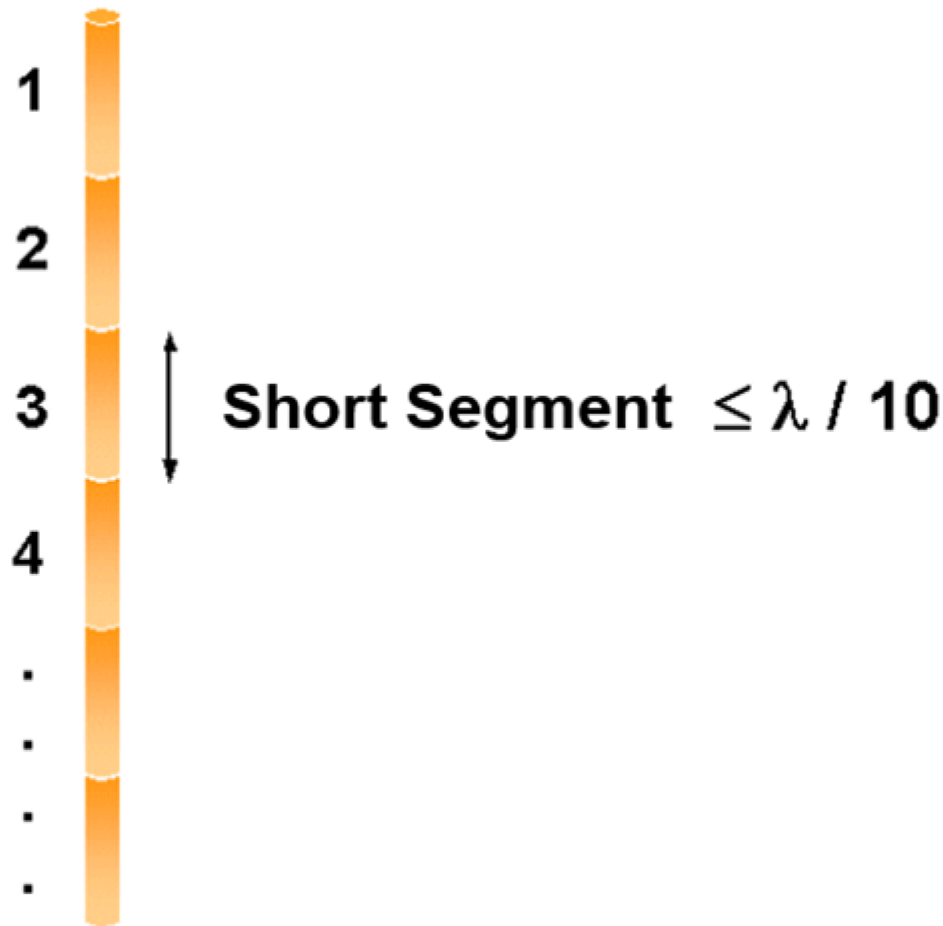


Fig. 2: A straight wire divided into short segments relative to the wavelength.

The flow of electric currents within the structure can be achieved by introducing a **voltage generator** at a specific location operating at a given frequency. **Current generators** can also serve as the excitation source, alongside **plane waves** impinging on the structure from distant sources. Once the geometry, materials, and sources of the structure are defined, the computation can be executed to determine the currents flowing through the **wire segments**. Generally, these electric currents exhibit varying intensities along and across the structure, collectively referred to as a **current distribution**. Fig. 3 showcases an example of the current distribution on a log-periodic antenna.

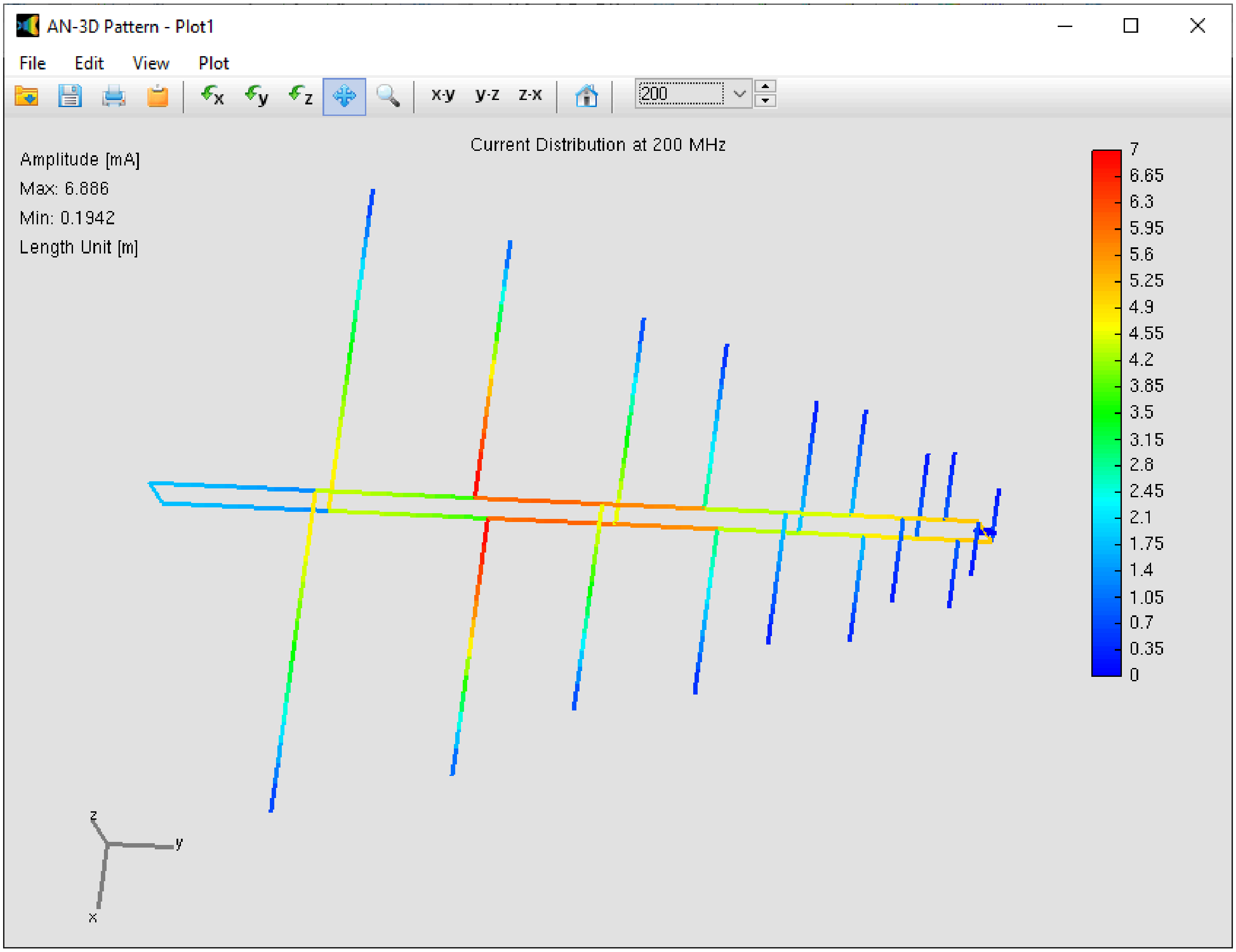


Fig. 3: Current distribution on a log-periodic antenna. The color map on the structure indicates the amplitude of the electric currents.

In the subsequent phase of the simulation process, the **electromagnetic field radiated by the current distribution** can be calculated. However, the current distribution itself provides valuable insights into the behavior of the structure, particularly when a **frequency sweep** is conducted. In the case of antennas, the feed point impedance can be analyzed as a function of frequency to assess the **bandwidth**. The **Voltage Standing Wave Ratio (VSWR)** can be plotted on a **Smith chart** for better interpretation of the results, as demonstrated in Fig. 4. The electric and magnetic fields in the proximity of the structure, known as the **near-field zone**, can be obtained and visualized as a color map, with intensities often resembling temperature maps used in weather forecasts, as shown in Fig. 5.

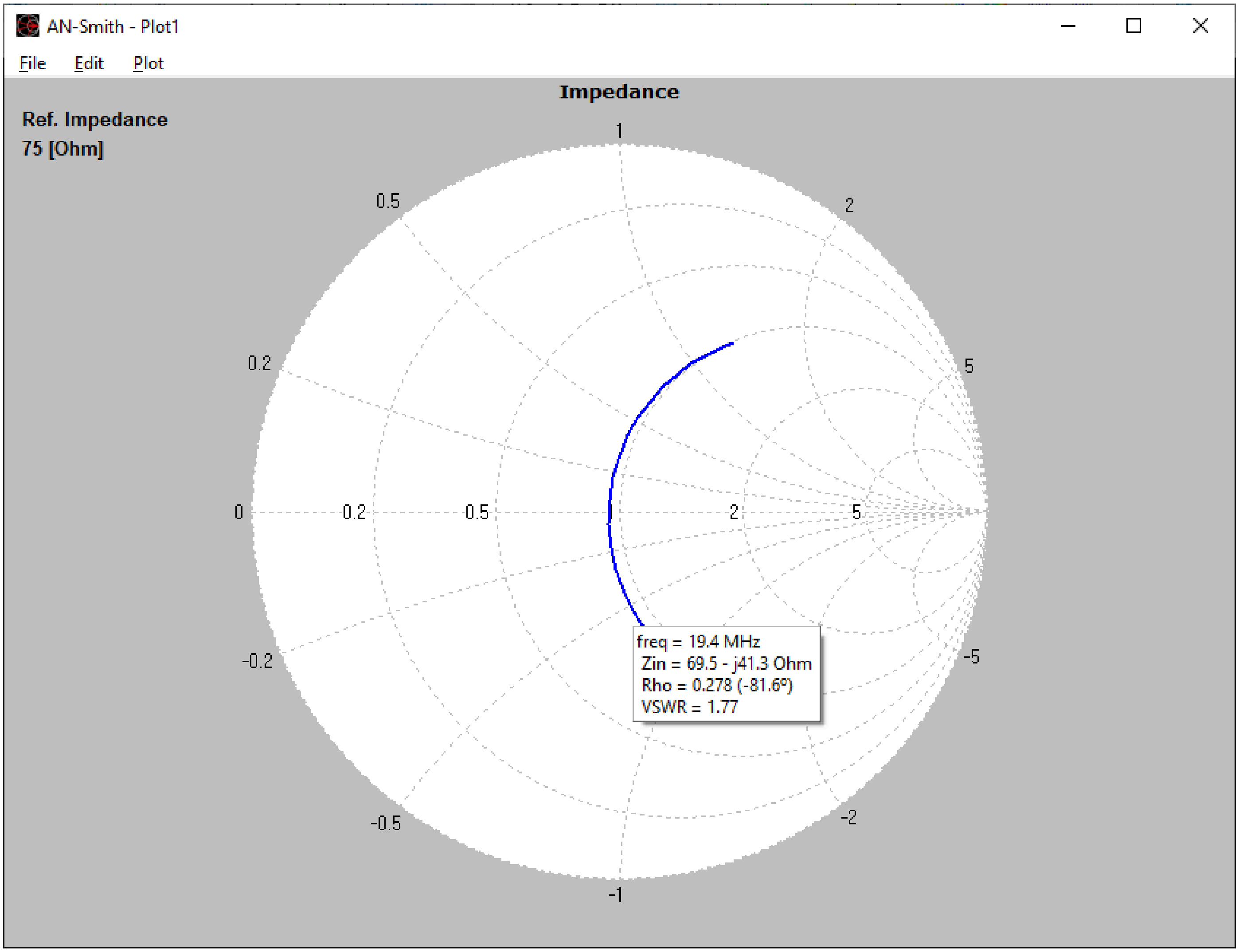


Fig. 4: Impedance plotted as a function of frequency on a Smith Chart, where the VSWR can be obtained by clicking on the curve.

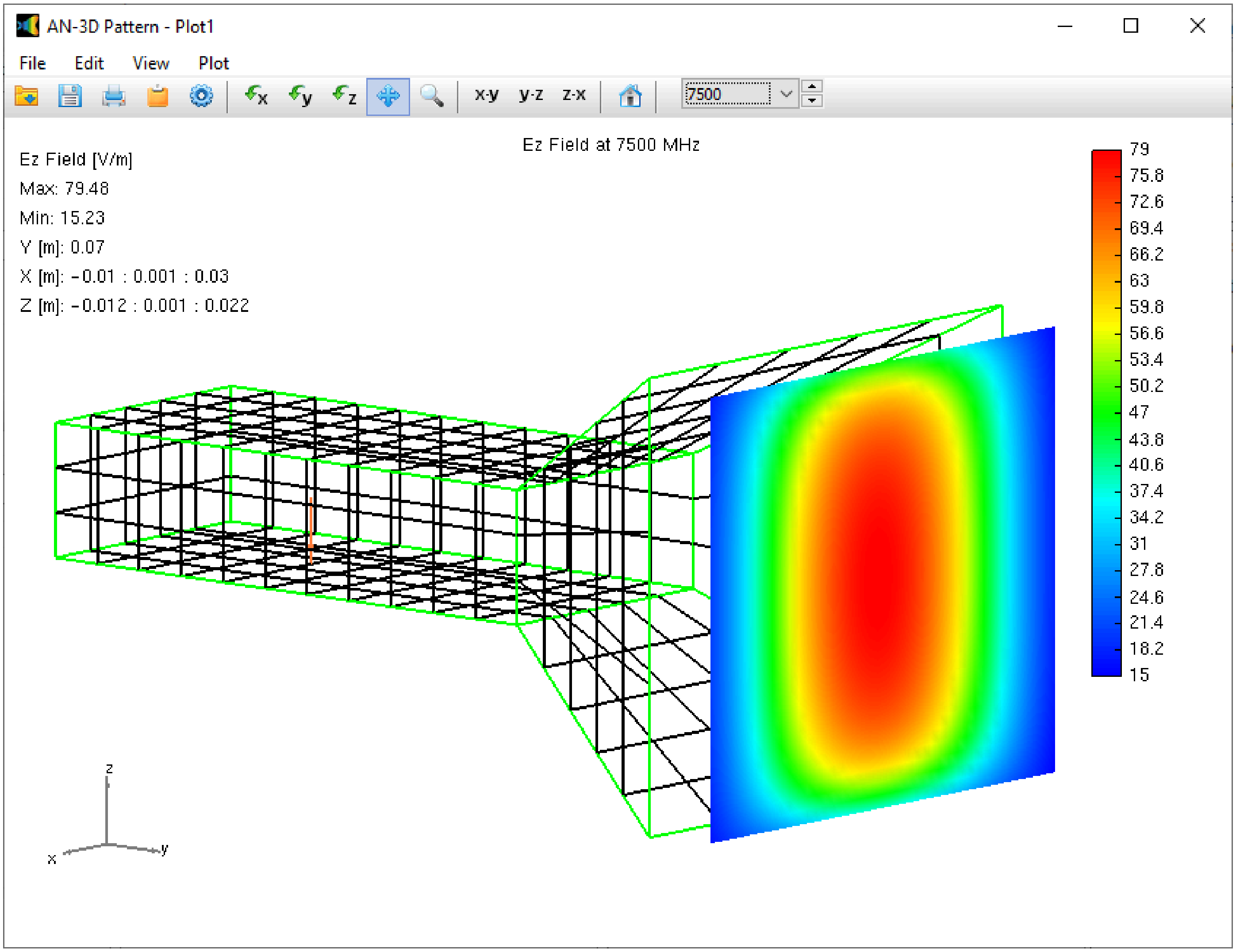


Fig. 5: Near electric field in the vicinity of a Horn antenna.

In the **far-field zone**, situated several wavelengths away from the structure, the magnetic field becomes proportional to the electric field. As a result, the electric field intensities are commonly used to analyze the results. This region is depicted in polar diagrams, as illustrated in Fig. 6, where the radiated field is represented as a function of direction. A more comprehensive representation can be achieved by plotting a **3D pattern**, where radiation lobes can be superimposed onto the structure's geometry, providing **enhanced visualization of its directional properties**, as exemplified in Fig. 7.

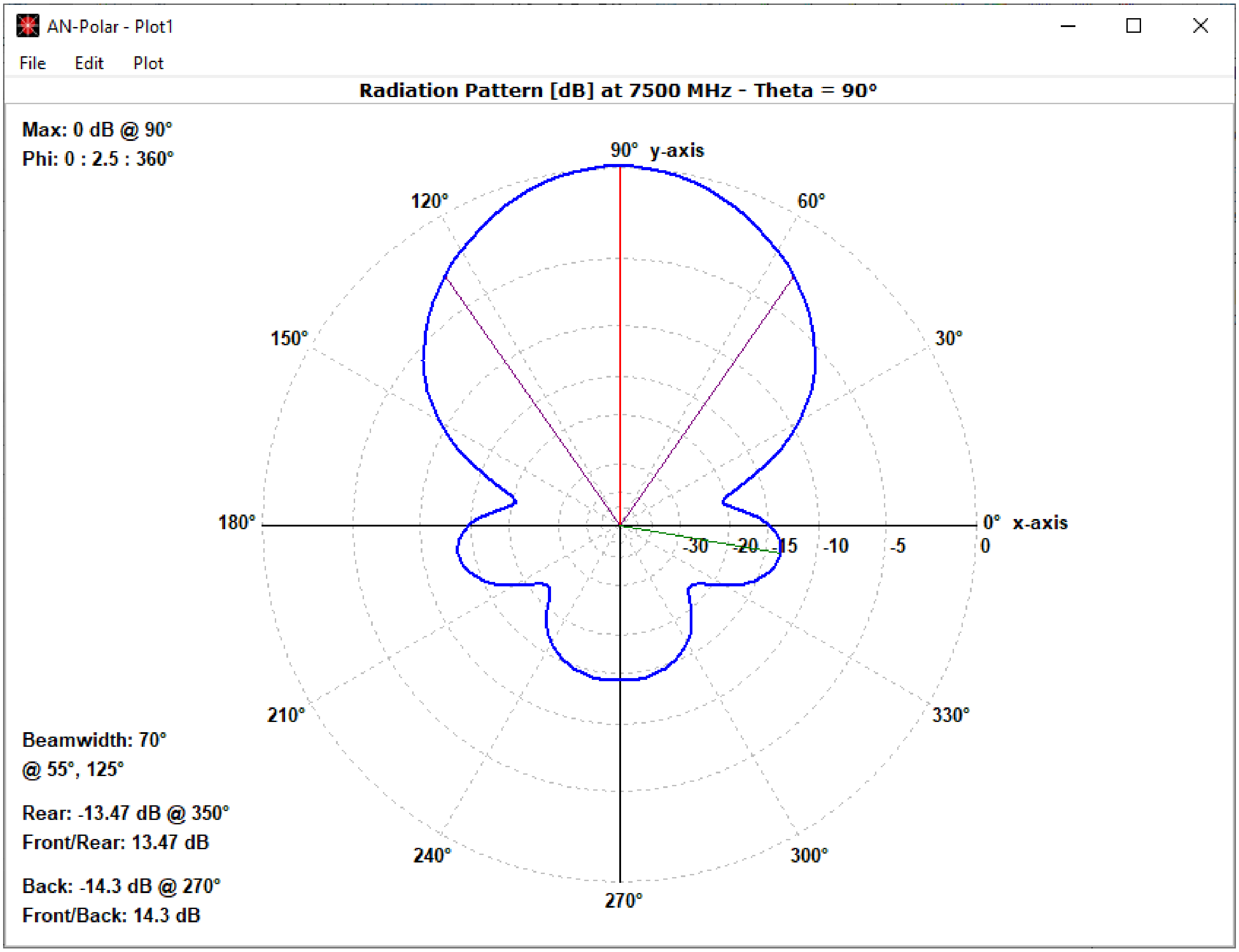


Fig. 6: Far-field pattern represented in a polar diagram, indicating beamwidth, front-to-rear ratio, and front-to-back ratio.

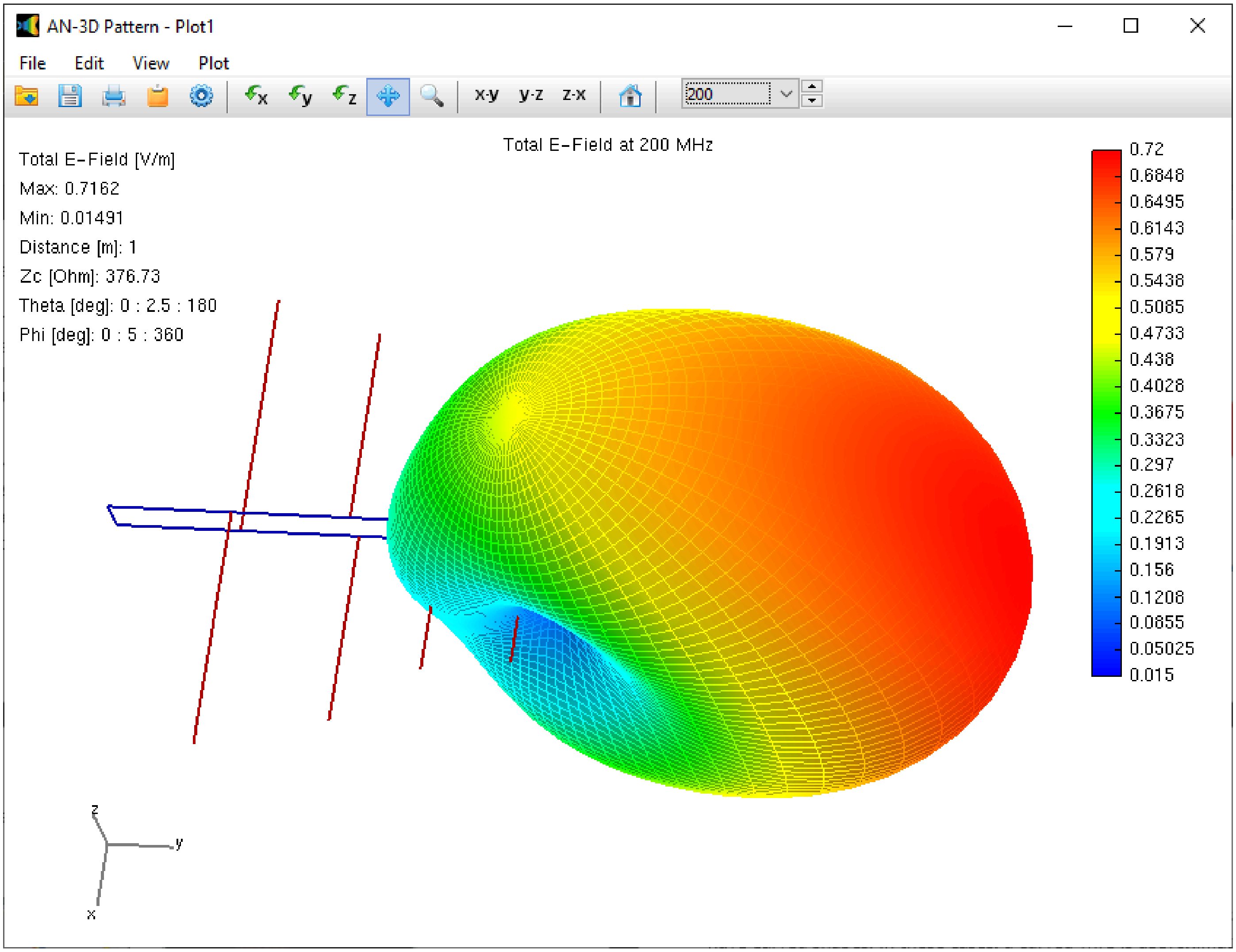


Fig. 7: Far-field pattern represented in a 3D plot, superimposed onto the antenna geometry.

AN-SOF stands out as the easiest-to-use software tool for simulating antennas, particularly those that can be modeled using conductive wires. Are you ready to embark on your first simulation? Let's get started!

Explore Our Pre-Computed Examples in the Models Tab

AN-SOF includes a collection of pre-calculated models that enable users to quickly load example projects directly into the interface. These models are organized into categories for easy navigation, making this feature especially useful for users exploring example designs and learning key antenna concepts.

In the **Models tab**, you'll find quick-access buttons for opening example models (see Fig. 8). Each model displays a 3D radiation pattern and includes a PDF guide with informational resources. Since all models are pre-calculated, they can be opened and explored using the **AN-SOF Trial version**.

The **total number of segments** used in each model is displayed after the “|” symbol on each button label in the **Models** tab, as well as in the **Status Bar** at the bottom of the AN-SOF window. This total includes:

- **Wire segments**
- **Wire-to-wire junctions**
- **Wire-to-ground connections** (labeled as “GNDs”)

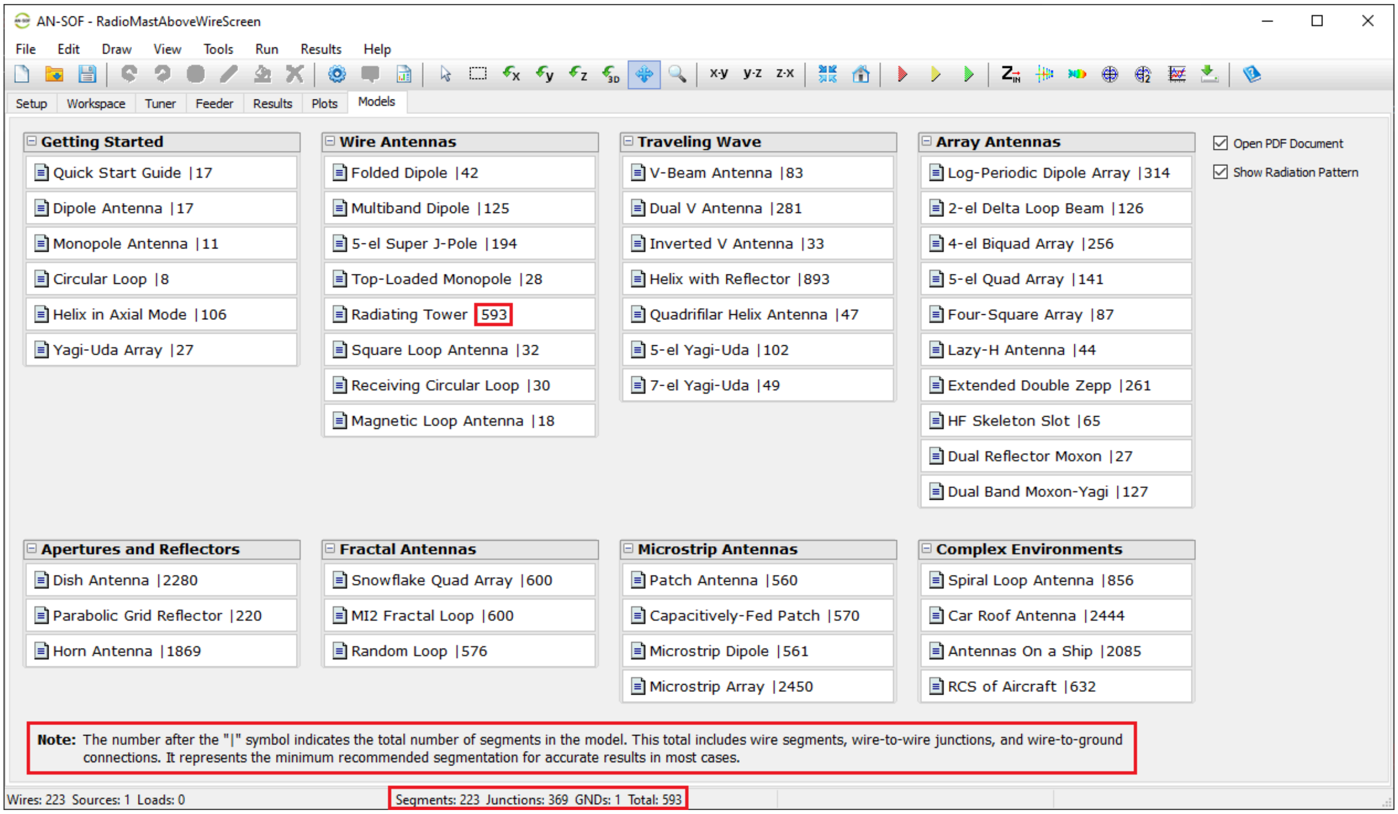


Fig. 8: The Models tab in the AN-SOF interface.

Note:

[The AN-SOF Trial version supports models with up to 50 total segments](). If you modify an example model and exceed this limit, you’ll need an active license for the **AN-SOF PRO Edition** to run additional simulations.

To explore the example models, simply click the buttons in the **Models tab**. On the right side of the screen, you can use the options **“Open PDF Document”** and **“Show Radiation Pattern”** to toggle the display of the PDF guide or the radiation pattern plot as needed.

Performing the First Simulation with AN-SOF

Several example files are included in the AN-SOF installation directory, located within a folder named “Examples”. When opening a file with the extension “.emm”, the wire structure will be displayed on the screen. To run the calculation, click on the **Run ALL** button on the toolbar. The main results can be plotted by clicking on the following buttons: **Plot Current Distribution**, **Far-Field 3D Plot**, and **Far-Field Polar 1 Slice**.

As a first experience using AN-SOF, let's simulate a standard **half-wave dipole**, which is one of the simplest antennas that can be modeled. A dipole is a straight wire that is fed at its center. When the wire's cross-section is circular, it is referred to as a **cylindrical antenna**. Since the wire is typically made of a highly conductive material, it can be considered a **perfect conductor** with **zero resistivity**. Therefore, we will model a cylindrical antenna with zero resistivity in this example. Follow the steps below to perform this simulation.

Step 1: Setup

The first step is to set the operating frequency. Navigate to the **Setup tab** in the AN-SOF main window. Within the **Frequency panel**, there are three options to choose from. Select **Single** and enter the operating frequency for the antenna (see Fig. 9). In this case, the frequency is given in megahertz (MHz), and lengths are measured in meters (m). If desired, you can change the unit system for frequencies and lengths by going to **Tools > Preferences**. Please note that for a frequency of 300 MHz, the wavelength is approximately 1 meter (0.999308 m).

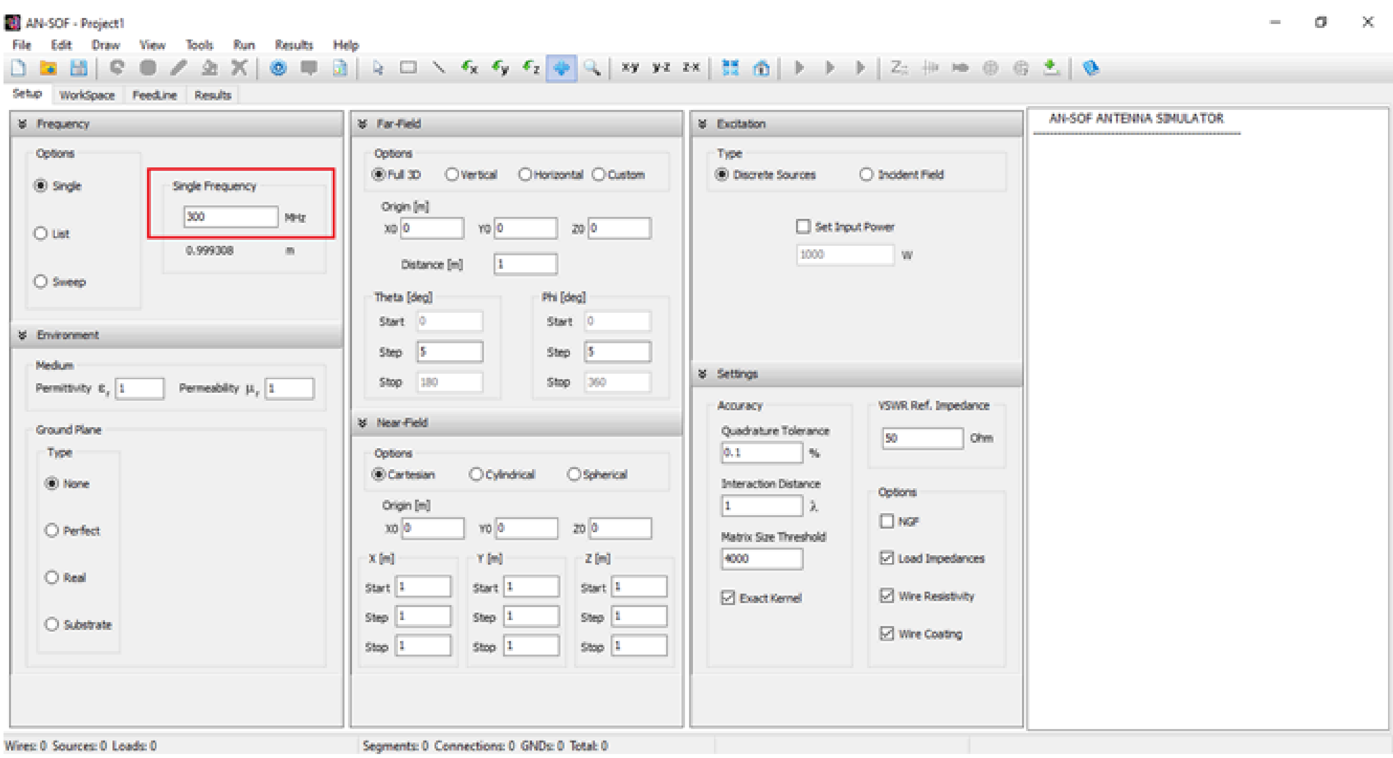


Fig. 9: The Single Frequency option in the Setup tab, where a frequency of 300 MHz is set.

Step 2: Draw

Once the operating frequency has been set, you can draw the antenna geometry on the **Workspace** tab. The workspace is where the wire structure is visualized, representing a 3D space that allows zooming, rotation, and movement.

In AN-SOF, a straight wire is referred to as a **Line**. To draw a line, go to the main menu and select **Draw > Line**. This will open the **Draw** dialog box. In the **Line** tab, you can set the coordinates of two distinct points.

For this example, we will create a line along the z-axis that is 0.5 meters long, corresponding to half a wavelength at 300 MHz. Figure 10 illustrates the chosen starting point of the line at (X1, Y1, Z1) = (0, 0, -0.25) m, and the ending point at (X2, Y2, Z2) = (0, 0, 0.25) m. Next, switch to the **Attributes** tab (see Fig. 11). To ensure accurate results, the line should be divided into segments that are relatively short compared to the wavelength. Generally, a segment length equal to or less than **one-tenth of a wavelength** is considered short. AN-SOF suggests a minimum number of segments to achieve reliable results automatically. If you require higher resolution, you can increase the number of segments.

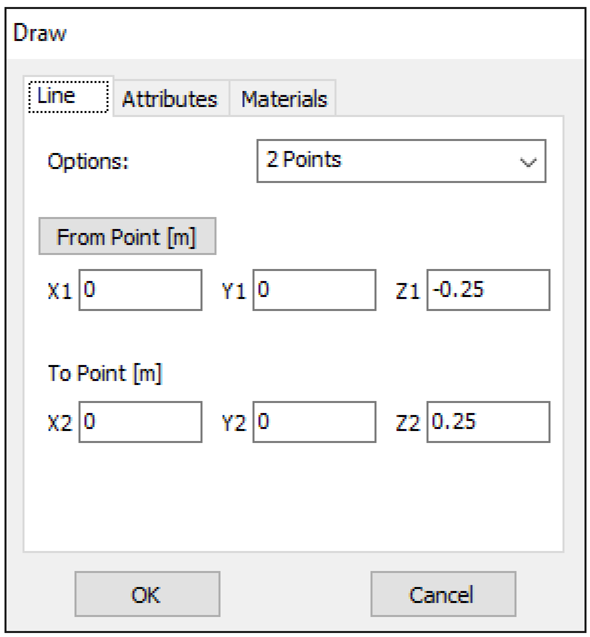


Fig. 10: The Line tab in the Draw dialog box for drawing a straight line.

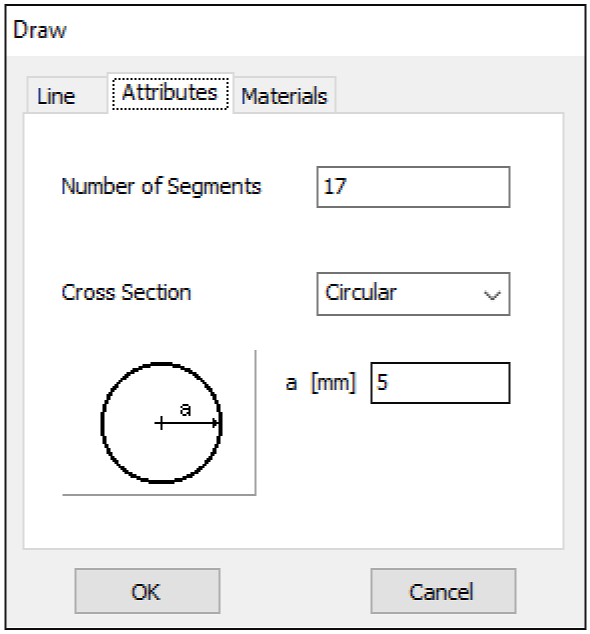


Fig. 11: The Attributes tab in the Draw dialog box, where you can set the number of segments and wire radius.

In this case, the line will be divided into 17 segments, and the wire cross-section will be circular with a radius of 5 millimeters. On the **Materials** tab (refer to Fig. 12), you can set the wire's resistivity to zero.

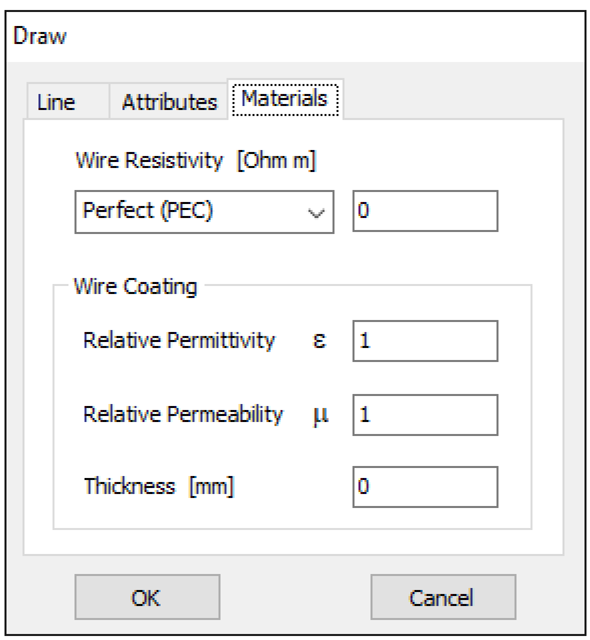


Fig. 12: The Materials tab in the Draw dialog box, used for setting the wire resistivity.

The next step is to feed the dipole. Right-click on the wire and select the **Source/Load** command from the **pop-up menu** that appears. A **toolbar** with a slider will be displayed at the bottom of the screen. Move the slider to the segment located at the center of the wire. Then, click the **Add**

Source button. Add a voltage source with an amplitude of 1 Volt and a phase of zero (see Fig. 13).

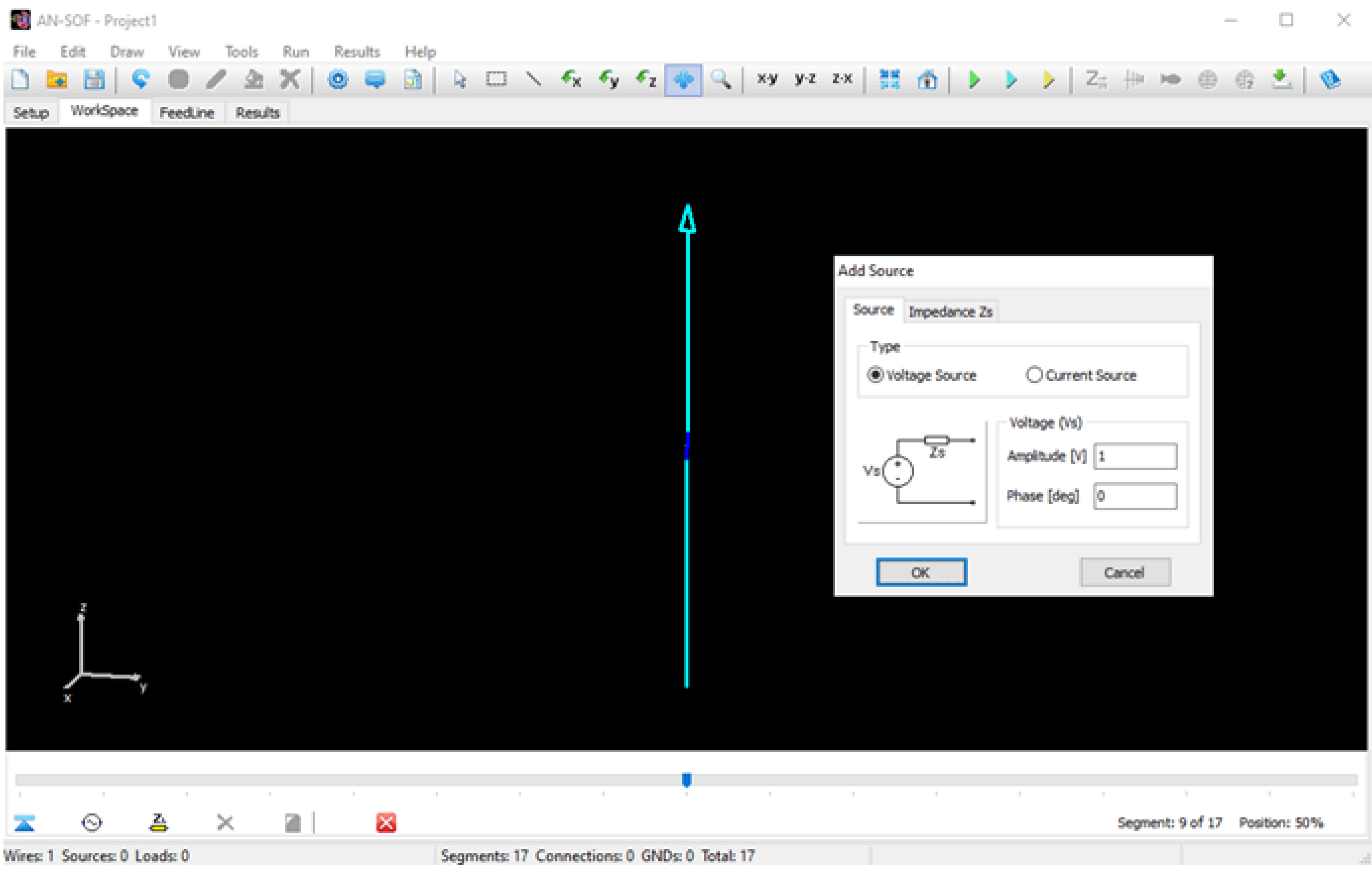


Fig. 13: The Add Source dialog box appears after clicking the Add Source button in the Source/Load toolbar at the bottom of the screen.

Step 3: Run

To run the calculation, go to **Run > Run Currents** in the main menu. Once the calculations are completed, proceed to **Run > Run Far-Field** in the main menu. This will calculate the current distribution on the dipole antenna and the radiated field.

AN-SOF provides **integrated graphical tools** for result visualization. Right-click on the wire and select **Plot Currents** from the displayed pop-up menu. A plot showing the current distribution in amplitude along the dipole antenna will be displayed (refer to Fig. 14). Since a half-wave dipole has been drawn, the resulting current distribution resembles a semi-cycle approaching a sine function.

You can obtain several parameters from the perspective of the voltage source connected to the antenna terminals. Right-click on the wire and select **List Currents** from the pop-up menu. Move the slider to the position of the voltage source and click on the **Input List** button. This will display the input impedance of the dipole antenna, along with many other parameters (see Fig. 15).

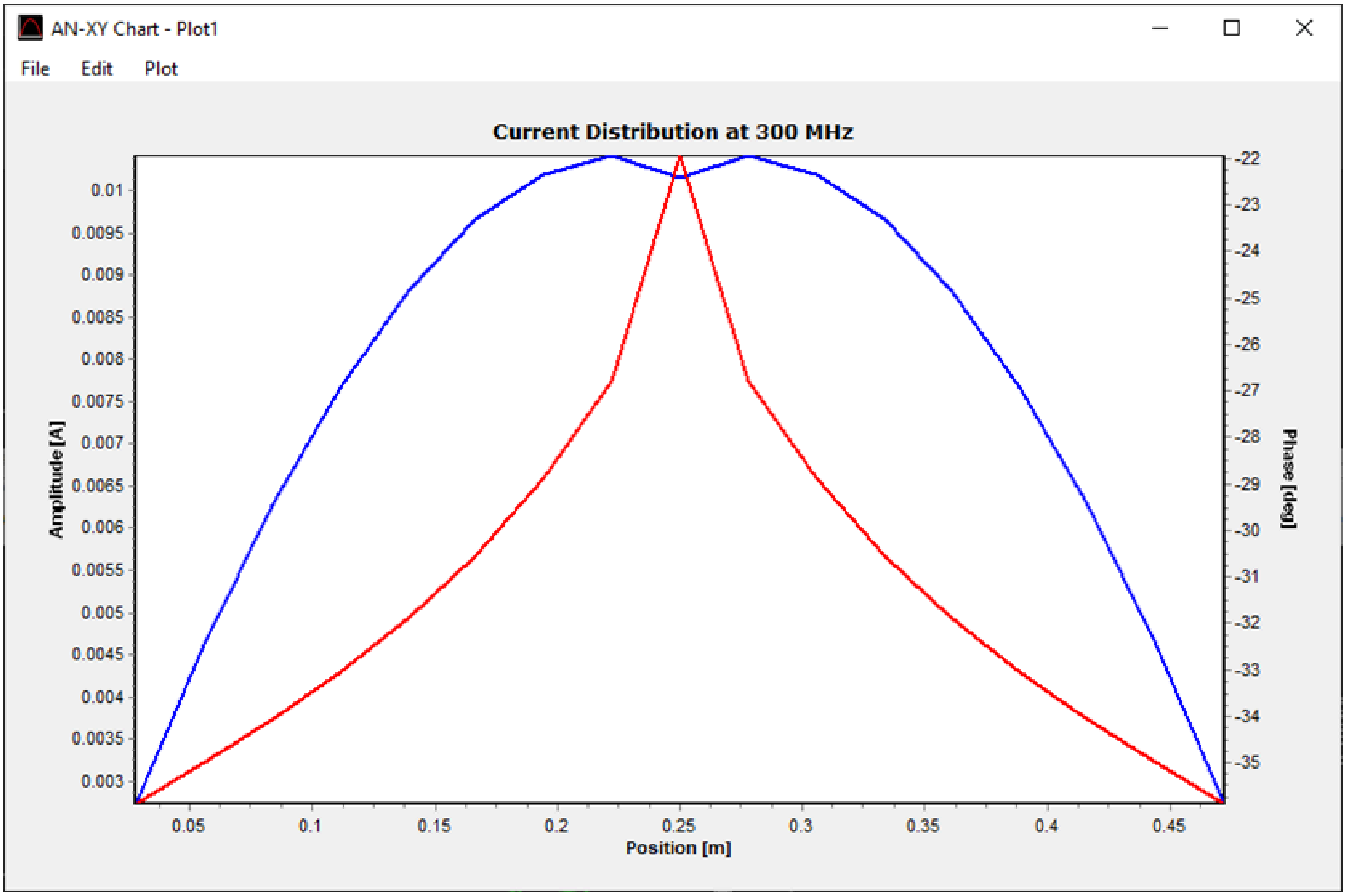


Fig. 14: Current distribution in amplitude and phase along a half-wave dipole.

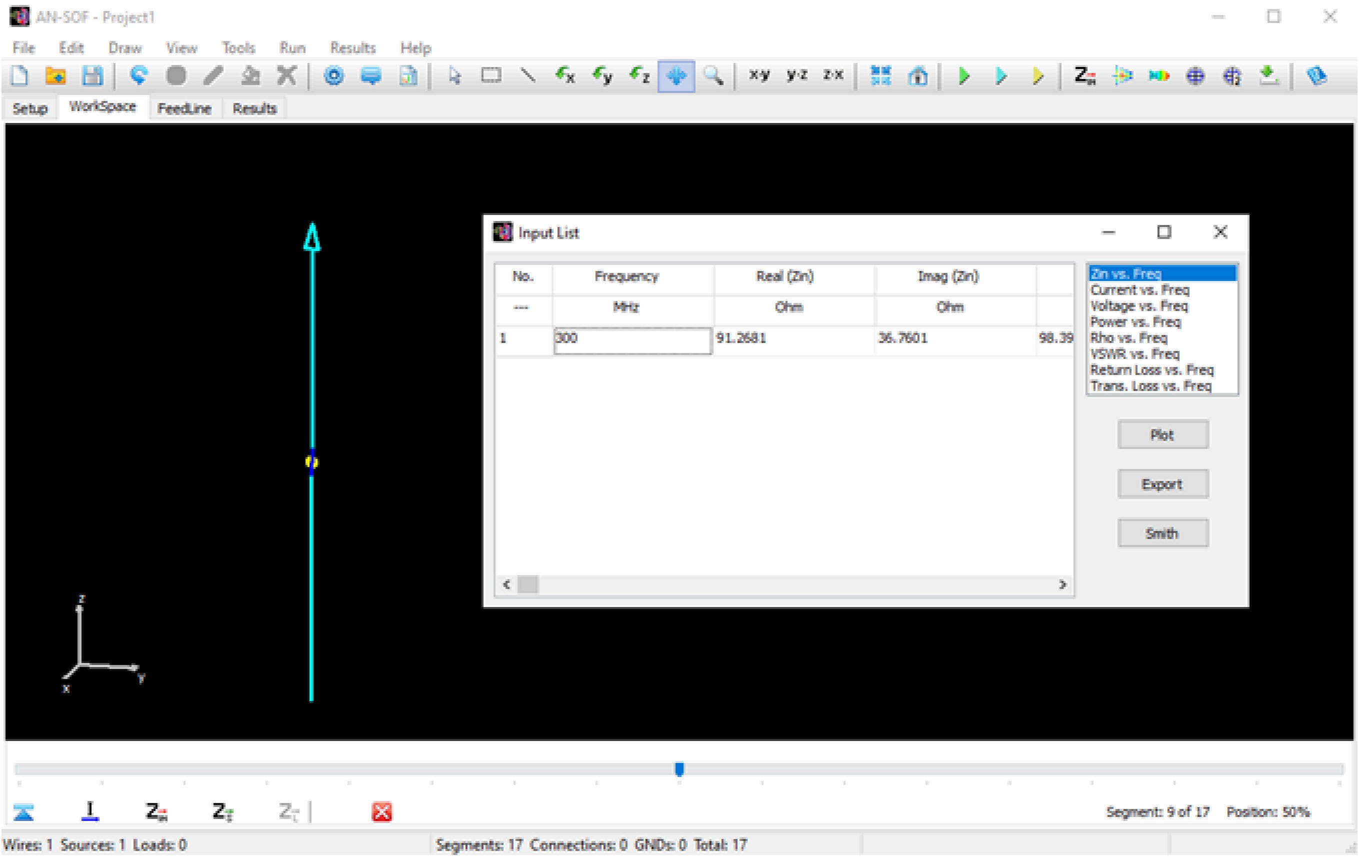


Fig. 15: The Input List dialog box displaying the input impedance.

Alternatively, you can obtain the input impedance by simply clicking on the **List Input Impedances (Zin)** button in the main toolbar. To represent the radiation pattern in a 3D plot, navigate to **Results > Plot Far-Field Pattern > 3D Plot** in the main menu. The normalized radiation pattern will be displayed in the **AN-3D Pattern** application. A color bar-scale indicates the field intensities over the radiation lobes. Additionally, you can plot the directivity, gain, and electric field patterns by accessing the **Plot** menu in AN-3D Pattern. In the case of a half-wave dipole, it exhibits omnidirectional characteristics in the plane perpendicular to the dipole axis (xy-plane) (refer to Fig. 16).

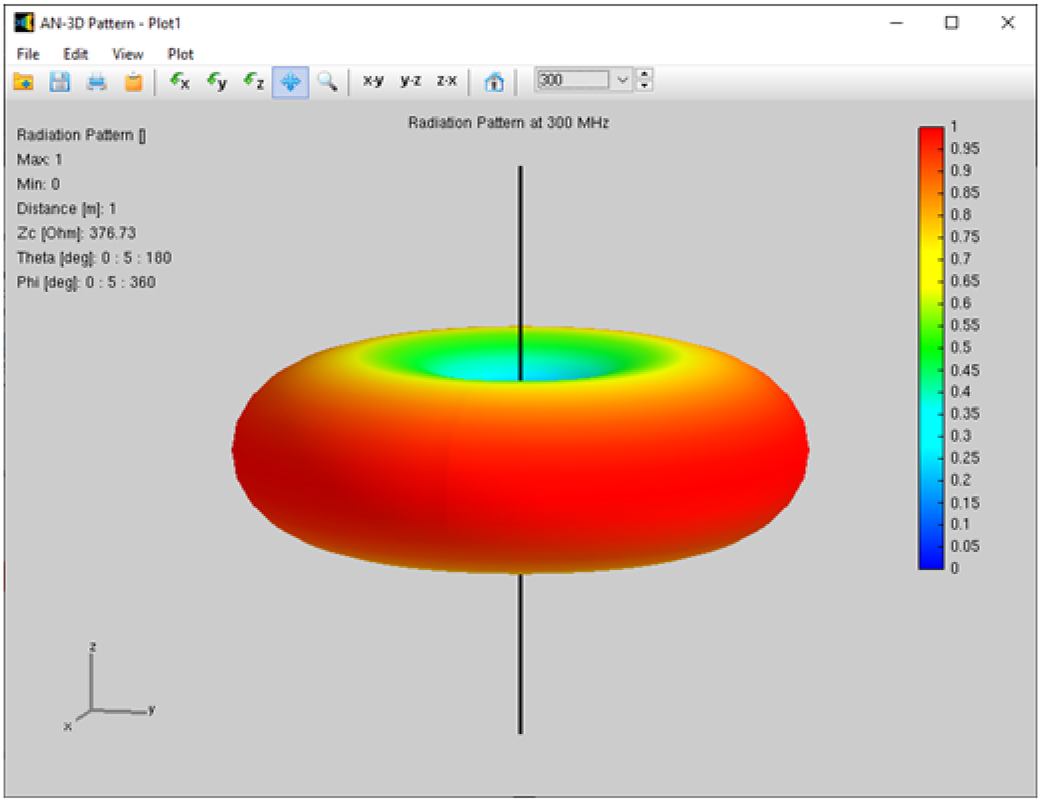


Fig. 16: The radiation pattern of a half-wave dipole exhibits a donut shape.

As you have just experienced, a simulation consists of **three simple steps**. We hope you have enjoyed this example. For additional step-by-step examples, please visit our section titled [Examples > Step by Step](#).

Summary

The key advantages of AN-SOF can be summarized as follows:

> **AN-SOF** is antenna modeling and design software that offers **fast** and **user-friendly** input and output graphical interfaces.
>
> **AN-SOF** employs the [Conformal Method of Moments](#) with **Exact Kernel**, resulting in enhanced accuracy and speed.
>
> **AN-SOF** provides an **extended frequency range**, enabling simulations from extremely low frequencies (such as 60 Hz circuits) to microwave antennas.

Simulating a wire structure involves a three-step procedure:

1. **Setup**: Set frequencies, environment, and desired results.
2. **Draw**: Draw the geometry, specify materials, and add sources.
3. **Run**: Perform the calculation and visualize the results.

At the beginning of the simulation, you can choose a convenient **unit system** for frequencies and lengths. This choice can be adjusted later by accessing **Tools > Preferences**. For instance, wire lengths are typically measured in meters (m) or feet (ft) for frequencies below 100 MHz, while millimeters (mm) or inches (in) are commonly used for higher frequencies.
AN-SOF Overview

Features and Capabilities

AN-SOF is a comprehensive software tool for the modeling and simulation of **antenna systems** and **radiating structures** in general.

AN-SOF is intended for solving problems in the following areas:

- Modeling and design of wire antennas.
- Antennas above a lossy ground plane.
- Broadcast antennas over radial wire ground screens.
- Single layer microstrip patch antennas.
- Radiated emissions from printed circuit boards (PCBs).
- Electromagnetic Compatibility (EMC) applications.
- Passive circuits, transmission lines, and non-radiating networks.

AN-SOF is based on an improved version of the so-called **Method of Moments (MoM)** for wire structures. Metallic objects like antennas can be modeled by a set of conductive **wires** and **wire grids**, as it is illustrated in Fig. 1. In the MoM formulation, the wires composing the structure are divided into **segments** that must be short compared to the wavelength. If a source is placed at a given location on the structure, an electric current will be forced to flow on the segments. The induced current on each individual segment is the first quantity calculated by AN-SOF.

Once the current distribution has been obtained, the radiated electromagnetic field can be computed in the far- and near-field zones. Input parameters at the position of the source or generator can also be obtained, such as the input impedance, input power, standing wave ratio (SWR), reflection coefficient, transmission loss, etc.

The modeling of the structure can be performed by means of the AN-SOF specific 3D CAD interface. Electromagnetic fields, currents, voltages, input impedances, consumed and radiated powers, directivity, gain and many more parameters can be computed in a frequency sweep and plotted in 2D and 3D graphical representations.

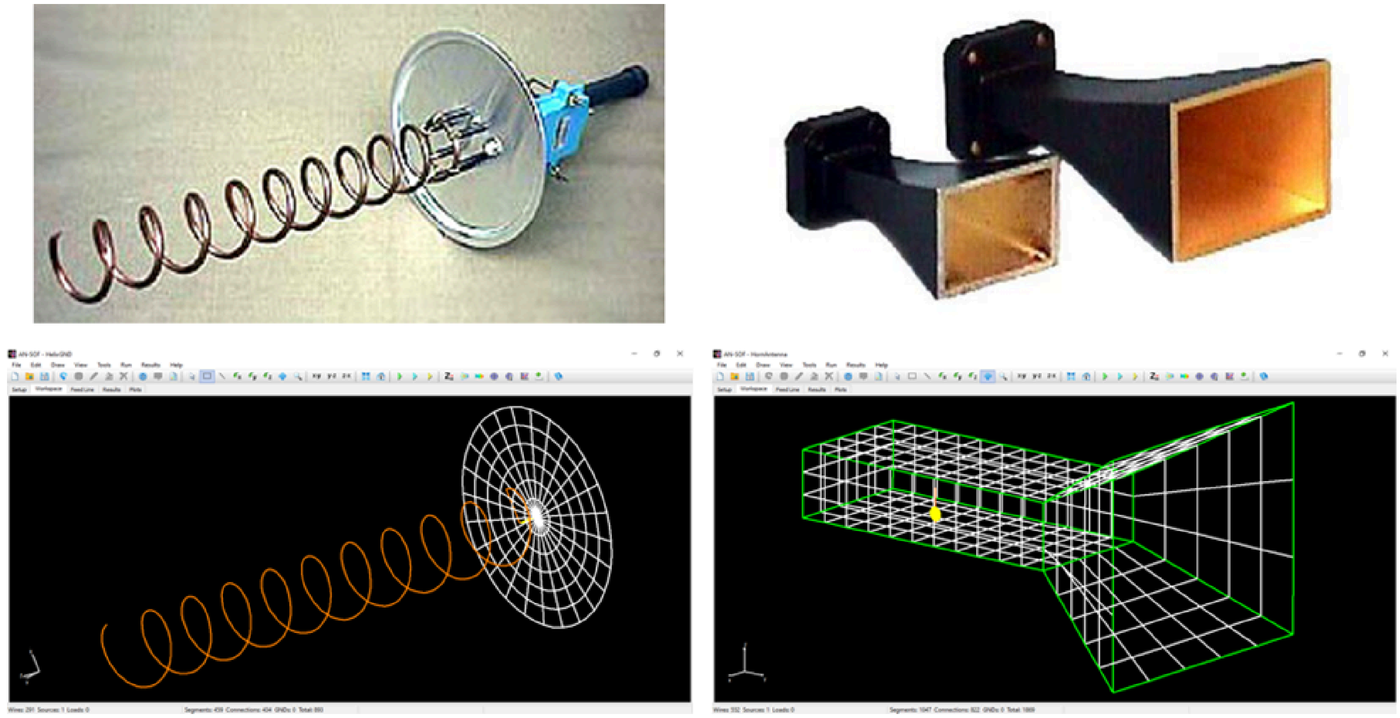


Fig. 1: Antennas modeled by means of wires and wire grids.

In the case of curved antennas like loops, helices, and spirals, the MoM in AN-SOF has been improved to accurately account for the wire's exact curvature. Traditional calculations often use straight-line segments to approximate curved antennas, resulting in many discontinuous wire junctions. This linear approximation can be inefficient in terms of computer memory and the number of calculations required, as it necessitates multiple straight segments to mimic the smooth curvature of wires. To address this issue, AN-SOF uses **curved segments** that precisely follow the contours of curved antennas. This innovative technique is known as the **Conformal Method of Moments (CMoM)**.

As an example, Fig. 2 shows the different approaches to a circular disc obtained by means of the MoM and CMoM methods. Both methods are available in AN-SOF since the MoM is a special case of the more general CMoM.

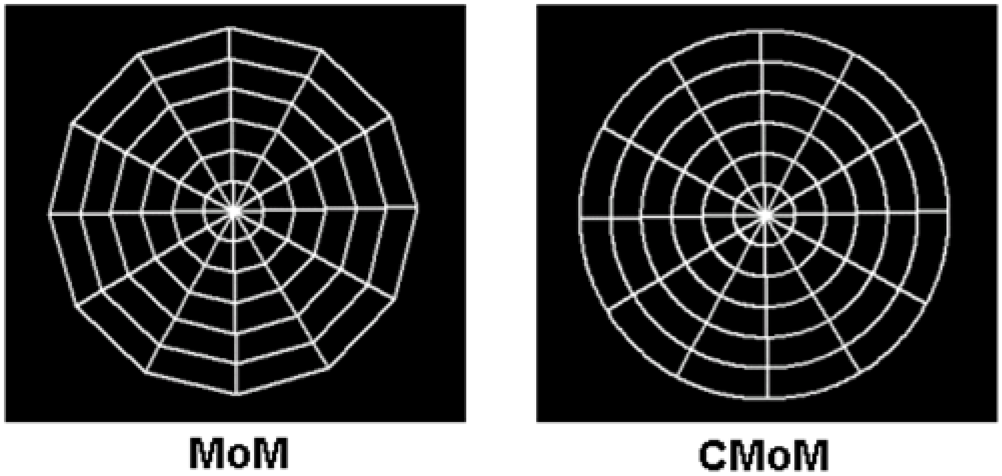


Fig. 2: Modeling of a disc by means of the MoM and CMoM methods.

In addition to the CMoM capabilities, advanced mathematical techniques have been implemented in the calculation engine making possible simulations from extremely low frequencies (e.g., electric circuits at 50-60 Hz) to very high ones (e.g., microwave antennas above 1 GHz).

In what follows, a summary of the modeling options and the simulation results that can be obtained from AN-SOF is presented.

Modeling of Metallic Structures

Metallic structures can be modeled by combining different types of **wires**, **grids**, and **surfaces**:

Wires

- Linear wire
- Circular arc
- Circular loop
- Helix
- Quadratic wire
- Archimedean spiral
- Logarithmic spiral

Wire Grids and Solid Surfaces

- Patch
- Plate
- Disc
- Flat ring
- Cone
- Truncated cone
- Cylinder

- Sphere
- Paraboloid

1. All types of **curved wires** can be modeled by means of **arced** or **quadratic segments**.
2. **Wire grids and solid surfaces** can be defined using either curved or straight wire segments. Curved segments follow the **exact curvature** of discs, rings, cones, cylinders, spheres, and parabolic surfaces. **Grids** are composed of cylindrical wires that leave holes between them, while **solid surfaces** are composed of flat wires or strips that cover the surface without leaving holes between them.
3. **Tapered wires** with stepped radii can be defined.
4. All wires can be loaded or excited at any segment.
5. The structure can also have **finite non-zero resistivities (skin effect)**.
6. Electrical connections of different wires and connections of several wires at one point are possible.
7. Metallic wires in either **dielectric or magnetic media** can be analyzed.
8. Wires with **insulation** can be modeled. Dielectric and magnetic coatings are available.
9. The structures can be placed in **free space**, over a **perfectly conducting ground plane** or over an **imperfect ground plane**.
10. Flat strip lines can be defined on a **dielectric substrate** for modeling planar antennas and printed circuit boards (PCB).
11. **Vias** in microstrip antennas and printed circuit boards can also be modeled.
12. The wire cross-section can either be **Circular, Square, Flat, Elliptical, Rectangular** or **Triangular.**
13. **Transmission lines** can be connected to the metal structure. There are over 160 cable models available, including two-wire and coaxial cables, with characteristic impedance, velocity factor, and loss parameters adjusted to actual datasheets.
14. The geometry modeling can be performed in suitable **unit systems** (um, cm, mm, m, in, ft). Different unit systems can also be chosen for inductance (pH, nH, uH, mH, H) and capacitance (pF, nF, uF, mF, F).

Excitation Methods

1. **Voltage sources** can be placed on the wires, as many as there are segments, with equal or different amplitudes (RMS values) and phases.
2. **Current sources** (e.g., representing impressed currents) can also be arranged at any segments.
3. The voltage and current sources can have **internal impedances**.
4. An **incident plane wave** of arbitrary polarization (linear, circular, or elliptical) and direction of incidence can also be used as the excitation.
5. **Hertzian electric and magnetic dipoles** can also be modeled and used as the excitation.
6. The antenna **input power** can be set to obtain the results (current distribution, near and far fields) scaled accordingly.

Frequency options

1. The simulation can either be performed for a **single frequency**, for frequencies taken from a **list** or for a **frequency sweep**.
2. The list of frequencies can either be created inside the program or loaded from a text file. It can also be saved to a txt file.
3. **Linear** and **logarithmic** frequency sweeps are possible.
4. A suitable **unit system** can be selected (Hz, KHz, MHz, GHz).

Data Input

1. **3D CAD tools** are implemented for drawing and modifying the structure geometry, including wires, grids, surfaces, discrete generators, and lumped loads.
2. The **segmentation** of wire geometry can be done **automatically** or **manually**.
3. Left-clicking on a wire selects and highlights it. Right-clicking on a wire reveals a **pop-up menu** with various options.
4. Wire connections are easily established by **copying** and **pasting** the endpoints of wires.
5. Special **3D symbols** indicate the positions of sources, load elements, and ground points.
6. All dialog boxes validate inputs for accuracy.
7. The program includes **mouse-supported functions** for rotating, moving, and zooming.
8. **Transmission lines** can be easily entered into a table, which serves as a library, for later use. A line is highlighted in the graphical interface for easy identification.
9. The program allows you to import geometrical data from text files. It supports three different file formats for importing wires, including the **NEC (Numerical Electromagnetics Code)** cards. Additionally, it can import DXF files containing 3D LINE entities.
10. The AN-SOF architecture integrates powerful numerical methods to achieve the **fastest calculation speed** and **the most accurate results**.

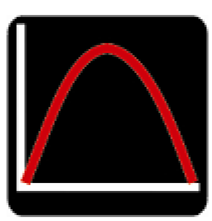
Data Output

1. All computed data is stored in files for subsequent graphical analysis.
2. **Input impedances**, currents, voltages, **VSWR**, **S_{11}**, return and transmission losses, radiated and consumed powers, efficiency, directivity, gain, and other system responses are presented as lists in text format and can be plotted against frequency. A **Smith chart** is available to represent impedances and admittances, as well as to display the reflection coefficient and VSWR at the selected point on the graph.
3. The **current distribution** on a selected wire can be plotted in amplitude, phase, real, and imaginary parts against position in a 2D representation. The currents flowing on a structure can also be plotted as a **color map on the wires**.
4. **Radiation and scattering fields** are obtained, including power density, directivity and gain patterns, total electric field, linearly and circularly polarized components, axial ratio, and Radar Cross Section (RCS). The **surface-wave field** can be determined as a function of distance in the case of a real ground with finite conductivity.

5. **Near-field components** can be calculated in Cartesian, cylindrical, and spherical coordinates. Field intensities can be plotted in 2D and 3D graphical representations and visualized as **color maps** in the proximity of a structure.
6. A 2D representation of radiated fields is available in Cartesian and polar coordinates. The **ARRL-style** log scale can be applied to **polar diagrams**.
7. **3D radiation patterns** can be viewed from arbitrary angles with zoom functions, colored mesh and surface representations, and a color bar scale. 3D patterns can be plotted with specially designed lighting and illumination for enhanced visualization of simulation results.
8. **Far-field patterns** can be separated into theta (vertical) and phi (horizontal) linearly polarized components, as well as right and left circularly polarized components. The **axial ratio** and the **front-to-rear** and **front-to-back** ratios are shown in polar plots and can be displayed as a function of frequency.
9. The **frequency spectrum** of **near- and far-fields** can be visualized in a 2D representation for all field components across different frequencies.
10. An average radiated power test, also known as **AGT (Average Gain Test)**, is conducted to verify the accuracy of the simulation.
11. The calculated data can be exported to **.csv**, **.dat**, or **.txt** files for use in other software programs.
12. An embedded **transmission line calculator** is included to simplify the design of **feed lines** for transmitting antennas. Actual cable part numbers can be selected from a wide range of manufacturers, thanks to data extracted from cable datasheets and integrated into the calculator.
13. A **Bulk Simulation** feature enables the automated calculation of multiple files, each with different geometric descriptions, to obtain results based on **variable geometric parameters**. The results are automatically exported to .csv files for further processing.
14. You can choose suitable **unit systems** for the plotted results, including current scaling (KA, A, mA, uA), voltage scaling (KV, V, mV, uV), electric field scaling (KV/m, V/m, mV/m, uV/m), magnetic field scaling (KA/m, A/m, mA/m, uA/m), decibel scales, and more.

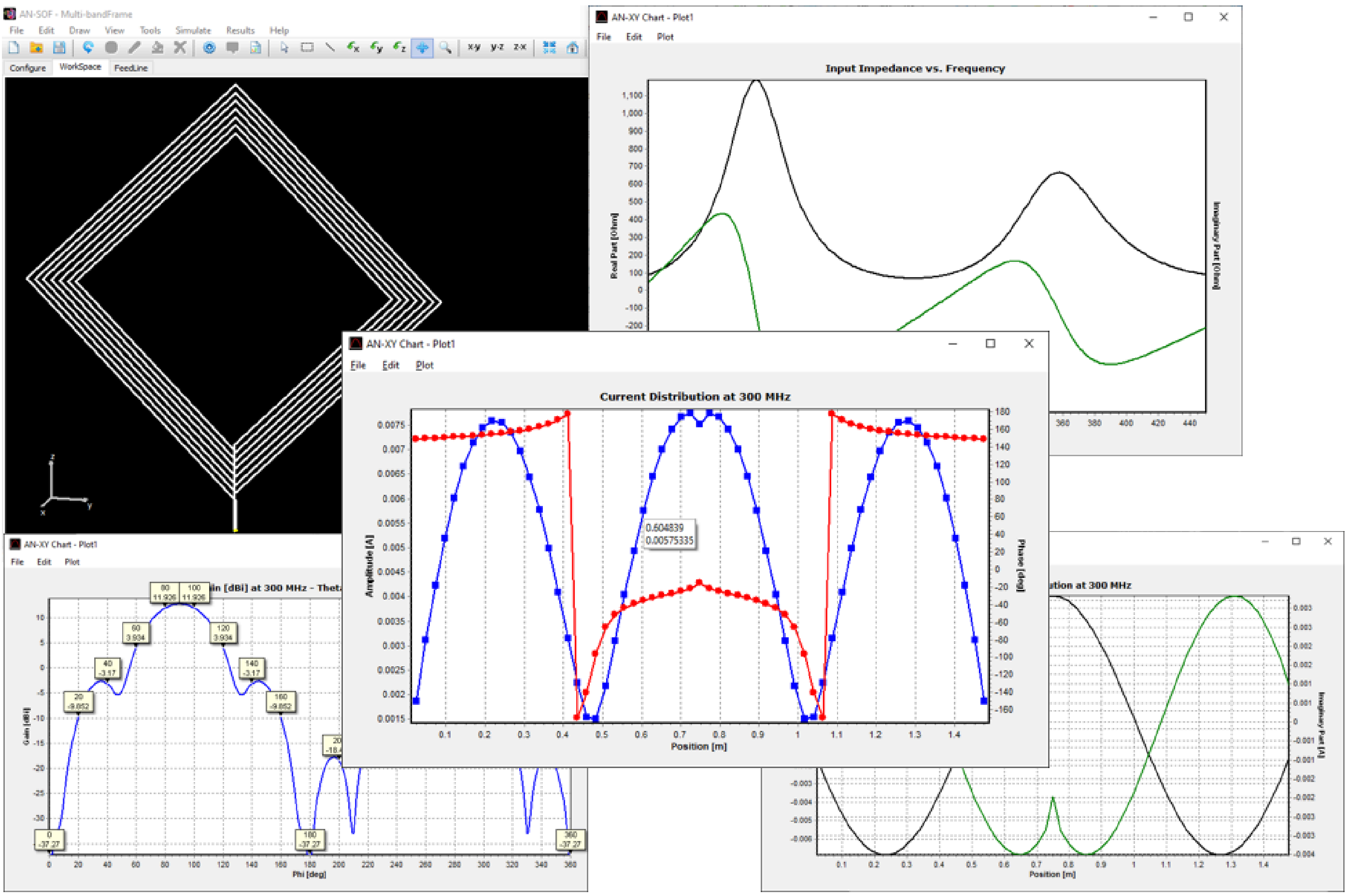
Integrated graphical tools

AN-SOF has a suite of integrated graphical tools for the convenient visualization of the simulation results. The following applications are installed automatically and used by the main program, AN-SOF:



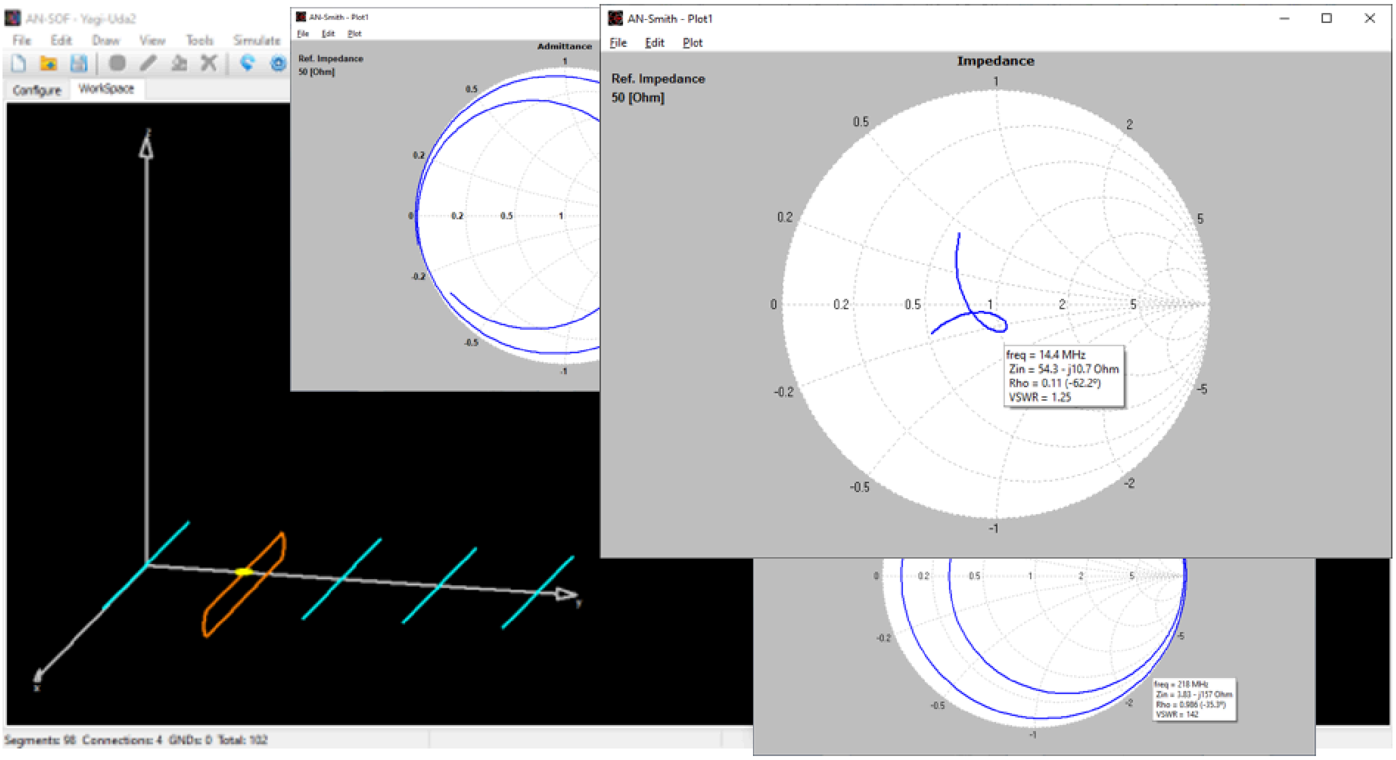
AN-XY Chart app

A friendly **2D chart** for plotting two related quantities, Y versus X. Use AN-XY Chart to plot parameters that depend on frequency, such as currents, voltages, impedances, reflection coefficient, VSWR, S_{11}, radiated power, consumed power, directivity, gain, radiation efficiency, radar cross section, field components, axial ratio, and many more. Also plot the current distribution along wires as a function of position, 2D slices of radiation lobes and near fields as a function of distance from an antenna. Choose different units to display results and use the mouse to easily zoom and scroll graphs.



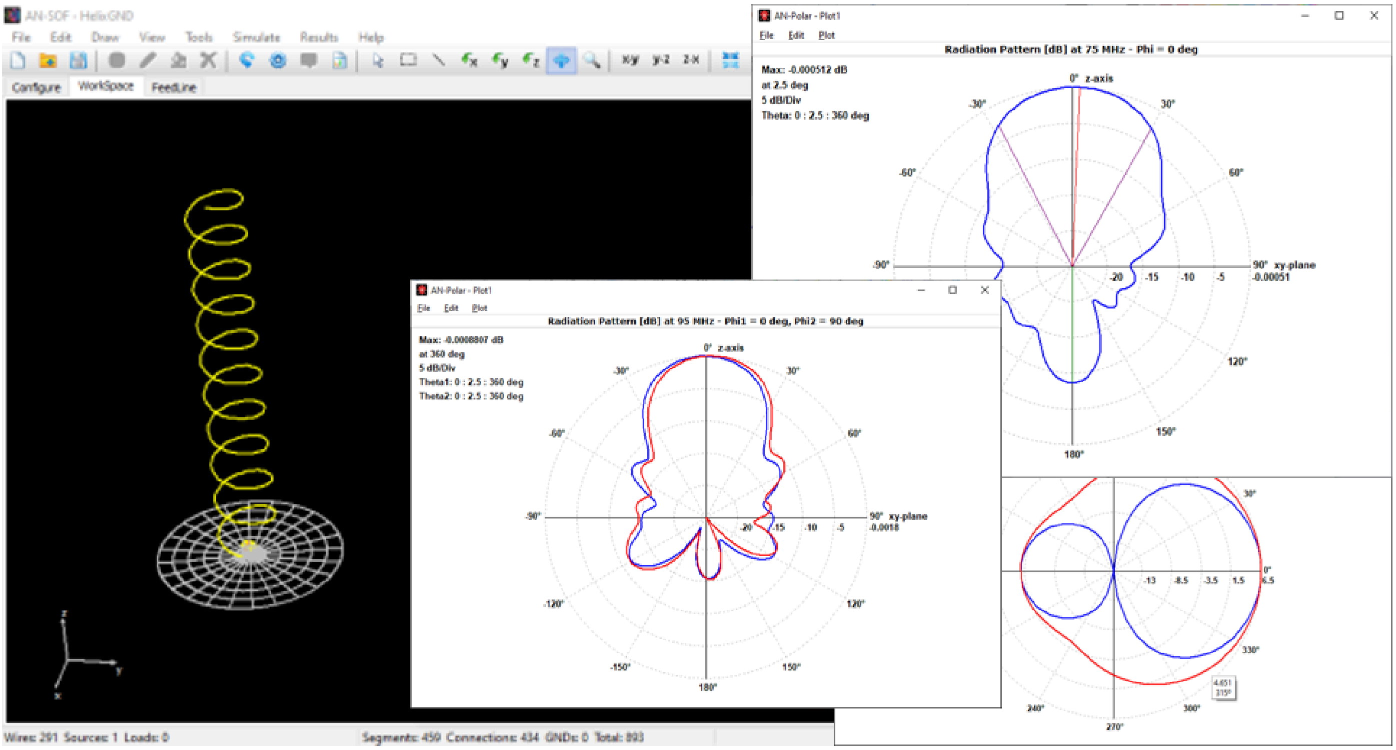
AN-Smith app

Plot **impedance** or **admittance curves** on the Smith chart with this tool. Just click on the graph to get the frequency, impedance, reflection coefficient, VSWR, and S_{11} that correspond to each point on the curve. Plots can be stored in independent files and opened later for a graphical analysis with AN-Smith.



AN-Polar app

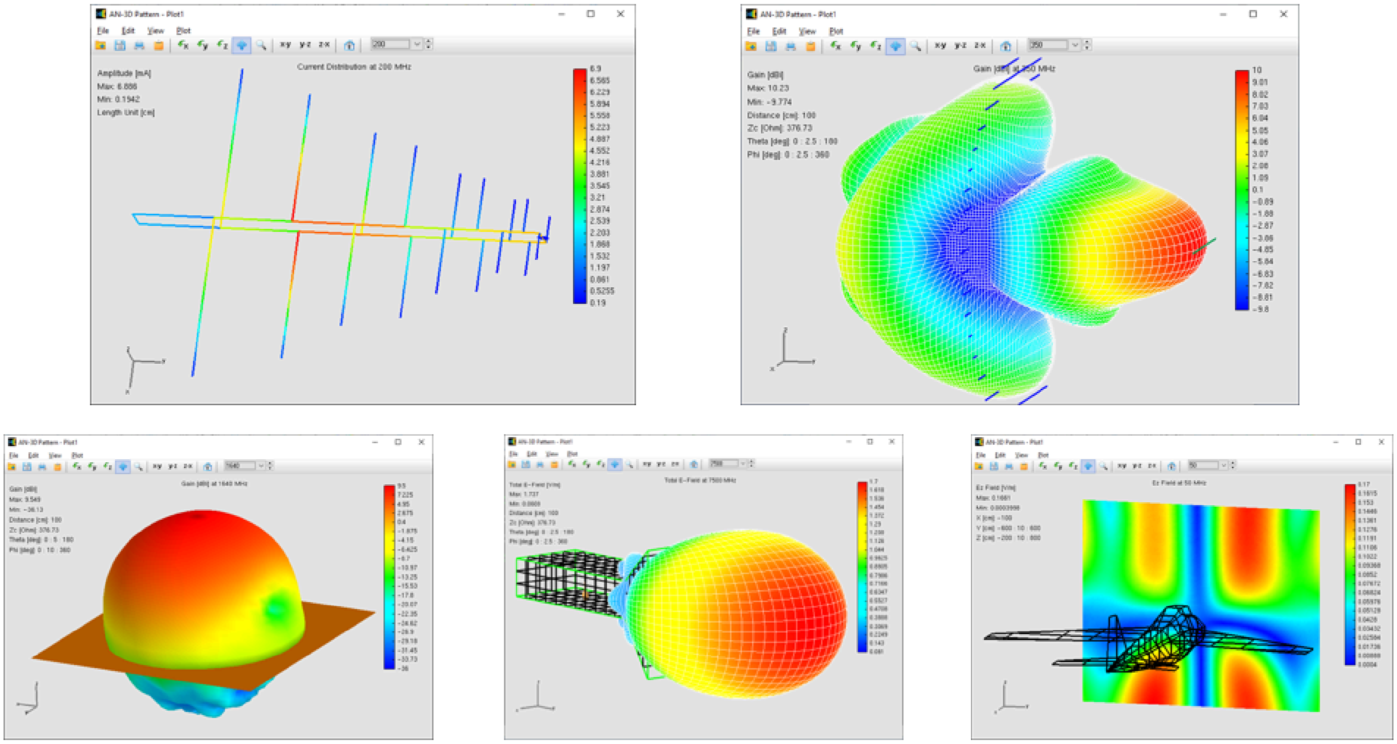
Plot on a **polar diagram** the radiation pattern versus the azimuth (horizontal) or zenith (vertical) angles. The maximum, -3dB and minimum radiation levels are shown within the chart as well as the **beamwidth** and **front-to-rear/back ratio**. Click on the graph to quickly obtain the values of the radiated field. The represented quantities include power density, directivity, gain, normalized radiation pattern, total electric field, linearly and circularly polarized components, axial ratio, and radar cross section (RCS).



AN-3D Pattern app

Get a complete view of the radiation properties of a structure by plotting a **3D radiation pattern**. AN-3D Pattern implements **colored mesh and surface** for the clear visualization of radiation lobes, including a color bar-scale indicating the field intensities over the lobes. Quickly rotate, move, and zoom the graph using the mouse. The 3D radiation pattern can be superimposed to the structure geometry to gain more insight into the directional properties of antennas.

The represented quantities include the power density, normalized radiation pattern, directivity, gain, total field, linearly and circularly polarized components, axial ratio, and Radar Cross Section (RCS). Choose between linear or decibel scales. Display **near fields** as **color maps** in the proximity of antennas in three different representations: Cartesian, cylindrical and spherical plots. Also plot the **current distribution** on the structure as a colored intensity map.



The AN-SOF Interface

Main Window and Menu

When AN-SOF starts, the main window displays several key components that form the core of the user interface (**Fig. 1**):

- **Title Bar** – Displays the name of the currently active project file (`.emm` format).
- **Main Menu Bar** – Provides access to the primary menus: *File*, *Edit*, *Draw*, *View*, *Tools*, *Run*, *Results*, and *Help*.
- **Toolbar** – Contains icons for quick access to commonly used commands and tools.
- **Tab Sheets** – Allow users to switch between different stages of the workflow, from **Setup** to **Plots**, with a single click.
- **Workspace** – The central 3D canvas where wire structures are drawn and visualized.
- **Status Bar** – Displays real-time information, such as the number of segments, wire-to-wire junctions, and wire-to-ground connections (GNDs).

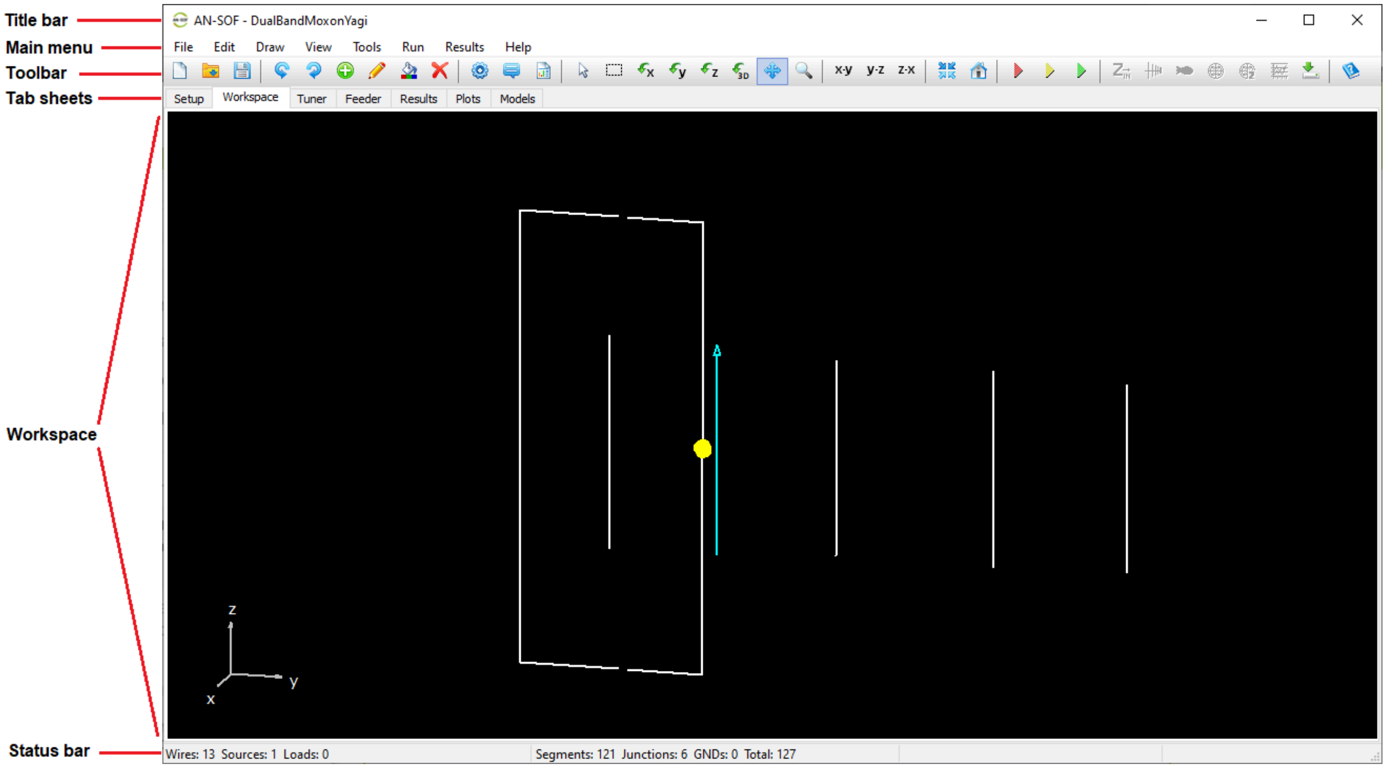


Fig. 1: Overview of the AN-SOF user interface.

File Menu

The **File** menu provides commands for managing AN-SOF projects, including creating, opening, saving, importing, and exporting files. The available commands are:

- **New… (Ctrl + N)**
  Creates a new project from scratch.
- **Open… (Ctrl + O)**
  Opens an existing project file (`.emm`) using the standard Open dialog.
- **Save (Ctrl + S)**
  Saves the current project under its existing name.
- **Save As…**
  Saves the current project with a new name. If the project is new, this command prompts the user to assign a name and location.
- **Import Wires**
  Opens a dialog to **import wire data** from various formats, including AN-SOF (`.wre`), NEC (`.nec`), and MM format (`.txt`).
- **Export Wires**
  Allows exporting the wire geometry to several formats:
  - **NEC** (`*.nec`)
  - **DXF** (`*.dxf`)
  - **Scilab** script (`*.sce`)
  - **MATLAB/Octave** script (`*.m`)

- **Copy Workspace**
  Copies the current workspace view to the clipboard as a bitmap image.
- **Recent Files List**
  Displays up to ten recently opened project files for quick access. Use **“Empty Recent Files List”** to clear the history.
- **Exit (Ctrl + Q)**
  Closes the current project and exits AN-SOF.

Edit Menu

The **Edit** menu provides tools for modifying, organizing, and managing wires in your project. The available commands include:

- **Undo (Ctrl + Z)**
  Reverts the project to the previous state before the last command was executed.
- **Redo (Ctrl + Y)**
  Reapplies the last undone action, restoring the project to the state before **Undo** was used.
- **Source / Load / TL (Ctrl + Ins)**
  Opens the **Source/Load/TL toolbar**, allowing you to add sources, loads, or connect transmission line (TL) ports to a selected wire segment. This command is enabled when a single wire is selected.
- **Modify (Ctrl + M)**
  Opens the **Modify dialog box** for editing properties of the selected wire(s). Available when one or more wires are selected.
- **Wire Color**
  Opens a standard color selection dialog to change the color of the selected wire(s). Enabled when one or more wires are selected.
- **Delete (Ctrl + Del)**
  Deletes the selected wire(s), including any attached sources or loads. Enabled when one or more wires are selected.
- **Copy Start Point**
  Copies the start point coordinates of the selected wire. This point can be used as the start of a new wire, enabling direct connection. Enabled when a wire is selected.
- **Copy End Point**
  Copies the end point coordinates of the selected wire for use as the starting point of a new, connected wire. Enabled when a wire is selected.
- **Start Point to GND**
  Draws a vertical wire from the start point of the selected wire down to the ground plane. This command is available only when a ground plane is defined in the model and a wire is selected.

- **End Point to GND**
  Draws a vertical wire from the end point of the selected wire down to the ground plane. Like the previous command, it is available when a ground plane exists and a wire is selected.
- **Move Wires… (Ctrl + Alt + M)**
  Opens the **Move Wires** dialog to reposition the selected wires. Enabled when at least one wire is selected.
- **Rotate Wires… (Ctrl + Alt + R)**
  Opens the **Rotate Wires** dialog to rotate selected wires around a specified axis. Enabled when at least one wire is selected.
- **Scale Wires… (Ctrl + Alt + S)**
  Opens the **Scale Wires** dialog to apply a scaling factor to the selected wires. Enabled when at least one wire is selected.
- **Copy Wires…**
  Opens the **Copy Wires** dialog to duplicate the selected wires, with options to apply translation or rotation to the copies. Enabled when at least one wire is selected.
- **Stack Wires…**
  Opens the **Stack Wires** dialog to generate multiple copies of the selected wires arranged along a specified direction. Enabled when at least one wire is selected.

Draw Menu

The **Draw** menu provides tools for creating a wide variety of wire geometries and wire grids. These commands allow users to model antenna elements with precision and flexibility. The available options include:

- **Line**
  Opens the **Line** dialog box to draw a straight wire between two defined endpoints.
- **Arc**
  Opens the **Arc** dialog to draw a circular arc segment.
- **Circle**
  Opens the **Circle** dialog to draw a complete circular loop.
- **Helix**
  Opens the **Helix** dialog to create a helical wire, often used in axial-mode antennas.
- **Quadratic**
  Opens the **Quadratic** dialog to draw a curved wire defined by a quadratic function.
- **Archimedean Spiral**
  Opens the **Archimedean Spiral** dialog to draw a spiral with uniform spacing between turns.
- **Logarithmic Spiral**
  Opens the **Logarithmic Spiral** dialog to create a spiral with exponentially increasing spacing—ideal for frequency-independent antenna designs.

Wire Grid / Solid Surface

This option creates a wire grid or a discretized surface geometry. It includes the following sub-commands:

- **Patch** – Draws a rectangular mesh on the **xy-plane**.
- **Plate** – Creates a bilinear surface or flat panel.
- **Disc** – Draws a flat circular disc.
- **Flat Ring** – Draws a ring-shaped disc with a central hole.
- **Cone** – Creates a conical surface structure.
- **Truncated Cone** – Draws a cone with a flat top (cut-off cone).
- **Cylinder** – Draws a cylindrical surface.
- **Sphere** – Creates a spherical grid or surface.
- **Paraboloid** – Draws a parabolic reflector surface.

Tapered Wire

This option allows drawing wires with **stepped-radius profiles**, useful for simulating tapered elements such as monopoles or matching sections. Each command mirrors the standard wire options above (e.g., Line, Arc, Helix), but with the ability to assign different radii along the length.

- **Tabular Input (Ctrl + T)**
  Opens a **spreadsheet-style table** for entering **linear wires, sources, loads**, and **transmission lines** numerically—ideal for precise control and batch input.
- **Transmission Lines (Ctrl + L)**
  Opens a dedicated table to define **transmission lines** by specifying their characteristic impedance, velocity factor, length, and loss parameters.

View Menu

The **View** menu provides options to control the display of interface elements, adjust zoom levels, and access additional project and wire information. Use these commands to customize your workspace and enhance your modeling experience:

- **Wire Properties… (Ctrl + W)**
  Opens the **Wire Properties** dialog, showing detailed information about the selected wire. This command is available when a wire is selected.
- **Project Details…**
  Opens the **Project Details** dialog, displaying a summary of the current project, including the number of segments, wires, sources, and loads.
- **Zoom In (Ctrl + I)**
  Magnifies the view within the workspace. You can also zoom in by scrolling the mouse wheel backward.

- **Zoom Out (Ctrl + K)**
  Reduces the view scale in the workspace. You can also zoom out by scrolling the mouse wheel forward.
- **Reset Zoom Scale**
  Resets the zoom level and resizes the workspace to fit the entire structure.
- **Axes… (Ctrl + A)**
  Opens the **Axes** dialog to adjust the appearance of the coordinate axes. Press **F7** to toggle between the **small axes** (shown in the lower-left corner) and the **main axes** (centered in the workspace).
- **X-Y Plane / Y-Z Plane / Z-X Plane**
  Switches the view to align the selected plane (XY, YZ, or ZX) parallel to the screen, offering easier modeling and inspection from different perspectives.
- **Center**
  Centers the view on the structure in the workspace without altering the zoom level.
- **Initial View (Home)**
  Resets the workspace to the default orientation and zoom level.
- **Drawing Panel**
  Displays a panel on the left side of the workspace with quick-access buttons for drawing **wires**, **wire grids/solid surfaces**, and **tapered wires**.

Tools Menu

The **Tools** menu provides access to plotting tools, wire checking utilities, and other helpful features for project analysis and customization. The commands available in this menu include:

- **3D Chart**
  Launches the **AN-3D Pattern** application to open and view 3D radiation pattern files (`.p3d`).
- **Polar Chart**
  Launches the **AN-Polar** application for displaying polar radiation patterns from `.plr` files.
- **Rectangular Chart**
  Launches the **AN-XY Chart** application to view rectangular (Cartesian) plots from `.plt` files.
- **Smith Chart**
  Launches the **AN-Smith** application to display impedance or reflection coefficient data from `.sth` files using a Smith chart.
- **Check Individual Wires**
  Verifies the segment length, cross-sectional size, and thin-wire ratio for each wire. Wires that fall outside acceptable ranges are highlighted in **yellow** (warning) or **red** (error).
- **Check Wire Spacing**
  Analyzes the spacing between adjacent wires to ensure they are sufficiently separated. Wires with spacing issues are also highlighted in **yellow** or **red**.
- **Delete Duplicate Wires**
  Automatically detects and removes duplicate or overlapping wires from the project.

- **RF Calculators**
  Opens a webpage with a **collection of online tools** for calculating antenna parameters, propagation metrics, component values, and unit conversions.
- **Calculator**
  Launches the built-in **Microsoft Windows® Calculator** for quick arithmetic operations.
- **Preferences…**
  Opens the **Preferences** dialog box, where users can configure unit systems, workspace appearance, pen width, confirmation prompts, and other interface settings.

Run Menu

The **Run** menu provides commands to execute the simulation of currents and electromagnetic fields. These options allow users to selectively perform calculations depending on their analysis needs:

- **Run Currents and Far-Field (F10)**
  Calculates the **current distribution** on the wire structure, from which **input impedance** and **VSWR** are derived, along with the **far-field radiation pattern**, used to compute **radiation efficiency**, **gain**, and **front-to-back ratio**.
- **Run Currents and Near-Field (F11)**
  Calculates the **current distribution** and the **near electric and magnetic fields** around the antenna structure.
- **Run ALL (F12)**
  Executes a complete simulation including **currents**, **far-field**, and **near-field** calculations in a single step.
- **Run Currents (Ctrl + R)**
  Calculates only the **current distribution** on the wire structure.
  This is useful when interested solely in **input parameters** (impedance, reflection coefficient, VSWR), avoiding the time cost of field computations.
  Note: This command is disabled when the currents have already been computed.
- **Run Far-Field**
  Calculates the **far-field radiation pattern** based on previously computed currents.
  Enabled only after the current distribution has been calculated.
- **Run Near E-Field**
  Computes the **near electric field** generated by the current distribution.
  Requires prior current calculation.
- **Run Near H-Field**
  Computes the **near magnetic field** generated by the current distribution.
  Requires prior current calculation.

- **Run Bulk Simulation**
  Opens a dialog to select multiple input files in **NEC format** (`.nec`). AN-SOF will import each file, run the simulation, and export the results as **CSV files** to the same directory. This is ideal for **batch processing multiple designs**.

Results Menu

The **Results** menu provides a set of commands for visualizing and exporting simulation results. These include current distributions, field patterns, power metrics, and spectral data. The available commands are as follows:

Plot Current Distribution

Launches the **AN-3D Pattern** application to visualize the current distribution as a color pattern mapped onto the wire structure.

Plot Currents

Opens the **AN-XY Chart** to plot the magnitude and phase of currents versus position along a selected wire. This command is enabled only when a wire is selected.

List Currents…

Displays the **List Currents** toolbar to show current values versus frequency at a specific segment on a selected wire. If the segment includes a source, additional data such as input impedance, voltage, and power versus frequency will also be available. This command is enabled only when a wire is selected.

Export Currents

Exports the current distribution along a wire to a CSV file, including position and frequency-dependent data. Enabled only when a wire is selected.

List Input Impedances…

Shows a **table of input impedances versus frequency**, including associated parameters such as reflection coefficient, VSWR, return loss, and transmission loss at the antenna terminals.

Plot Far-Field Pattern

Opens a sub-menu with commands for plotting the radiation pattern in various formats:

- **3D Plot**: Launches **AN-3D Pattern** to visualize the **full 3D far-field radiation pattern**.
- **Polar Plot 1 Slice**: Opens a dialog to select a single **2D slice of the far-field pattern**, displayed in polar coordinates using **AN-Polar**.
- **Polar Plot 2 Slices**: Allows selection of two **2D slices of the far-field pattern**, displayed in polar coordinates using **AN-Polar**.

- **2D Rectangular Plot**: Allows selection of a **2D slice of the far-field pattern**, displayed in rectangular (Cartesian) coordinates using **AN-XY Chart**.

Plot Far-Field Spectrum

Opens a dialog to select a point in space where **far-field components will be analyzed across frequency**. The spectrum is displayed using **AN-XY Chart**.

List Far-Field Pattern

Displays a table of the total electric field and its components (E-theta, E-phi, right-hand and left-hand circular polarization) at the theta–phi angular grid defined in the **Far-Field panel**. **The data can be exported as a CSV file**.

List Far-Field Spectrum

Opens a dialog to select a spatial point where the far-field components will be listed versus frequency. The table includes the total E-field and its components: E-theta, E-phi, right-hand and left-hand polarized components. **The data can be exported as a CSV file**.

Power Budget / RCS

Displays the **Power Budget** dialog showing total input power, radiated and consumed power, power density, efficiency, directivity, and gain as functions of frequency. For simulations with **plane wave excitation**, the **Radar Cross Section (RCS)** versus frequency is shown.

Plot Near E-Field Pattern

Opens a sub-menu for near-field electric field visualization:

- **3D Plot**: Launches **AN-3D Pattern** to show a **3D visualization of near E-field components**.
- **2D Plot**: Opens the **Near-Field Cut** dialog for selecting and plotting a 2D cut using **AN-XY Chart**.

Plot Near E-Field Spectrum

Opens a dialog to select a spatial point and plots the **frequency response of the near E-field components** using **AN-XY Chart**.

List Near E-Field Pattern

Displays a table of the total near electric field and its components at the grid defined in the **Near-Field panel**. **The data can be exported as a CSV file**.

List Near E-Field Spectrum

Opens a dialog to select a point in space and displays the near E-field spectrum in table format, including the individual field components versus frequency. **The data can be exported as a CSV file**.

Following these commands, similar options are available for **Near H-Field** and **Power Density** data. To compute power density, both the near E-field and H-field must have been previously calculated, as power density is obtained from the vector (cross) product of these fields at each spatial point.

Help Menu

Use the **Help** menu to access the user guide, technical support, license activation, and version information. This menu includes:

- **User Guide**
  Opens the **AN-SOF User Guide** in PDF format.
- **AN-SOF Home Page**
  Opens the official website at **www.antennasimulator.com** in your default browser.
- **Knowledge Base**
  Opens the **Knowledge Base** where you can search categorized help articles and how-to guides.
- **Email to Tech Support**
  Launches your default email client to send a support request to **info@antennasimulator.com**.
- **Chat to Tech Support**
  Opens the **live chat support page** in your default browser.
- **Activate AN-SOF**
  Runs the AN-Key application, which shows your PC's serial number and allows you to enter an activation key to license the software.
- **Check for Updates**
  Opens the webpage with **the latest AN-SOF software releases**.
- **About AN-SOF…**
  Displays version number, license, and copyright.

Main Toolbar

The main toolbar provides quick access to essential commands, many of which are also found in the main menu. In addition, it includes extra tools not available in the menu.

One such group consists of **3D View Controls**, which help users interact with the 3D workspace (see **Fig. 2**):

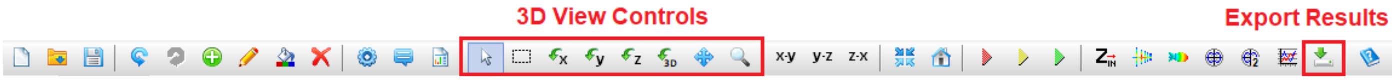


Fig. 2: The Main Toolbar with 3D View Controls and Export Button

3D View Controls

- **Select Wire** (arrow icon)
  Activates selection mode. Click a wire to select it. **To select multiple wires**, hold **Ctrl** and click each one. Selected wires are highlighted in light blue. To deselect, **Ctrl + click** again, or **double-click** in the workspace or press **ESC** to clear all selections.
- **Selection Box**
  Allows **selecting multiple wires** by dragging a box with the left mouse button:
  - Drag from **top to bottom** to select only wires fully enclosed.
  - Drag from **bottom to top** to select any wires that intersect the box.
  - Previously selected wires will be **deselected** if included again.
  - Double-click the workspace or press **ESC** to deselect all.
- **Rotate around X/Y/Z**
  Rotates the view around the selected axis (X, Y, or Z) by moving the mouse in the workspace.
- **3D Rotation**
  Enables free rotation around all axes by moving the mouse in the workspace.
- **Move** (cross icon)
  Pans the view by dragging the mouse with the left button held down.
- **Zoom** (magnifying glass icon)
  Lets you zoom into a selected area by dragging a rectangle. After zooming, use the mouse wheel to adjust the zoom level.

Other Toolbar Commands

Export Results
Located next to the results-related buttons, this command exports the data in the **Results** tab to a **CSV** file.

Custom Preferences

Preferences

Preferences in AN-SOF allow users to customize the unit system for input and output data, adjust the workspace appearance, and configure various miscellaneous options. To access preferences, navigate to **Tools > Preferences** from the main menu.

Units

On the **Units** page of the Preferences dialog box (see Fig. 1), users can select suitable units for frequencies, lengths, wire cross-section, inductances, and capacitances. Apart from standard SI units, options such as inches (in) and feet (ft) are available for lengths and cross-sections.

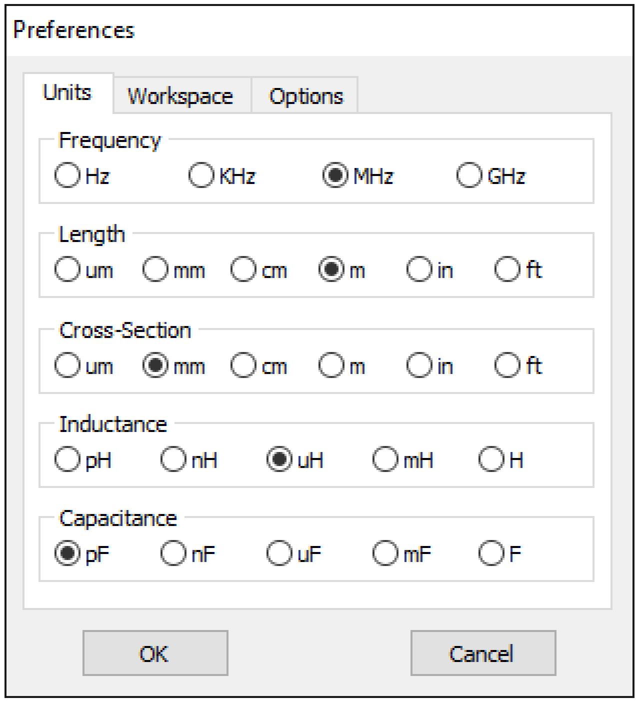


Fig. 1: Units tab in the Preferences dialog box where frequencies, lengths, wire cross-sections, inductances, and capacitances can be set.

Workspace

In the **Workspace** tab (Fig. 2), users can toggle the workspace background color between black and white. Additionally, there are three levels for the pen width used to draw objects on the workspace: Thin, Medium, and Thick. This option applies to axes, wires, and wire grids. Users can also customize the size and color of source symbols and loads. Enabling the **Show Segments** option displays the segments in the workspace.

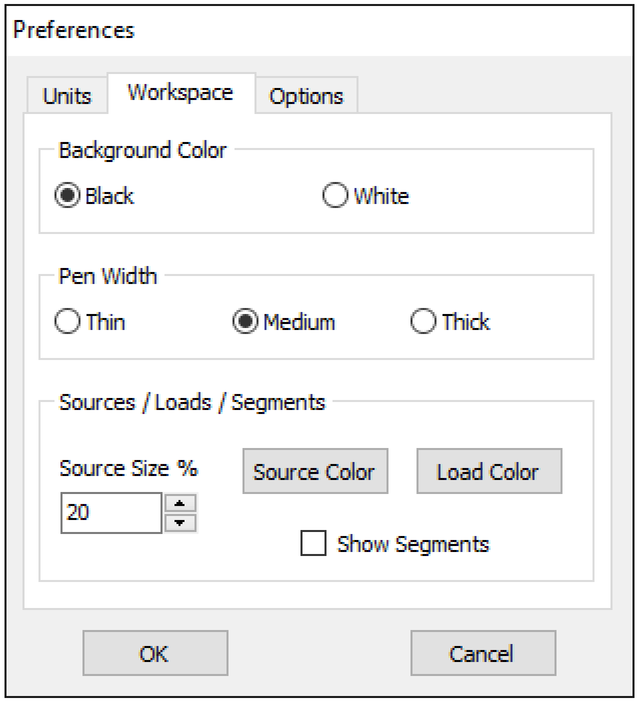


Fig. 2: Preferences dialog box showing the Workspace tab, where the workspace background color, pen width, and appearance of sources/loads can be set.

Options

In the **Options** tab, users can check the **Show Main Toolbar** option to display the toolbar (Fig. 3). Two "Ask before…" questions can be set to avoid mistakes. If the option **"Run ALL" also calculates the H-Field** is checked, the near H-field will be calculated after clicking on the "Run ALL" button. Users can also choose to close the chart windows after exiting AN-SOF. Additionally, the option **"The comma is set as the decimal symbol"** should be selected if the comma is used as the decimal separator in the Windows® regional settings. Users can also set the number of significant digits shown in results, although this option does not modify the **double precision** used in the internal algorithms.

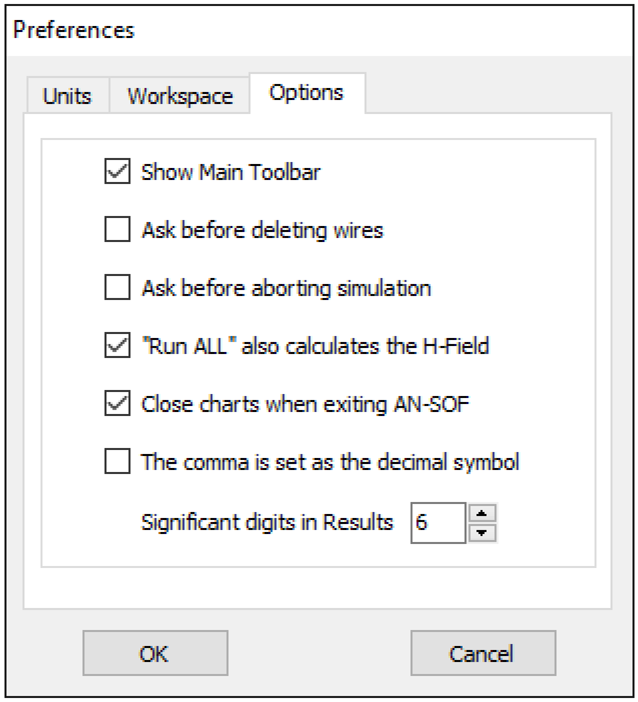


Fig. 3: Options tab in the Preferences dialog box, where various additional settings can be configured.

> **Note**
>
> All preferences can be configured at any time, either before or after performing a calculation.

Tools in the Workspace

Display Options

The workspace background can be set to **white** or **black**. When a white (black) background is chosen, all wires will default to **black (white)** unless a different color is specified for specific wires. To set the workspace color, navigate to **Tools >** **Preferences** **> Workspace tab**. The color of **selected wires** can be changed at any time via **Edit > Wire Color** in the main menu.

The line width used for drawing wires and axes in the workspace can be adjusted by selecting a **Pen Width** option in the **Workspace tab** of the **Preferences dialog box**. There are three levels: **Thin**, **Medium**, and **Thick**. Figure 1 illustrates the different combinations of workspace color and pen width that can be achieved.

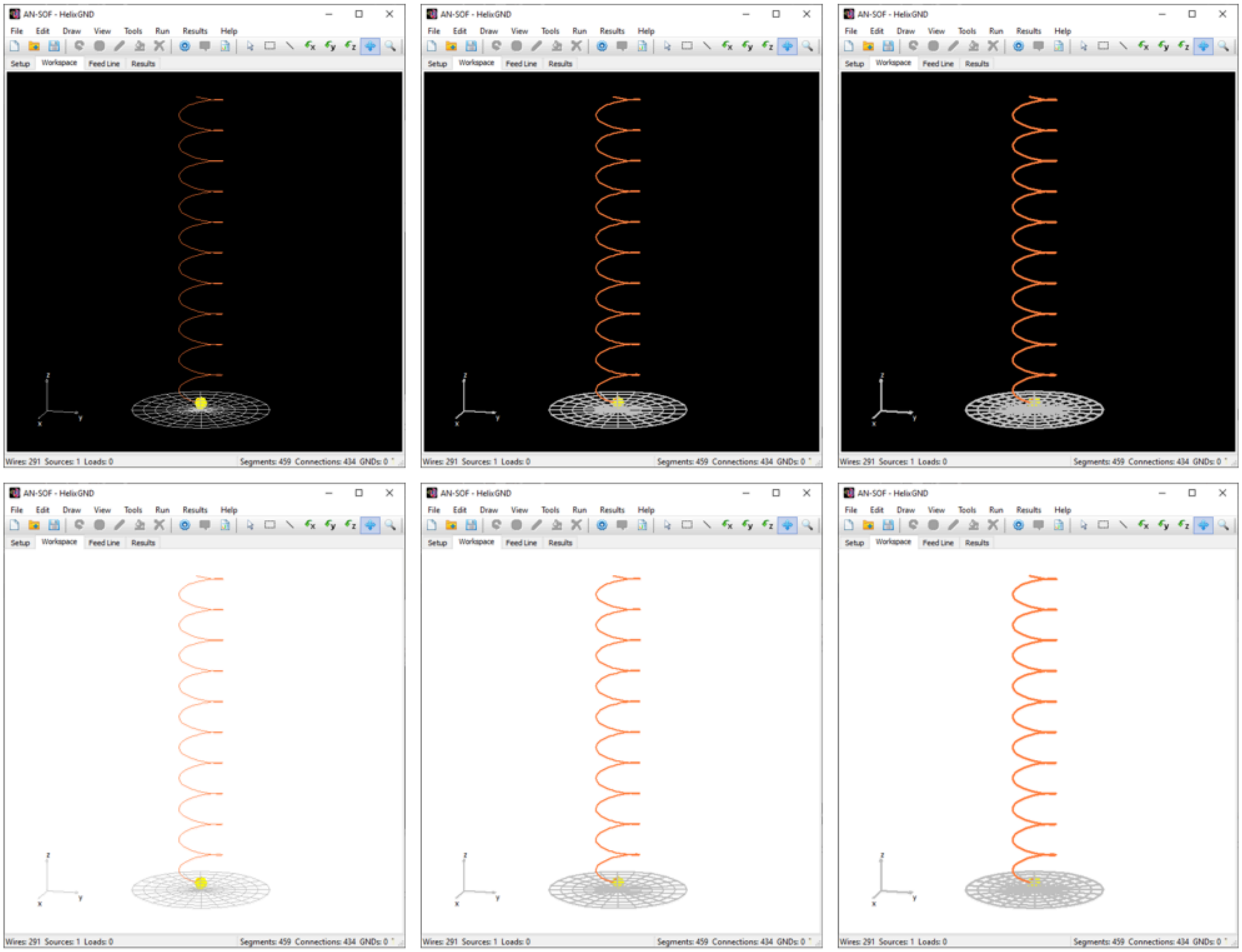


Fig. 1: Display options in the workspace.

Viewing 3D Axes

To customize the appearance of the **X, Y, Z** axes in the workspace, navigate to **View > Axes (Ctrl + A)** in the main menu to open the **Axes** dialog box (see Fig. 2). There are two types of axes:

- **Small Axes**: Displayed in the lower-left corner of the workspace.
- **Main Axes**: Displayed at the center of the screen.

Both positive and negative axes can be displayed. The color of the main axes can be changed by clicking the **Color** button. Check the **Show Ticks** option to add a specified number of ticks to the **Main Axes**.

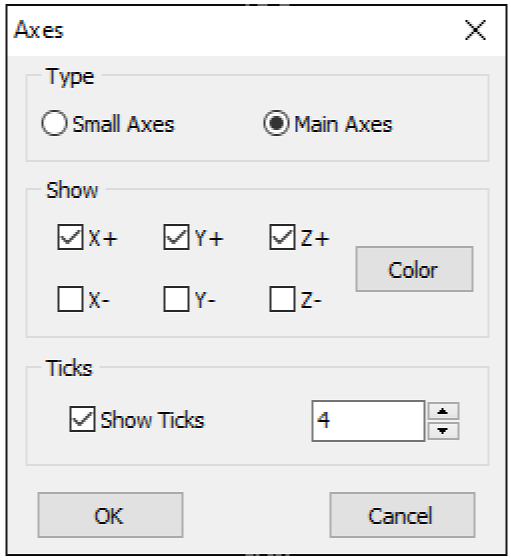


Fig. 2: Axes dialog box, showing options to display positive and negative axes.

> **Tip**
>
> Press **F7** to switch between small and main axes.

Zooming the View

To zoom in or out of the structure in the workspace:

- Use the **mouse wheel**.
- On a laptop touchpad, use two fingers (similar to zooming an image).
- Alternatively, use the **Zoom In (Ctrl + I)** and **Zoom Out (Ctrl + K)** commands in the **View** menu.

For a more specific zoom on a particular area, click the **Zoom** button on the toolbar and drag a rectangle over the desired area. To return to the initial view, click the **Initial View (Home)** button on the toolbar.

Rotating the View

To rotate the view of the structure around a specific axis:

1. Press one of the following toolbar buttons:

- **Rotate around X**
- **Rotate around Y**
- **Rotate around Z**
- **3D Rotation**

2. Move the mouse while holding the left button.

Alternatively, use the following keyboard shortcuts:

- **F1**: Right-handed rotation around the x-axis.
- **F2**: Left-handed rotation around the x-axis.
- **F3**: Right-handed rotation around the y-axis.
- **F4**: Left-handed rotation around the y-axis.
- **F5**: Right-handed rotation around the z-axis.
- **F6**: Left-handed rotation around the z-axis.

Moving the View

To move the view of the structure in the workspace:

1. Click the **Move** button on the toolbar.
2. Move the mouse while holding the left button.

The Conformal Method of Moments

Introduction

The **Method of Moments (MoM)** is widely recognized as one of the most reliable techniques for modeling and simulating antennas and radiating systems. However, traditional implementations of MoM suffer from several issues primarily stemming from **approximations** used in numerical calculations to reduce computational requirements. While these approximations were justified in the 1970s and 1980s due to limited processor speeds and memory capacities, the present-day computing power, even on personal computers, allows for **more accurate calculations**. The limitations imposed by these approximations in traditional MoM models restrict their validity and applicability.

The fundamental principle of MoM involves representing metal surfaces through **wire segments**, which is a suitable approximation for many **metallic antennas**, particularly wire-type antennas like linear antennas, dipoles, monopoles, yagis, log-periodic arrays, quads, antenna arrays of all types, traveling wave antennas, fractals, aperture antennas, and reflectors. It is essential for each wire segment to have a **small length and cross-section relative to the wavelength**. The MoM seeks to determine the unknown current flowing through each wire segment, as depicted in Fig. 1.

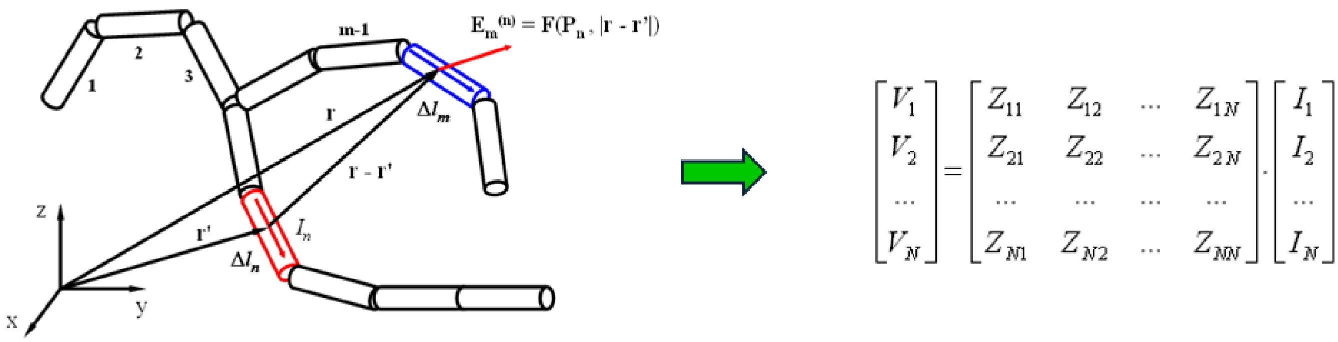


Fig. 1: In the traditional Method of Moments (MoM), linear approximation is applied to the structure's geometry using straight segments. The MoM enables the conversion of Maxwell's equations from their integral form into a matrix equation, which in turn allows for the determination of currents in the segments.

The Thin-Wire Approximation

In the modeling of antennas using **cylindrical wire segments**, the initial approximation commonly employed is known as the "thin-wire approximation," as illustrated in Fig. 2. This approximation is based on the following assumptions:

1. The electric current flowing through a wire can be represented as a **filament along the wire axis**, disregarding the fact that it actually flows on the wire's surface.
2. **Variations** in the current along the circular contour of the wire's cross-section **can be ignored**.
3. The component of the **current perpendicular** to the wire axis **can be disregarded**.
4. It is sufficient to enforce the boundary condition of **zero total tangential electric field** on the surface of an ideal conducting wire **along its axis**.

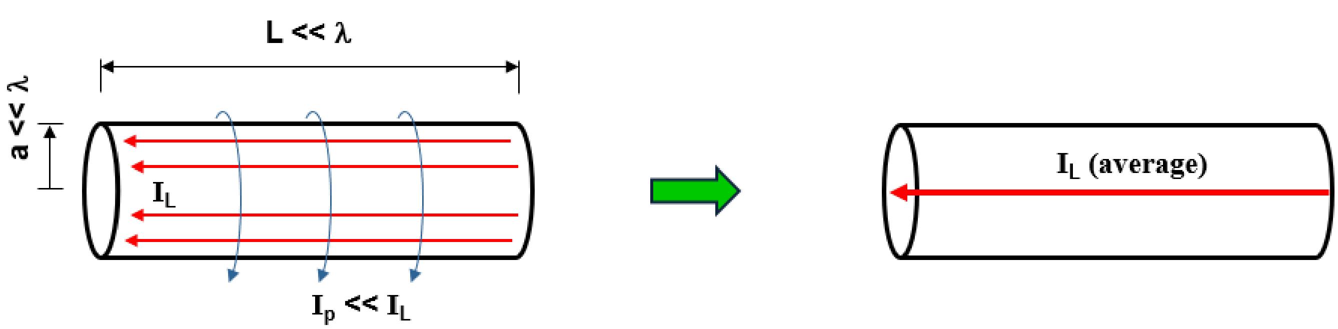


Fig. 2: Illustration of the thin-wire approximation for a wire segment in the Method of Moments.

When dealing with a wire segment with a cross-section significantly smaller than the wavelength, assumptions 2, 3, and 4 are reasonably valid and align with experimental observations and theoretical predictions in the quasi-electrostatic regime for metal surfaces. However, **assumption**

1, regarding the **current filament along the wire axis**, has sparked debates throughout **the history of linear antennas**.

Assumption 1 only holds as a **limiting case** when the wire's cross-section approaches zero size, such as when the wire has a circular cross-section and **its radius tends to zero**. This assumption relates to the crucial aspect known as the **Kernel** of the problem. The Kernel represents the core of the integral equation that the MoM solves to determine the currents flowing along the wires. Instead of employing the "thin-wire Kernel" utilized in traditional MoM, which is based on assumption 1, **AN-SOF employs the exact Kernel**. In the exact Kernel, it is considered that the current flows on the **surface of the wires** rather than being confined to a filament along the wire axis.

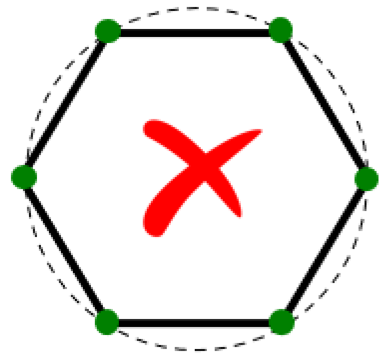
Eliminating assumption 1 has a significant impact on the **accuracy** of calculations, particularly in the current distribution near the antenna's feed point or terminals, where obtaining precise values for input impedance and standing wave ratio (SWR) is crucial. In addition to discarding assumption 1 in AN-SOF, the use of the exact Kernel and **curved wire segments** helps overcome other issues inherent in traditional MoM, as described below.

Overcoming the 7 Limitations of the Traditional MoM

In AN-SOF, we have departed from the traditional MoM and embraced innovation by implementing a new method called the **Conformal Method of Moments (CMoM)** with an **exact Kernel** formulation. This decision stems from the lack of substantial improvements in traditional methods over several decades, despite advancements in computational power. By adopting CMoM with an exact Kernel, we have successfully addressed the main limitations of the traditional MoM, which can be categorized into seven key areas:

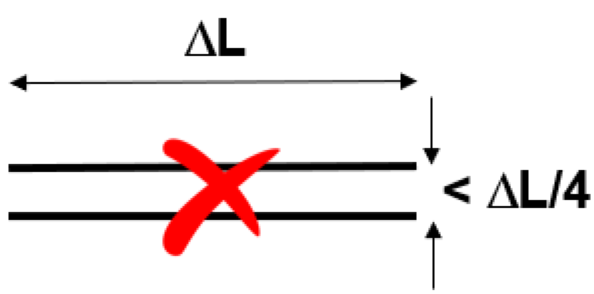
1. No curved wires:

Traditional MoM models rely on **straight wire segments**, which are suitable for linear antennas such as dipoles and their arrays. However, many antennas and structures have **curved shapes**. In traditional MoM, curved wires are approximated using a series of straight-line segments, leading to modeling errors that persist throughout the simulation. This approximation often produces inaccurate results for curved antennas like loops, helices, and spirals, particularly in terms of feed point impedances.



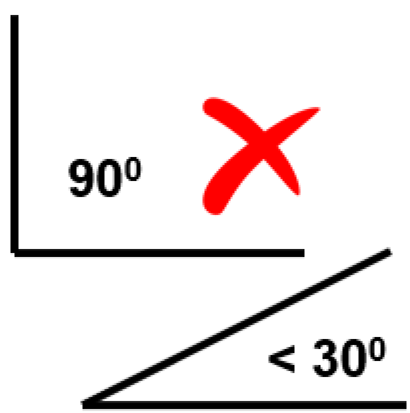
2. Wire spacing limitation:

Another limitation of traditional MoM is the **spacing between parallel wires**. Misleading results occur when the spacing between segments is less than a quarter of the segment length. As a result, the traditional MoM becomes less applicable when modeling configurations with **close parallel wires**, such as in two-wire transmission lines.



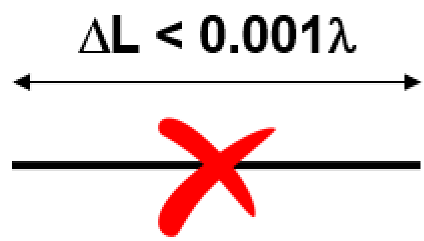
3. Issues with bent wires:

The thin-wire Kernel employed in traditional MoM leads to erratic numerical oscillations when wires are bent at **right angles** or have angles less than 30 degrees between adjacent segments.



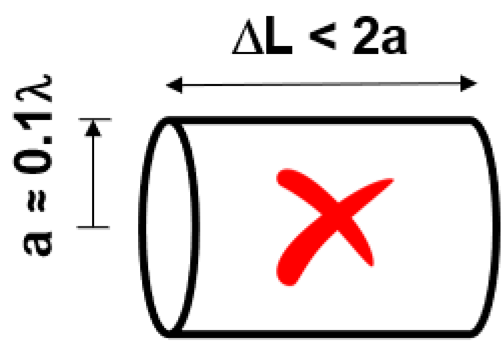
4. Short segment constraint:

Traditional MoM imposes a constraint on the segment length, requiring it to be greater than 0.001 of a wavelength. Consequently, the traditional MoM cannot be effectively applied at **very low frequencies**. For instance, when modeling an electric circuit of around 1 meter operating at 60 Hz, the segment length needed to accurately represent the circuit becomes at least 5,000 times shorter than the minimum segment length supported by traditional MoM. Therefore, the traditional MoM implementation falls short when modeling wire antennas at low frequencies.



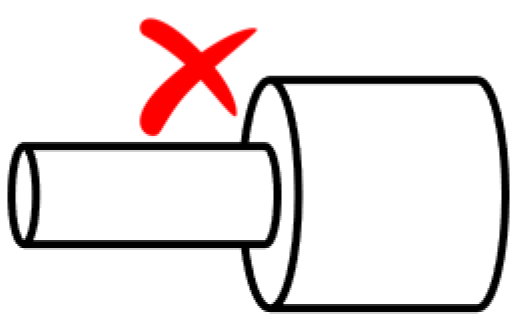
5. Thin wire requirement:

Thick wires deviate from the thin-wire approximation assumption, where current flow is limited to the wire axis rather than its surface. This deviation introduces significant errors in the results.



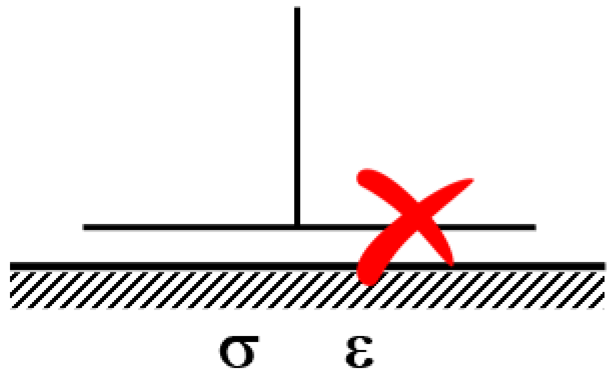
6. Tapered wire issues:

Changes in radius between adjacent segments create **non-physical discontinuities** in traditional MoM simulations.

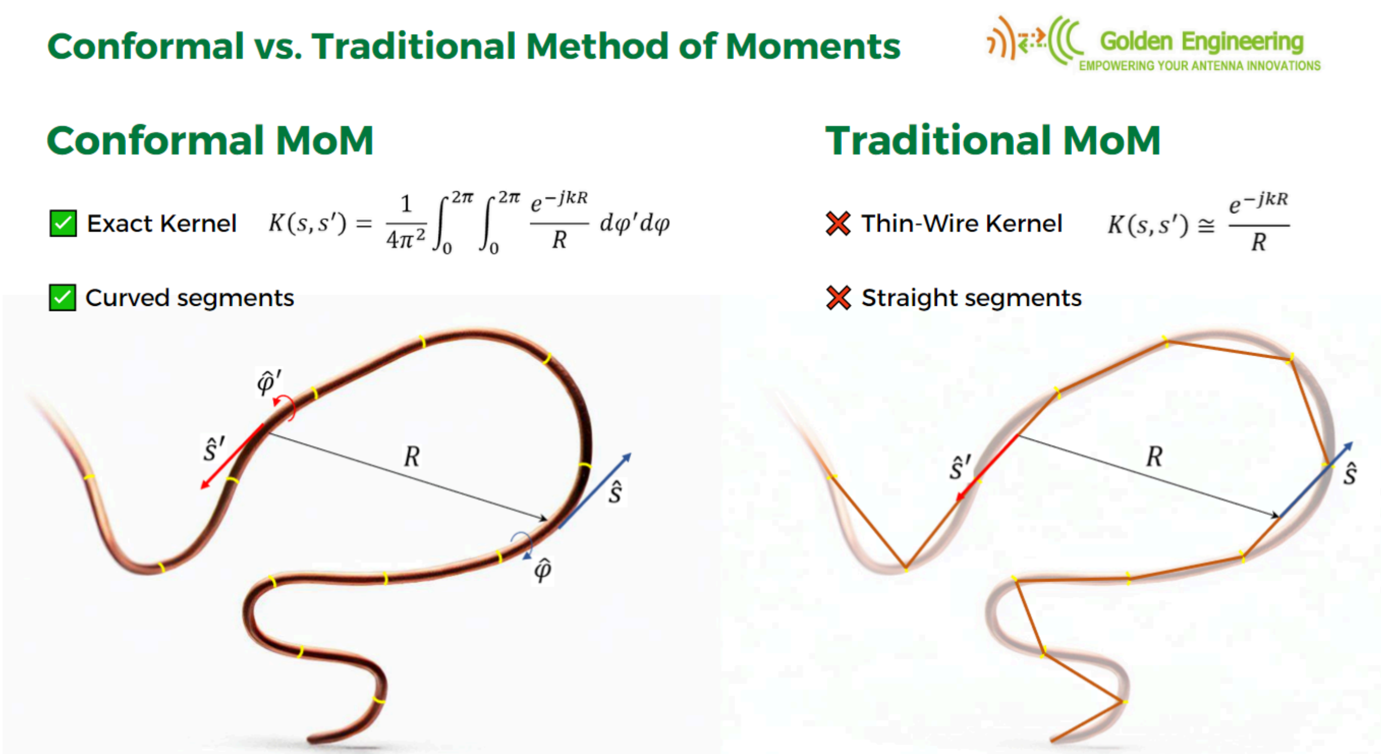


7. Proximity to lossy ground plane affects horizontal wires:

Antennas such as monopoles positioned above ground screens with **elevated radial wires** exhibit diverging input impedance and inaccurate antenna efficiency due to the influence of the lossy ground plane.



Thanks to the **Conformal Method of Moments (CMoM) with Exact Kernel**, AN-SOF has successfully **eliminated these limitations**. CMoM employs **conformal segments** that accurately capture the structure's contour, enabling an exact representation of geometric details. Conformal segments, resembling curved cylindrical tubes, enable precise modeling of **curved wires**. By employing the **exact Kernel** instead of the thin-wire approximation, AN-SOF overcomes limitations associated with bent wires, small wire spacings, and segment lengths. This approach facilitates highly accurate calculations compared to the traditional method.



With the implementation of CMoM and an exact Kernel formulation, AN-SOF achieves **enhanced accuracy**, **reduced computational requirements**, and **more efficient simulations**. The improved method enables AN-SOF to simulate a wide frequency range, spanning from extremely low frequencies (e.g., 60 Hz circuits) to microwave antennas.

> **AN-SOF** stands as the only antenna modeling software that offers a calculation engine based on the **Conformal Method of Moments** with an **Exact Kernel**.

Simulation Setup

The Setup Tab

Before **drawing wires** or **running a simulation**, it's recommended to configure the simulation parameters in the **Setup** tab. This tab contains the following panels: **Frequency**, **Environment**, **Far-Field**, **Near-Field**, **Excitation**, and **Settings** (see **Fig. 1**).

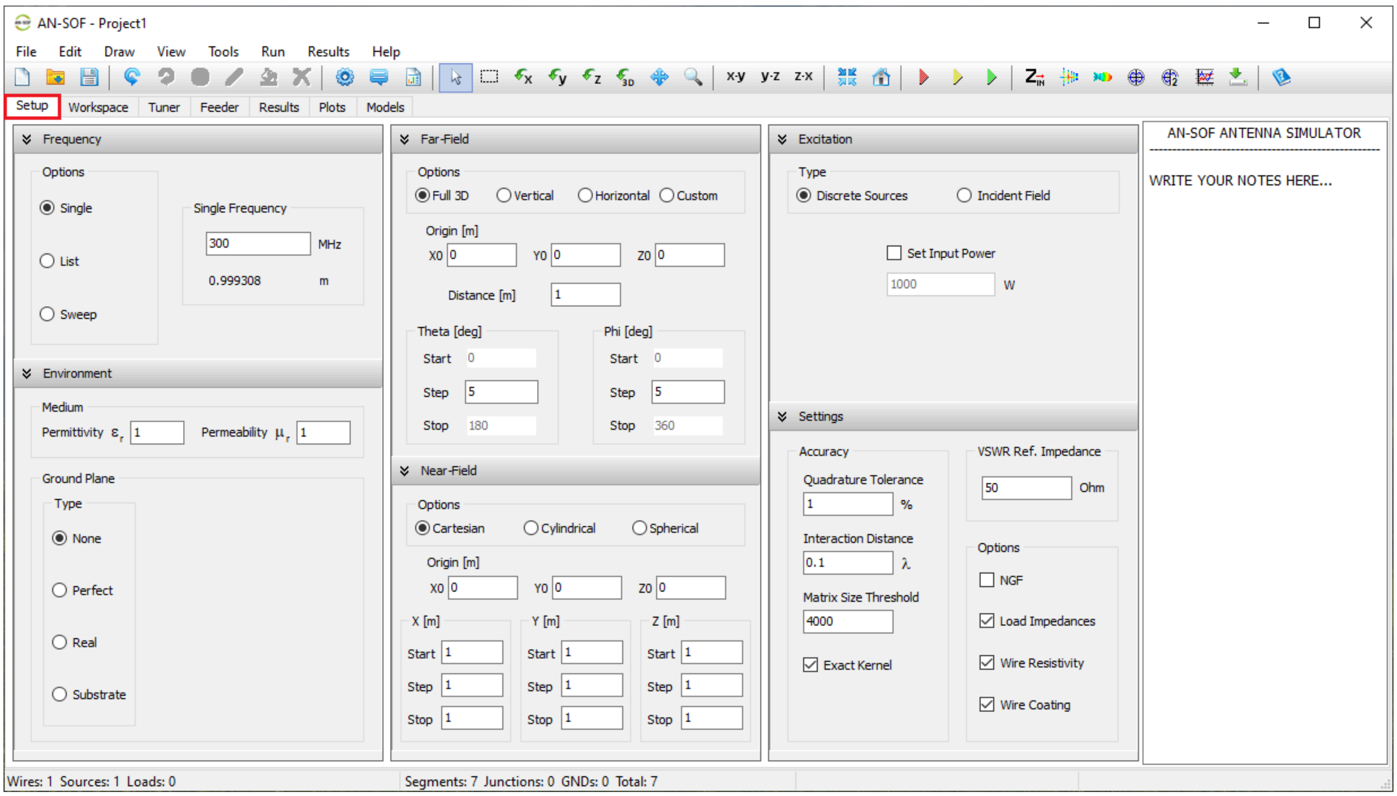


Fig. 1: Setup tab for configuring simulation parameters.

- **Frequency panel**: Specify the operating frequencies for the project.
- **Environment panel**: Set the relative permittivity and permeability of the surrounding medium, as well as the ground plane type.
- **Far-Field panel**: Define the angular ranges for far-field calculations.
- **Near-Field panel**: Define the observation points for near-field calculations.
- **Excitation panel**: Choose the excitation type for the structure. If using discrete sources, set the total input power. If using an incident field, define its direction and polarization.

[**Settings panel**](#): Set the reference impedance for VSWR and adjust parameters to fine-tune calculation accuracy.

Additionally, a **Note** panel on the right side of the Setup tab allows you to add project-related notes. These notes are saved in a `.txt` file in the same folder as the project file, using the same filename.

Specifying the Frequencies

To define the operating frequencies, go to the **Setup** tab in the main window and select the **Frequency** panel. You can choose from three frequency modes: **Single**, **List**, or **Sweep**.

- **Single**: Enter a single frequency value in the *Single Frequency* box (see **Fig. 1**). The corresponding wavelength is displayed automatically below the frequency field.
- **List**: Enter multiple frequencies in the *Frequency List* box (**Fig. 2**). You can import a list from a text file using the **Open** button, or export the current list using the **Save** button.
- **Sweep**: Perform a frequency sweep using either a **linear** or **logarithmic** scale.
  - For a *linear sweep*, specify the start, step, and stop frequencies (**Fig. 3**).
  - For a *logarithmic sweep*, enter the start and stop frequencies, along with a multiplication factor (**Fig. 4**).

To change the frequency unit (e.g., Hz, kHz, MHz, GHz), go to **Tools > Preferences** in the main menu, and select the desired unit on the **Units** page of the *Preferences* dialog box. See **Preferences** for more details.

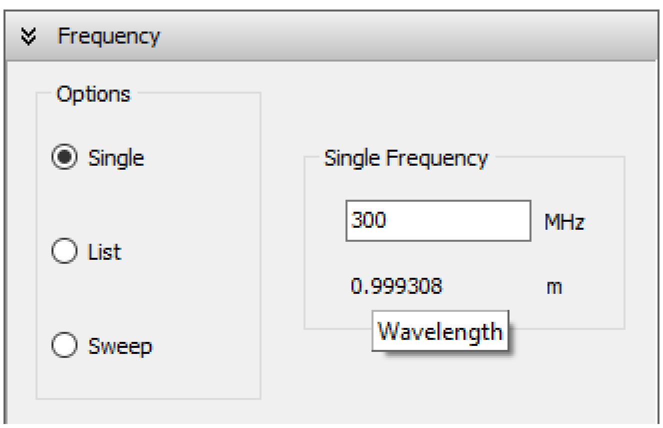


Fig. 1: Frequency panel in the Setup tab with a single frequency specified.

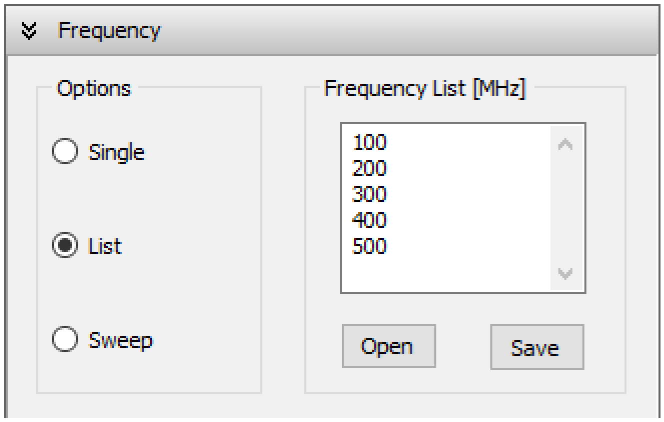


Fig. 2: Frequency panel in the Setup tab with a list of frequencies specified.

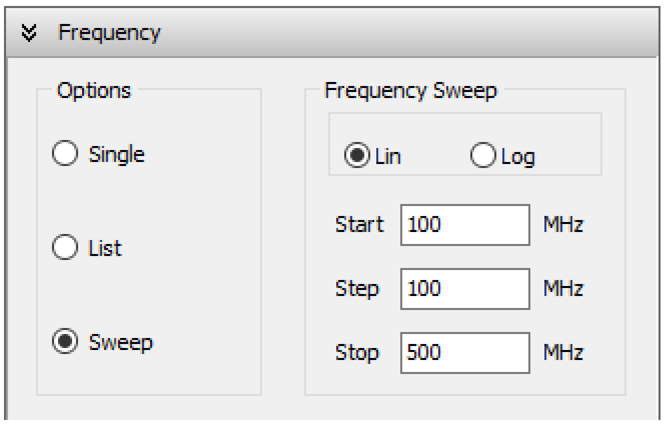


Fig. 3: Frequency panel in the Setup tab with a linear frequency sweep set.

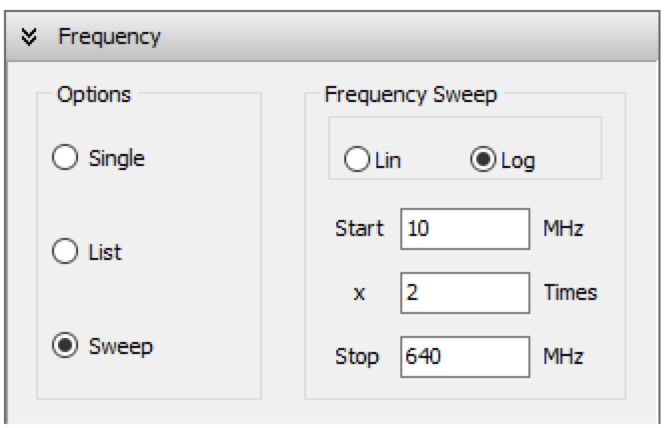


Fig. 4: Frequency panel in the Setup tab configured for a logarithmic frequency sweep.

Defining the Environment

Environment and Ground Plane Options

The environment surrounding an antenna or metallic structure can be either free space or include a ground plane. In the case of free space, you can define the relative permittivity (dielectric constant ε_r) and magnetic permeability (μ_r) of the medium, with vacuum corresponding to $\varepsilon_r = 1$

and μ_r = 1.

Three ground plane options are available:

- **Perfect Electric Conductor (PEC):** An ideal, lossless ground plane.
- **Real (Lossy) Ground:** A ground plane with user-defined conductivity (σ) and relative permittivity (ε_r). Optionally, a radial ground screen can be added.
- **Dielectric Substrate Slab:** Used for modeling microstrip lines and printed antennas.

All ground planes lie in the **xy-plane**. The available environment settings will be described in detail in the sections below.

Configuring the Medium Properties

To configure the medium, navigate to the **Setup tab > Environment panel**. Use the **Medium box** (**Fig. 1**) to set the **relative permittivity (ε_r)** and **permeability (μ_r)** of the surrounding medium. When a **ground plane** is used, these values represent the permittivity and permeability of the medium **above** the ground.

Note:

The **wavelength** displayed below the frequency set in the **Frequency panel** (when the **Single** option is selected) and the wavelength used internally by AN-SOF for calculations will be adjusted based on the specified permittivity and permeability of the medium. Specifically, these properties cause **the wavelength to shorten** accordingly relative to the wavelength in a vacuum.

None (Free Space)

Selecting the **None** option simulates the antenna in free space. The relative permittivity (ε_r) and permeability (μ_r) specified in the **Medium** box (see **Fig. 1**) define the properties of the surrounding environment.

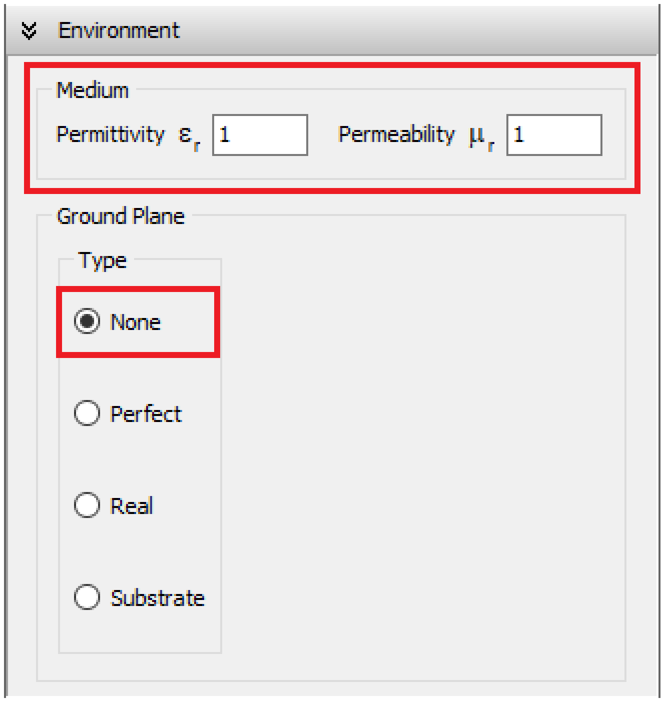


Fig. 1: "Medium" and "Ground Plane" boxes in the "Environment" panel. The "None" ground plane option is selected, indicating free space.

Perfect (PEC Ground)

Selecting the **Perfect** option places an infinitely large, **perfectly electrically conducting (PEC)** ground plane at a specified height relative to the xy-plane (see "Z Position" in **Fig. 2**). The ground plane remains parallel to the xy-plane, and its position is defined by the **Z** coordinate: a negative Z places the ground plane below the xy-plane, while a positive value places it above.

When using a perfect ground plane, all wires must be located above it—that is, each wire must have a Z-coordinate greater than or equal to the specified ground plane position. Wires crossing through or lying below the ground plane are not allowed. Additionally, horizontal wires placed directly on the ground plane are unsupported. However, **connections from wire ends to the ground plane are permitted**.

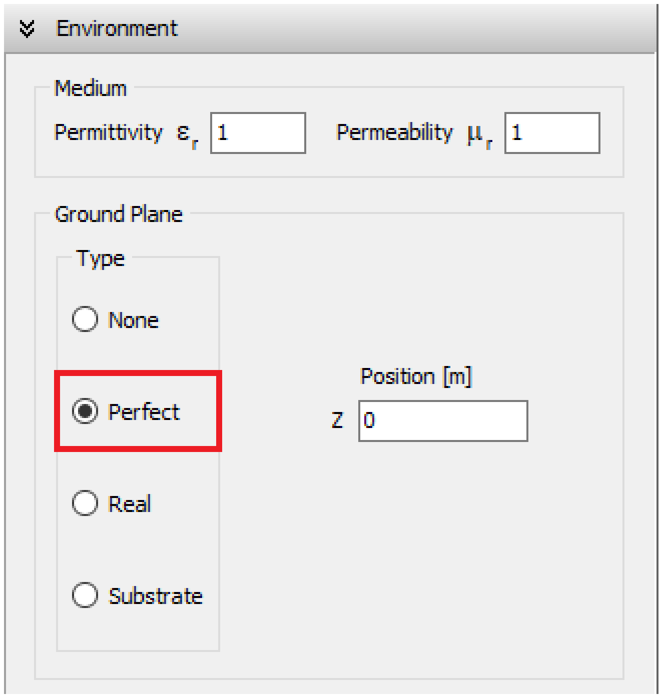


Fig. 2: Perfect ground plane positioned at Z = 0 (i.e., on the xy-plane).

Real (Lossy Ground)

Selecting the **Real** option places a lossy ground plane at the xy-plane (z = 0), with user-defined conductivity (σ) and relative permittivity (ε_r), as shown in **Fig. 3**. Four modeling methods are available for real ground calculations:

- **Sommerfeld–Wait Asymptotic**
- **Reflection Coefficients**
- **Radial Wire Ground Screen**
- **Sommerfeld–Norton**

These options are described in the following sections.

All wires must be positioned **above** the ground plane (z > 0). Horizontal wires **directly on** the ground plane are **not supported**.

Wire end connections to the ground are allowed when using either the **Sommerfeld–Wait Asymptotic** or **Radial Wire Ground Screen** options.

The **Reflection Coefficients** and **Sommerfeld–Norton** models assume perfect (zero-Ohm) wire connections to ground and yield reasonable results **only for vertical wires** connected to good conducting grounds—not dielectric surfaces. These models are not suitable when vertical wire-to-ground connections are made through dielectric materials.

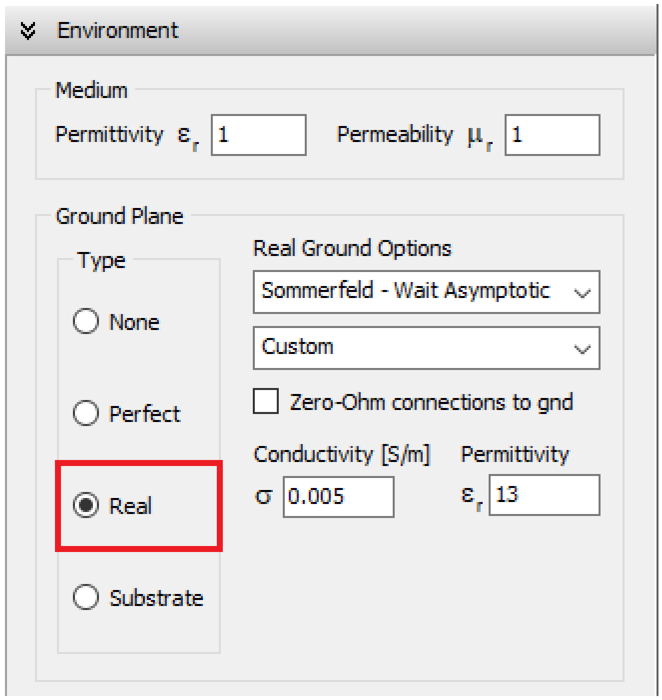


Fig. 3: "Real" option selected, showing the parameters of a real ground plane.

Real Ground: Sommerfeld – Wait Asymptotic

This option models the antenna/wire currents using a **hybrid approach** that combines a perfect ground plane with loss impedances to account for power dissipation in the real ground—particularly important when wires are close to or connected to the ground. Developed by [Prof. James R. Wait](), this model is especially effective for low-frequency (LF) and medium-frequency (MF) antennas, where ground conductivity tends to be high. However, it remains applicable at higher frequencies **as long as the ground behaves as a good conductor**.

The model uses the ground's conductivity (σ) and permittivity (ε_r) to compute both **near-field** and **far-field** radiation, applying **Norton's asymptotic approximations** to the exact Sommerfeld solution. The **far-field** results match those predicted by **Fresnel reflection coefficients**.

Wire connections to ground are supported. If a wire's starting or ending point is located at **z = 0**, it will automatically be connected to the ground. These connections are treated as imperfect (lossy) by default, meaning power can be dissipated at the connection point due to current flow. If the **Zero-Ohm connections to ground** option is enabled, these connections are instead treated as perfect (lossless), though the presence of the lossy ground still influences the overall near and far fields.

In summary, the *Sommerfeld – Wait Asymptotic* model is appropriate when the ground can be considered a good conductor at the operating frequency and when the structure includes wire-to-ground connections.

Real Ground: Reflection Coefficients

When this option is selected, the ground conductivity (σ) and relative permittivity (ε_r) influence the current distribution on the antenna or wire structure above the ground. As a result, the input impedance of a transmitting antenna is also affected by real ground conditions. This influence is determined using **a generalization of Fresnel's reflection coefficients**, which is theoretically valid when **wires are positioned several wavelengths above the ground**. In practice, however, good results have been reported for heights between one-quarter and one-half of a wavelength.

Near fields are calculated using **Norton's asymptotic approximations** to Sommerfeld's exact solution, allowing the electric and magnetic fields to be computed as a function of distance from the antenna. This enables analysis of field attenuation due to ground losses. The **far field**, by contrast, is computed using the standard **Fresnel reflection coefficients**.

Vertical wire connections to ground may be added when the ground behaves as a **good conductor at the operating frequency**, even though these connections theoretically violate the height requirement. These wire-to-ground connections are treated as **lossless** in this model.

In summary, the *Reflection Coefficients* ground model is best suited for wire structures located several wavelengths above ground, ideally without any wire-to-ground connections. However, if the ground is a good conductor at the operating frequency, vertical wire connections may be included—but results under such conditions should be interpreted with caution.

Real Ground: Radial Wire Ground Screen

When this option is selected, a ground screen made of **buried radial wires** is placed on the ground plane, which has the specified conductivity (σ) and relative permittivity (ε_r). The screen is centered at the origin (0,0,0), and its configuration is defined by user-specified parameters: the **number of radial wires**, their **length (or screen radius)**, and **wire radius**. These wires are assumed to be laid on the ground surface or buried at a depth less than one soil skin depth.

This model is based on **Prof. James R. Wait's theory** for **good conducting grounds**. It affects the current distribution on the antenna or wire structure by accounting for the power dissipated in the combined ground plane and wire screen system. As a result, both the presence of the screen and the ground properties (conductivity and permittivity) influence the input impedance of a transmitting antenna located above the screen. The same parameters are used to compute the **radiated near and far fields** using **Norton's asymptotic approximations** to Sommerfeld's solution and **Fresnel reflection coefficients**, respectively.

Wire-to-ground connections may be defined at wire endpoints located at **z = 0**. These are treated as *imperfect* by default, meaning that power is dissipated in the ground-screen system due to currents flowing between the ground and the wires. If the **Zero-Ohm connections to ground** option is enabled, these connections are treated as *perfect*, with no power dissipation at the contact point.

In summary, the *Radial Wire Ground Screen* model is suitable when the ground behaves as a good conductor at the operating frequency and when the structure includes wire-to-ground connections. The screen serves to increase the effective conductivity of the combined ground-screen system, reducing power losses beneath antennas placed above it. Typical use cases include monopole antennas in the form of poles or radiating towers used in broadcasting applications.

Real Ground: Sommerfeld – Norton

Unlike the previous ground models, which apply various approximations to compute the current distribution on wires above a lossy ground plane, the **Sommerfeld–Norton** model numerically solves **Sommerfeld's exact solution**. This enables simulations where the ground does **not** need to be a good conductor—it may instead be a dielectric medium. To approximate a purely dielectric ground, enter a very low conductivity (e.g., σ = 1E-6 S/m) along with the desired relative permittivity (ε_r).

Vertical wires can be placed very close to the ground, with a minimum height of **one wire radius** above ground. **Horizontal wires** can be simulated as low as **0.005λ** above ground. For vertical or horizontal wires at heights below **0.01λ**, it is important to increase the accuracy of the calculations. To do this, go to **Setup tab > Settings panel** and set the **Quadrature Tolerance** to **0.1%**, instead of the default 1% (see **Fig. 4**).

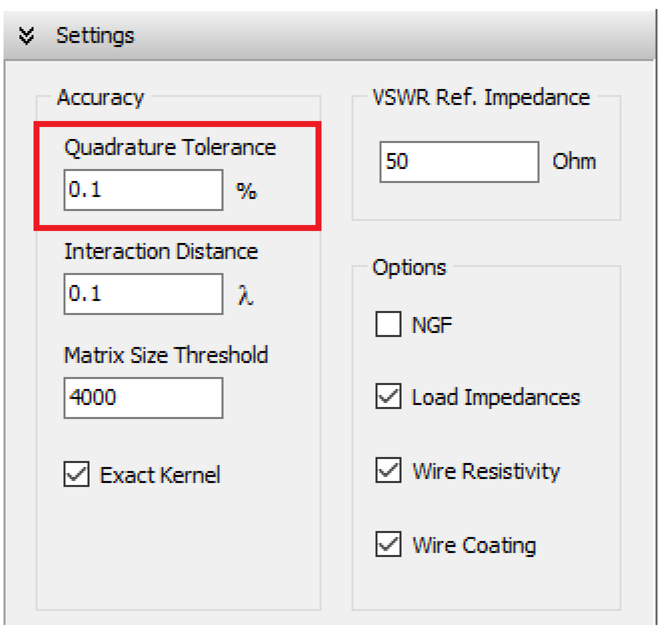


Fig. 4: Settings panel where the "Quadrature Tolerance" is set to 0.1% to enable accurate calculations for wire heights below 0.01λ.

The **near fields** are computed from the current distribution using **Norton's asymptotic approximation** to Sommerfeld's solution. This is accurate at distances **greater than about one-quarter wavelength** from the wire structure, which is sufficient for most practical applications. The

far field, in contrast, is calculated using **Fresnel reflection coefficients**, which yield the correct asymptotic expressions.

Wire-to-ground connections are theoretically **not permitted** in this model. However, they can be used with caution **only for vertical wires**, and **only if the ground behaves as a good conductor** at the operating frequency. In such cases, the model assumes **lossless connections** and employs **specular reflection** of the fields to compute the current distribution. These assumptions are valid only when the ground is effectively conductive.

In summary, the *Sommerfeld–Norton* model is the most accurate among the available ground models, but it comes with certain restrictions. It is recommended for **highly dielectric grounds** or situations where the ground **cannot** be considered a good conductor. Horizontal wires **must not** be placed below **0.005λ** above ground. Vertical wires may be connected to ground **only when** the ground is a good conductor, and the results in such cases should be interpreted with care.

Substrate (Grounded Dielectric Slab)

When this option is selected, a dielectric substrate with user-defined relative permittivity (ε_r) is placed beneath the *xy*-plane ($z = 0$), as shown in **Figs. 5** and **6**. The slab may extend infinitely or have finite dimensions in the *xy*-plane, where you can define its width along the X and Y directions. The substrate thickness, denoted as h, must also be specified. A **perfectly electrically conducting (PEC) ground plane** is located at $z = -h$, just below the dielectric slab (see **Fig. 6**). A drop-down list is available to select common substrate materials (e.g., FR4, RT/Duroid, Rogers RO). Note that **the dielectric loss tangent cannot be specified**—it is assumed to be **zero**, meaning the substrate is treated as lossless.

When the **Substrate** option is active:

- All wires must lie **horizontally** in the *xy*-plane ($z = 0$). These wires typically represent traces of printed antennas, microstrip lines, or PCB tracks.
- The only exception is for **vertical wires (vias)**, which may connect traces at $z = 0$ to the ground plane at $z = -h$. These are commonly used to feed the structure with voltage or current sources.

Important constraints:

- The PEC ground plane beneath the substrate is **mandatory** and cannot be omitted. As a result, **ungrounded dielectric substrates are not supported**.
- Wires above the *xy*-plane ($z > 0$) or below the ground plane ($z < -h$) are not allowed in this model.

The **Substrate** model is an extension of the **Conformal Method of Moments** tailored for printed wire structures. It has the following limitations:

Model Limitations

Single-Layer, Lossless Substrate Only

- Only one dielectric layer is supported (multilayer stacks are not).
- The substrate must be lossless (loss tangent is assumed to be zero).
- No holes or cutouts are permitted in the substrate.

Finite-Size Substrate Constraints

- The substrate must be rectangular in shape.
- Traces must be placed at least 5× their width away from the substrate edges.

Ground Plane and Vias

- A PEC ground plane is always present and cannot be disabled.
- Vertical wires (vias) can be added to connect traces to the ground, typically for feeding.

No Slot-Based Designs

Slot antennas or patches with slots cannot be modeled due to software limitations.

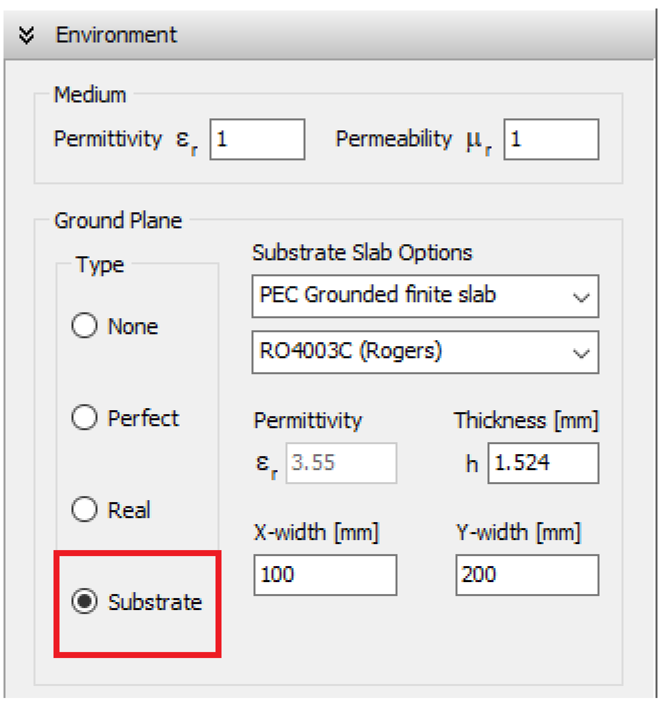


Fig. 5: Finite dielectric substrate slab configured using the "Substrate" option.

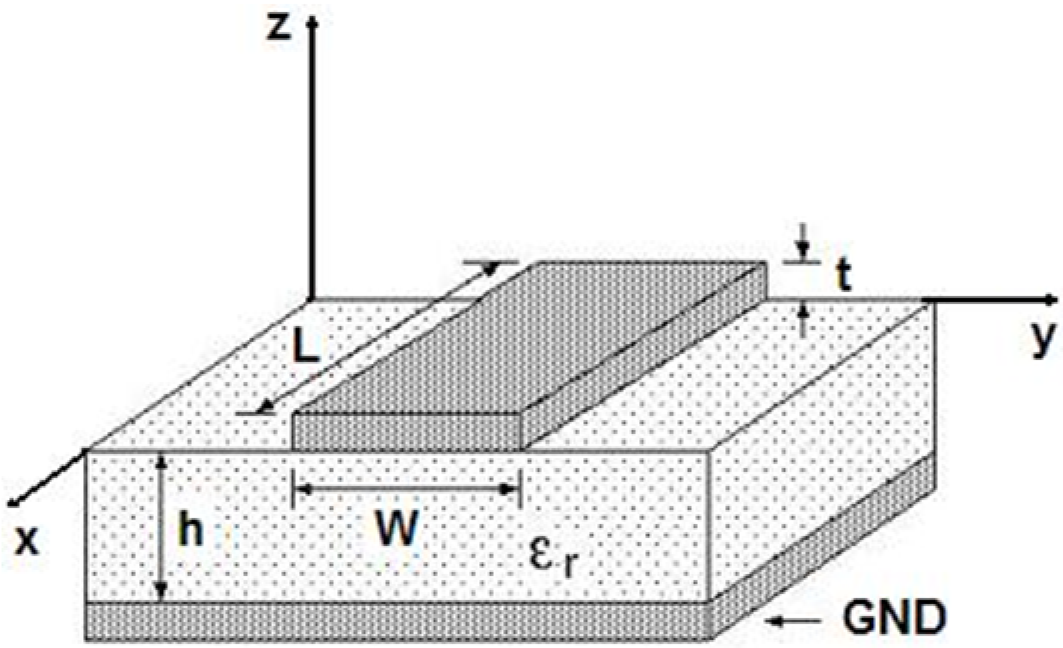


Fig. 6: Sketch of a trace on a grounded dielectric substrate, showing the Cartesian axes for reference.

Far Field Parameters

To configure radiation pattern parameters, navigate to the **Setup tab** in the main window and select the **Far-Field** panel (**Fig. 1**).

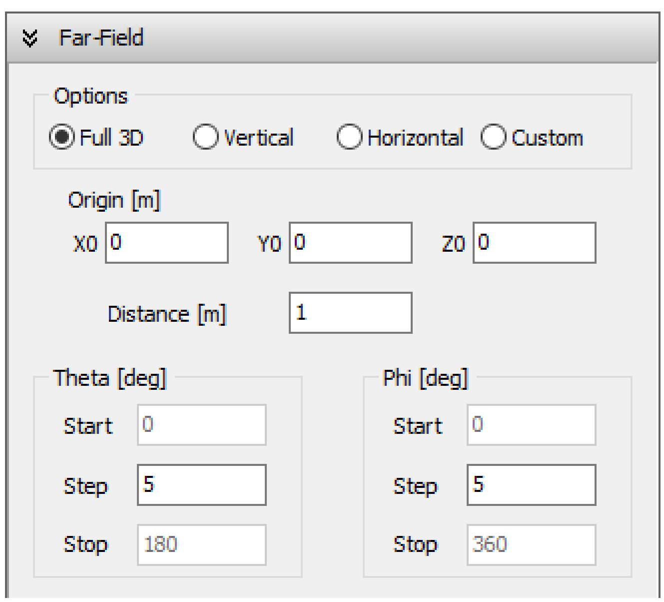


Fig. 1: Far-Field panel in the Setup tab, used to configure radiation pattern parameters.

The far field can be computed **after calculating the current distribution**. Thus, the parameters set in the **Far-Field panel** have no effect on the determination of the currents and can be modified at any time. However, **the far field must be recalculated** every time these parameters are modified.

There are **four options** for radiation pattern calculations:

Full 3D

The far field is calculated in angular ranges that cover the entire 3D space, allowing you to obtain **3D radiation lobes**. The steps for the *Theta (zenith)* and *Phi (azimuth)* angles can be set in the **Theta [deg]** and **Phi [deg]** boxes.

Vertical

The far field is calculated at a vertical slice for a given *Phi (azimuth)* angle. The step for the *Theta (zenith)* angle can be set in the **Theta [deg]** box, while the fixed *Phi* can be set in the **Phi [deg]** box.

Horizontal

The far field is calculated at a horizontal slice for a given *Theta (zenith)* angle. The step for the *Phi (azimuth)* angle can be set in the **Phi [deg]** box, while the fixed *Theta* can be set in the **Theta [deg]** box.

Custom

The far field is calculated for the specified ranges of angles *Theta (zenith)* and *Phi (azimuth)*. The start, step, and stop values for *Theta* and *Phi* can be set in the **Theta [deg]** and **Phi [deg]** boxes.

Additionally, the following parameters can be set:

Origin (X0,Y0,Z0)

This is any point used as a phase reference. Its coordinates do not affect the shape of the radiation pattern. The **3D radiation pattern** will be plotted centered at this point.

Distance

This represents the distance from **(X0,Y0,Z0)** to an observation point in the far-field region. A **normalized far-field pattern** can be obtained by setting **Distance = 1 meter**.

The zenith and azimuth angles, **Theta** (θ) and **Phi** (ϕ), are shown in **Fig. 2**. The figure also illustrates the **Distance** *R* from the structure to an observation point in the far-field zone. These three numbers (R, θ, ϕ) define the spherical coordinates of the far-field point.

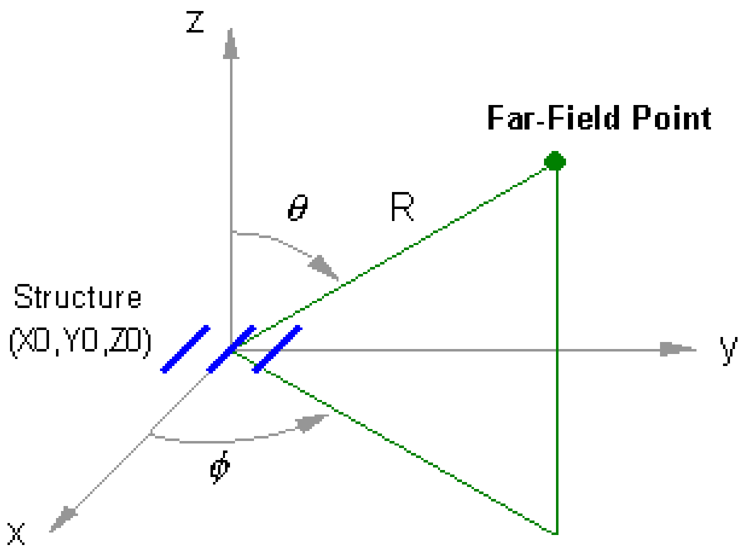


Fig. 2: Spherical coordinates (R, θ, ϕ) defining the position of a Far-Field observation point.

A frequently asked question is **how to displace the center of the radiation pattern** in a [3D plot displaying the radiation lobes](). The answer lies in setting the **Origin (X0, Y0, Z0)** in the **Far-Field panel**. This allows you to adjust the reference point for the radiation pattern, enabling better visualization of the results. For more details, refer to this article:

> [How to Adjust the Radiation Pattern Reference Point for Better Visualization]()

Near Field Parameters

To configure the observation points for the calculation of near fields, navigate to the [Setup tab]() in the main window and select the **Near-Field panel**.

The near field can be computed [after calculating the current distribution](). Thus, the parameters set in the **Near-Field panel** have no effect on the determination of the currents and can be adjusted at any time. However, [the near field must be recalculated]() whenever these parameters are modified.

The **Near-Field panel** provides three coordinate system options for near-field calculations:

- **Cartesian**
- **Cylindrical**
- **Spherical**

By selecting one of these options, near fields can be calculated in **Cartesian**, **Cylindrical**, or **Spherical coordinates**, depending on your preference or the requirements of your analysis.

Cartesian Coordinates

If the **Cartesian option** is selected, the following parameters can be configured for near-field calculations (**Fig. 1**):

Origin (X0, Y0, Z0)

This is the origin of the Cartesian coordinate system used to define the observation points where near fields will be calculated.

X

This box is used to specify the **x-coordinates** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the x-coordinates. The start and stop values are measured relative to **X0**.

Y

This box is used to specify the **y-coordinates** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the y-coordinates. The start and stop values are measured relative to **Y0**.

Z

This box is used to specify the **z-coordinates** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the z-coordinates. The start and stop values are measured relative to **Z0**.

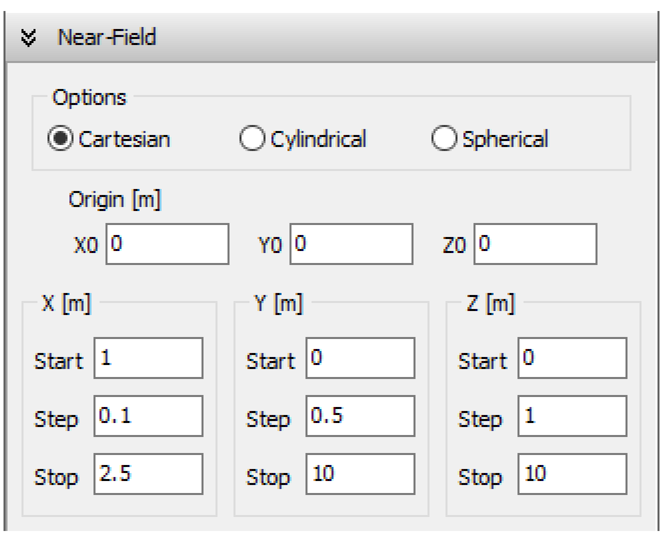


Fig. 1: Near-Field panel in the Setup tab with the Cartesian option selected.

Cylindrical Coordinates

If the **Cylindrical option** is selected, the following parameters can be configured for near-field calculations (**Fig. 2**):

Origin (X0, Y0, Z0)

This is the origin of the cylindrical coordinate system used to define the observation points where near fields will be calculated.

R

This box is used to specify the **radial distances (R-coordinates)** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the radial distances. The start and stop distances are measured relative to the origin **(X0, Y0, Z0)**.

Phi

This box is used to specify the **azimuth angles (phi-coordinates)** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the azimuth angles in degrees.

Z

This box is used to specify the **z-coordinates** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the z-coordinates.

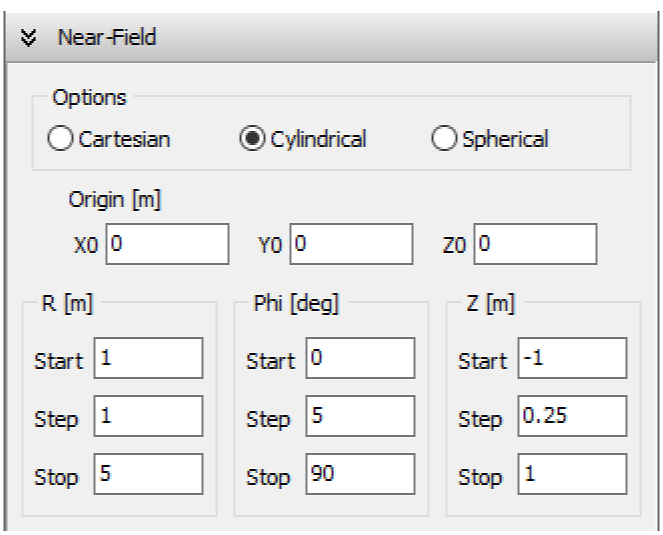


Fig. 2: Near-Field panel in the Setup tab with the Cylindrical option selected.

Spherical Coordinates

If the **Spherical option** is selected, the following parameters can be configured for near-field calculations (**Fig. 3**):

Origin (X0, Y0, Z0)

This is the origin of the spherical coordinate system used to define the observation points where near fields will be calculated.

R

This box is used to specify the **radial distances (R-coordinates)** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the radial distances. The start and stop distances are measured relative to the origin **(X0, Y0, Z0)**.

Theta

This box is used to specify the **zenith angles (theta-coordinates)** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the zenith angles in degrees.

Phi

This box is used to specify the **azimuth angles (phi-coordinates)** of the observation points where near fields will be calculated. You must set the start, step, and stop values for the azimuth angles in degrees.

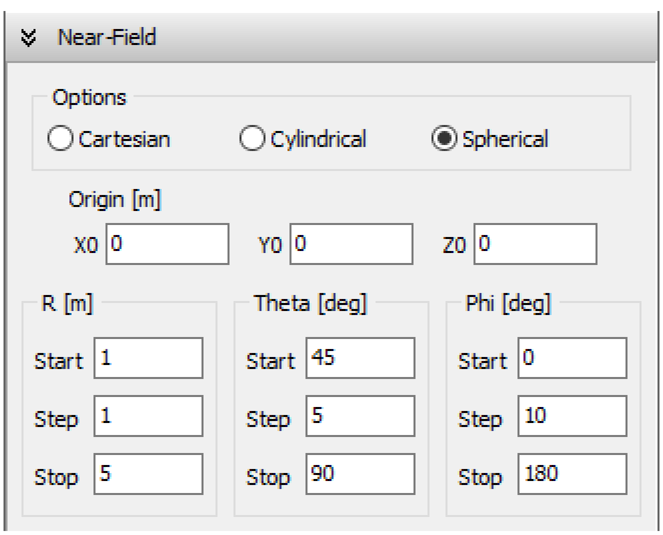


Fig. 3: Near-Field panel in the Setup tab with the Spherical option selected.

Near-Field Calculation Workflow

The **Cartesian**, **Cylindrical**, and **Spherical options** described above define a grid of observation points where the near field will be computed. The near **electric field (E-field)** and near **magnetic field (H-field)** can be calculated separately, or you can compute them sequentially by clicking **Run Currents and Near-Field (F11)** in the **Run menu** or toolbar. This command first calculates the current distribution and then computes the near fields.

Alternatively, the calculation of the current distribution, far-field, and near-field can be executed in a single step by selecting the **Run ALL (F12)** command in the **Run menu**.

If the near **H-field** is not required, you can disable its automatic calculation by turning off the **"Run ALL also calculates the H-Field"** option. This setting can be found in the **Options tab** of the **Preferences window**. To access the Preferences window, click the **gear button** in the main toolbar or navigate to **Tools > Preferences** in the main menu.

> **Note:**
>
> When defining the grid of points for near-field calculations, the closest distance from a near-field point to a wire with a circular cross section must be at least **five times the wire radius**. For flat strip conductors (such as those used in metallic surfaces), the near-field point must be located at a distance of **at least one strip width** from the strip.

Defining the Excitation

Accessing Excitation Settings

Navigate to:
Setup tab > Excitation panel
Two excitation types available (**Fig. 1**):

1. **Discrete Sources**
2. **Incident Field**

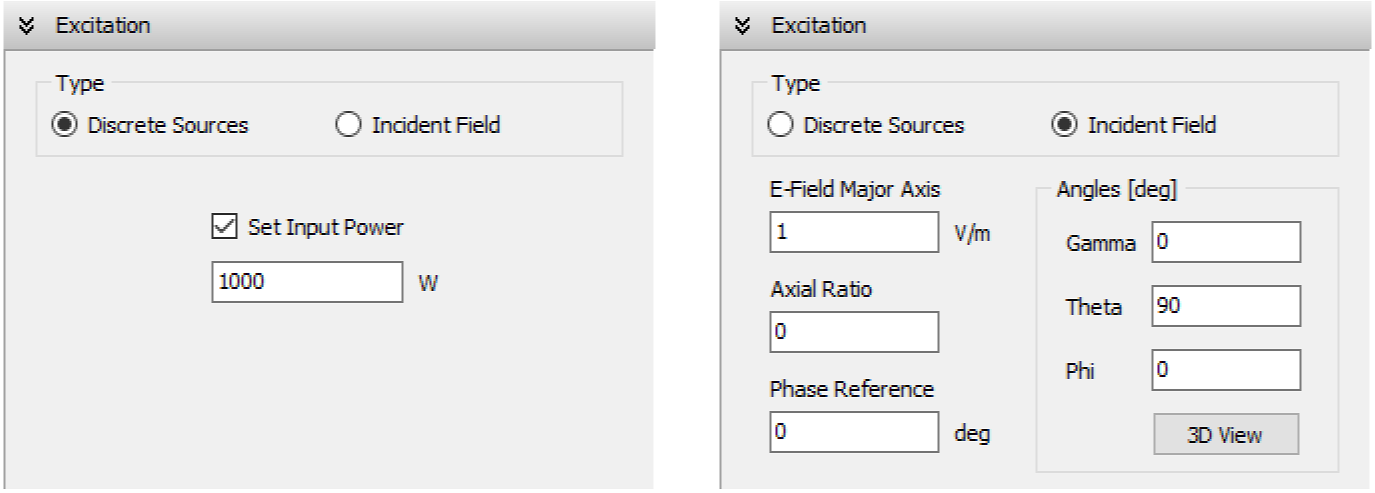


Fig. 1: Excitation panel showing Discrete Sources and Incident Field options.

Discrete Sources

- **Purpose**: Calculate current distribution using voltage/current sources on wires
- **Power Configuration**:
  - *Specified Input Power*: Sources auto-adjust to achieve target power (Watts)
  - *Unspecified Power*: Sources remain constant; power becomes output result

Incident Field (Plane Wave Excitation)

Define an incident plane wave's direction and polarization:

Key Parameters

Parameter	Description	Units/Values
E-Field Major Axis	Linear: RMS amplitude (V/m) Elliptical: Major axis of polarization ellipse	V/m
Axial Ratio	Minor/major axis ratio: – Positive: Right-handed ellipse – Negative: Left-handed ellipse – Zero: Linear polarization	Unitless
Phase Reference	Phase shift at coordinate origin	Degrees
Gamma	Linear: Polarization angle from incidence plane Elliptical: Major axis angle from incidence plane	Degrees
Theta	Zenith angle of incidence	Degrees
Phi	Azimuth angle of incidence	Degrees

Visual reference: Fig. 2 shows parameter definitions

3D Visualization

Click **3D View** (**Fig. 3**) to interactively set:

- Wave direction
- Polarization state

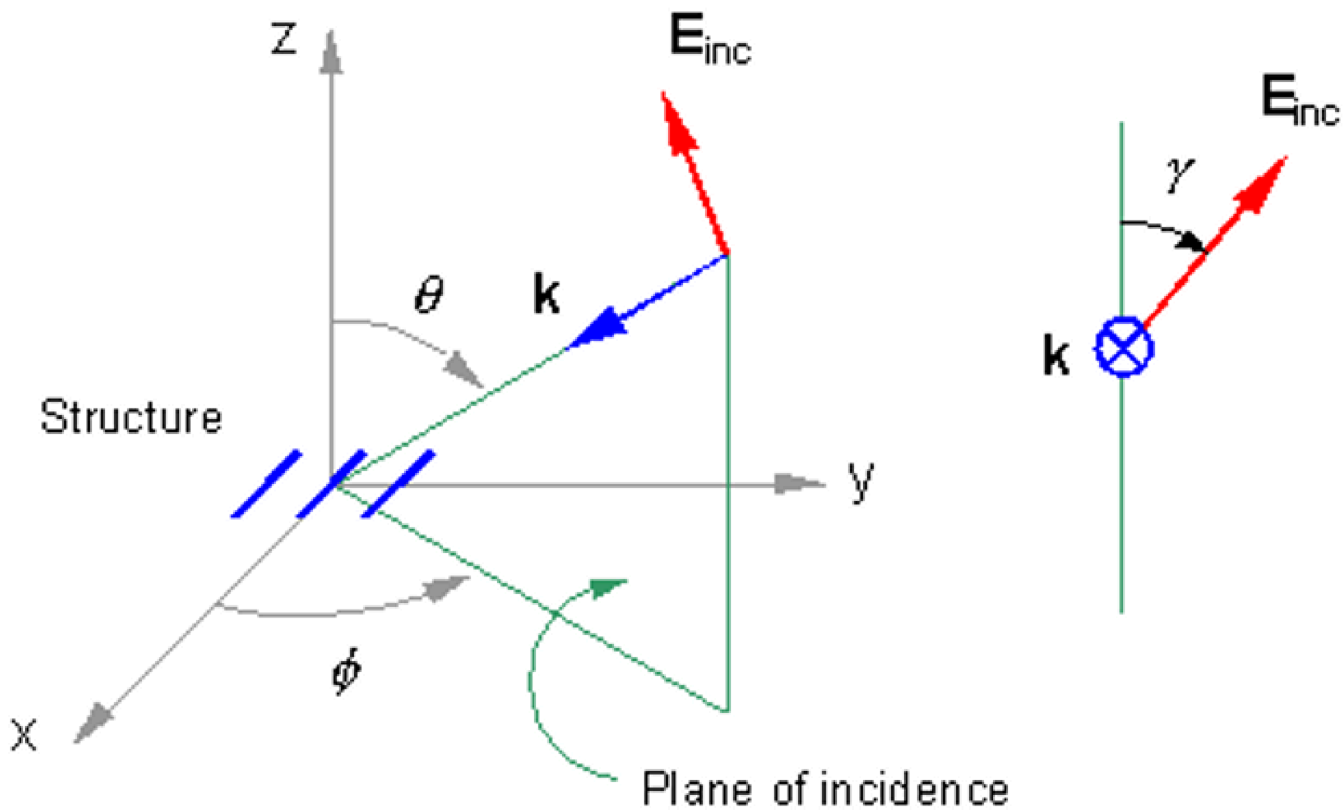


Fig. 2: Incident plane wave parameters (Theta, Phi, Gamma) and polarization definition.

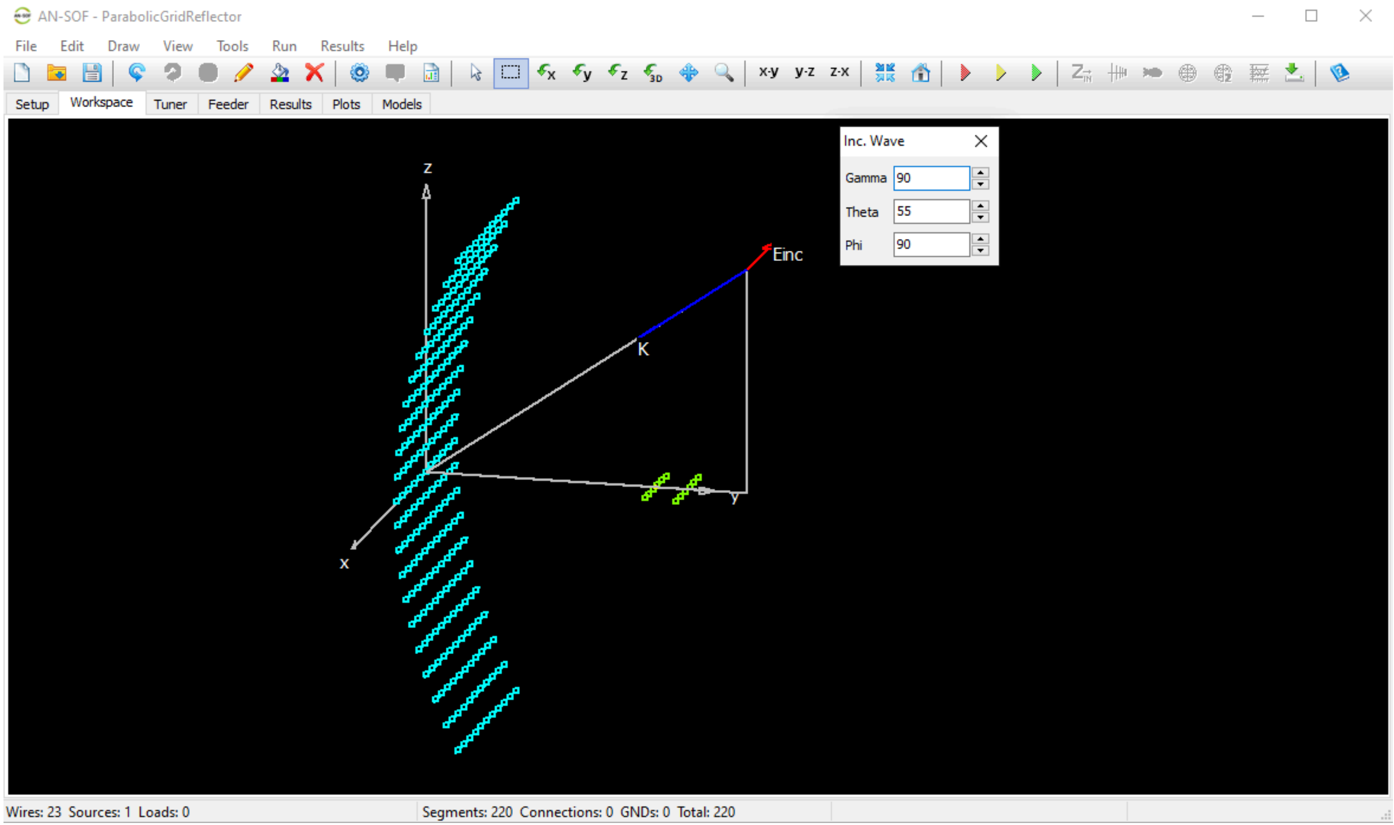


Fig. 3: 3D View interface for incident field definition, showing Einc (major axis of polarization ellipse).

> **Note**
>
> When an incident plane wave is used as the excitation, **any discrete sources present will be ignored during the simulation**.

> **Note**
>
> When an incident plane wave is used as the excitation, the calculated [far-field](#) and [near-field](#) results represent the **scattered fields**. The resulting radiation pattern is the **scattered field pattern**, observed in the far-field region where the scattered field amplitude decays with the inverse of the distance (1/r), and the scattered power density decays with the inverse square of the distance ($1/r^2$) from the structure.

The Settings Panel

To access advanced simulation options, go to the [Setup](#) tab and select the **Settings** panel. Here, you can:

- Fine-tune the **accuracy** of the simulation,
- Set the **reference impedance** for VSWR calculations, and
- Adjust various **options that affect the calculation engine**.

These settings provide greater control over simulation precision and convergence behavior. See **Fig. 1**.

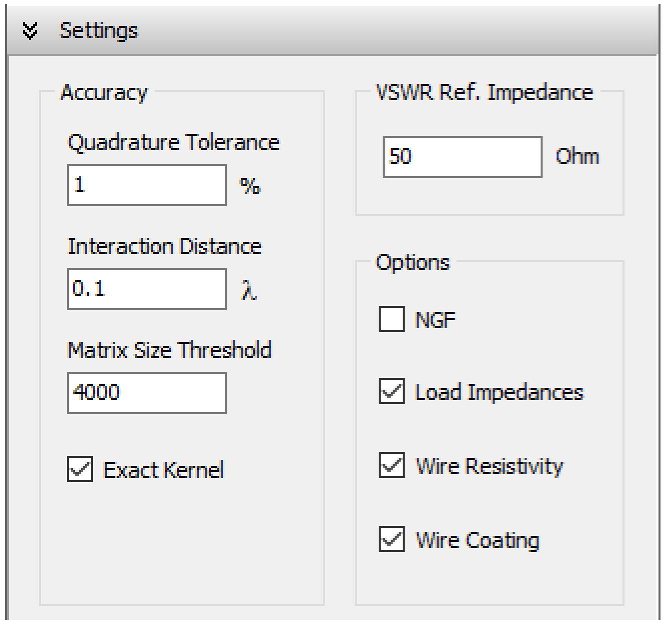


Fig. 1: Settings panel in the Setup tab.

Controlling Numerical Precision

The **Accuracy** section in the **Settings** panel allows you to control the precision of the simulation and manage memory usage for large models.

> **Quadrature Tolerance** defines the acceptable numerical error in the evaluation of integrals between wire segments that are closer than the **Interaction Distance**. This parameter governs the precision of electromagnetic interaction calculations. The default value is **1%**.
>
> **Interaction Distance** sets the maximum separation (in wavelengths) between segments for which the numerical integration guarantees an error below the *Quadrature Tolerance*. For segments farther apart than this distance, a **third-degree polynomial approximation** is used instead of the traditional **Hertzian dipole approximation**, offering improved accuracy for **curved segments**. The default value is **0.1λ**.
>
> **Matrix Size Threshold** determines the maximum size of the impedance matrix (N × N, where N is the number of wire segments) that is computed in **double precision**. For models with more segments than the set threshold, **single precision** is used to reduce memory usage. While this may slightly reduce accuracy, it enables the simulation of larger models on systems with limited Windows on-board memory. The default threshold is **4,000**.
>
> **Exact Kernel** enables the use of the exact formulation of the **Electric Field Integral Equation (EFIE)**, significantly improving accuracy for antennas made with **thicker wire segments**. When this option is disabled, an **extended thin-wire approximation** is used. See **The Exact Kernel** for more information.

VSWR Reference Impedance Configuration

The reference impedance serves as the normalization value for VSWR calculations throughout the simulation. This parameter is particularly important when matching simulation conditions to real-world measurement setups.

Key features:

- Defaults to **50 Ω** (standard for most RF systems)
- Commonly adjusted to **75 Ω** for television/video applications
- Requires refreshing the results (**Ctrl + R**) to update all VSWR-dependent data

Simulation Control Options

These global toggles enable efficient comparison of different simulation scenarios without modifying individual element properties. They are especially valuable for parametric studies and troubleshooting.

Option	Technical Implementation	Practical Application
NGF	Stores LU-decomposed impedance matrix after first calculation	Accelerates repeated simulations with varying source excitations
Load Impedances	Master switch for all lumped load elements in the model	Quickly compare loaded vs. unloaded system performance
Wire Resistivity	Toggles Ohmic loss calculation for all conductors	Evaluate conductor loss impact on system efficiency
Wire Coating	Enables/disables dielectric coating effects	Analyze insulation impact on antenna performance

Implementation Note:

These options provide non-destructive alternatives to physically removing/readding model elements when running comparative analyses.
Project Details

A summary of the project information can be displayed by navigating to **View > Project Details** in the main menu. This opens the **Project Details window** (**Fig. 1**). Alternatively, you can access this window using the dedicated button on the toolbar.

The text in the **Project Details window** can be selected and copied to the clipboard in the usual way (**Ctrl + C** to copy from the window and **Ctrl + V** to paste elsewhere).

The **Project Details window** provides a comprehensive summary of the following project information:

- **Project Name**: The name of the project.
- **Date and Time**: The date and time when the project was last saved.
- **Structure**: Includes the number of wires, sources, loads, transmission lines, and segment counts.
- **Excitation**: Indicates whether **discrete sources** or **an incident field** are set.
- **Frequencies**: Displays the **frequency range configured for the simulation**.
- **Medium**: Lists the **permittivity, permeability, and wavelength** (of the upper medium if a ground plane is present).
- **Ground Plane**: Specifies the **type of ground plane** and its associated parameters, if applicable.
- **Far-Field Parameters**: Displays **settings related to far-field calculations**.
- **Near-Field Parameters**: Shows **settings for near-field calculations**.
- **Numerical Accuracy**: Provides details about **the settings employed for numerical accuracy**.
- **Units**: Lists **the units used** for frequency, length, wire radius, inductance, and capacitance.

Fig. 1: The Project Details window showing a summary of project information.

File Formats

When a project is saved in AN-SOF, multiple files that share the same name as the project are saved within the same directory. Each file has a unique extension that corresponds to its specific content.

IMPORTANT: When requesting support, please compress all the project files into a ZIP archive and attach it to your support request email.

These files include:

File type	Description
***.emm**	Main file with configuration data
***.wre**	Geometric description of the wire structure
***.cur**	Current distribution
***.phi**	E-phi component of the far-field.
***.the**	E-theta component of the far-field.
***.pwr**	Radiation pattern data
***.nef**	Near electric field
***.nhf**	Near magnetic field
***.ngf**	Numerical Green's function
***.txt**	Notes written by the user

Shortcut Keys

Pressing ALT with the underlined letter of a menu item will execute the command associated with the item.

The following keys and associated actions are available:

Key	Action
Home	Return the structure to the initial view
ESC	Deselect all wires
F1	Rotate view around +X axis
F2	Rotate view around -X axis
F3	Rotate view around +Y axis
F4	Rotate view around -Y axis
F5	Rotate view around +Z axis
F6	Rotate view around -Z axis
F7	Show/hide main and small axes
F8	Select a wire in order of creation
F9	Select a wire in reverse order of creation
F10	Run current distribution and far-field
F11	Run current distribution and near-field
F12	Run full simulation (all options)
Ctrl + A	Open Axes dialog box
Ctrl + I	Zoom in
Ctrl + K	Zoom out
Ctrl + L	Open Transmission Lines dialog
Ctrl + M	Modify selected wire(s)
Ctrl + N	Create a new project
Ctrl + O	Open an existing project
Ctrl + Q	Exit AN-SOF
Ctrl + R	Run current distribution only
Ctrl + S	Save the project
Ctrl + T	Open tabular input for linear wires

Key	Action
Ctrl + W	Show properties of selected wire
Ctrl + Y	Redo
Ctrl + Z	Undo
Ctrl + Del	Delete selected wire(s)
Ctrl + Ins	Show Source/Load/TL toolbar
Ctrl + Alt + M	Move selected wire(s)
Ctrl + Alt + R	Rotate selected wire(s)
Ctrl + Alt + S	Scale selected wire(s)

Drawing Wires

Types of Wires

AN-SOF provides several wire types for creating antenna geometries. Each wire type has its own *geometric parameters*, *attributes*, and *material* settings, which can be configured through a dedicated **Draw** dialog box. This dialog box allows you to insert and customize new wires directly in the workspace.

Selecting **Draw** from the main menu displays the following wire types:

- **Line** – Opens the dialog box for drawing a straight (linear) wire.
- **Arc** – Opens the dialog box for drawing a circular arc.
- **Circle** – Opens the dialog box for drawing a circular loop.
- **Helix** – Opens the dialog box for drawing a helical wire.
- **Quadratic** – Opens the dialog box for drawing a wire following a quadratic curve.
- **Archimedean Spiral** – Opens the dialog box for drawing an Archimedean spiral.
- **Logarithmic Spiral** – Opens the dialog box for drawing a logarithmic spiral.

Accessing Drawing Commands

The wire drawing tools can be accessed through any of the following:

- **Main Menu** → Draw
- **Right-click Menu** – Right-click on the workspace to open the context menu
- **Main Menu** → View → Drawing Panel (see **Fig. 1**)

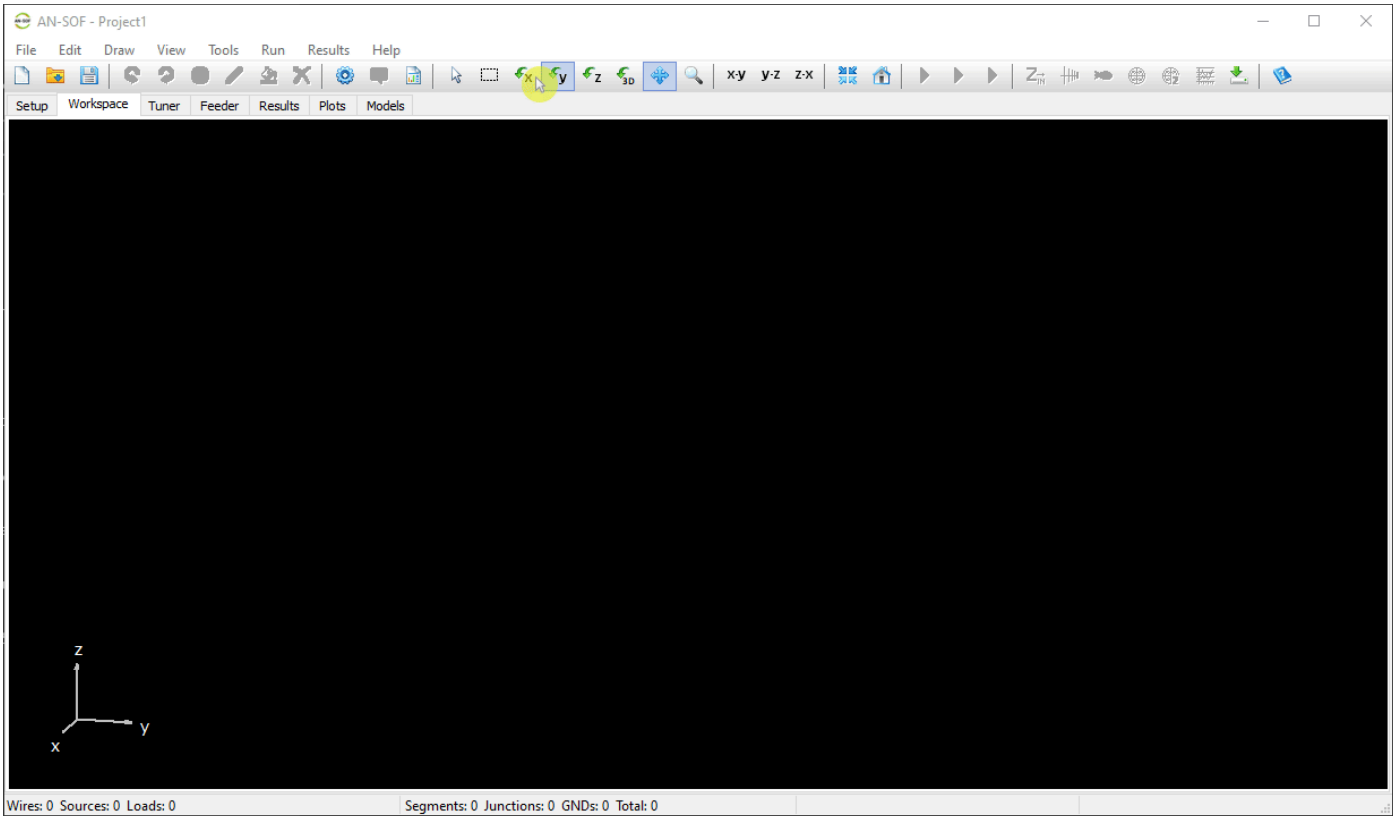


Fig. 1: Options for accessing different wire types for drawing in the workspace.

Wire Attributes

The **Attributes** tab is a core component of the **Draw** dialog box (see **Fig. 1**). It defines the physical and computational properties of each wire in your model, ensuring the simulation is both accurate and mathematically stable.

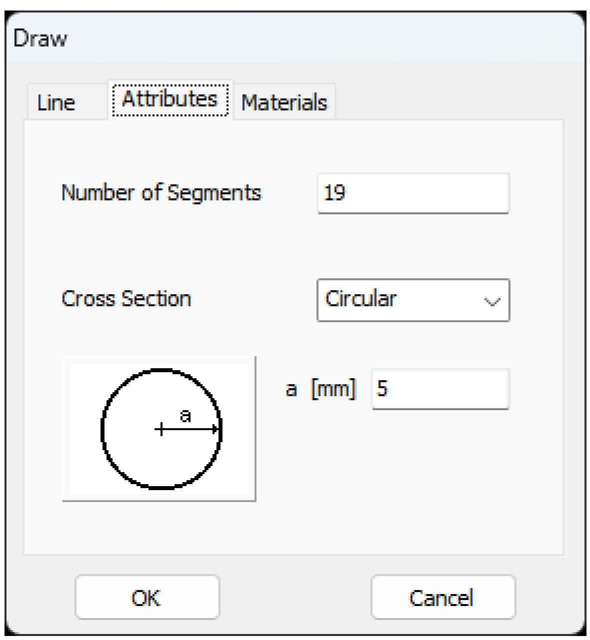


Fig. 1: *Attributes tab in the Draw dialog box for the Line.*

Number of Segments

To solve for the electromagnetic behavior, AN-SOF divides every wire into smaller pieces called **segments**. The software calculates the unknown current on each segment using the **Method of Moments (MoM)**.

- **Default Calculation:** A default number is suggested based on the wire's physical length relative to the shortest wavelength (λ) in your **frequency range**.
- **Automatic Optimization:** If you set the **Number of Segments to zero**, AN-SOF will automatically apply the minimum recommended density.
- **Standard Rule:** The automatic calculation generally targets **10 segments per wavelength** (at the highest frequency) to ensure convergence and precision.

Cross-Section

Because infinitesimally thin wires do not exist in the physical world and would cause mathematical singularities, every wire must have a defined thickness.

- **Available Shapes:** Circular, Square, Flat, Elliptical, Rectangular, and Triangular.
- **Equivalent Radius:** For all non-circular shapes, AN-SOF calculates an **equivalent radius**. This allows the solver to treat the wire as a cylinder while maintaining the electromagnetic characteristics of the original shape (such as a flat PCB trace or a square structural member).
- **Constraint:** The cross-section radius or width must always be **greater than zero**.

Universal Application

The **Attributes** tab is consistent across all wire creation tools. Whether you are drawing a simple **Line**, an **Arc**, or a complex **Helix**, you will find these same features available to define your geometry.
Wire Materials

In AN-SOF, the physical properties of the wire, such as the metal it is made of and whether it has insulation, are defined in the **Materials** tab of the **Draw** dialog box (**Fig. 1**). This allows for realistic simulations that account for ohmic losses and dielectric loading.

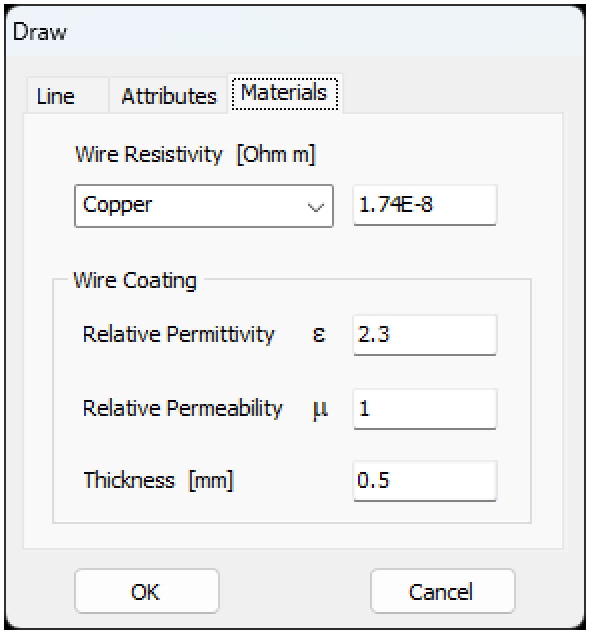


Fig. 1: *Materials tab in the Draw dialog box for the Line.*

Wire Resistivity (Metallic Losses)

You can specify the resistivity (ϱ) in $\Omega{\cdot}m$ to account for real-world conductor losses. AN-SOF uses this value to compute a distributed impedance per unit length, which includes the **skin effect**, where current tends to flow on the outer surface of the conductor at high frequencies.

Common Metals in AN-SOF:

Material (Metals)	Resistivity [Ω m]
Aluminum (Pure)	2.65E-8
Aluminum (6061-T6)	4.01E-8
Aluminum (6063-T832)	3.25E-8
Brass	6.41E-8
Carbon Steel	1.67E-7
Constantan	4.42E-7
Copper	1.74E-8
German Silver	3.33E-7
Germanium	4.55E-7
Gold	2.44E-8
Iron	9.71E-8
Manganin	4.41E-7
Nichrome	1.00E-6
Nickel	6.90E-8
Phosphor Bronze	1.10E-7
Silver	1.59E-8
Solder	1.43E-7
Stainless Steel	9.09E-7
Stainless Steel 302	7.19E-7
Tin	1.14E-7
Tungsten	5.49E-8
Zinc	5.90E-8

- **Custom:** Manually enter a value for materials not listed.
- **Perfect (PEC):** Sets the wire as a Perfect Electrical Conductor (zero loss).

> IMPORTANT: To include these losses in your simulation, ensure the **Wire Resistivity** option is checked in the **Settings** panel of the **Setup** tab.

Wire Coating (Insulation)

If your antenna uses insulated wires, you can define the **dielectric coating**. AN-SOF assumes a **circular cross-section** for the coating.

Coating Parameters:

- **Relative Permittivity** (εr): The dielectric constant of the insulation (e.g., ~2.3 for polyethylene).
- **Relative Permeability** (μr): The magnetic constant of the coating (usually 1.0 for non-magnetic insulators).
- **Thickness:** The physical depth of the coating layer. Set to **0** if the wire is bare.

> IMPORTANT: To account for the velocity factor and loading effects of the insulation, ensure the **Wire Coating** option is checked in the **Settings** panel of the **Setup** tab.

Geometry Note

For wires with non-circular cross-sections (like flat strips), AN-SOF utilizes an **equivalent radius** calculation to determine the metallic impedance behavior.
Enabling/Disabling Resistivity

In AN-SOF, you can globally control whether the software accounts for metallic losses or treats the entire structure as a **Perfect Electric Conductor (PEC)**. This is a powerful feature for benchmarking, as it allows you to quickly compare the idealized performance of your antenna against its real-world performance with ohmic losses.

How to Toggle Resistivity

Instead of manually editing every wire in your model, use the global toggle:

1. Navigate to the **Setup** tab in the main AN-SOF window.
2. Locate the **Settings** panel (**Fig. 1**).
3. **To Enable Losses:** Check the **Wire Resistivity** box. AN-SOF will now use the specific resistivity values (e.g., for copper or aluminum) you assigned to each wire in the Draw dialog.
4. **To Disable Losses:** Uncheck the **Wire Resistivity** box. The software will ignore all individual wire resistivity settings and treat every conductor as having zero resistance (PEC).

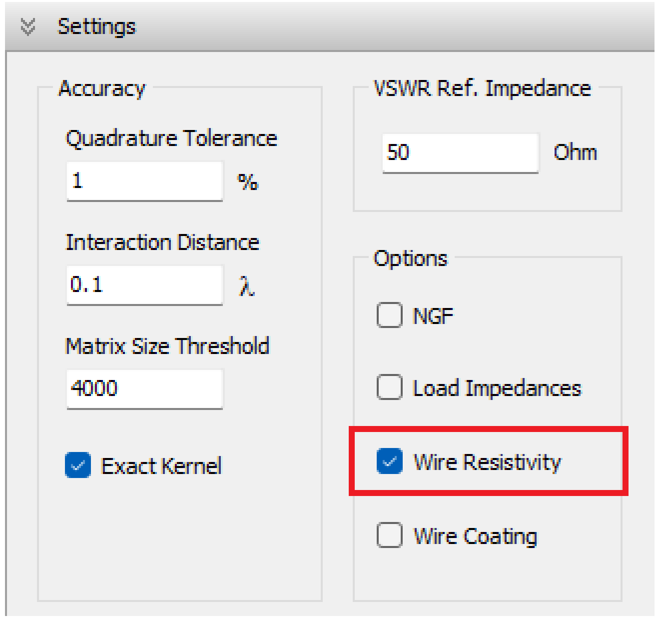


Fig. 1: *Wire Resistivity option in the Settings panel of the Setup tab.*

Why Use This Toggle?

- **Efficiency Analysis:** Running a simulation twice, once with resistivity enabled and once without, provides a clearer understanding of how ohmic losses affect the antenna's **radiation efficiency**.
- **Troubleshooting:** If your antenna's gain is lower than expected, disabling resistivity helps you determine if the issue is due to high ohmic losses or poor impedance matching.
- **Fast Prototyping:** Use the PEC (disabled) setting during the initial design phase to speed up optimization before adding the complexity of skin-effect losses.

Enabling/Disabling Coating

Similar to the **resistivity toggle**, AN-SOF provides a global master switch for wire insulation. This allows you to quickly assess the **velocity factor** and **dielectric loading** effects of your coating without having to manually remove the insulation from every wire in your model.

How to Toggle Coating

To switch between bare wires and insulated wires:

1. Navigate to the **Setup** tab in the main AN-SOF window.
2. Locate the **Settings** panel (**Fig. 1**).
3. **To Enable Insulation:** Check the **Wire Coating** box. AN-SOF will apply the permittivity, permeability, and thickness values you assigned in the **Materials** tab of the Draw dialog.
4. **To Disable Insulation:** Uncheck the **Wire Coating** box. The software will treat all wires as bare conductors, effectively setting the coating thickness to zero for the entire simulation.

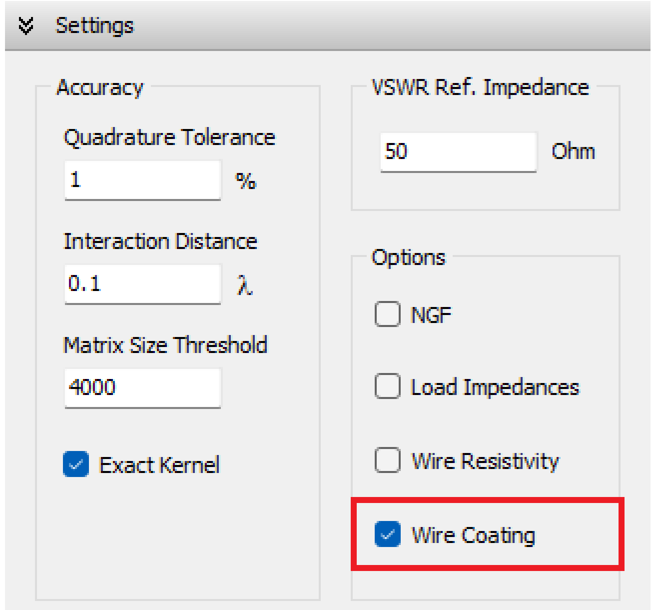


Fig. 1: *Wire Coating option in the Settings panel of the Setup tab.*

Technical Impact

> **Resonant Frequency:** Insulation typically increases the capacitance of the wire, lowering the resonant frequency. By toggling this setting, you can see exactly how much the insulation "electrically lengthens" your antenna.

Cross-Section Equivalent Radius

While AN-SOF is based on wire modeling, it allows you to simulate structures with various physical cross-sections. This is achieved by calculating an **equivalent radius**: the radius of a circular wire that would produce the same average electromagnetic fields as the actual shape.

You can select the cross-section type from the **Attributes** page of the **Draw** dialog box (**Fig. 1**).

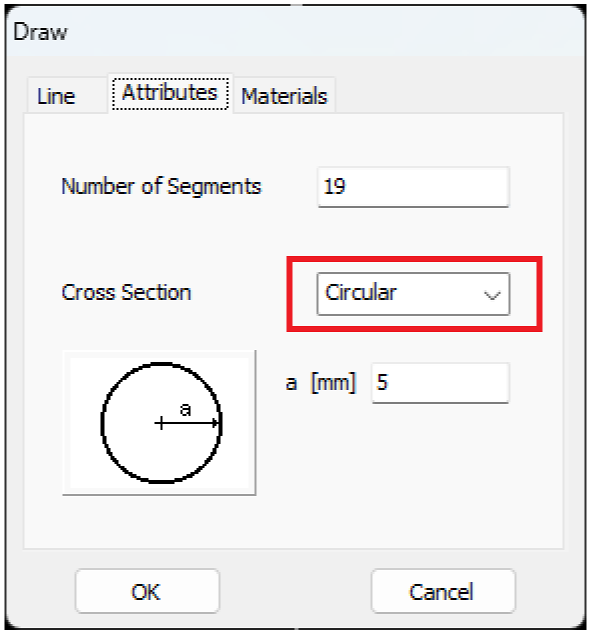
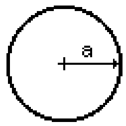
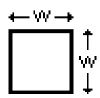
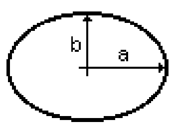
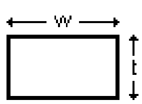


Fig. 1: *Cross-Section dropdown in the Attributes tab of the Draw dialog box. A circular cross-section of radius “a” is selected.*

Cross-Section Types and Formulas

AN-SOF provides six geometry options, automatically converting them to an equivalent radius aeq for the simulation engine:

Cross-Section	Shape	Parameters	Equivalent Radius
Circular		Radius a	a
Square		Width w	$0.59017w$
Flat		Width w	$w/4$
Elliptical		Semi-axes a,b	$(a+b)/2$
Rectangular		Width w, Thickness t	[Numerical solution to an integral equation]()
Triangular		Width w (equilateral)	$0.42w$

Key Technical Insights

- **The Flat Strip Approximation:** The equivalent radius for a flat strip ($w/4$) is a classic result in antenna theory, often used to model PCB traces or metal ribbons.
- **Rectangular Precision:** For rectangular cross-sections, AN-SOF doesn't use a simple linear factor. Instead, it employs a sophisticated **polynomial and logarithmic approximation** to solve an integral equation, ensuring high accuracy even for high aspect ratios.
- **Physical Meaning:** The equivalent radius ensures that the **capacitance per unit length** and the **inductance per unit length** of the modeled wire accurately represent the chosen physical shape.

Selecting the Attribute

To change the wire cross-section type for an existing wire, follow these steps:

1. **Select the Wire:** Right-click on the wire in the workspace and choose **Modify** from the pop-up menu (**Fig. 2**).
2. **Navigate to Attributes:** In the Draw dialog box that appears, click on the **Attributes** tab.
3. **Choose Cross-Section:** Locate the **Cross-Section** dropdown menu and select the desired shape.
4. **Enter Dimensions:** Provide the required dimensions for that specific shape to allow AN-SOF to calculate the equivalent radius.

To modify the cross-section of wires in bulk, follow the instructions in this guide to select and modify the wires:

[**Selecting and Modifying Multiple Wires**]

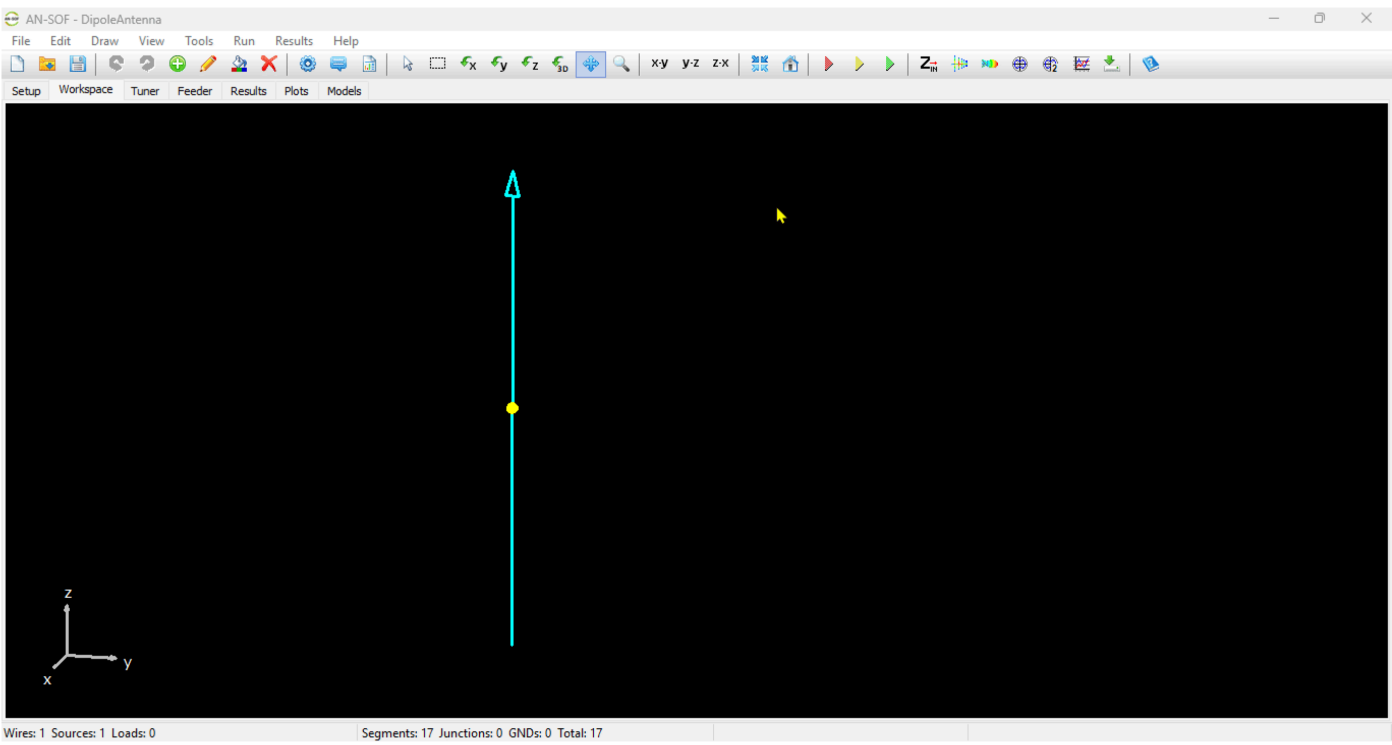


Fig. 2: *Modifying the cross-section of a selected wire.*

Exporting Wires

AN-SOF provides several ways to export your wire geometry and simulation settings for use in other software, ranging from legacy solvers to CAD tools and programming environments.

1. NEC Format (.nec)

Exporting to the **Numerical Electromagnetics Code (NEC)** format is ideal for benchmarking or using alternative solvers. To do this, navigate to **File > Export Wires** and select the **.nec** file type.

The software automatically translates your model into standard NEC “cards”:

- **GW Lines:** All linear wires are converted into geometry cards.
- **Physics & Setup:** The export includes cards for ground conditions (**GN/GE**), transmission lines (**TL**), loads and conductivity (**LD**), and insulation (**IS**).
- **Simulation Parameters:** Frequency (**FR**), excitation (**EX**), the “Exact Kernel” toggle (**EK**), and radiation pattern requests (**RP**) are also preserved.

2. CAD Exchange (DXF)

If you need to incorporate your antenna design into a mechanical drawing or a professional CAD environment (like AutoCAD or SolidWorks):

- Navigate to **File > Export Wires** and choose **.dxf**.
- **Note:** While the 3D paths and positions of the wires are maintained, the specific **cross-section attributes** (like the equivalent radius or shape) are lost in this format, as DXF treats the segments as basic lines.

3. Programming Scripts (.m / .sce)

For users who prefer automation and parametric modeling, you can export your project as a script for **MATLAB/Octave (.m)** or **Scilab (.sce)**.

- **Parametric Control:** The exported file contains code that defines the wire lengths, positions, and frequencies.
- **Customization:** You can modify variables within the script to create loops, allowing you to sweep geometry parameters or generate complex, repeatable project configurations automatically.

Adding Wires

Line

A **Line** represents a linear (straight) wire in AN-SOF.

Accessing the Line Dialog Box

To open the **Line** dialog box:

1. Navigate to **Draw > Line** in the main menu.
2. The dialog box contains three tabs: **Line**, **Attributes**, and **Materials** (**Fig. 1**).

Line Tab: Setting Geometrical Parameters

Two options are available for defining the line:

1. **2 Points** (**Figs. 1 and 2**)

    Define the line by specifying two points:

    - **From Point**: Starting coordinates (X1, Y1, Z1).
    - **To Point**: Ending coordinates (X2, Y2, Z2).

2. **Start – Direction – Length** (**Figs. 3 and 4**)

    Define the line by:

    - **Start Point**: Initial coordinates (X1, Y1, Z1).
    - **Direction**: Spherical angles (**Theta**, **Phi**).
    - **Wire Length**: Total length of the line.

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

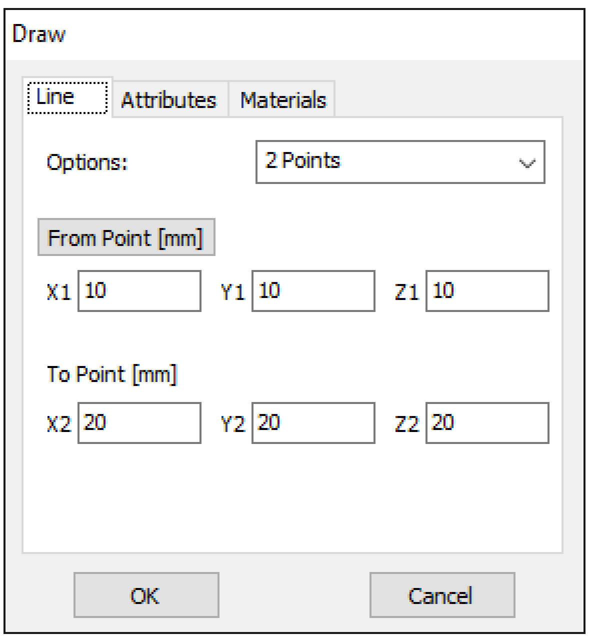


Fig. 1: *Line dialog box – "2 Points" option selected in the Line tab.*

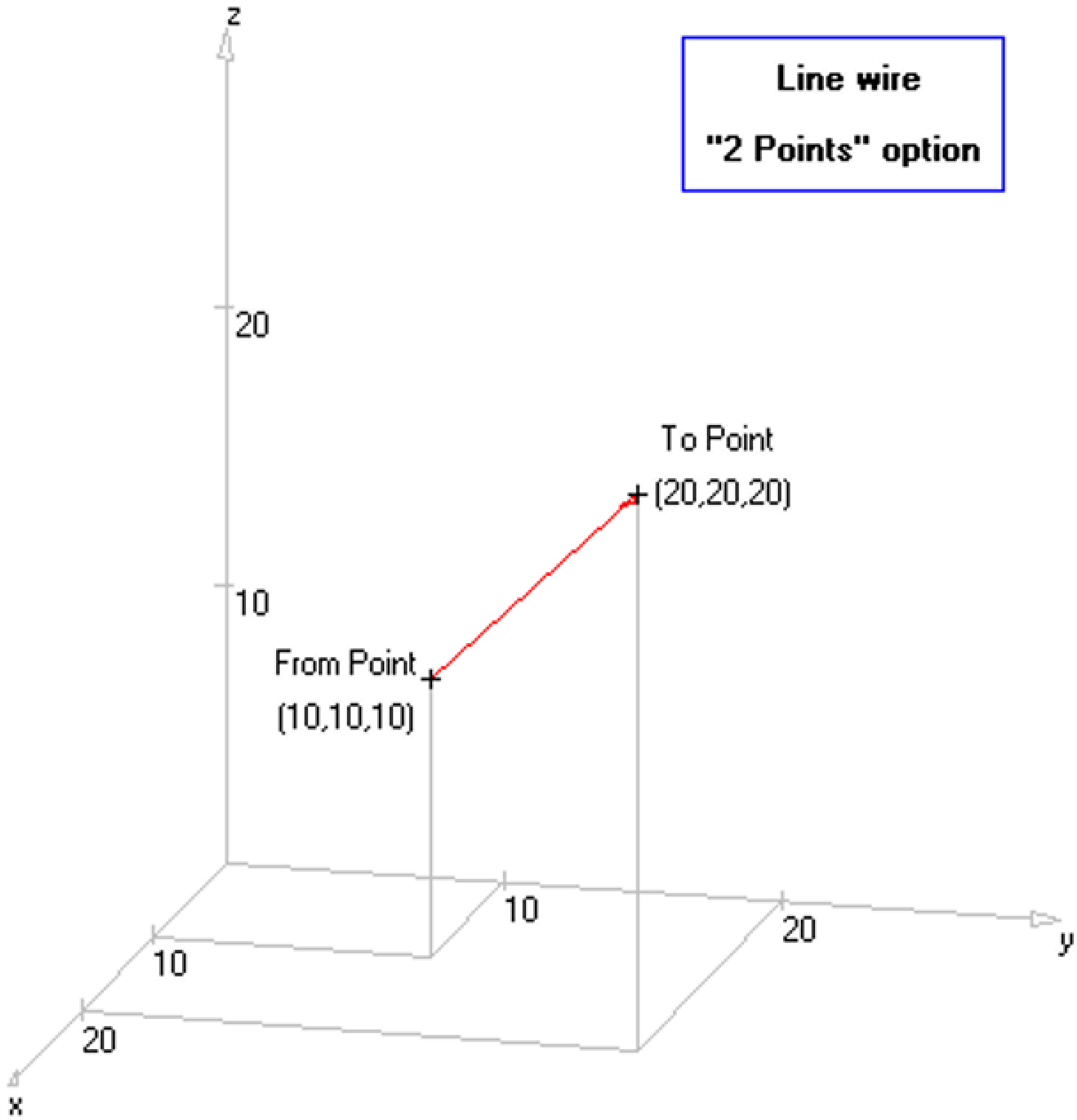


__Fig. 2:__ Line created using the 2 Points option (parameters from Fig. 1).

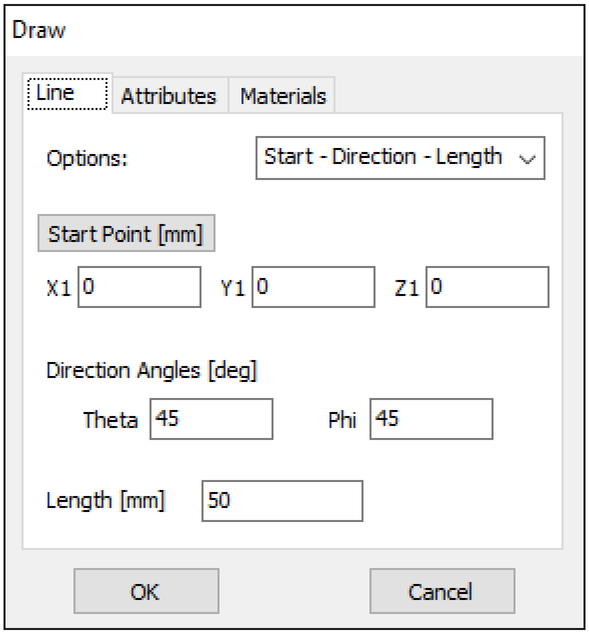


Fig. 3: *Draw Line dialog box – Start-Direction-Length option in Line tab.*

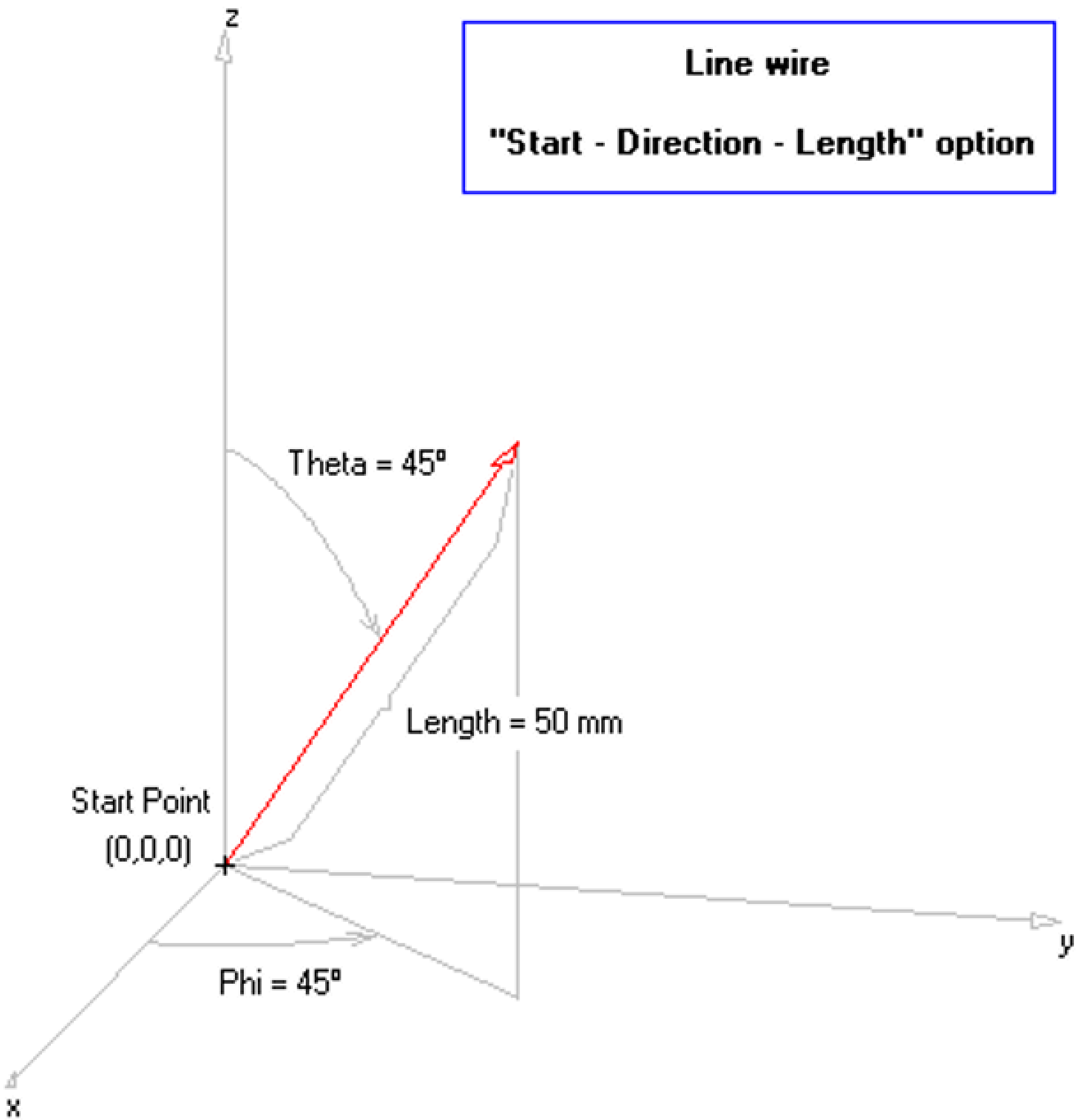


Fig. 4: *Line created using Start-Direction-Length option (parameters from Fig. 3).*

Arc

An **Arc** represents a circular arc in AN-SOF.

Accessing the Arc Dialog Box

To open the **Arc** dialog box:

1. Navigate to **Draw > Arc** in the main menu.
2. The dialog box contains three tabs: **Arc**, **Attributes**, and **Materials** (**Fig. 1**).

Arc Tab: Setting Geometrical Parameters

Two options are available for defining the arc:

1. **3 Points** (**Figs. 1 and 2**)
   Define the arc by specifying three points:
   - **Start Point**: Starting coordinates (X1, Y1, Z1).
   - **Second Point**: Second point coordinates (X2, Y2, Z2).
   - **End Point**: Ending coordinates (X3, Y3, Z3).
2. **Start – Center – End** (**Figs. 3 and 4**)
   Define the arc by:
   - **Start Point**: Initial coordinates (X1, Y1, Z1).
   - **Center**: Coordinates of the arc center (Cx, Cy, Cz).
   - **End Point**: Ending coordinates (X2, Y2, Z2), which may not coincide with the actual ending point of the arc.

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

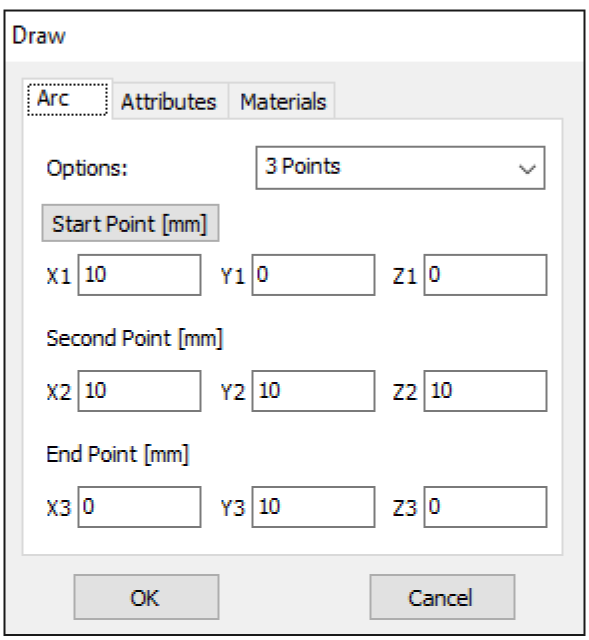


Fig. 1: *Arc dialog box – "3 Points" option selected in the Arc tab.*

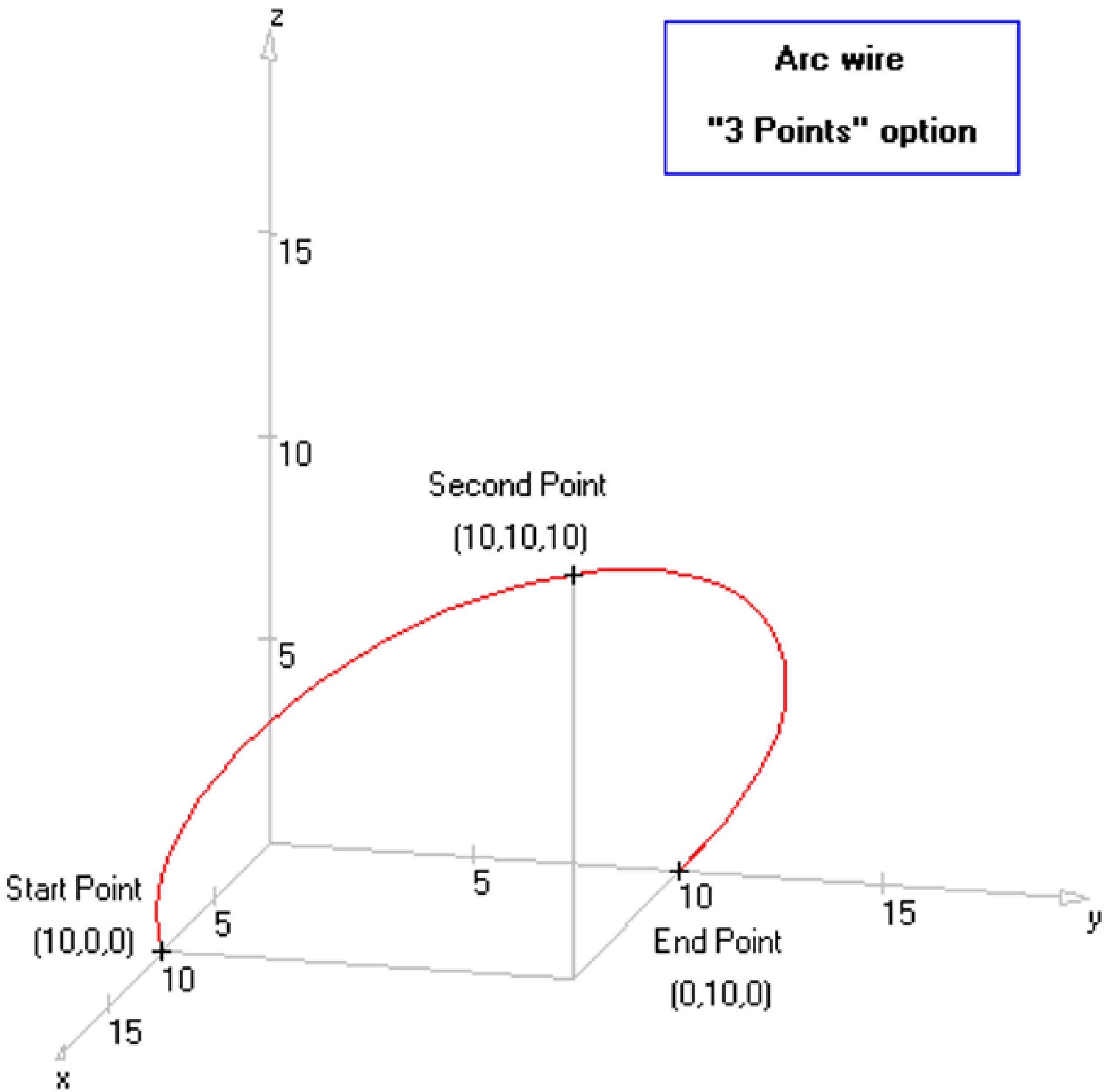


Fig. 2: *Arc created using the 3 Points option (parameters from Fig. 1).*

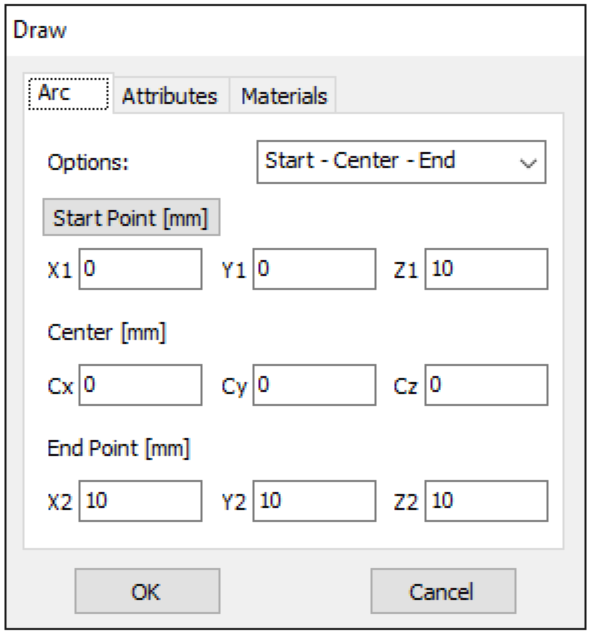


Fig. 3: *Draw Arc dialog box – Start-Center-End option in Arc tab.*

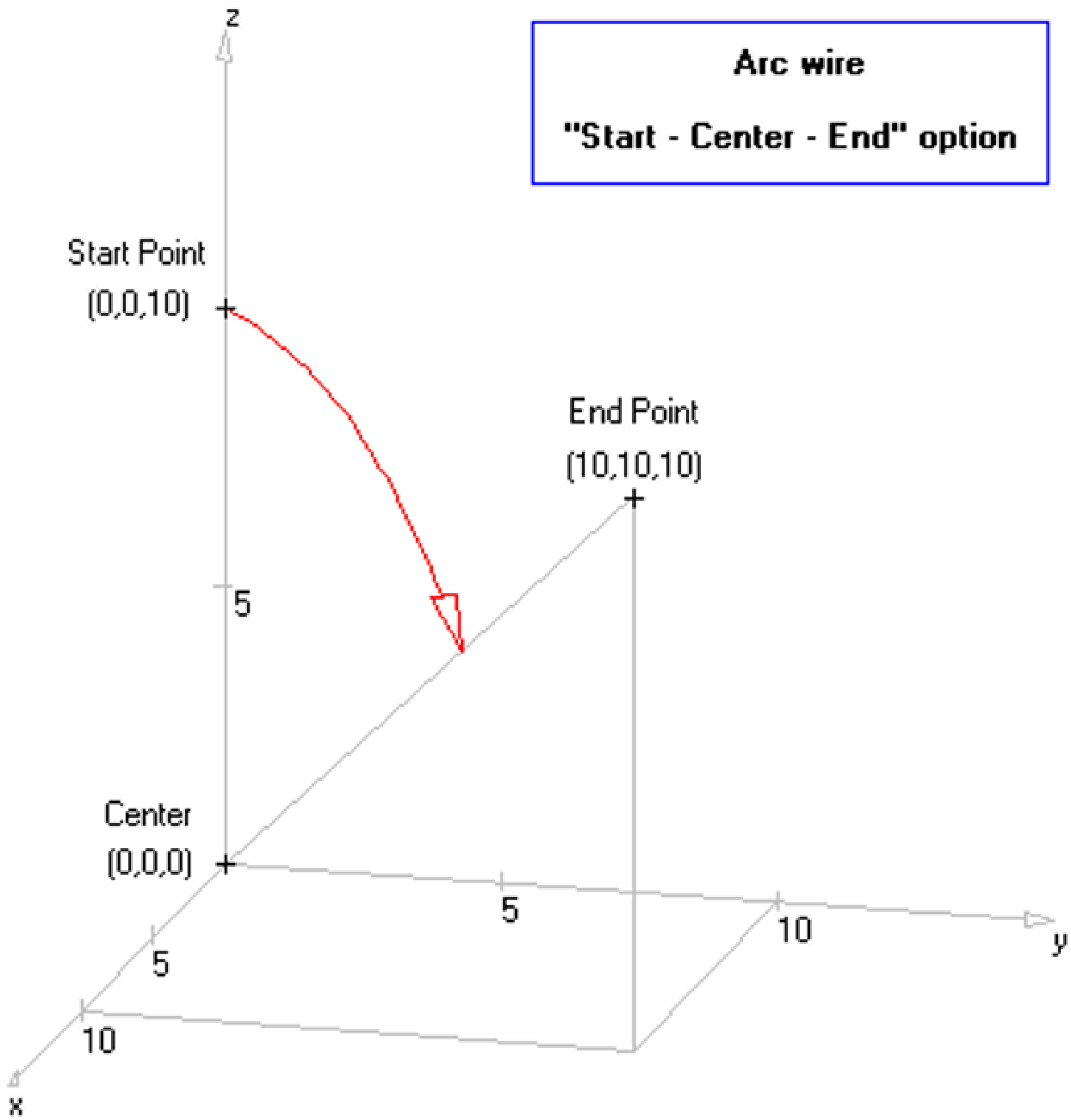


Fig. 4: *Arc created using Start-Center-End option (parameters from Fig. 3).*

Circle

A **Circle** represents a circular loop in AN-SOF.

Accessing the Circle Dialog Box

To open the **Circle** dialog box:

1. Navigate to **Draw > Circle** in the main menu.
2. The dialog box contains four tabs: **Circle**, **Orientation**, **Attributes**, and **Materials** (**Fig. 1**).

Circle Tab: Setting Geometrical Parameters

Two options are available for defining the circle:

1. **Center – Radius – Orientation** (**Figs. 1, 2, and 3**)
   Define the circle by specifying:
   - **Center**: Coordinates of the circle center (Cx, Cy, Cz).
   - **Radius**: Circle radius.
   - **Orientation Tab Enabled**: Direction of the circle axis, set by the spherical angles (Theta, Phi) or a normal vector (Nx, Ny, Nz). The "rotation angle" rotates the circle around its axis.
2. **3 Points** (**Figs. 4 and 5**)
   Define the circle by:
   - **First Point**: Initial coordinates (X1, Y1, Z1).
   - **Second Point**: Second point coordinates (X2, Y2, Z2).
   - **Third Point**: Third point coordinates (X3, Y3, Z3).

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

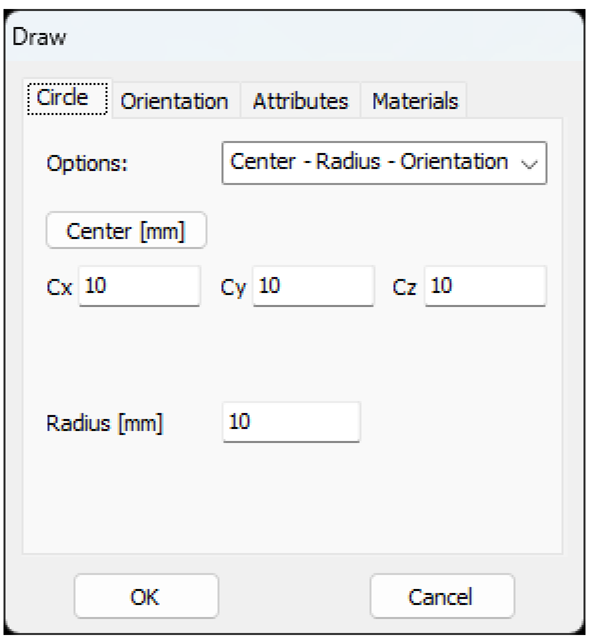


Fig. 1: *Circle dialog box – "Center – Radius – Orientation" option selected in the Circle tab.*

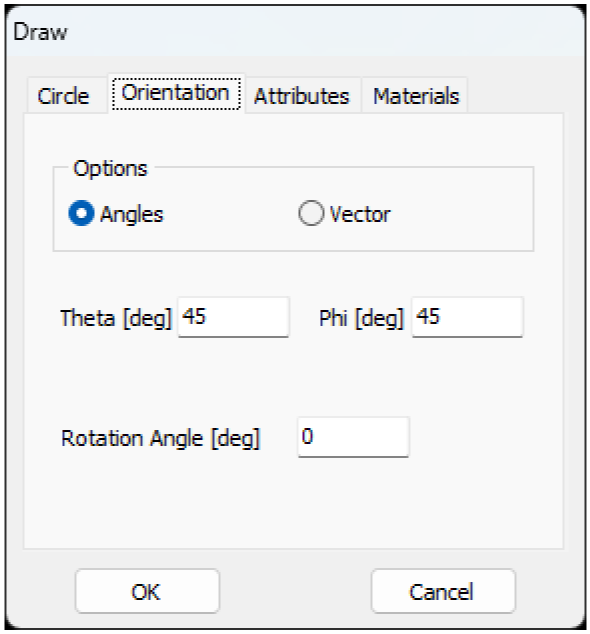


Fig. 2: *Orientation tab where the circle axis direction can be set.*

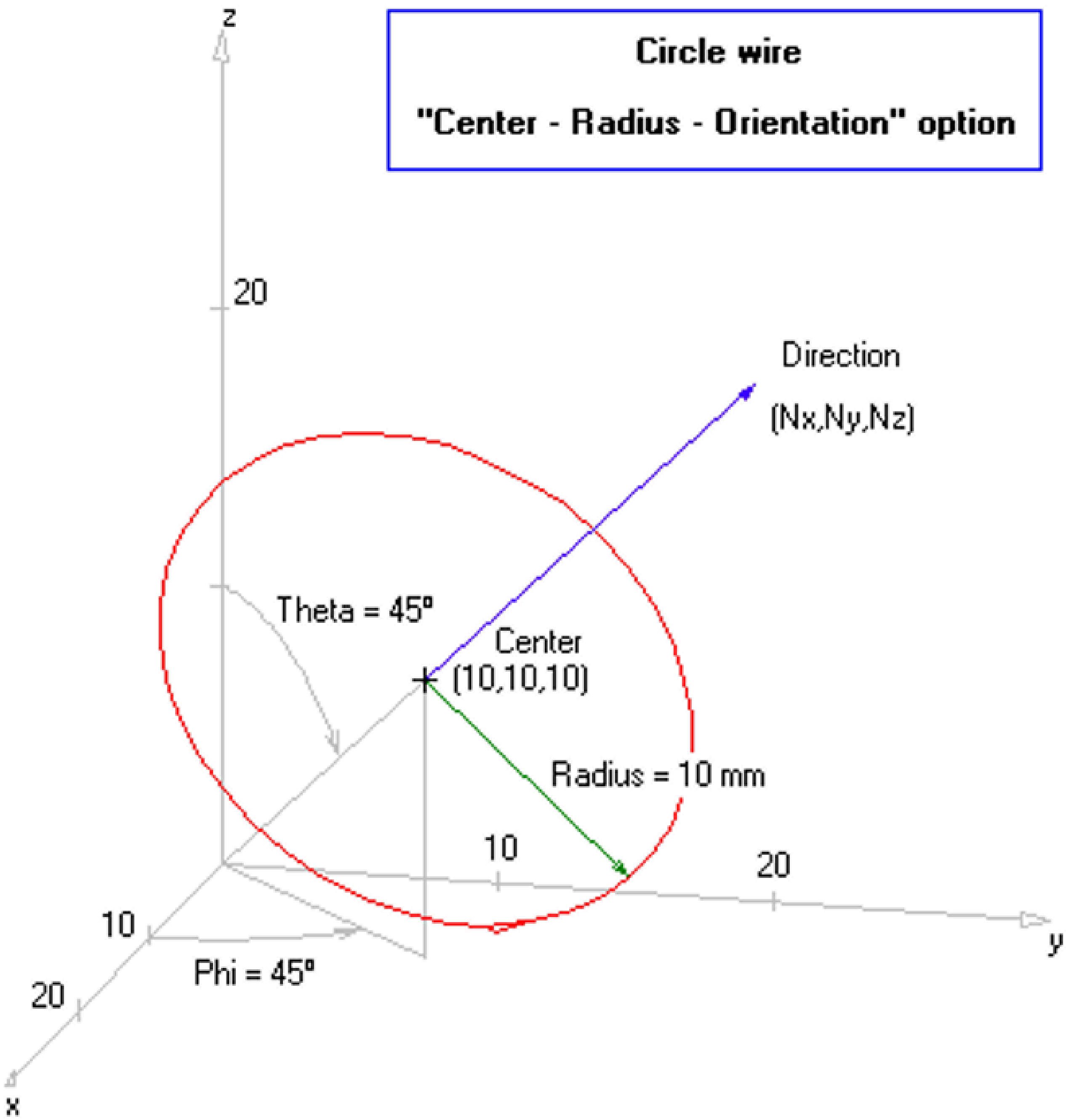


Fig. 3: *Circle created using the "Center – Radius – Orientation" option (parameters from Figs. 1 and 2).*

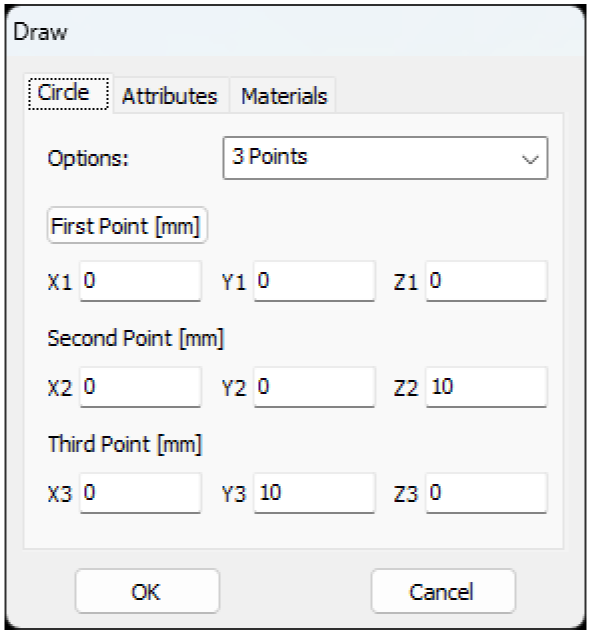


Fig. 4: *Draw Circle dialog box – "3 Points" option in Circle tab.*

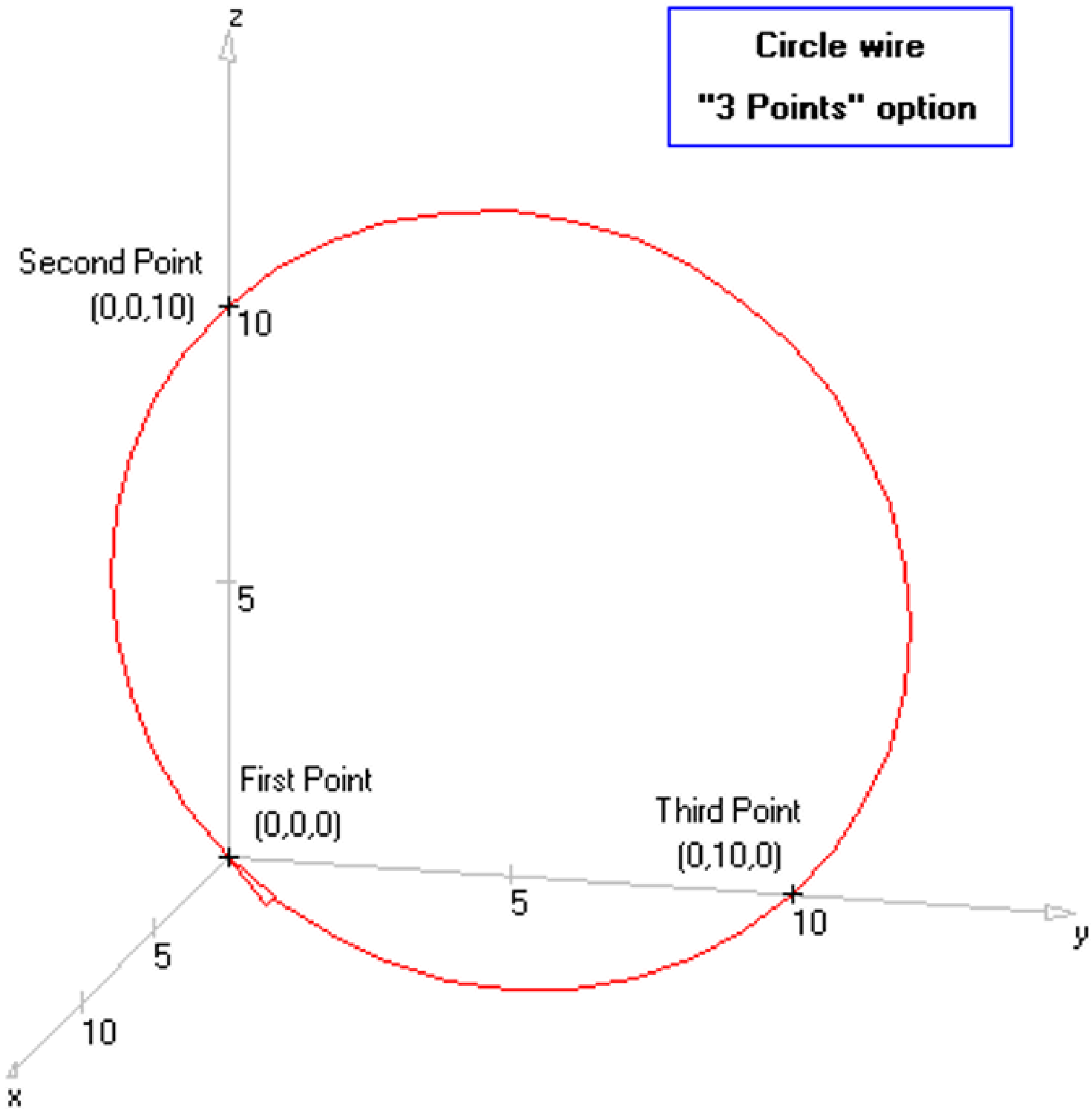


***Fig. 5:** Circle created using "3 Points" option (parameters from Fig. 4).*

Helix

A **Helix** represents a circular helix in AN-SOF.

Accessing the Helix Dialog Box

To open the **Helix** dialog box:

1. Navigate to **Draw > Helix** in the main menu.
2. The dialog box contains four tabs: **Helix**, **Orientation**, **Attributes**, and **Materials** (**Fig. 1**).

Helix Tab: Setting Geometrical Parameters

Two options are available for defining the helix:

1. **Start – Radius – Pitch – Turns** (**Figs. 1, 2, and 3**)
   Define the helix by specifying the following:
   - **Start Point**: Starting coordinates (X1, Y1, Z1).
   - **Radius** or **Diameter**: The helix radius or diameter; entering one will automatically calculate the other.
   - **Pitch** (turn spacing) or **Pitch Angle**: The helix pitch or pitch angle; entering one will automatically calculate the other. A **positive pitch** results in a **right-handed** helix, while a **negative pitch** results in a **left-handed** helix.
   - **Nr. of Turns** or **Length**: The number of turns or the filar length; entering one will automatically calculate the other. The number of turns does not need to be an integer, allowing for fractional turns.
   - **Height**: The axial height from the starting point to the ending point. This value is calculated automatically and cannot be edited.
   - **Orientation Tab Enabled**: Sets the direction of the helix axis using spherical angles (Theta, Phi) or a vector (Nx, Ny, Nz). The “rotation angle” rotates the helix around its axis.
2. **Start – End – Radius – Turns** (**Figs. 4 and 5**)
   Define the helix by specifying the following:
   - **Start Point**: Starting coordinates (X1, Y1, Z1).
   - **End Point**: Ending coordinates (X2, Y2, Z2).
   - **Radius:** The helix radius.
   - **Number of Turns:** This must be an integer. A **positive** value results in a **right-handed** helix, while a **negative** value results in a **left-handed** helix.
   - **Rotation Angle:** Rotates the helix around its axis.
   - **Height:** The axial height from the starting point to the ending point. This value is calculated automatically and cannot be edited.

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

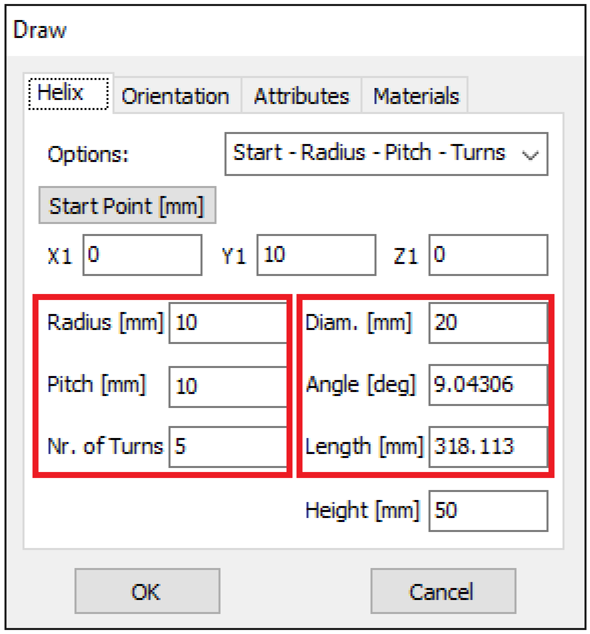


Fig. 1: *Helix dialog box – "Start – Radius – Pitch – Turns" option selected in the Helix tab.*

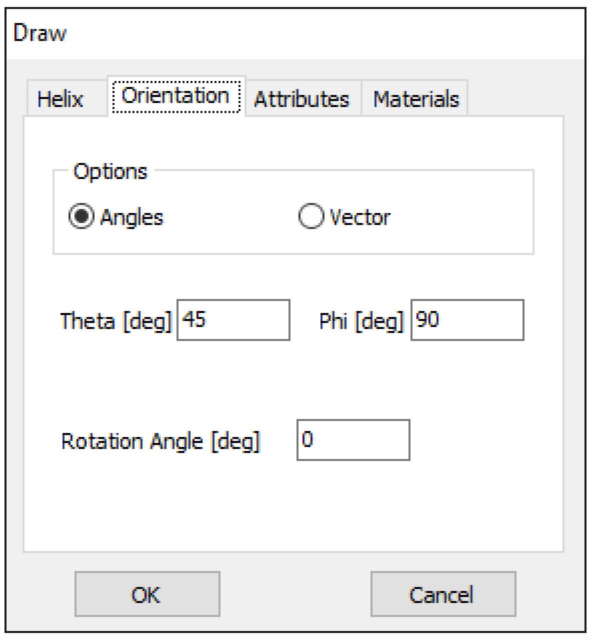


Fig. 2: *Orientation tab where the helix axis direction can be set.*

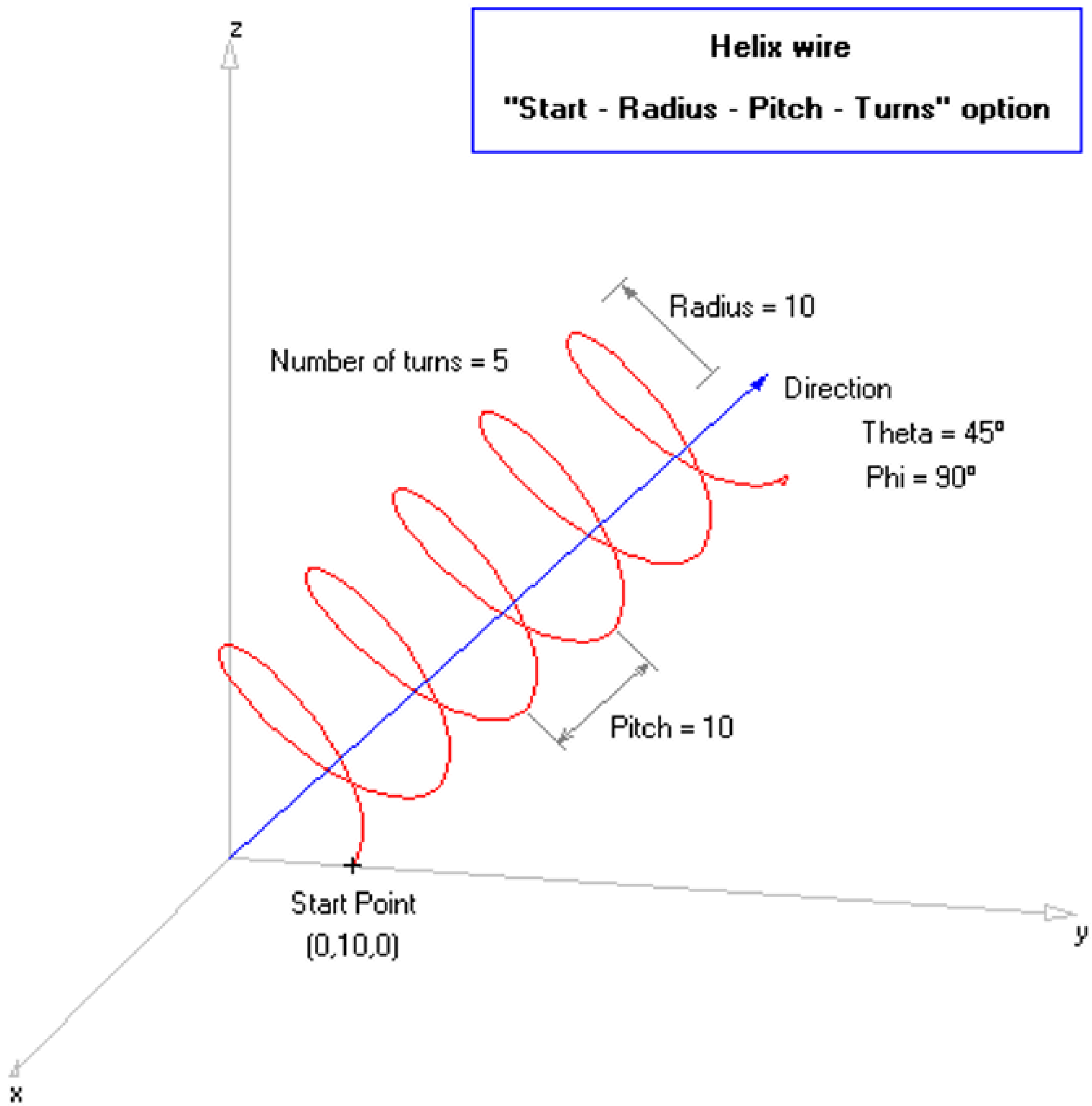


Fig. 3: *Helix created using the "Start – Radius – Pitch – Turns" option (parameters from Figs. 1 and 2).*

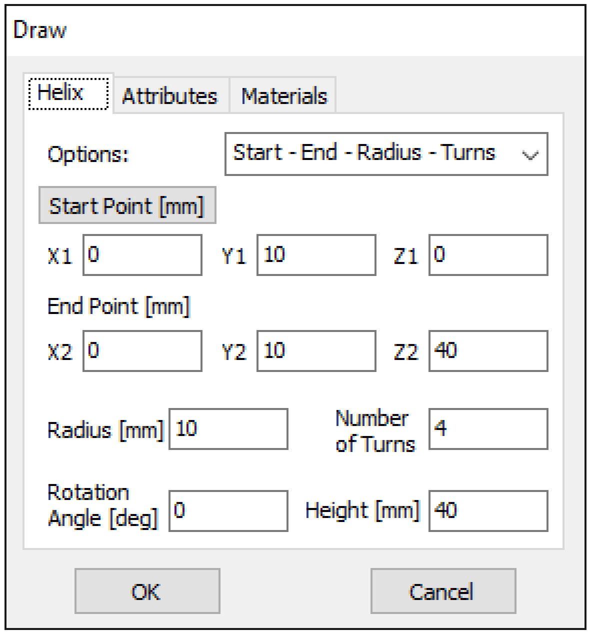


Fig. 4: *Draw Helix dialog box – "Start – End – Radius – Turns" option in Helix tab.*

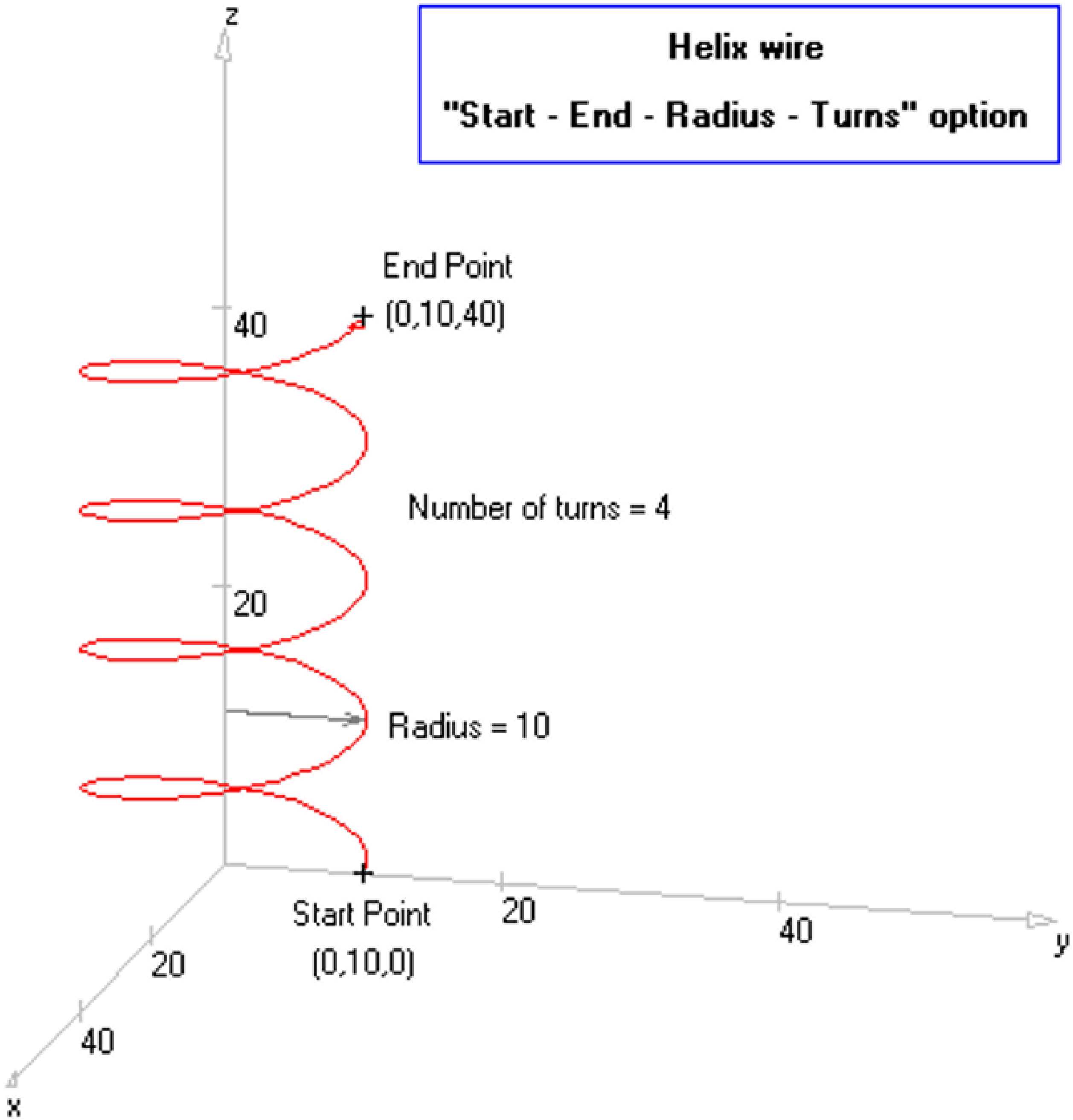


Fig. 5: *Helix created using "Start – End – Radius – Turns" option (parameters from Fig. 4).*

Quadratic

A **Quadratic** represents a parabola in AN-SOF.

Accessing the Quadratic Dialog Box

To open the **Quadratic** dialog box:

1. Navigate to **Draw > Quadratic** in the main menu.
2. The dialog box contains three tabs: **Quadratic**, **Attributes**, and **Materials** (**Fig. 1**).

Quadratic Tab: Setting Geometrical Parameters

Define the quadratic by specifying (**Fig. 1**):

- **Start Point**: Starting coordinates (X1, Y1, Z1).
- **Second Point**: Second point coordinates (X2, Y2, Z2).
- **End Point**: Ending coordinates (X3, Y3, Z3).

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

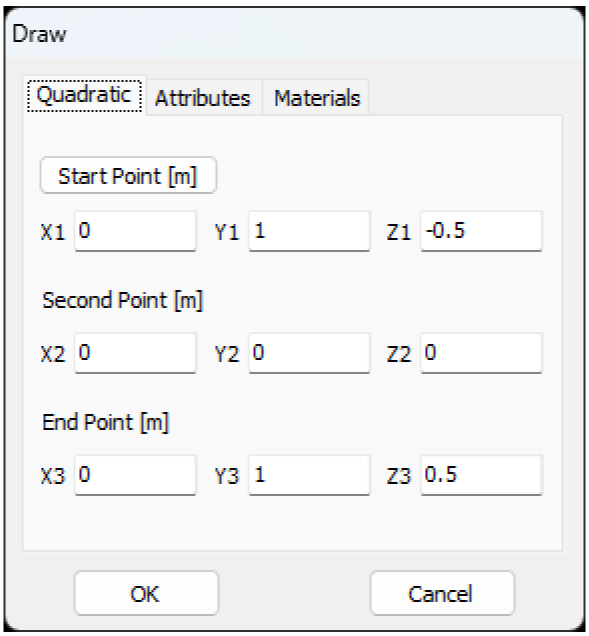


Fig. 1: *Quadratic dialog box.*

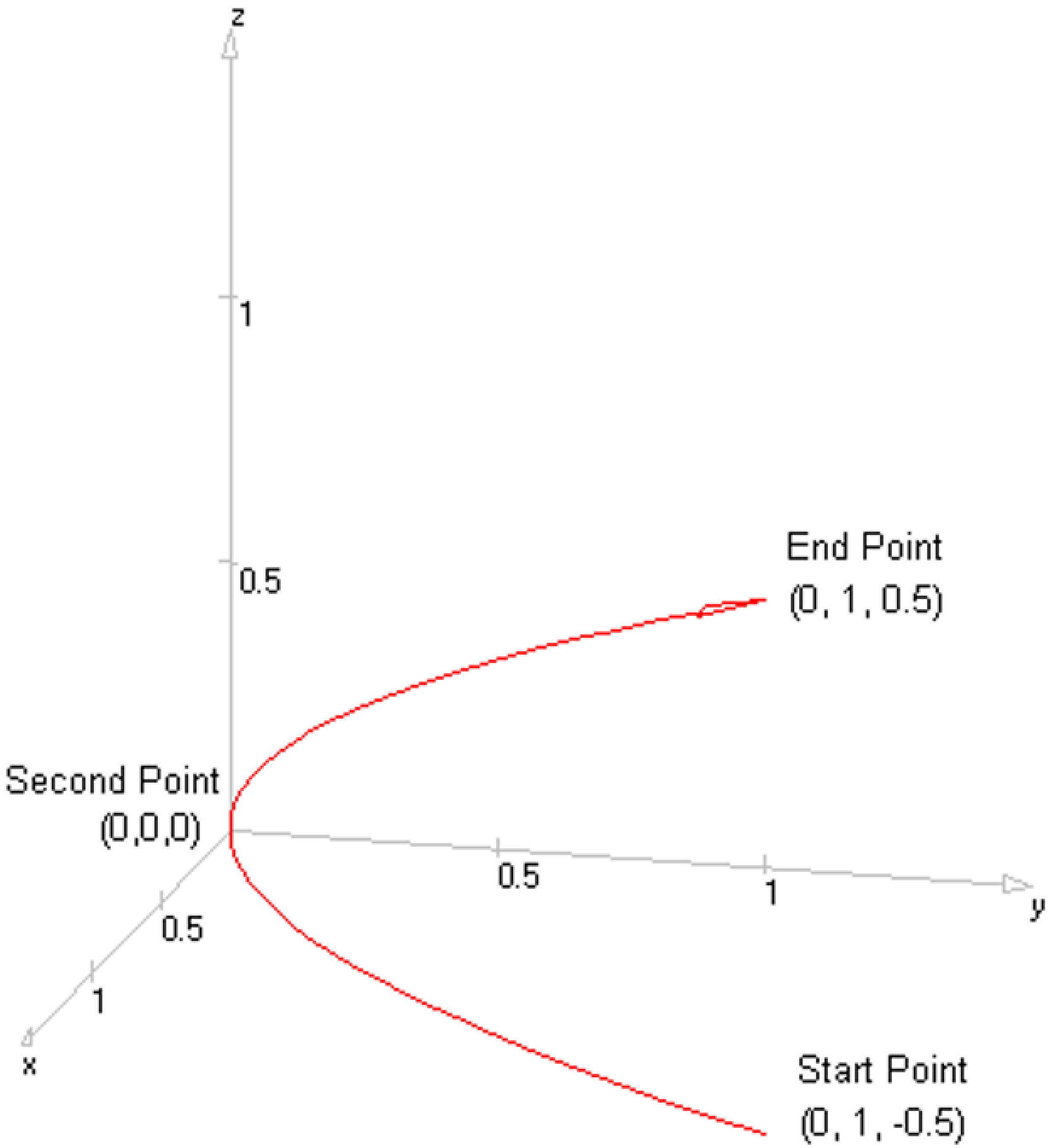


Fig. 2: *Quadratic created using the parameters from Fig. 1.*

Archimedean Spiral

An **Archimedean Spiral** represents an Archimedes' spiral in AN-SOF.

The Archimedean spiral is a planar spiral defined by the polar equation:

$r(\alpha) = r0 + p2\pi\alpha$

where $r0$ is the **starting radius**, p is the **pitch**, and α is the angle in the plane of the spiral, ranging from 0 to $2\pi M$, with M representing the **number of turns**.

The spiral's starting radius is $r(0)=r0$ and its ending radius is $r(2\pi M)=r0+pM$. Consequently, the pitch p represents the constant spacing between consecutive turns. Additionally, the pitch is equal to the constant growth rate of the spiral radius $r(\alpha)$ per turn, expressed as:

$p=2\pi drd\alpha$

Accessing the Archimedean Spiral Dialog Box

To open the **Archimedean Spiral** dialog box:

1. Navigate to **Draw > Archimedean Spiral** in the main menu.
2. The dialog box contains three tabs: **Archimedean Spiral**, **Attributes**, and **Materials** (**Fig. 1**).

Archimedean Spiral Tab: Setting Geometrical Parameters

Define the Archimedean spiral by specifying the following (**Fig. 1**):

- **Start Point**: Starting coordinates (X1, Y1, Z1).
- **Start Radius**: The initial radius of the spiral.
- **Pitch**: The constant spacing between turns (can be positive or negative).
- **Number of Turns**: This value does not need to be an integer, allowing for fractional turns.
- **Orientation Angles**: Sets the direction of the spiral axis using spherical angles (Theta, Phi).
- **Rotation Angle**: Rotates the spiral around its axis.

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

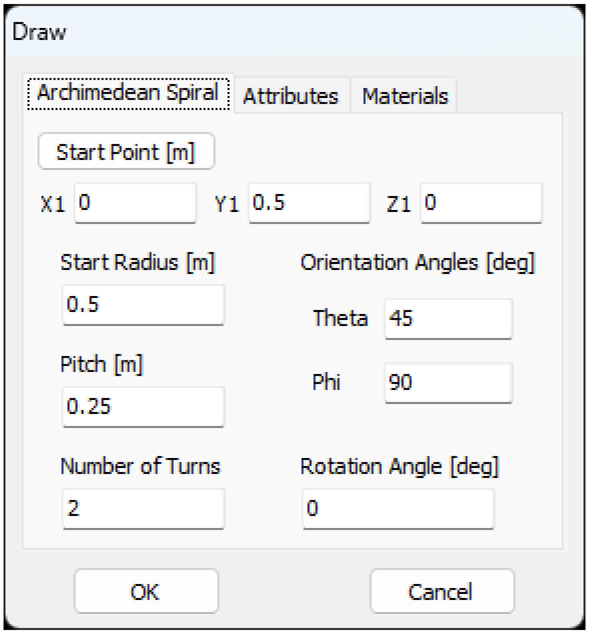


Fig. 1: *Archimedean Spiral dialog box.*

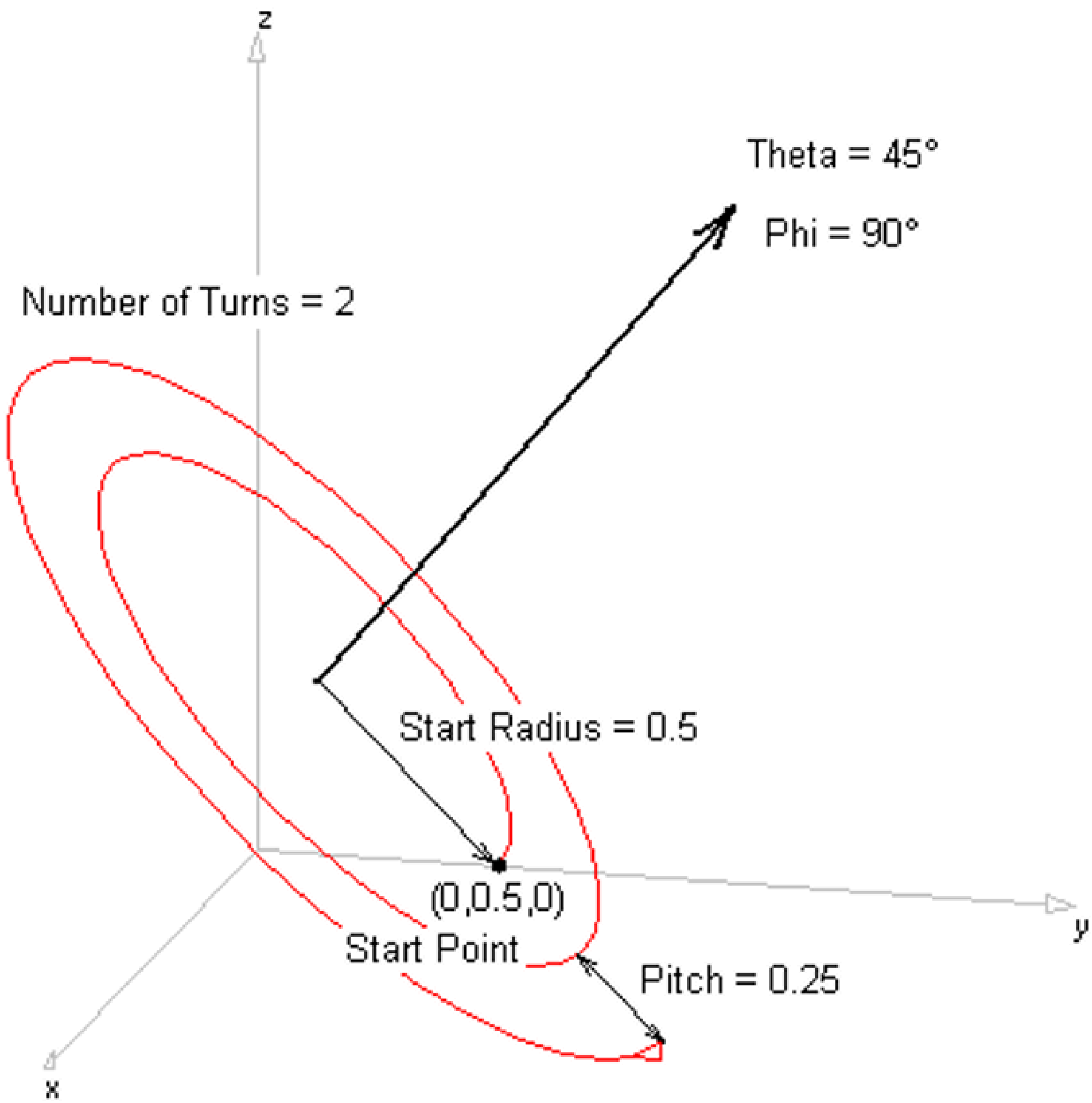


***Fig. 2:** Archimedean spiral created using the parameters from Fig. 1.*

Logarithmic Spiral

A **Logarithmic Spiral** represents an equiangular spiral in AN-SOF.

The logarithmic spiral (*spira mirabilis*) is defined by the polar equation:

$$r(\alpha)=r_0 e^{b\alpha}$$

where r_0 is the **starting radius** (the value of r at $\alpha=0$) and b is a constant given by:

$$b=\frac{p}{2\pi r_0}$$

In this expression, p represents the **starting pitch**, which is the derivative of the radius at the origin:

$$p=2\pi \frac{dr}{d\alpha}\Big|_{\alpha=0}$$

This value p corresponds to the initial growth rate of the spiral radius $r(\alpha)$ per turn. The angle α ranges from 0 to $2\pi M$, where M is the **number of turns**.

The first three terms of the Taylor expansion of the radius around $\alpha=0$ are:

$$r(\alpha)=r_0+\frac{p}{2\pi}\alpha+r_0\frac{(b\alpha)^2}{2}+\ldots$$

The linear portion of this expansion ($r_0+p/(2\pi)\alpha$) provides the polar equation of an **Archimedean spiral**.

Accessing the Logarithmic Spiral Dialog Box

To open the **Logarithmic Spiral** dialog box:

1. Navigate to **Draw > Logarithmic Spiral** in the main menu.
2. The dialog box contains three tabs: **Logarithmic Spiral**, **Attributes**, and **Materials** (**Fig. 1**).

Logarithmic Spiral Tab: Setting Geometrical Parameters

Define the logarithmic spiral by specifying the following (**Fig. 1**):

- **Start Point**: Starting coordinates (X1, Y1, Z1).
- **Start Radius**: The initial radius of the spiral at the starting point.
- **Start Pitch**: The initial pitch (turn spacing) of the spiral; this value can be positive or negative.
- **Number of Turns**: The total number of turns; this does not need to be an integer, allowing for fractional turns.
- **Orientation Angles**: Sets the direction of the spiral axis using spherical angles (Theta, Phi).
- **Rotation Angle**: Rotates the spiral around its axis.

Attributes Tab

Specify the **Number of Segments** and **Cross-Section** properties (refer to **Wire Attributes**).

Materials Tab

Set the **Resistivity** and **Coating** properties of the wire (refer to **Wire Materials**).

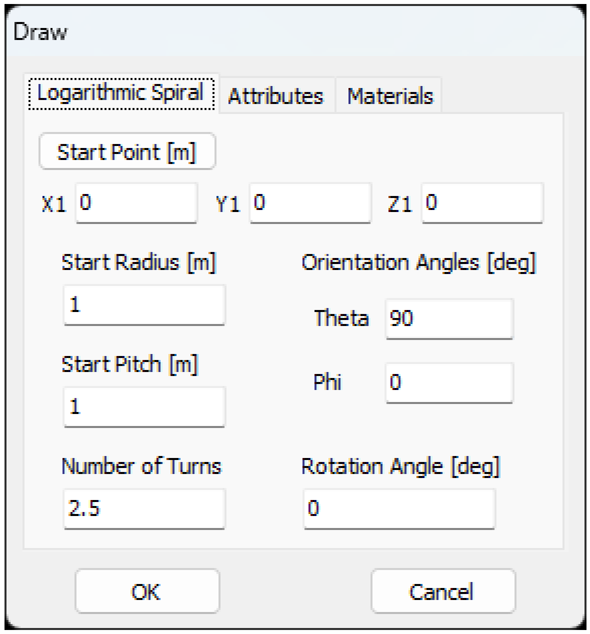


Fig. 1: *Logarithmic Spiral dialog box.*

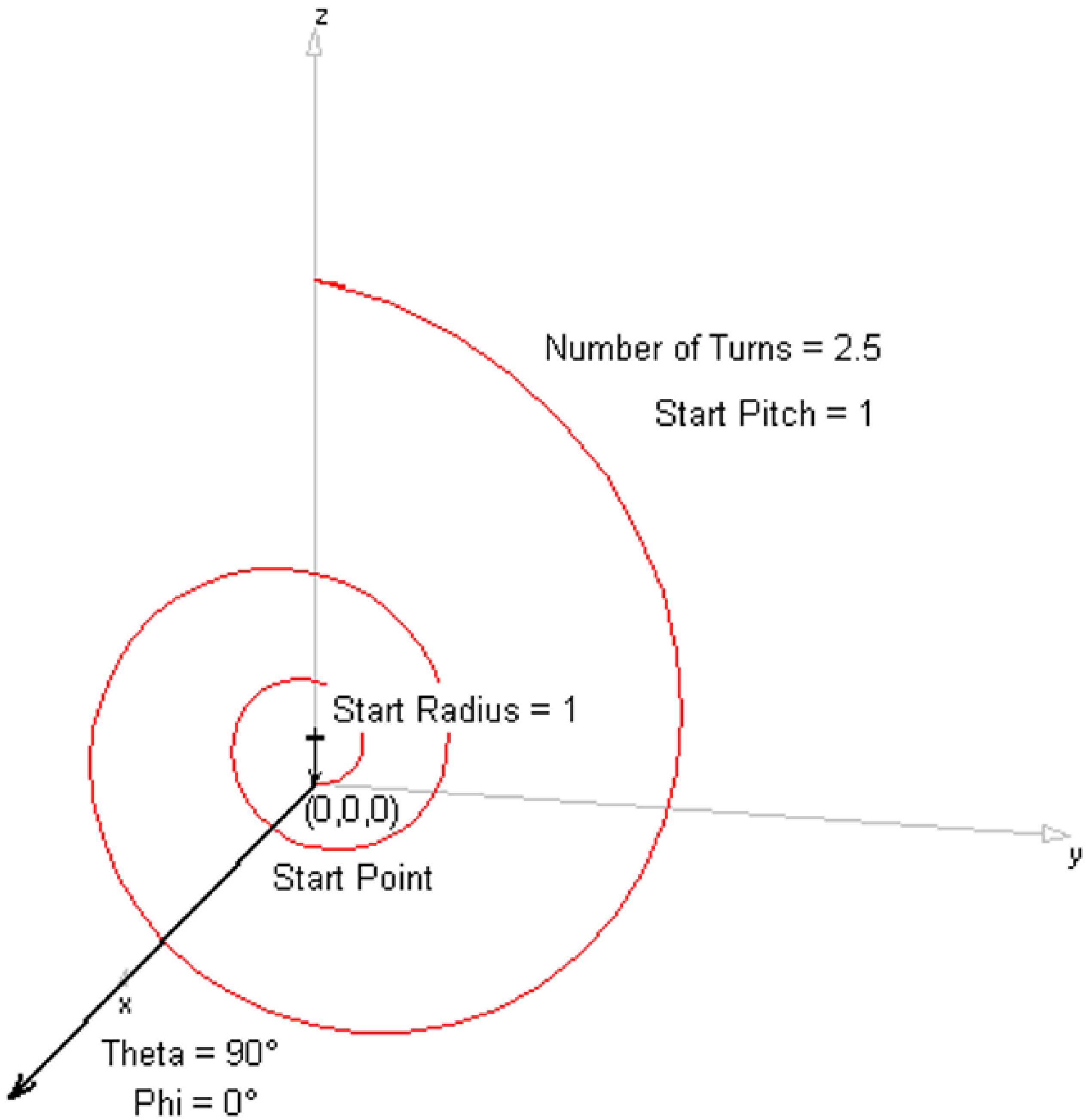


Fig. 2: *Logarithmic spiral created using the parameters from Fig. 1.*

Tapered Wires

A **tapered wire** is a wire whose radius varies along its length. Because the change in radius occurs in discrete increments, it is also known as a **stepped-radius wire**. Tapered wires always have a circular cross section, and the radius typically changes linearly in defined steps—resulting in a wire with sections of constant radius, as illustrated in **Fig. 1**.

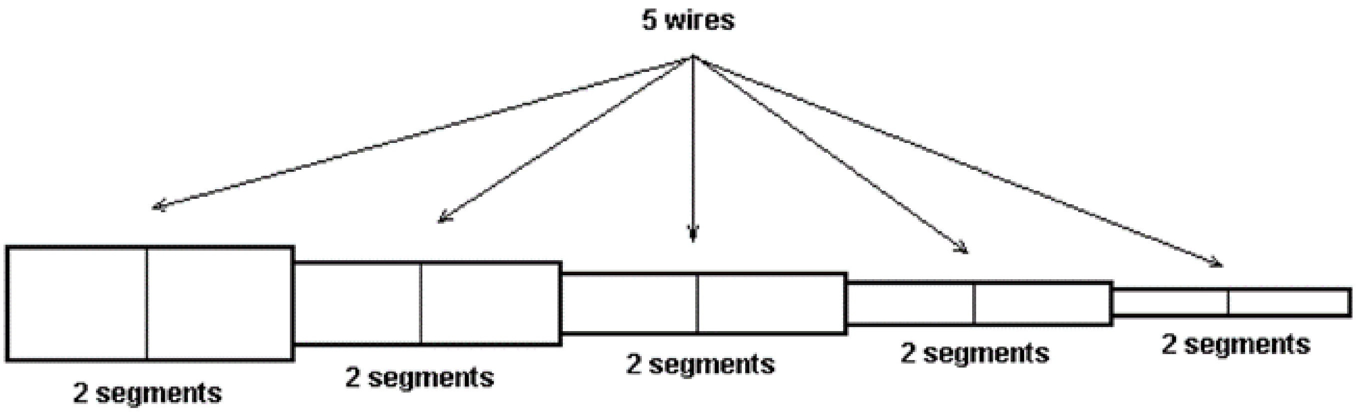


Fig. 1: Sketch of a tapered wire divided into 5 wire portions, each further subdivided into 2 segments.

To draw a tapered wire, go to **Draw > Tapered Wire** in the main menu and select a wire type. The available types (e.g., Line, Arc, Circle) are the same as those found under the **Draw** menu. For example, **Fig. 2** shows the *Line* page of the Draw dialog box when a linear wire type is selected.

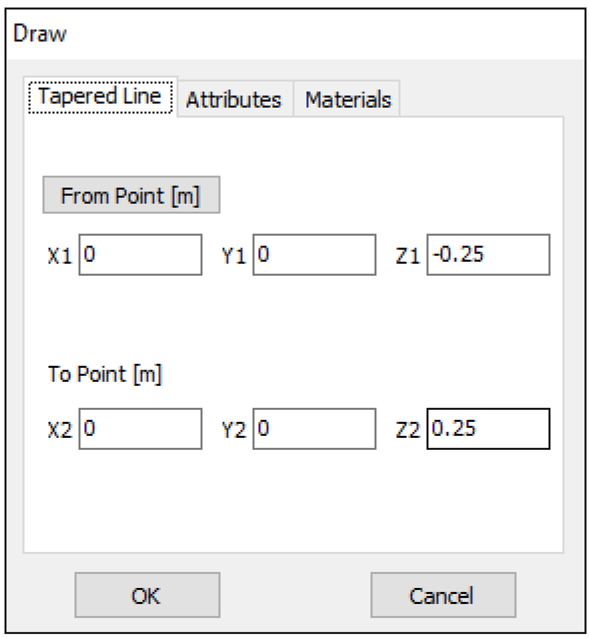


***Fig. 2:** Tapered Line page in the Draw dialog box. Access it via Main Menu > Draw > Tapered Wire > Tapered Line.*

The wire must be divided into **wire portions** based on the desired number of radius steps, as indicated in **Fig. 1**. Each wire portion with a constant radius should then be subdivided into **segments** as required by the **Method of Moments** used in the simulation.

You can define the number of wire portions and the number of segments per portion in the **Attributes** tab (see **Fig. 3**). Here, you can also specify the start and end radii of the taper. To set the material properties—including the resistivity of the wire and any coating—use the **Materials** tab (see **Fig. 4**). A tapered coating can be specified by assigning start and end thickness values.

In the workspace, **wire portions are shown in alternating colors for easier identification**.

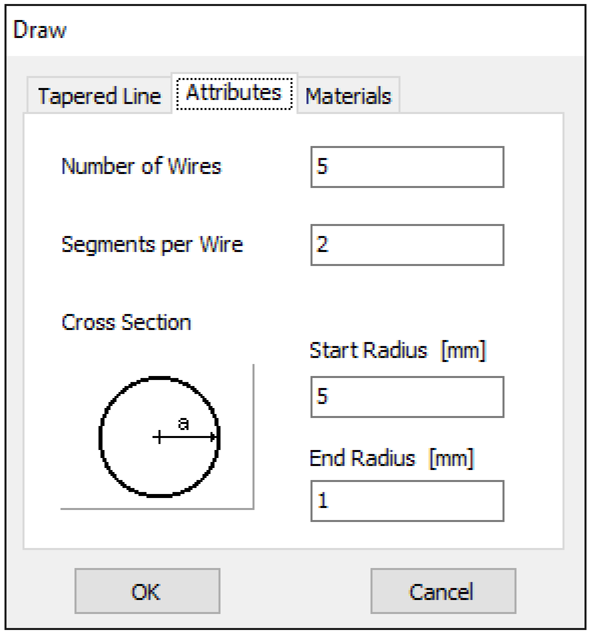


Fig. 3: *Attributes page where you can set the number of wire portions, segments per portion, and the start and end radii.*

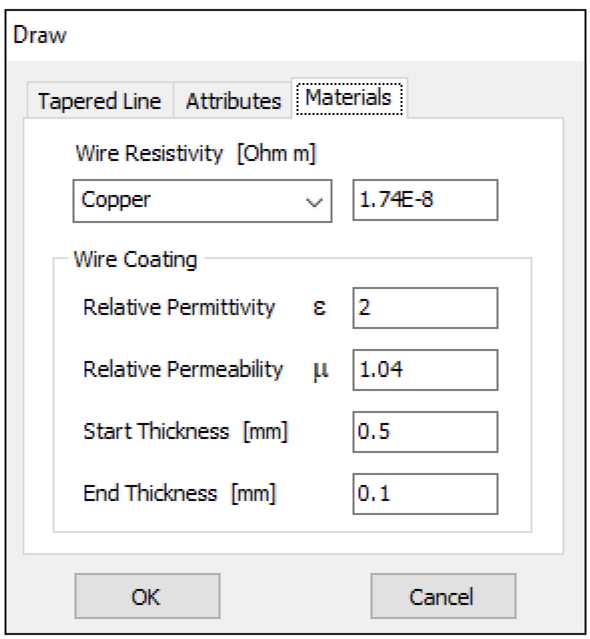


Fig. 4: *Materials page where the wire resistivity and coating properties are defined. A tapered coating can be specified by setting the start and end thickness values.*

Importing Wires

Supported Formats

To import wires from an external file into AN-SOF, follow these steps:

1. Navigate to File > **Import Wires** in the main menu.
2. A sub-menu with three options will be displayed: **AN-SOF**, **NEC**, and **MM** formats.
3. Note that the MM format should contain only **linear wires** in **ASCII** text format.

AN-SOF Format

Wires can be imported into the AN-SOF workspace from another AN-SOF project. When a project is saved, a corresponding file with a **.wre** extension is created in the same directory. This file, named after the project, contains the geometrical description of all wires within the project. For details on files generated during project saves, refer to **File Formats**.

To import wires into your project, navigate to the main menu and select **File > Import Wires > AN-SOF Format**. Then, choose the specific **.wre** file you wish to import. You can import multiple **.wre** files, one at a time, as needed.

NEC Format

There are slight differences between the commands supported by AN-SOF and the standard NEC cards. To maintain compatibility with the NEC format, originally designed for data entry using punch cards, some fields appear repeating, and others must be entered with a zero, having no meaning. **Lengths and wire radii are assumed to be in meters.** If errors are found while importing a file, an error report will be shown in the **Note panel** of the **Setup tab**.

The `SY` command for symbolic variables **is not supported**. To run simulations with variable geometric parameters, you can write a script to sweep those parameters and generate the corresponding NEC files, then use the **Run Bulk Simulation** feature (see **Running a Bulk Simulation**).

For **optimization** tasks, you'll need an optimizer script and should work with the **AN-SOF Engine**. Examples of parameter sweeps and optimization scripts are available in the **Scripts & Optimizers** category.

GW – Linear Wire

One linear wire per line must be set, beginning with "GW" and ending with an Enter, as follows:

GW Tag Segments X1 Y1 Z1 X2 Y2 Z2 Radius

[Enter]

Tag: Tag number for the linear wire (Tag > 0). The space between "GW" and Tag is optional. A single tab or comma can also be used as a separator between the command name and the first data field.

Segments: Number of segments for the wire. If zero is entered, the minimum recommended number of segments will be computed.

X1 Y1 Z1: Cartesian coordinates of the start point of the linear wire.

X2 Y2 Z2: Cartesian coordinates of the end point of the linear wire.

Radius: Wire radius.

Fields can be separated by up to two spaces, a single tab, a single comma, or a comma and space. Each GW line, including the last one in a set of linear wires to be imported, must end with an Enter (press Enter on the keyboard for a carriage return). The text lines above the GW lines will be ignored, so comments can be added at the beginning of the file.

The following are equivalent examples:

Write comments here

GW 1 12 5.42 0.38 1.262 5.425 -0.378 1.261 0.01*[Enter]*

GW 2 5 7.45 0 1.122 7.45 0 1.49 0.015*[Enter]*

GW 3 2 8.3 0.0 1.12 8.37 0.0 1.595 0.01*[Enter]*

Write comments here

GW1,12,5.42,0.38,1.262,5.425,-0.378,1.261,0.01*[Enter]*

GW2,5,7.45,0,1.122,7.45,0,1.49,0.015*[Enter]*

GW3,2,8.3,0.0,1.12,8.37,0.0,1.595,0.01*[Enter]*

CM and Other Commands

The following commands: **CM** (comment lines), **GH** (helical wire), **GA** (arc), **GM** (coordinate transformation), **GS** (scale dimensions), **GE** (ground connections), **GN** (real ground parameters), **TL** (transmission line), **LD** (load impedances and wire conductivity), **IS** (insulated wire), **FR** (frequency), **EX** (excitation), **EK** (exact kernel), and **RP** (radiation pattern), will also be read.

CM lines will be added to the Note panel of the Setup tabsheet after the NEC file is imported. The comment termination card, “CE”, is not needed in AN-SOF. Comments without the CM command at the beginning of the file will be ignored and not imported. The **command names**—“CM”, “GW”, “GH”, etc.—are **reserved words** in AN-SOF and are used to recognize the fields between these commands and the final Enter in each text line, so **the command names should not be used in comments**.

IMPORTANT: CM lines must always be placed at the beginning of a .nec file and kept separate from other commands.

The rest of the AN-SOF commands in NEC format are listed below, where **all the indicated fields are mandatory**.

GH – Helix

The GH command is used to define a helix in AN-SOF with the following syntax:

GH Tag Segments Spacing Length R R R R Radius

[Enter]

Tag: A positive number representing the tag for the helix. The space between "GH" and the Tag is optional. Note that the helix begins at the origin and develops along the positive z-axis. To adjust the helix's position or rotation, use the GM command described below. It's important to mention that the GH command differs in NEC-4.

Segments: The number of segments for the helix. If zero is entered, AN-SOF will compute the minimum recommended number of segments. Unlike NEC, AN-SOF uses conformal segments that precisely follow the helix contour.

Spacing: Spacing between turns.

Length: Total length of the helix. A positive Length value results in a right-handed helix, while a negative Length value produces a left-handed helix.

R: Radius of the helix (repeated four times).

Radius: Wire radius.

Note: AN-SOF uses **conformal segments** that exactly follow the helix contour, distinguishing it from NEC.

GA – Arc

The GA command is utilized to define an arc in AN-SOF with the following syntax:

GA Tag Segments R Ang1 Ang2 Radius

[Enter]

Tag: A positive number serving as the tag for the arc. The space between "GA" and the Tag is optional. The arc is situated on the **xz-plane**, centered at the origin, making the **y-axis** the axis of the arc. To manipulate the position or rotation of the arc, use the GM command described below.

Segments: The number of segments for the arc. If zero is entered, AN-SOF will compute the minimum recommended number of segments. It's worth noting that, unlike NEC, AN-SOF uses conformal segments that precisely follow the arc contour.

R: Arc radius.

Ang1: The angle of the first end of the arc measured from the x-axis in a left-handed direction about the y-axis, given in degrees.

Ang2: The angle of the second end of the arc, measured in degrees.

Radius: Wire radius.

Note: AN-SOF uses **conformal segments** that exactly follow the arc contour, distinguishing it from NEC.

GB – AN-SOF's Arc

The GB command is utilized to define an arc in AN-SOF with the following syntax:

GB Tag Segments Type X1 Y1 Z1 X2 Y2 Z2 X3 Y3 Z3 Radius

[Enter]

Tag: A positive number serving as the tag for the arc. The space between "GB" and the Tag is optional.

Segments: The number of segments for the arc. If zero is entered, AN-SOF will compute the minimum recommended number of segments. It's worth noting that, unlike NEC, AN-SOF uses conformal segments that precisely follow the arc contour.

Type: [Type of arc](). Set Type = 0 for entering three points, and Type = 1 for entering the start point, center, and end point.

X1 Y1 Z1: Cartesian coordinates of the start point of the arc.

X2 Y2 Z2: Cartesian coordinates of the second point of the arc if Type = 0, or the arc center if Type = 1.

X3 Y3 Z3: Cartesian coordinates of the end point of the arc.

Radius: Wire radius.

Note: AN-SOF uses **conformal segments** that exactly follow the arc contour, distinguishing it from NEC. The "GB" command is exclusive to AN-SOF and cannot be found in any NEC version.

GM – Coordinate Transformation

The GM command in AN-SOF facilitates coordinate transformations with the following syntax:

GM 0 N rotX rotY rotZ DX DY DZ 0

[Enter]

N: If N is set to **0**, it implies that the entire structure above the GM command must undergo rotation and translation based on the specified values for (rotX, rotY, rotZ) and (DX, DY, DZ). The coordinate transformations are applied sequentially in that order. If N is set to **1**, it indicates that the structure above the GM command must be copied, and the copy should be moved to a new position (DX, DY, DZ) from the origin. You can use the "GM" command below the "GW," "GH," and "GA" commands to rotate, move, and copy linear wires, helices, and arcs as needed.

rotX: Angle of rotation about the X-axis, specified in degrees.

rotY: Angle of rotation about the Y-axis, specified in degrees.

rotZ: Angle of rotation about the Z-axis, specified in degrees.

DX: Translation along the X-axis, moving the structure by an amount DX.

DY: Translation along the Y-axis, moving the structure by an amount DY.

DZ: Translation along the Z-axis, moving the structure by an amount DZ.

GS – Scale Structure Dimensions

The GS command in AN-SOF is used for scaling structure dimensions. The syntax is as follows:

GS 0 0 Scale

[Enter]

Scale: This represents the scaling factor. Applying this command results in the multiplication of all structure dimensions, including wire radii, by the specified scale value.

GE – Ground Connections

The GE command in AN-SOF is used for defining ground connections. The syntax is as follows:

GE Type

[Enter]

Type = 0: No ground plane is present. If a “GE” command is used without specifying a type, it will be interpreted as “GE 0”.

Type = 1: A PEC ground plane is placed at z = 0, and wires ending on the ground plane will be connected to the ground. If a **real ground plane** has been chosen, Type = 1 indicates that the wire connections to the ground must be considered as **zero-Ohm connections**.

Type = -1: The wire connections to the ground are imperfect and produce power losses when a real ground plane has been chosen.

GN – Real Ground

The GN command in AN-SOF is used for defining real ground parameters. The syntax is as follows:

GN Type Screen 0 0 Epsilon Sigma Length WireRadius

[Enter]

Type: Type of ground plane.

> **Type = -1:** Free space simulation; all ground parameters are ignored. “GN -1” can be used in this case.
>
> **Type = 0:** Reflection Coefficients/Asymptotic option.
>
> **Type = 1:** PEC ground plane at z = 0; other parameters are ignored. “GN 1” can be used in this case.
>
> **Type = 2:** Sommerfeld-Wait/Asymptotic option.

Screen: Number of radials in a radial wire ground screen. Set Screen = 0 if no ground screen is present.

Epsilon: Ground plane relative permittivity or dielectric constant.

Sigma: Ground plane conductivity in [S/m].

Length: Length of radial wires if a radial wire ground screen is used. Enter zero if no ground screen is used.

WireRadius: Radius of radial wires if a screen is used. Enter zero if no ground screen is used.

TL – Transmission Line

The TL command in AN-SOF is used to define a transmission line. The syntax is as follows:

TL Tag1 Seg1 Tag2 Seg2 Zc Length Y1r Y1i Y2r Y2i

[Enter]

Tag1: Wire tag number to which the first port of the transmission line connects.

Seg1: Segment number of wire Tag1 to which the first port of the transmission line connects.

Tag2: Wire tag number to which the second port of the transmission line connects.

Seg2: Segment number of wire Tag2 to which the second port of the transmission line connects.

Zc: Characteristic impedance of the transmission line in Ohms. A negative Zc can be entered to set a "crossed" transmission line with a 180° phase reversal relative to the reference directions of the segments. The characteristic impedance of the line is |Zc|.

Length: Length of the transmission line in meters. If Length = 0, the linear distance between the transmission line ports will be considered as the length for the line. To simulate a zero-length transmission line, enter 1E-10.

Y1r: Real part of the shunt admittance across end one of the transmission line [S].

Y1i: Imaginary part of the shunt admittance across end one of the transmission line [S].

Y2r: Real part of the shunt admittance across end two of the transmission line [S].

Y2i: Imaginary part of the shunt admittance across end two of the transmission line [S].

Refer to **Adding Transmission Lines** for a review of considerations when setting transmission lines, including advanced settings not available with the TL command.

LD – Load Impedance

The LD command in AN-SOF is used to define a load impedance. The syntax is as follows:

LD Type Wire# Seg# Seg# R L C

[Enter]

Type: Type of load. Series/parallel/trap RLC loads, fixed impedances R+jX, and wire conductivity can be set.

- Set **Type = 0** for a series RLC load.
- Set **Type = 1** for a parallel RLC load.
- Set **Type = 4** for a fixed impedance R+jX. The reactance "X" must be entered in the position of "L" (the "C" field will be ignored). The reactance is fixed, so it does not scale with frequency.

- Set **Type = 5** and **Seg# = 0** to specify a wire conductivity [S/m] in the "R" field for the wire number "Wire#". Use the command **LD 5 0 0 0 R 0 0** to set a conductivity "R [S/m]" on all wires. **"LD 5" command for setting wire conductivity must be below all LD 0, LD 1, LD 4, and LD 6 lines.**
- Set **Type = 6** for a trap RLC load (series RL + parallel C).

Wire#: Wire tag number where the load or conductivity is placed.

Seg#: Segment number where the load is placed. Note that it appears twice due to a NEC convention not used in AN-SOF, so the second Seg# will be ignored. Set Seg# = 0 if a wire conductivity is to be entered.

R: Resistance in Ohms or conductivity in S/m.

L: Inductance in Henries when Type = 0, or reactance in Ohms when Type = 4 (it does not scale with frequency). The "L" field is ignored if R is a conductivity, so a zero can be entered.

C: Capacitance in Farads; if none, enter zero. It is ignored if R is a conductivity, so enter zero.

IS – Insulated Wire

The IS command in AN-SOF is used to define an insulated wire. The syntax is as follows:

IS 0 Wire# 0 0 Epsilon 0 Radius

[Enter]

Wire#: Wire tag number where the insulation or coating will be applied.

Epsilon: Relative permittivity of the dielectric sheath.

Radius: Radius of the insulating sheath. Ensure it is greater than the wire radius.

FR – Frequencies

The FR command in AN-SOF defines the frequencies used in a simulation. Its syntax is:

FR Type Num 0 0 Freq Df

[Enter]

Type: Specifies the type of frequency sweep. For a linear sweep, set Type = 0; for a logarithmic sweep, set Type = 1.

Num: Number of frequency steps (**Num ≥ 1**).

Freq: Frequency in MHz, or the starting frequency for a range.

Df: If Type = 0, it represents the frequency stepping increment in MHz. If Type = 1, it is the multiplication factor for a logarithmic sweep.

To simulate at a **single frequency**, use: **FR 0 1 0 0 Freq 0**

EX – Excitation

The EX command in AN-SOF is used to define excitation sources for simulations. The syntax is as follows:

EX Type Wire# Seg# 0 Real Imag

[Enter]

Type: Type of source. Use Type = **0** or **5** (the "5" corresponds to an old source model used in NEC) for a **voltage source**. Set Type = **6** for a **current source**. Note that current sources in AN-SOF automatically have a non-zero internal impedance set in parallel with the source (1E6 Ohm).

Wire#: Wire tag number where the source is placed.

Seg#: Segment where the source is located.

Real: Real part of the source voltage or current.

Imag: Imaginary part of the source voltage or current.

EK – Exact Kernel

The EK command in AN-SOF is used to force the use of the Exact Kernel. The syntax is as follows:

EK

[Enter]

This command ensures that the Exact Kernel is utilized, even if this option is disabled. It's important to note that AN-SOF has the Exact Kernel enabled by default.

RP – Radiation Pattern

The RP command in AN-SOF is used to set the radiation pattern parameters. The syntax is as follows:

RP 0 Ntheta Nphi 1001 Theta Phi Dtheta Dphi R

[Enter]

Ntheta: Number of values of Θ at which the field is to be computed.

Nphi: Number of values of φ at which the field is to be computed.

(***Note:*** The value "1001" is a NEC variable and will be ignored since AN-SOF always computes the average power gain.)

Theta: Initial Θ angle in degrees.

Phi: Initial φ angle in degrees.

Dtheta: Increment for Θ in degrees.

Dphi: Increment for φ in degrees.

R: Radial distance in meters of the field point from the origin. R = 0 is taken as R = 1 m.

MM Format

One linear wire per line must be defined as follows:

X1,*[TAB]***Y1,***[TAB]***Z1,***[TAB]***X2,***[TAB]***Y2,***[TAB]***Z2,***[TAB]***Radius,***[TAB]**Segments***

[Enter]

X1 Y1 Z1 = Cartesian coordinates of the wire start point.

X2 Y2 Z2 = Cartesian coordinates of the wire end point.

Radius = Wire radius.

Segments = Number of segments.

The last text line must end with an Enter (press Enter in the keyboard for a carriage return).

Example:

5.42, 0.38, 1.262, 5.425, -0.378, 1.261, 0.01, 12

7.45, 0, 1.122, 7.45, 0, 1.49, 0.015, 5

8.3, 0.0, 1.12, 8.37, 0.0, 1.595, 0.01, 2

[Enter]

In the MM format, a wire can be automatically segmented by specifying any number less than or equal to zero for the segment count. The coordinates of each wire's start and end points must use the same length unit selected in the **Preferences** dialog box in AN-SOF. Similarly, the radius of

any imported wire must also be expressed in the selected unit.
Tabular Input of Linear Wires

Linear wires can be entered and edited in a table using the **Tabular Input** window. To access this feature, navigate to **Draw > Tabular Input (Ctrl + T)** in the main menu. This opens the **Tabular Input** window (see **Fig. 1**), which contains four tabs:

1. **Wires**:

   Allows you to enter and edit linear wires by specifying their end coordinates, number of segments, wire radius, and materials.

2. **Sources**:

   Enables you to connect sources to the wires listed in the **Wires** tab. A source must be connected to a specific wire segment.

3. **Loads**:

   Allows you to connect loads to the wires listed in the **Wires** tab. A load must be connected to a specific wire segment.

4. **Trans. Lines**:

   Enables you to connect transmission lines between two wire segments listed in the **Wires** tab.

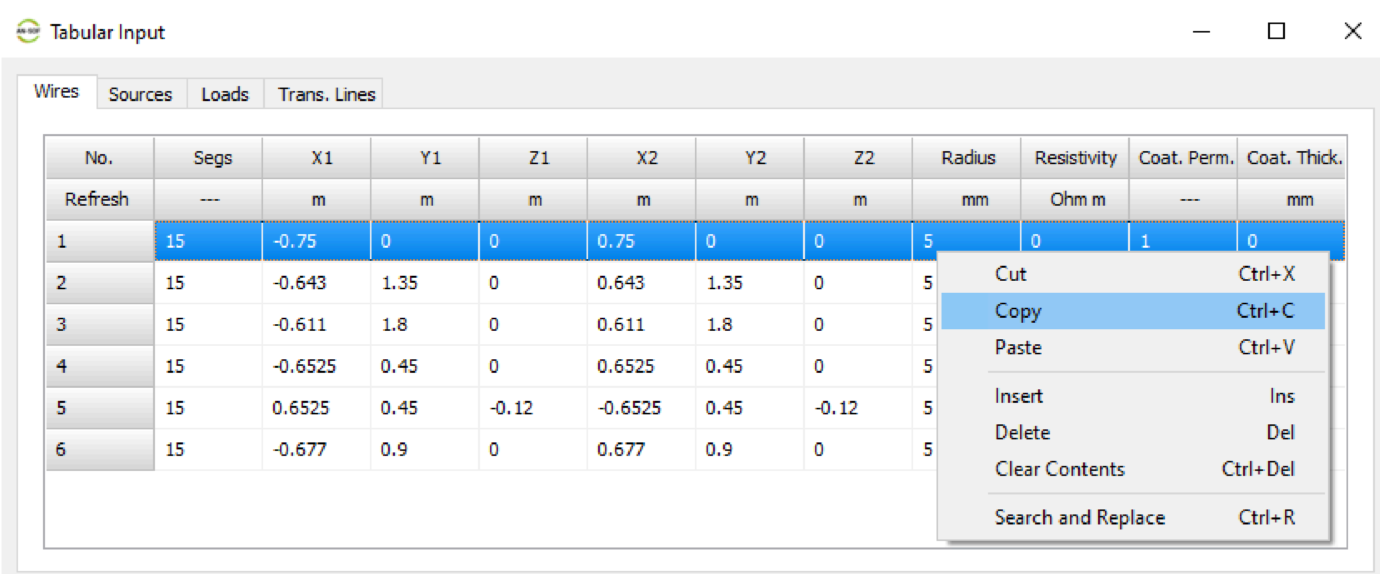


***Fig. 1:** Tabular Input window – Wires tab, for entering and editing linear wires.*

Wires Tab

Select the **Wires** tab and enter values as specified in the column titles (see **Fig. 1**). Each row corresponds to a linear wire, and you can input details such as:

- Number of segments (**Segs**).
- Coordinates of the starting point (**X1, Y1, Z1**) and ending point (**X2, Y2, Z2**).
- Wire radius.
- Resistivity.
- Coating (dielectric insulation).

Note: Only wires with a circular cross-section can be entered.

Table Interaction

- **Right-click** on the table to open a pop-up menu with standard options such as **Cut (Ctrl + X)**, **Copy (Ctrl + C)**, and **Paste (Ctrl + V)**.
- **Single cells** can be selected by left-clicking on them or by using the **TAB** and **arrow keys** on the keyboard.
- **Rows** can be selected by clicking on the row number in the left column (**No. column**). Use the mouse or the **up/down arrow keys** to select a single row. The selected wire (row) is **highlighted in red** in the workspace. Double-click on a cell to exit row selection mode.

Row Operations

- Use **Cut (Ctrl + X)**, **Copy (Ctrl + C)**, and **Paste (Ctrl + V)** to manipulate selected rows.
- Use the **Insert (Ins key)** and **Delete (Del key)** options to add or remove rows.
- The **Clear Contents (Ctrl + Del)** option clears the content of a selected cell or row.
- The **Search and Replace (Ctrl + R)** option allows for bulk edits to wire end coordinates.

Wire Numbers in the Workspace

While the **Tabular Input** window is open, wire numbers are displayed in the workspace next to the corresponding wires (see **Fig. 2**). These numbers indicate the order of the wires in the table.

Note:

- Wires do not have permanent tags in AN-SOF. If a wire is deleted, the numbers will adjust automatically.
- Wire numbers are used solely for identification in the workspace while the **Tabular Input** window is open.

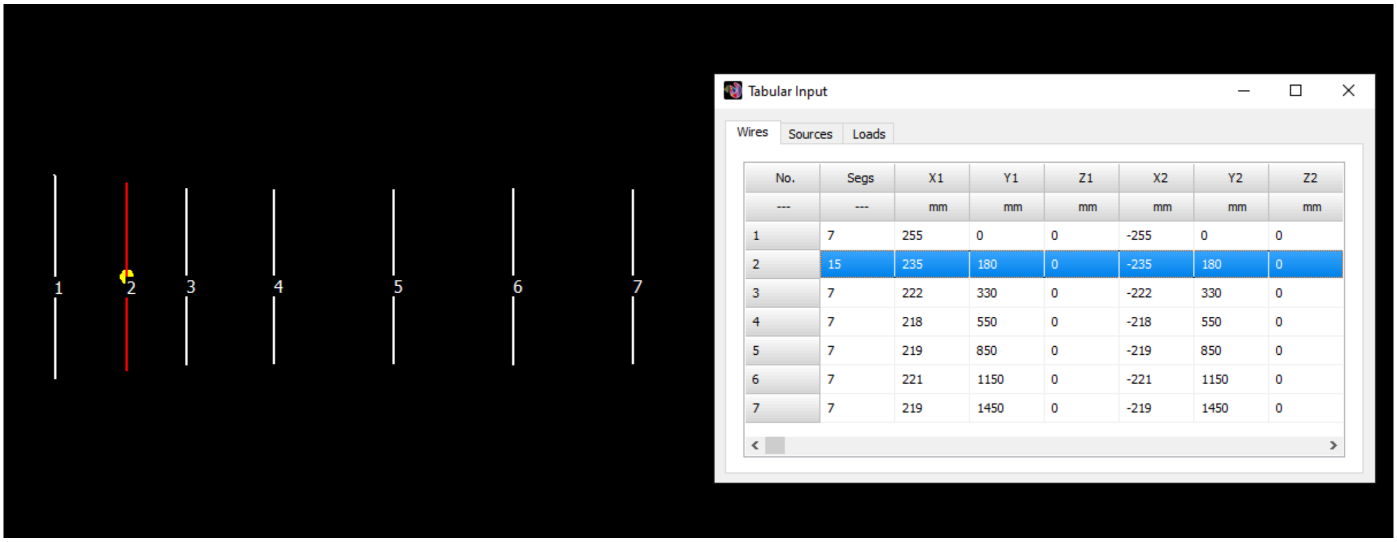


Fig. 2: *Tabular Input window, showing wire numbers displayed in the workspace.*

Sources Tab

Use the **Sources** tab to enter sources (see **Fig. 3**).

Entering Source Details

- **Type Column**: Enter **"V"** for a voltage source or **"I"** for a current source.
- **Wire No. Column**: Specify the wire on which the source is placed. Refer to the wire numbering in the **No.** column of the **Wires** tab.
- **Position Column**: Indicate the segment number where the source is connected. The segment number ranges from **1** to the number of segments (**Segs**) specified for the wire in the **Wires** tab.
- **Amplitude Column**: Enter the amplitude of the source in **Volts** or **Amperes**.
- **Phase Column**: Specify the phase of the source in **degrees**.

Table Interaction

Right-click on the table to open a pop-up menu with standard options:

- **Cut (Ctrl + X)**
- **Copy (Ctrl + C)**
- **Paste (Ctrl + V)**
- **Insert (Ins key)**
- **Delete (Del key)**
- **Clear Contents (Ctrl + Del)**

These options function the same way as in the **Wires** tab, allowing you to manipulate cells and rows.

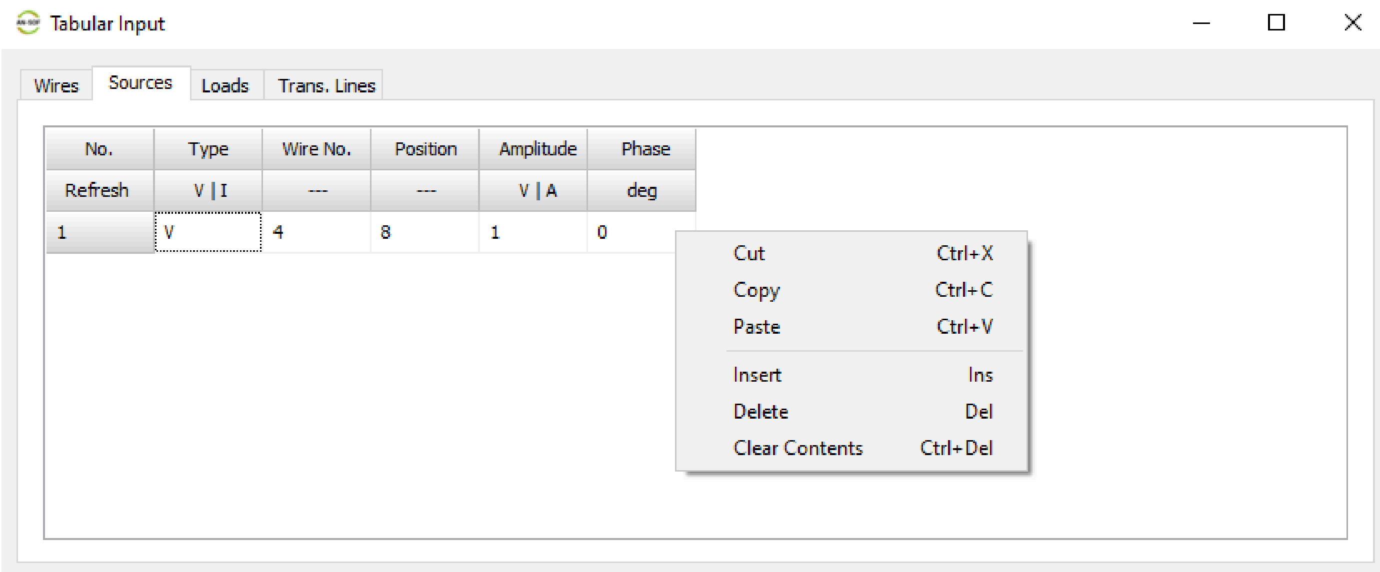


Fig. 3: *Tabular Input window – Sources tab, for entering and editing sources.*

Loads Tab

Use the **Loads** tab to enter load impedances (see **Fig. 4**).

Entering Load Details

- **Type Column**: Enter one of the following:
  - **"L"** for an inductor.
  - **"C"** for a capacitor.
  - **"Z"** for an impedance (**R + jX**).
- **Wire No. Column**: Specify the wire on which the load is placed. Refer to the wire numbering in the **No.** column of the **Wires** tab.
- **Position Column**: Indicate the segment number where the load is connected. The segment number ranges from **1** to the number of segments (**Segs**) specified for the wire in the **Wires** tab.
- **R Column**: Enter the resistance value (**R**) in Ohms.
- **Last Column**: Depending on the option entered in the **Type** column, input one of the following:
  - Inductance (**L**) in the displayed unit.
  - Capacitance (**C**) in the displayed unit.
  - Reactance (**X**) in Ohms.

Table Interaction

Right-click on the table to open a pop-up menu with standard options:

- **Cut (Ctrl + X)**
- **Copy (Ctrl + C)**
- **Paste (Ctrl + V)**
- **Insert (Ins key)**
- **Delete (Del key)**
- **Clear Contents (Ctrl + Del)**

These options function the same way as in the **Wires** tab, allowing you to manipulate cells and rows.

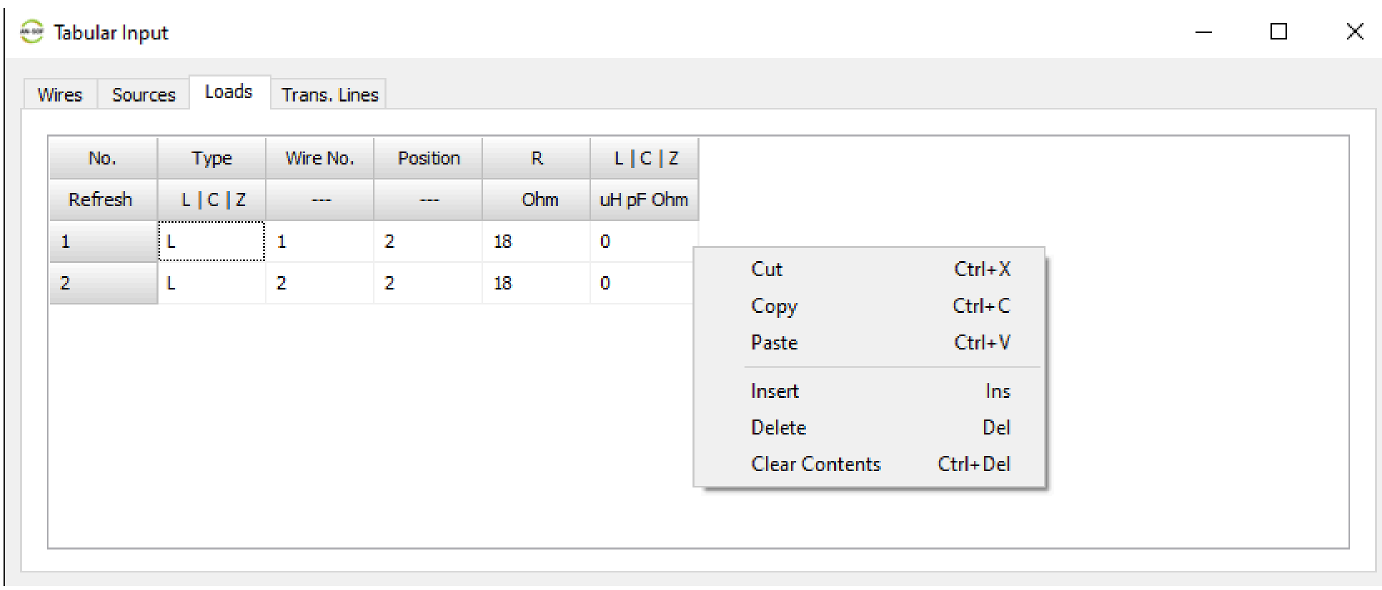


Fig. 4: Tabular Input window – Loads tab, for entering and editing load impedances.

Trans. Lines Tab

Use the **Trans. Lines** tab to enter transmission lines and connect them to wire segments (see **Fig. 5**).

Entering Transmission Line Details

- **Port 1 Columns**:
  Specify the wire segment where **Port 1** of the transmission line is connected by entering the **Wire No.** and **Position**. Refer to the numbering and number of segments specified for each wire in the **Wires** tab.
- **Port 2 Columns**:
  Similarly, specify the wire segment where **Port 2** of the transmission line is connected by entering the **Wire No.** and **Position**.

- **Additional Columns**:
  - Complete the **Type**, **Z0** (characteristic impedance), **VF** (velocity factor), **Length**, and other columns as explained in the section [Adding Transmission Lines](#).
  - To simplify the process, select a row and double-click on an option in the right panel, which contains a collection of **transmission line types with preloaded parameters**.

Table Interaction

Right-click on the table to open a pop-up menu with standard options:

- **Cut (Ctrl + X)**
- **Copy (Ctrl + C)**
- **Paste (Ctrl + V)**
- **Insert (Ins key)**
- **Delete (Del key)**
- **Clear Contents (Ctrl + Del)**

These options function the same way as in the **Wires** tab, allowing you to manipulate cells and rows.

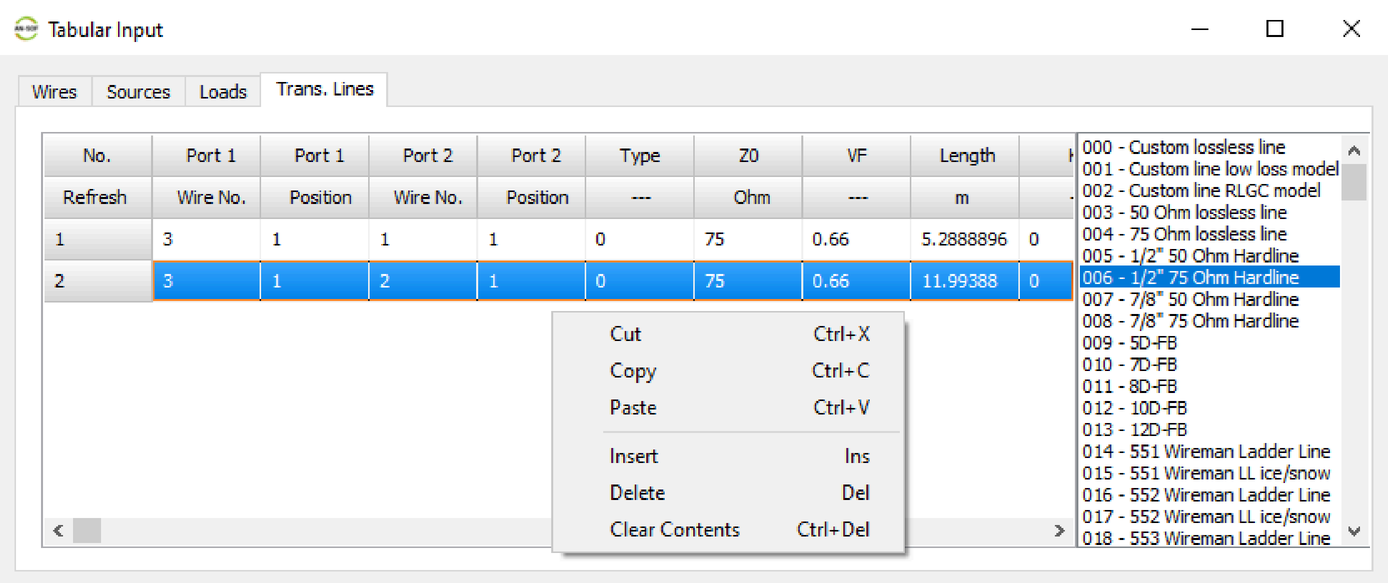


***Fig. 5:** Tabular Input window – Trans. Lines tab, for entering and editing transmission lines.*

Note:

- Entering a zero in **Wire No.**, regardless of the **Position** entered, disconnects the port from the wire (putting the transmission line port in **FREE** status).
- Only transmission lines with both ports connected to wire segments will be considered in a simulation.
- Clicking on a row number (first column of the table) in the **Trans. Lines** tab **highlights the transmission line in red** in the AN-SOF workspace.

Refresh Button

A **Refresh** button is located just below the **No.** cell in all tabs of the **Tabular Input** window (see **Fig. 6**). Click the **Refresh** button to instantly apply changes. This eliminates the need to close and reopen the **Tabular Input** window to apply modifications.

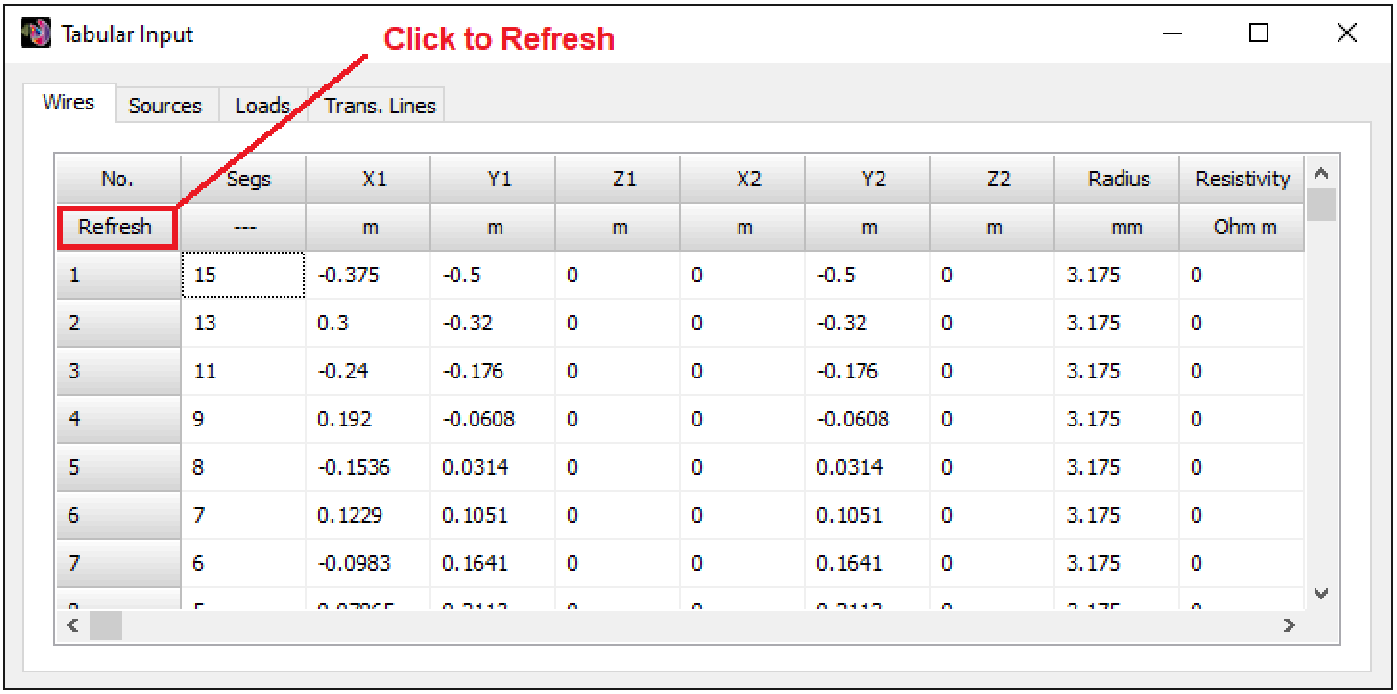


Fig. 6: *Refresh button in the Tabular Input window, for applying changes instantly.*

Editing Wires

Selecting a Wire

Ways to Select a Wire

Selecting a wire in the workspace allows you to edit it, visualize its properties, or view simulation results. You can select any wire using one of the following methods:

1. **Using the Select Wire Tool**: Click the **Select Wire** button (arrow icon) on the toolbar, and then left-click on the desired wire.
2. **Right-Clicking the Wire**: Right-click on the wire to open a pop-up menu (see **Fig. 1**).
3. **Using Keyboard Shortcuts**: Press **F8** or **F9** to select wires one by one, either forwards or backwards, in the order they were created.

Double-click on the workspace or press **ESC** to deselect the wire.

When a wire is selected, it will be highlighted in **light blue** for easy identification. Once selected, you can:

- **Edit the wire**.

- **View its properties**, such as **geometrical details**, **electrical length**, **number of segments**, **wire radius**, and **materials**.
- Visualize simulation results, such as **current distribution** or **input impedance** (if the wire has a source on one of its segments).

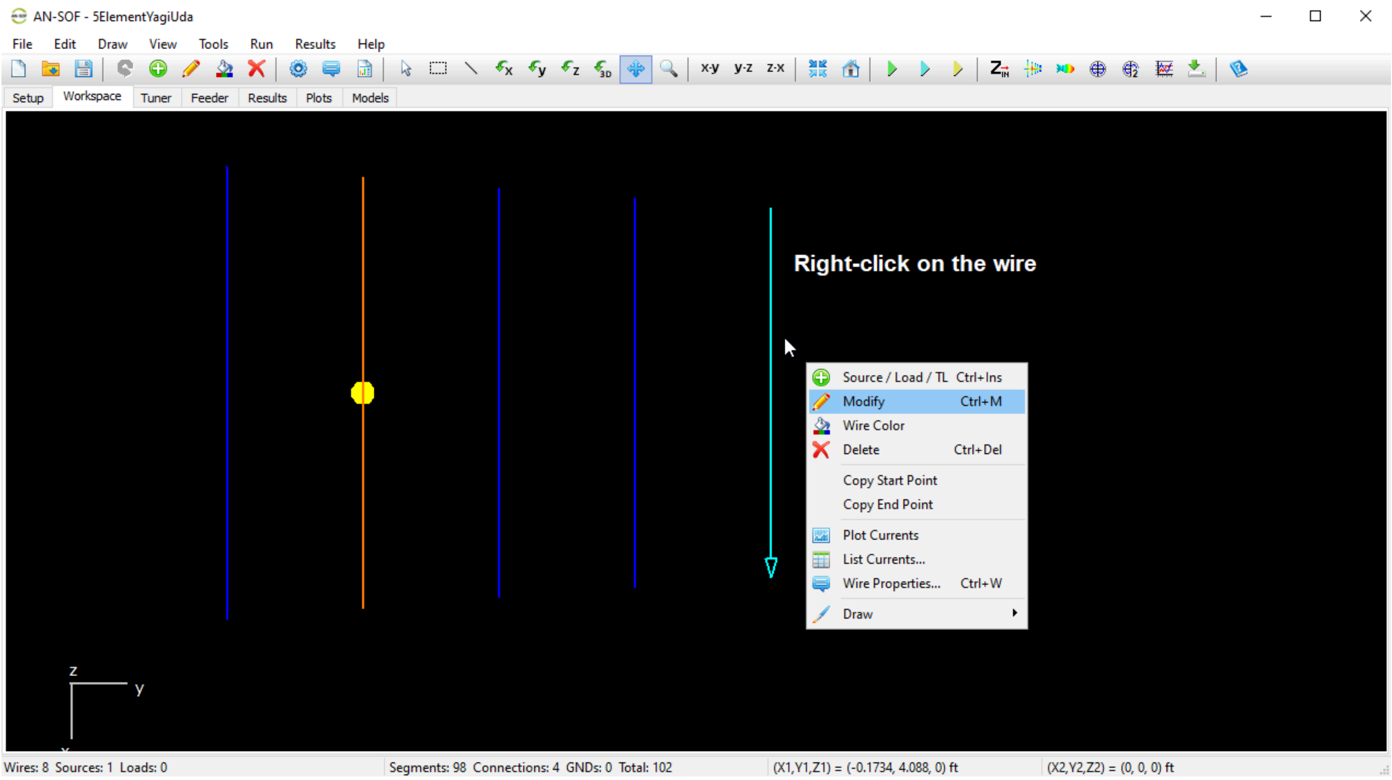


Fig. 1: *Pop-up menu displayed when right-clicking on a selected wire.*

The Pop-Up Menu

Right-clicking on a wire opens a pop-up menu with the following commands:

Source / Load / TL (Ctrl + Ins)

Opens the **Source / Load / TL** toolbar, allowing you to connect a source, load impedance, or transmission line to a segment of the selected wire.

Modify (Ctrl + M)

Opens the **Modify** dialog box to edit the selected wire.

Wire Color

Opens a dialog box to change the color of the selected wire.

Delete (Ctrl + Del)

Deletes the selected wire, including all sources and loads placed on it. Transmission line connections will also be removed.

Copy Start Point

Copies the start point of the selected wire, enabling you to connect it to the end of another wire.

Copy End Point

Copies the end point of the selected wire, enabling you to connect it to the end of another wire.

Plot Currents

Opens a chart in the **AN-XY Chart** application, displaying the current distribution along the selected wire. This option is enabled only after currents have been computed.

List Currents

Opens the **List Currents** toolbar, allowing you to select a wire segment and tabulate its current versus frequency. This option is enabled only after currents have been computed.

Wire Properties (Ctrl + W)

Opens the **Wire Properties** dialog box, where you can view the geometry, attributes, and material data of the selected wire.

Draw

Contains a sub-menu with commands to draw various types of wires, including:

- **Line**
- **Arc**
- **Circle**
- **Helix**
- **Quadratic**
- **Archimedean Spiral**
- **Logarithmic Spiral**

Modifying a Wire

To modify an existing wire, right-click on it and choose **Modify** from the **pop-up menu**. This opens the **Modify** dialog box, where you can adjust the wire’s geometric parameters, **attributes**, and **materials**.

Alternatively, you can use the **Select Wire** button (arrow icon) from the main toolbar. After clicking this button, left-click on the wire you wish to modify. Once selected, go to **Edit > Modify** in the main menu. Note that this option is only enabled when **a wire is selected**.

Figure 1 shows how to access the **Modify** command for a linear wire.

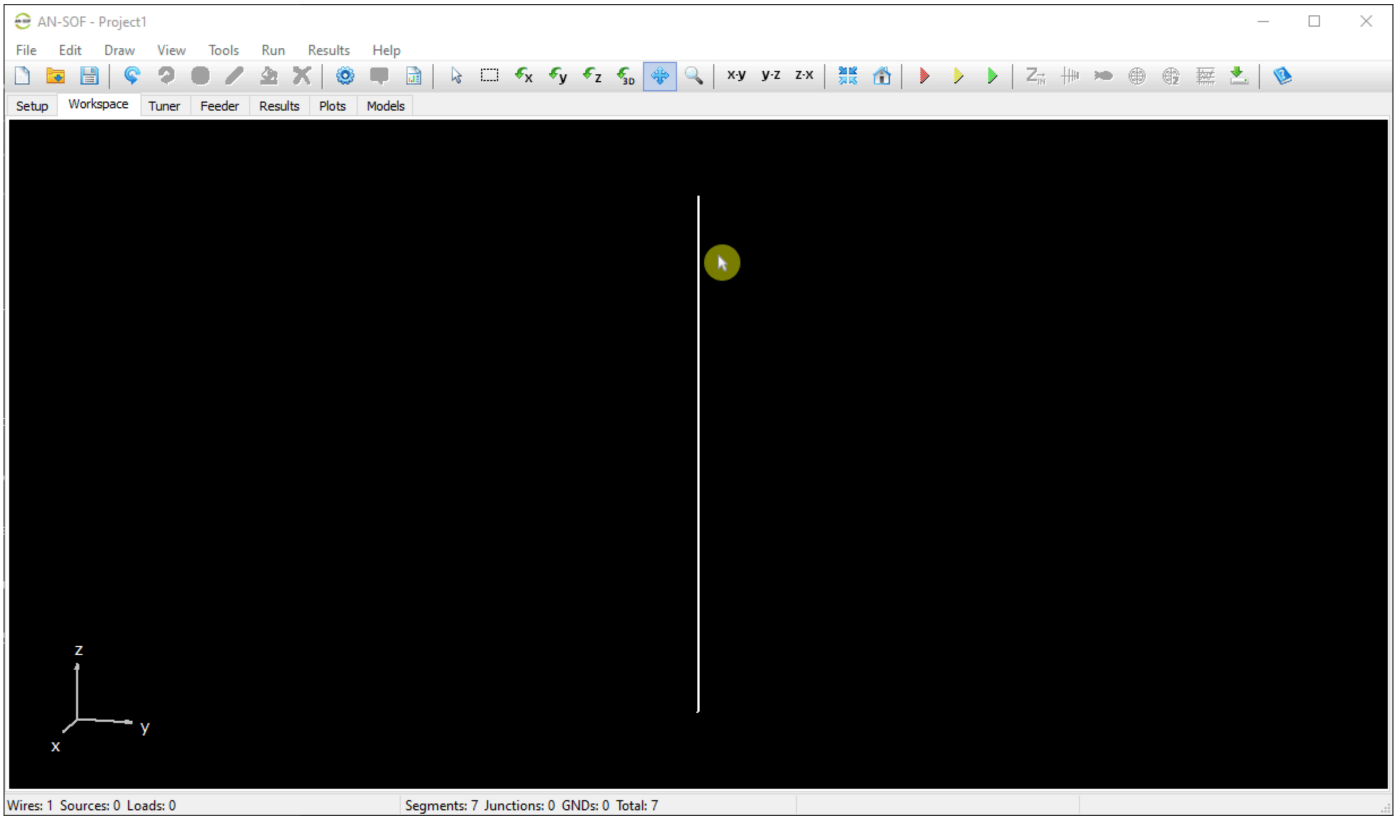


Fig. 1: *How to select a wire and access the Modify command.*

Deleting a Wire

To delete a wire, right-click on it and select **Delete** from the [pop-up menu](). This will remove the selected wire from the model. Note that any sources or loads attached to the wire will also be deleted.

Alternatively, you can delete a wire using the **Select Wire** button (arrow icon) in the main toolbar. Click the button, then left-click on the wire to select it. Once selected, go to **Edit > Delete** in the main menu. This option is only enabled when **a wire is selected**.

Figure 1 shows how to delete a linear wire using these options.

To enable a confirmation prompt before deleting a wire, go to **Tools > Preferences > Options** and check the option **"Ask before deleting wires."**

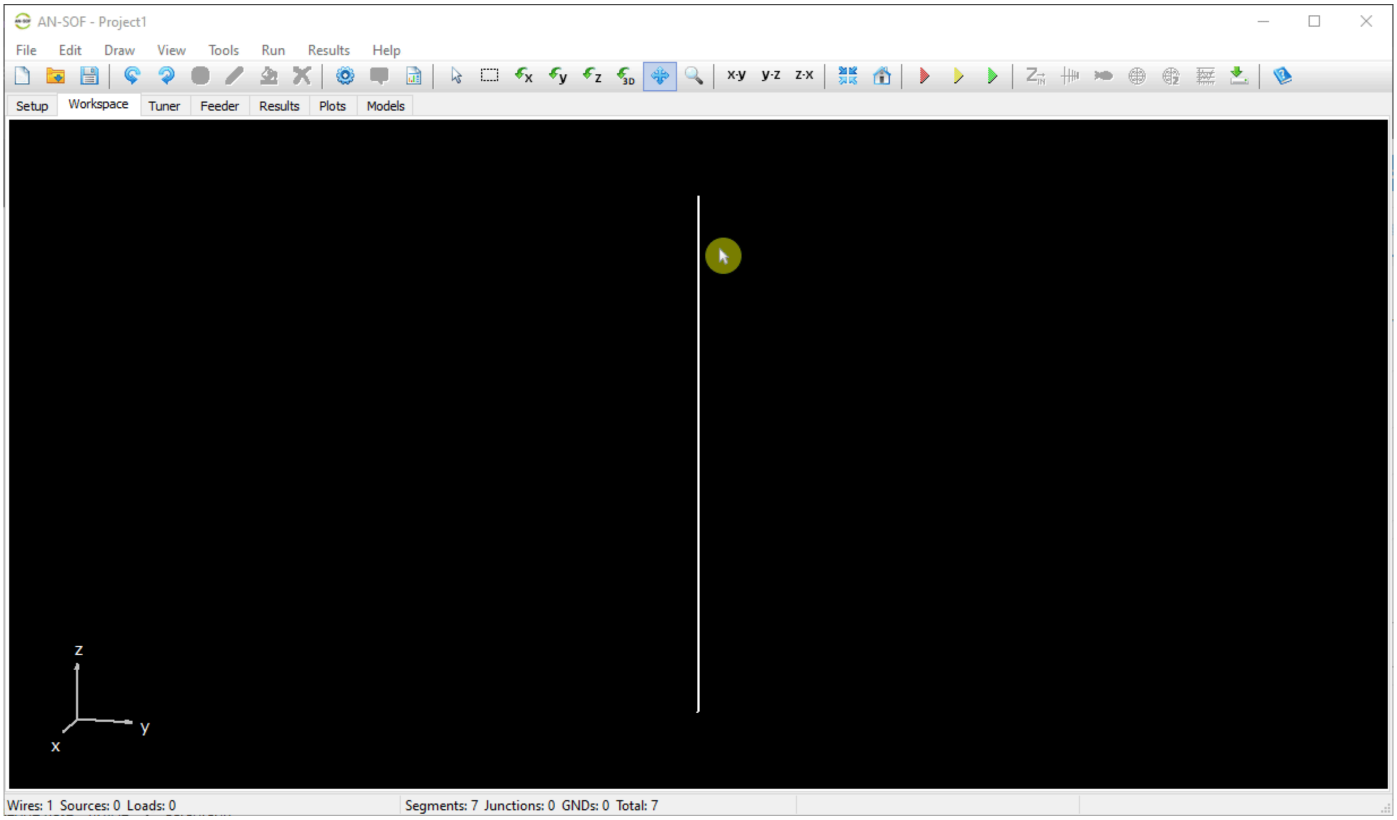


Fig. 1: *How to select a wire and access the Delete command.*

Selecting and Modifying Multiple Wires

AN-SOF allows you to simultaneously edit a group of wires. There are three ways to select multiple wires for editing:

1. **Using the Selection Box Tool**: Drag a rectangular box to select multiple wires.
2. **Selecting Wire by Wire**: Left-click on individual wires while holding the **Ctrl** key.
3. **Combination of Both Methods**: Use a mix of the Selection Box tool and wire-by-wire selection.

Using the Selection Box Tool

1. Click the **Selection Box** button on the main toolbar.
2. Left-click on the workspace and drag a box to select multiple wires (see **Fig. 1**).
   - All wires within the selection box will be highlighted in **light blue**.
   - Dragging the box **from top to bottom** selects only fully enclosed wires.
   - Dragging the box **from bottom to top** selects partially enclosed wires as well.
   - Enclosing already selected wires with the selection box will **deselect** them.
3. **To deselect all wires**, **double-click** on the screen or press **ESC**.

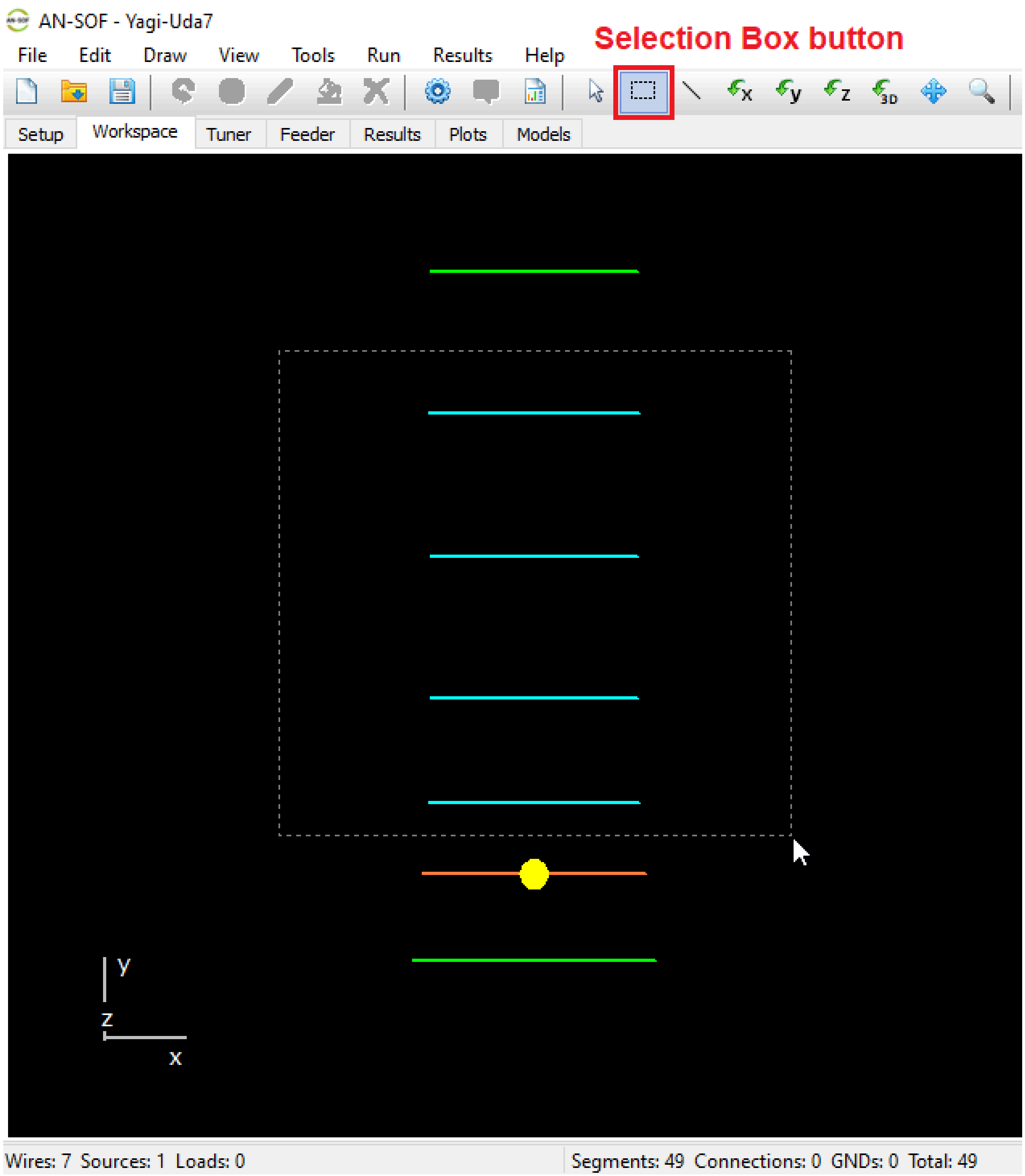


Fig. 1: *Selecting multiple wires using the Selection Box.*

Selecting Wire by Wire

1. Click the **Select Wire** button (arrow icon) on the main toolbar (**Fig. 2**).
2. Hold the **Ctrl** key and left-click on individual wires to select them.

   To **deselect** a wire, hold the **Ctrl** key and click on it again.
3. **To deselect all wires**, **double-click** on the screen or press **ESC**.

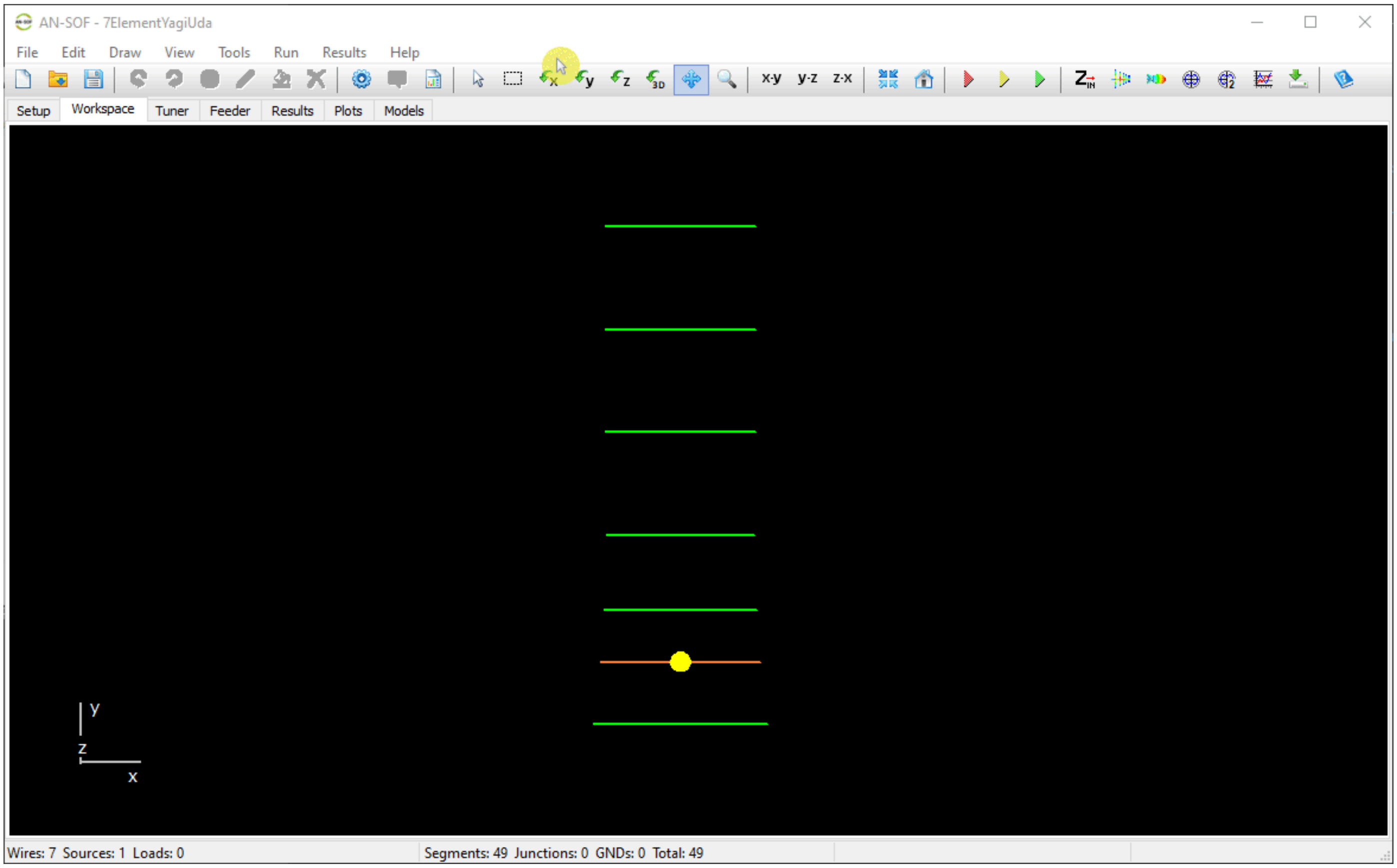
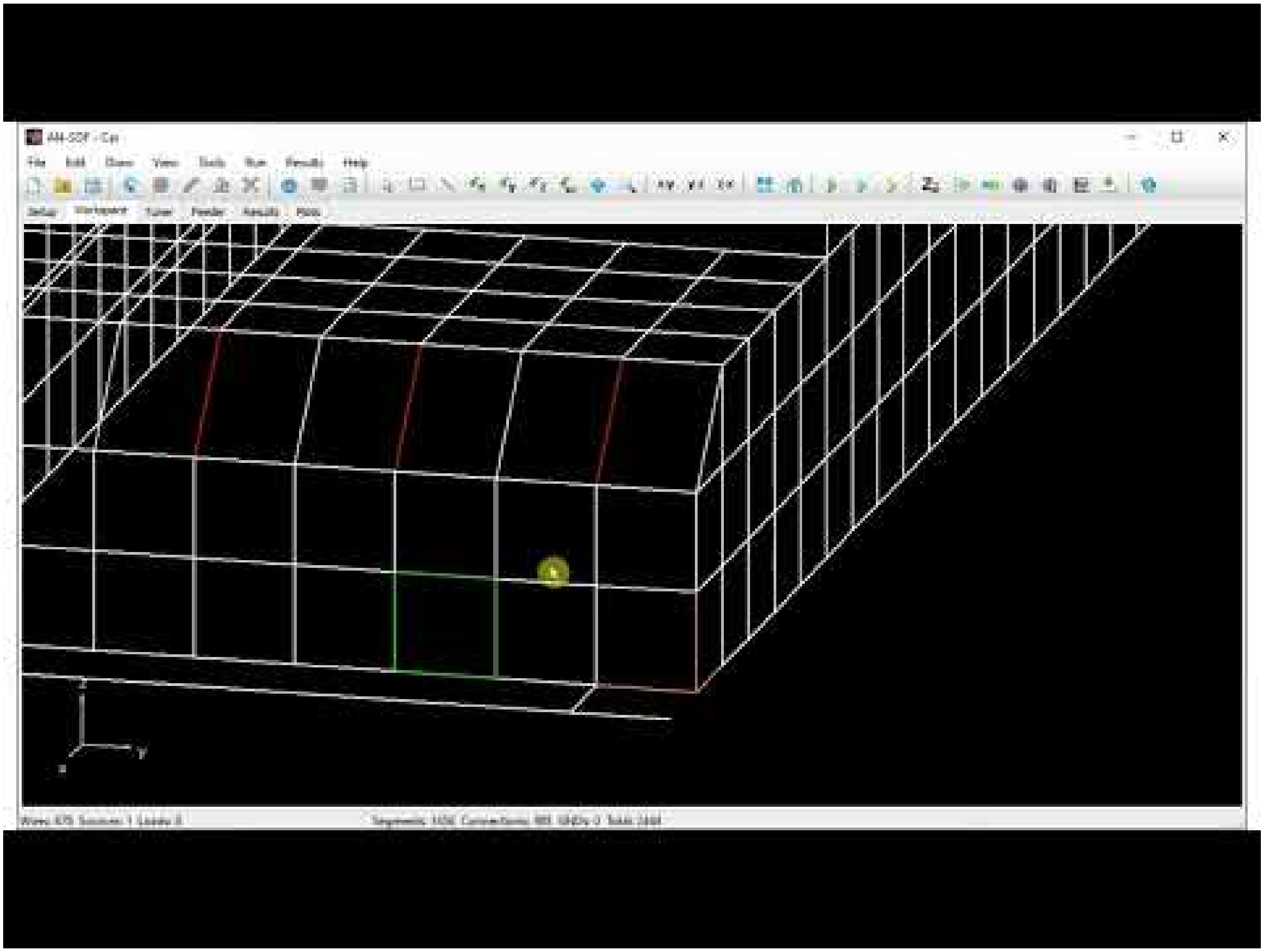


Fig. 2: *Selecting individual wires by clicking the "Select Wire" button on the toolbar and left-clicking on wires while holding the Ctrl key.*

Wire Selection Methods – Video Tutorial

Watch the video below to learn how to efficiently select and edit wires in your antenna models. This tutorial demonstrates various selection techniques, including the **'Select Wire'** and **'Selection Box'** tools, using **Ctrl + Left-Click** for multi-selection, and combining these methods to streamline your workflow.



[Watch on YouTube](#)

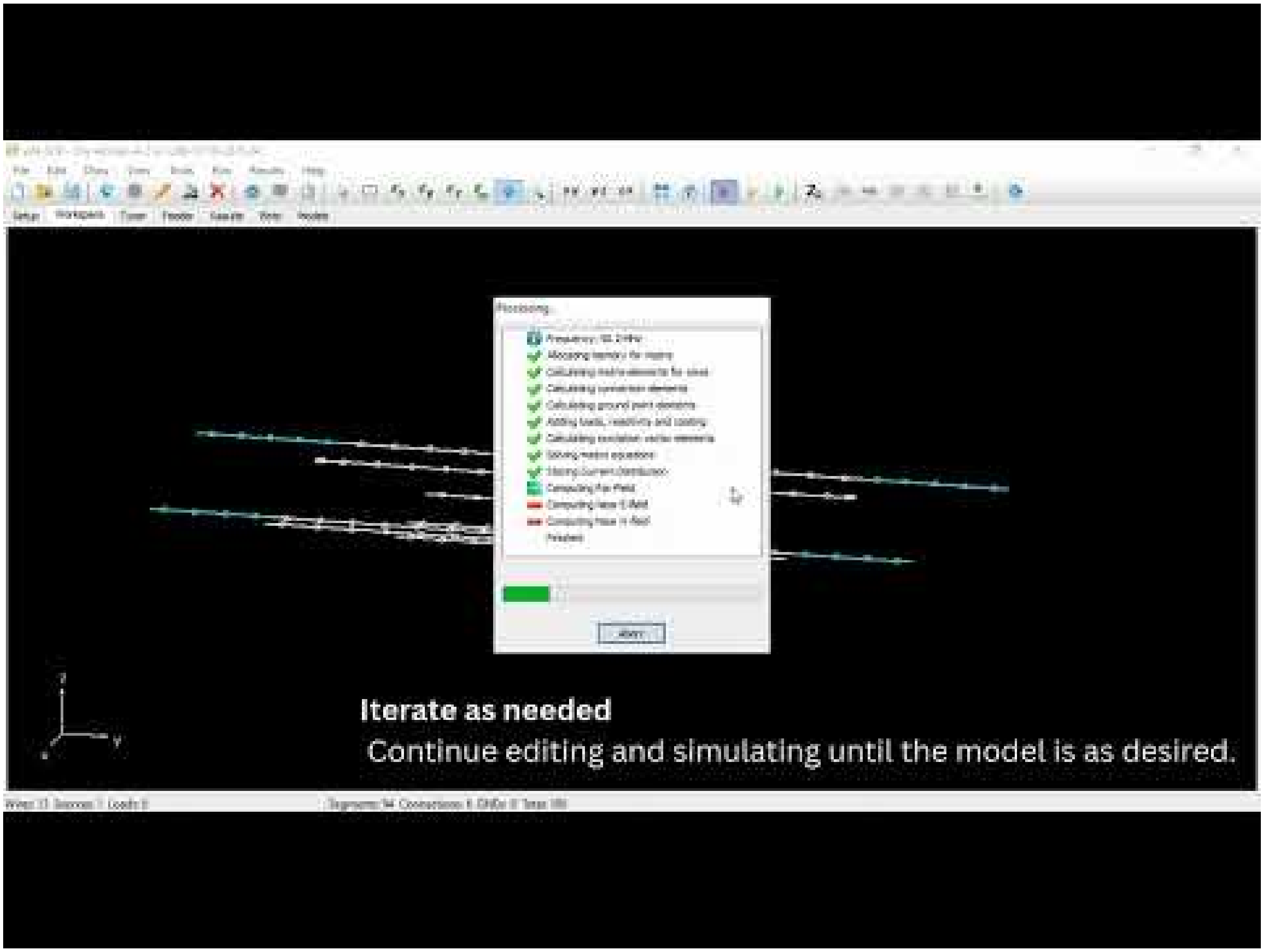
Tutorial: *Combining individual and multiple wire selection in AN-SOF*

Persistent Wire Selections

Wire selections in AN-SOF **persist** until you explicitly **deselect them**—by **double-clicking** or pressing **ESC**—making it easier to apply consecutive modifications and run simulations between changes without the need to reselect elements. Even after saving and reopening a project, selected wires remain highlighted, allowing you **to seamlessly resume editing exactly where you left off**.

AN-SOF also includes **Undo/Redo** functionality, with up to 10 levels of action history. These options are available via the **[Edit menu](#)** and the main toolbar, or using the shortcuts **Ctrl + Z (Undo)** and **Ctrl + Y (Redo)**. Together with persistent wire selections, this enables a smooth and efficient **iterative workflow**, where you can make changes, simulate, and refine your model until the desired outcome is achieved.

This process is illustrated in the video below and the accompanying flowchart (**Fig. 3**), with a detailed explanation provided beneath the diagram.



[Watch on YouTube]

Iterative Workflow in AN-SOF: *Combining persistent wire selection, undo/redo, and simulations to streamline modeling until the desired result is achieved.*

Step-by-Step Iterative Workflow

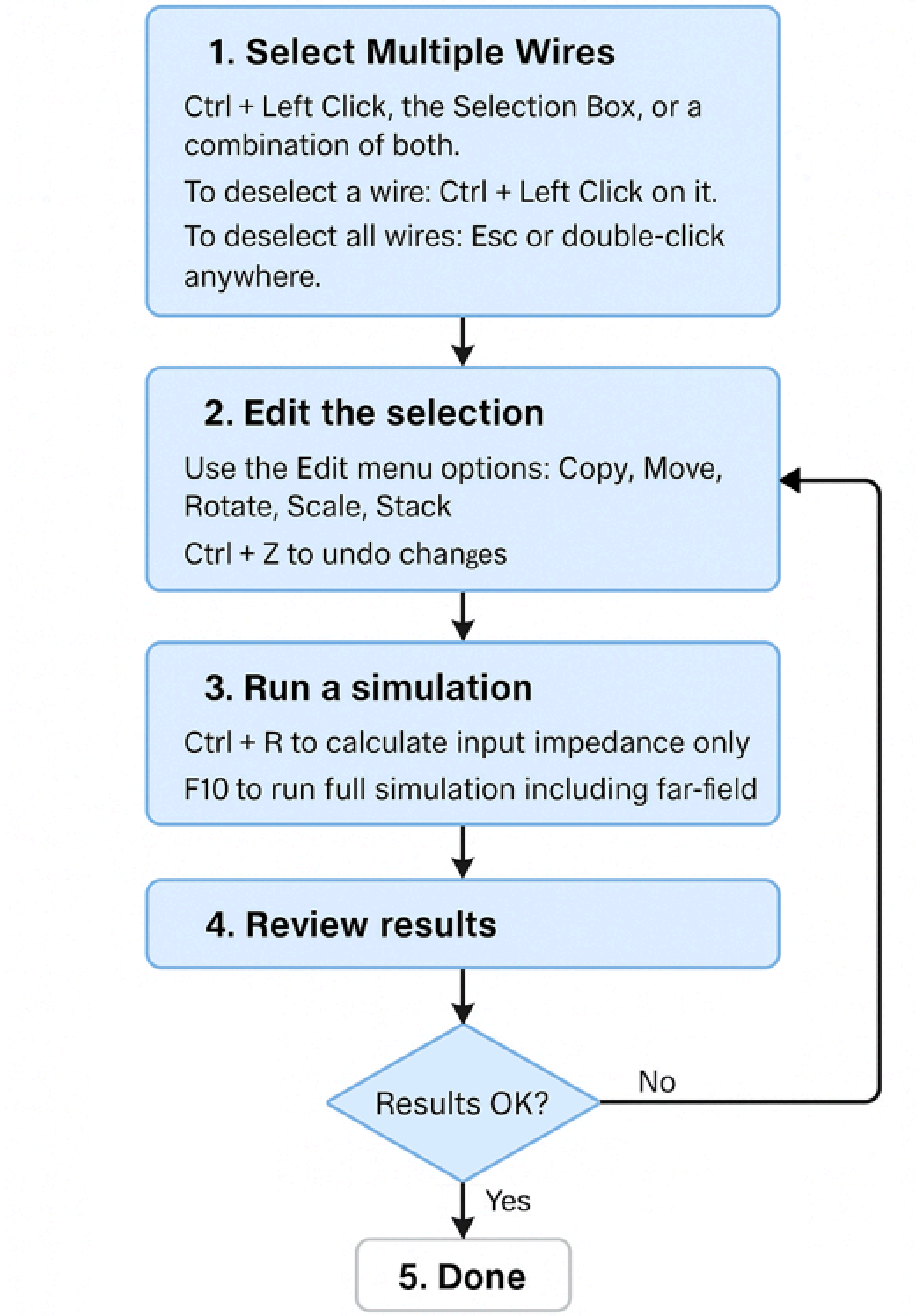


Fig. 3: *Flowchart of the iterative workflow combining multiple wire selection and editing to run simulations repeatedly until the desired result is achieved.*

Select Multiple Wires

- Use **Ctrl + Left Click**, the **Selection Box**, or both.
- Selected wires are highlighted in *light blue*.
- To deselect a wire: **Ctrl + Left Click** on it.

- To deselect all wires: Press **ESC** or **double-click** anywhere on the workspace.

Edit the Selection

Use the **Edit menu** options: **Modify**, **Move, Rotate, Scale**, **Copy, Stack**.

Run a Simulation

- Press **Ctrl + R** to calculate **input impedance only**.
- Press **F10** to run the **full simulation** including **far-field** results.

Review Results

Switch to the **Results** and **Plots** tabs.

Continue Editing

- Selected wires remain highlighted after simulation.
- Resume scaling, moving, rotating, etc., without reselecting.
- Use **Ctrl + Z** / **Ctrl + Y** to **undo/redo** changes.

Iterate as Needed

Edit, simulate, and fine-tune your model continuously until you achieve the desired result.

Improved View Adjustments

Panning, rotating, and zooming the view now retain the selection, streamlining the workflow.

Modifying the Selected Wires

Once multiple wires have been selected, navigate to **Edit > Modify** in the main menu to modify them. The **Modify** command opens a dialog box (see **Fig. 4**) with three tabs: **Attributes**, **Materials**, and **Sources/Loads**. Use the checkboxes to specify which parameters you want to modify.

In the **Attributes tab**, the **Segments per Wire** and **Segments per Wavelength** options allow for bulk editing of wire segments. These options are mutually exclusive:

- **Segments per Wire** sets a fixed number of segments for all selected wires.
- **Segments per Wavelength** sets the number of segments for each wire based on its length in wavelengths, using the shortest wavelength corresponding to the highest frequency specified.

Entering **“0”** (zero) in the **Segments per Wire** field will automatically set the number of segments for each wire based on **10 segments per wavelength**.

In the **Sources/Loads tab**, you can remove sources and loads in bulk by selecting **“Delete Sources”** or **“Delete Loads”**.

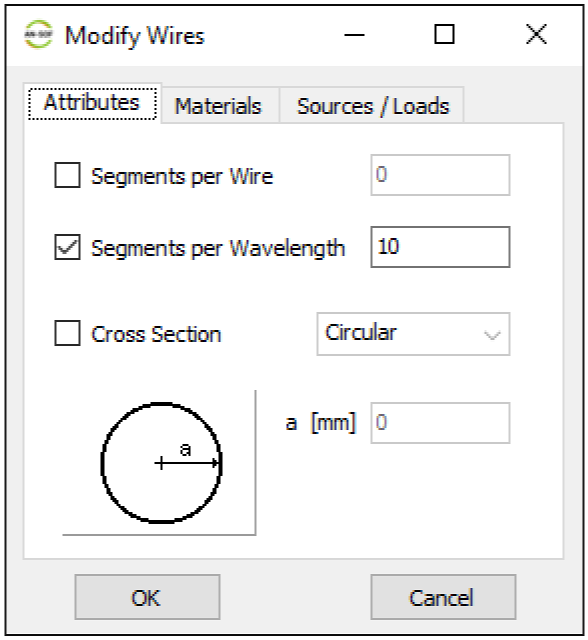


Fig. 4(a): *The Attributes tab in the Modify Wires dialog box.*

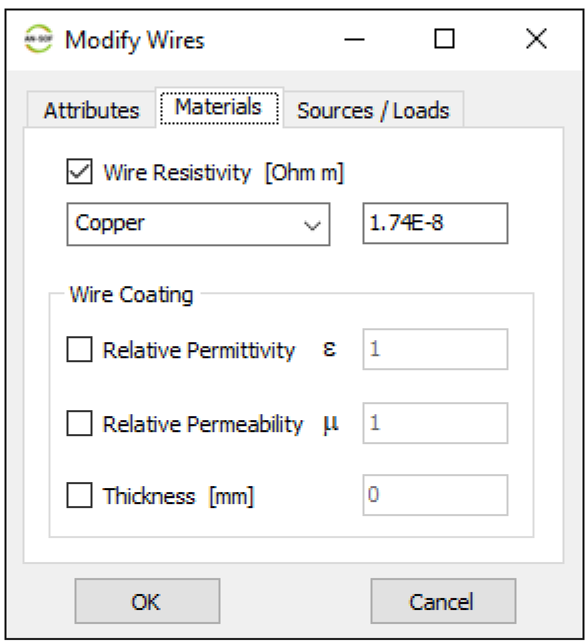


Fig. 4(b): *The Materials tab in the Modify Wires dialog box.*

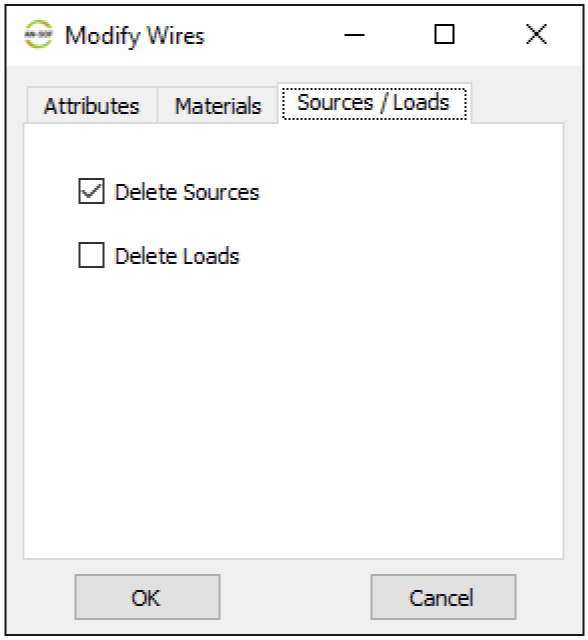


Fig. 4(c): *The Sources/Loads tab in the Modify Wires dialog box.*

If you need to manipulate the **position or arrangement of selected wires**—such as moving, rotating, scaling, copying, or stacking them—rather than editing their attributes like the number of segments, cross-section, or material properties, refer to the following articles:

Moving, Rotating, and Scaling Wires

Copying and Stacking Wires

Deleting Multiple Wires

You can delete multiple wires at once by first selecting them using one of the following methods:

- **Selection Box** – Click and drag a rectangular box around the wires you want to select.
- **Wire-by-Wire** – Hold down the **Ctrl** key and left-click on each wire individually.
- **Combination** – Combine both methods for more flexible selection.

For step-by-step instructions on selecting multiple wires, see the article:

Selecting and Modifying Multiple Wires

Once the wires are selected, you can delete them using one of the following methods:

- Go to **Edit > Delete** in the main menu
- Press **Ctrl + Del** on your keyboard
- Click the **Delete** button in the toolbar

Figure 1 shows how to select three wires using the **Select Wires** tool (arrow icon) and **Ctrl + left-click**, followed by clicking the **Delete** button in the toolbar.

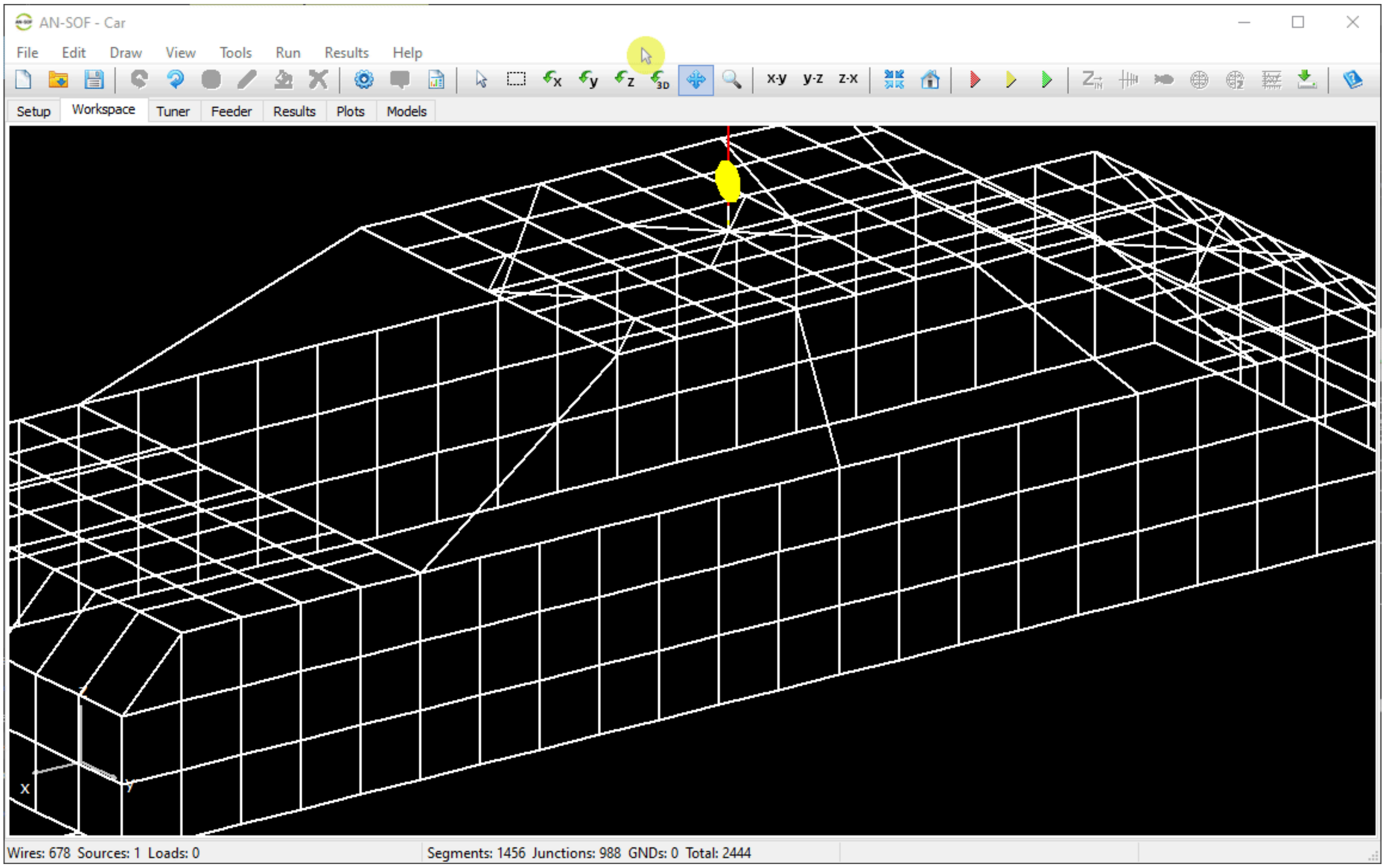


Fig. 1: *Selecting multiple wires using Ctrl + left-click, followed by pressing the Delete button on the toolbar.*

To enable a confirmation prompt before deleting wires, go to **Tools >** **[Preferences](#)** **> Options** and check the option **"Ask before deleting wires."**
Wire Color

To change the color of a wire, right-click on it to bring up the **[pop-up menu](#)**. Select the **Wire Color** command to open a dialog box where you can choose a color for the wire. This command is enabled only when **[a wire is selected](#)**.

Alternatively, you can access the **Wire Color** command by:

1. Clicking the **Select Wire** button (arrow icon) on the toolbar.
2. Left-clicking on the wire to select it.
3. Navigating to **Edit > Wire Color** in the main menu.

The **Wire Color** command is also available as a button on the toolbar.

To change the color of a **group of wires** (**Fig. 1**):

1. Select the wires using the **Selection Box** or by selecting them individually (as explained in the section **[Selecting and Modifying Multiple Wires](#)**).
2. Navigate to **Edit > Wire Color** in the main menu.

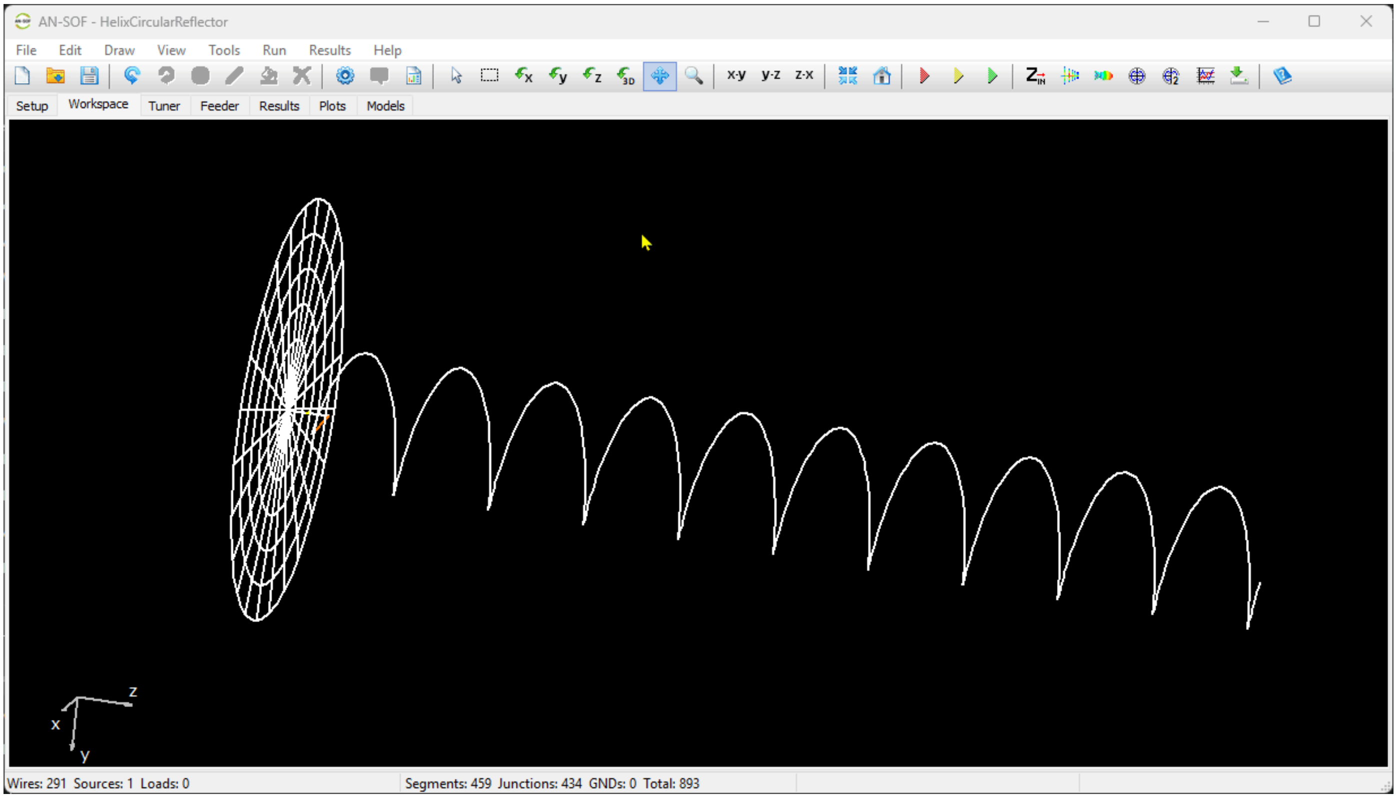


***Fig. 1:** Demonstration of color editing for an individual wire (a helix) and groups of wires (a circular ground plane).*

Wire Properties

Right-clicking on a wire displays a [pop-up menu](), where you can select the **Wire Properties** command.

Alternatively, you can access the **Wire Properties** command by:

1. Clicking the **Select Wire** button (arrow icon) on the toolbar.
2. Left-clicking on the wire to select it.
3. Navigating to **Edit > Wire Properties** in the main menu.

The **Wire Properties** command is also available as a button on the toolbar.

Executing the **Wire Properties** command opens the **Wire Properties** window, which contains three tabs: **Geometry**, **Attributes**, and **Materials**. This window is designed for **viewing** wire properties only. To edit a wire, refer to the section **Modifying a Wire**.

The Geometry Tab

This tab displays the geometrical properties of the selected wire (see **Fig. 1**), including:

- **Start Point**: Cartesian coordinates of the wire's start point.
- **End Point**: Cartesian coordinates of the wire's end point.
- **Wire Length**: Length of the wire.

- **Segment Length**: Length of a wire segment. For curved wires with non-uniform segments, this is the average segment length.
- **Shortest Wavelength (λ)**: Wavelength corresponding to the **highest frequency** specified in the **Frequency panel**.
- **Wire Length/λ**: Wire length measured in wavelengths (based on the shortest wavelength).
- **Segment Length/λ**: Length of a wire segment in wavelengths (based on the shortest wavelength).
- **Segments Per Wavelength**: Number of segments the wire would have if its length were one wavelength. This is the inverse of the segment length measured in wavelengths: 1/(Segment Length/λ).
- **Hallen's Parameter (Ω)**: A parameter that measures wire thickness, defined as $\Omega = 2 \ln(L/a)$, where L is the wire length and a is the wire radius.

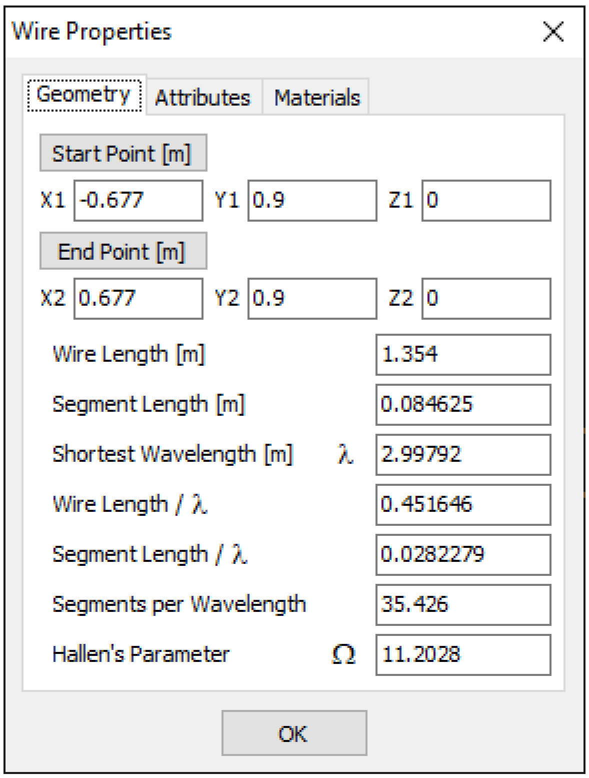


Fig. 1: *Wire Properties window – Geometry tab, displaying the geometrical properties of the selected wire.*

The Attributes Tab

This tab displays the electrical properties of the selected wire (see **Fig. 2**), including:

- **Number of Segments**: Number of segments into which the wire is divided.
- **Number of Sources**: Number of sources placed on the wire.
- **Number of Loads**: Number of loads placed on the wire.
- **Cross-Section**: Type and dimensions of the wire's cross-section.
- **Equivalent Radius**: **Equivalent radius of the cross-section**.

- **Equivalent Radius/λ**: Equivalent radius as a fraction of the shortest wavelength.
- **Thin-Wire Ratio**: Ratio of the wire diameter to the segment length. This must be less than 3 when the **Exact Kernel** option is unchecked in the **Settings panel** of the **Setup** tab. If the **Exact Kernel** option is checked, any value of the thin-wire ratio is allowed. For non-circular cross-sections, the wire diameter is twice the equivalent radius.

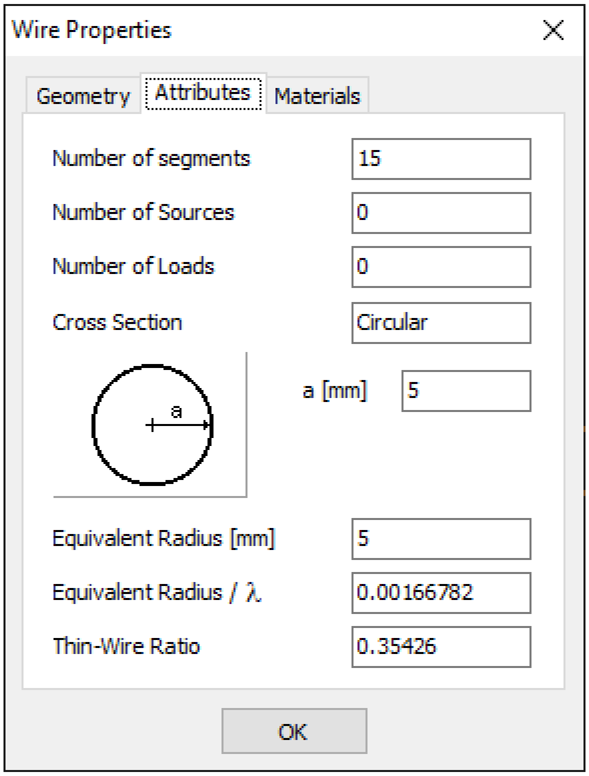


***Fig. 2:** Wire Properties window – Attributes tab, displaying the segmentation, number of sources and loads, and cross-section type of the selected wire.*

The Materials Tab

This tab displays the material properties of the selected wire (see **Fig. 3**), including:

- **Wire Resistivity**: Resistivity of the wire in [Ohm·m]. If the wire is coated, this refers to the resistivity of the internal conductor.
- **Wire Coating**: Parameters of the wire's coating shield.
- **Relative Permittivity**: Permittivity (dielectric constant) of the coating material relative to the permittivity of vacuum.
- **Relative Permeability**: Magnetic permeability of the coating material relative to the permeability of vacuum.
- **Thickness**: Thickness of the coating shield.

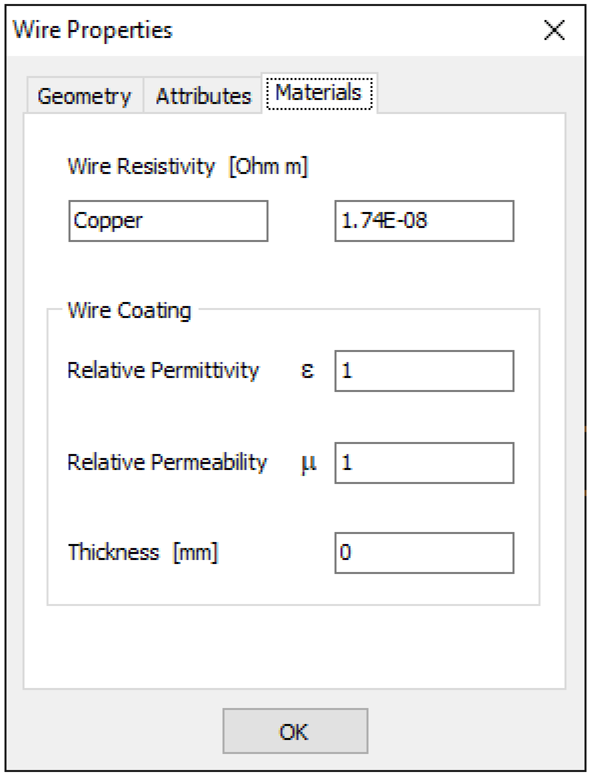


Fig. 3: *Wire Properties window – Materials tab, displaying the material parameters of the conductive wire and its coating shield or insulation.*

Connecting Wires

A wire junction is automatically established when the coordinates of a wire end match the end coordinates of a previously added wire. Wire junctions are essential to satisfy **Kirchhoff's current law** at the connection point.

Figure 1 illustrates the correct and incorrect ways to connect two wires. To connect the end of **wire 1** to a point on **wire 2** that is not an end, you must split **wire 2** into two wires. This means **three wires** will be needed instead of two to make the connection.

Fig. 1: *Incorrect and correct methods for connecting wires.*

Connecting Wires by Copying and Pasting Ends

Two wires can be connected by copying and pasting their ends:

1. **Right-click** on a wire to select it.
2. From the **pop-up menu**, choose **Copy Start Point** or **Copy End Point** to copy the coordinates of the wire end to an internal clipboard.
3. Paste the copied point in the **Draw** dialog box when adding a new wire by clicking the **From Point** button, located just above the start point coordinates **(X1, Y1, Z1)** (see **Fig. 2**).

Note:

- When a wire is selected in the workspace, it will be highlighted in **light blue**, and an **arrow** will appear at the **End Point**. The opposite end is the **Start Point**.
- Wire orientation serves as the electrical reference for the phase of the current distribution. However, it does not affect observables such as **input impedance** (and thus **VSWR** or $\mathbf{S_{11}}$) or **radiation pattern metrics** (directivity, gain, efficiency, etc.).

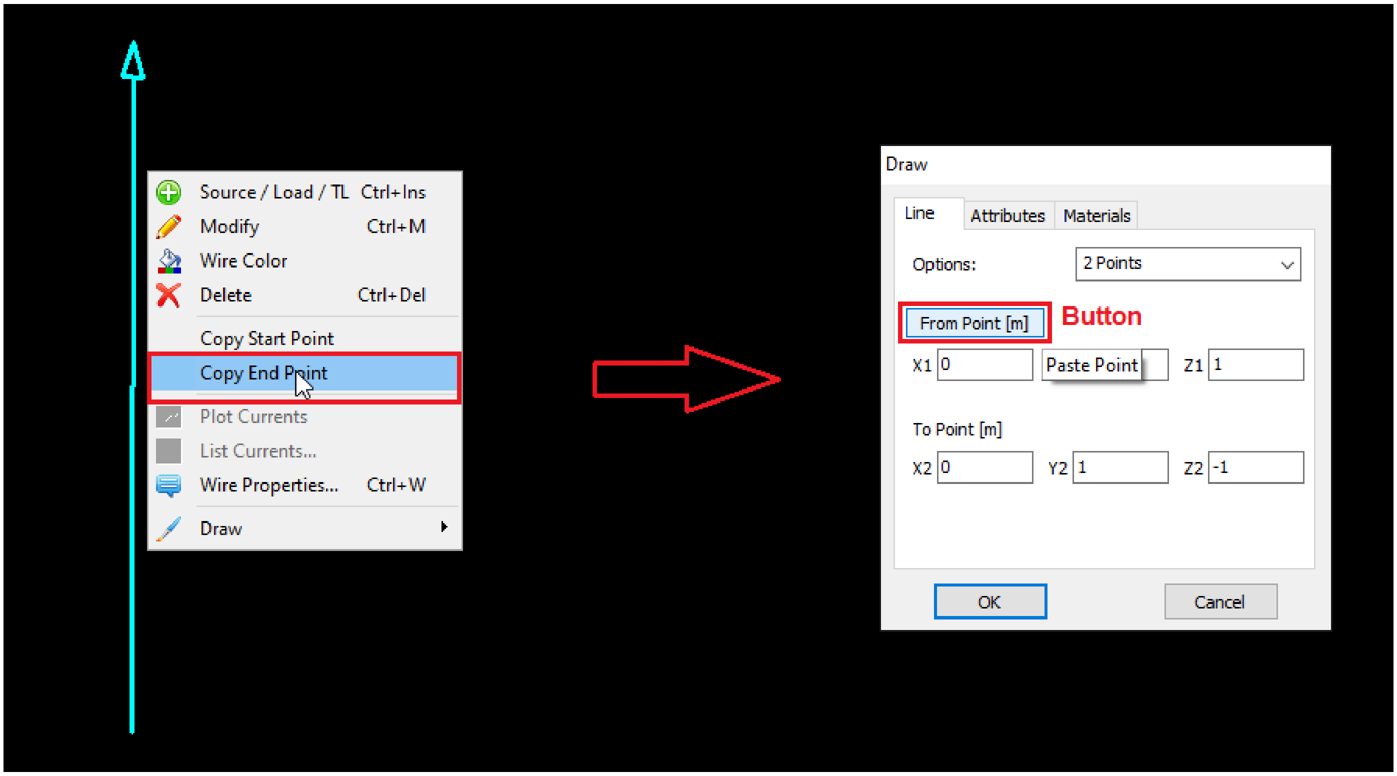
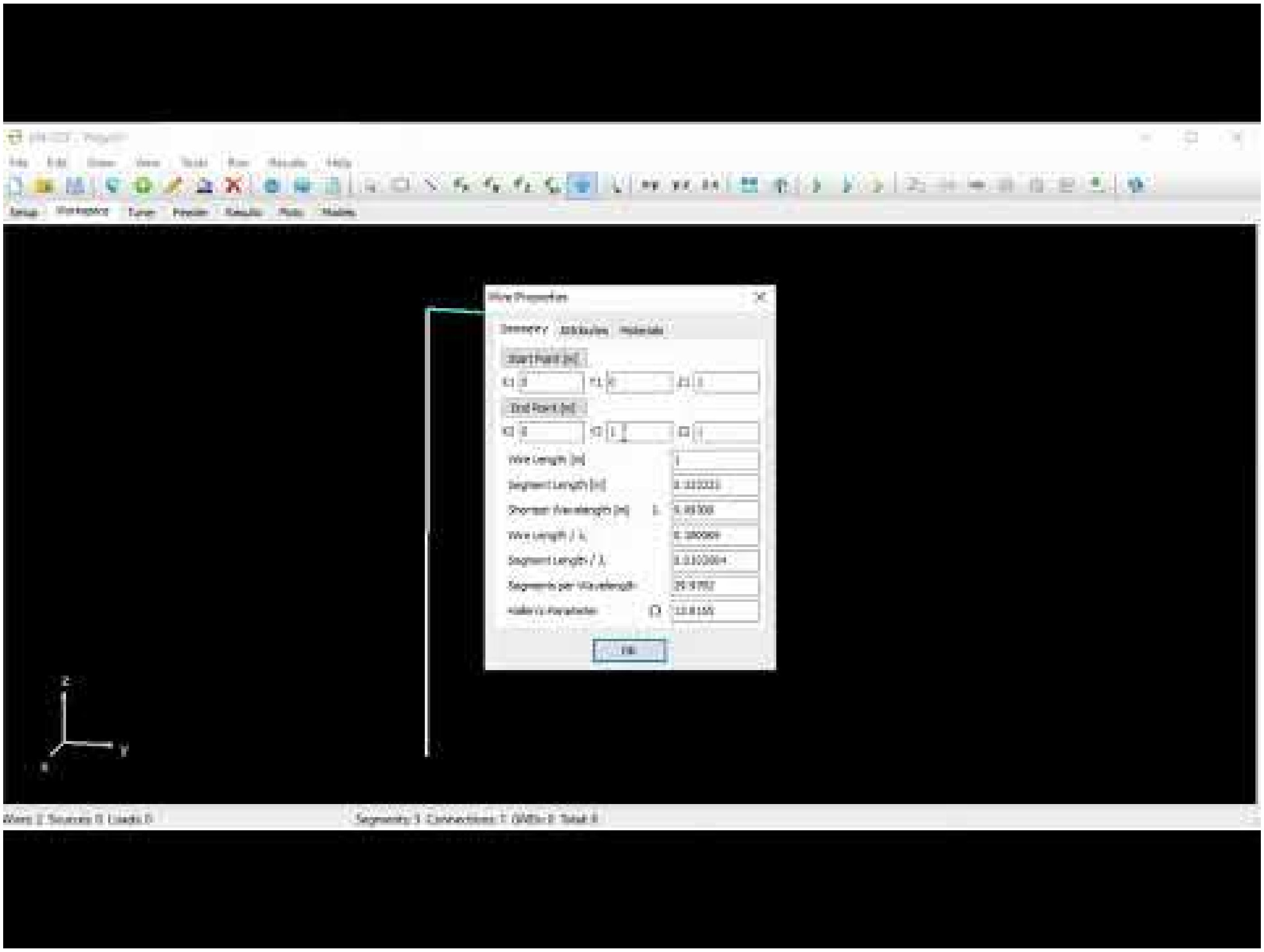


Fig. 2: *Connecting wires by copying and pasting ends.*

Copying Start and End Points via the Wire Properties Window

The **Start** and **End Points** of a wire can also be copied to the AN-SOF internal clipboard for use in a dialog box to draw another wire, using buttons in the **Wire Properties window**. This procedure is demonstrated in the video below and explained in detail in the following section.



[Watch on YouTube](#)

Procedure for Connecting Two Wires at Their Ends

This procedure demonstrates how to connect the **Start** or **End Point** of **wire #1** to the **Start Point** of **wire #2**:

1. **Right-click** on **wire #1** to display a **[pop-up menu](#)**.
2. Choose the **Copy Start Point** or **Copy End Point** command. This command is also available in the **[Wire Properties window](#)** of the selected wire (see **Fig. 3**).
3. In this example, **wire #2** will be a **[Line](#)**. Navigate to **Draw > Line** in the main menu to open the **Draw** dialog box for the Line.
4. Click the **From Point** button to paste the copied point (see **Fig. 4**). Then, complete the definition of **wire #2**.

Using this procedure, any number of wires can be connected at the same point.

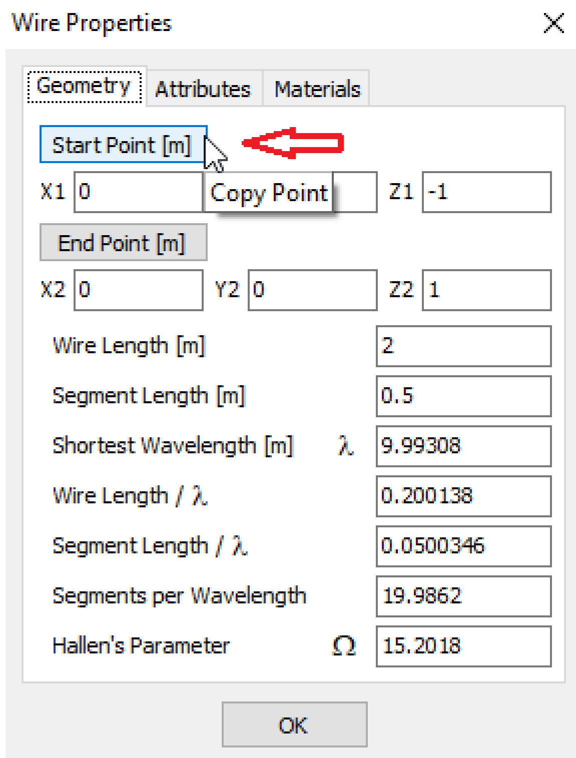


Fig. 3(a): *Wire Properties window for wire #1 – Click the Start Point button to copy this wire end.*

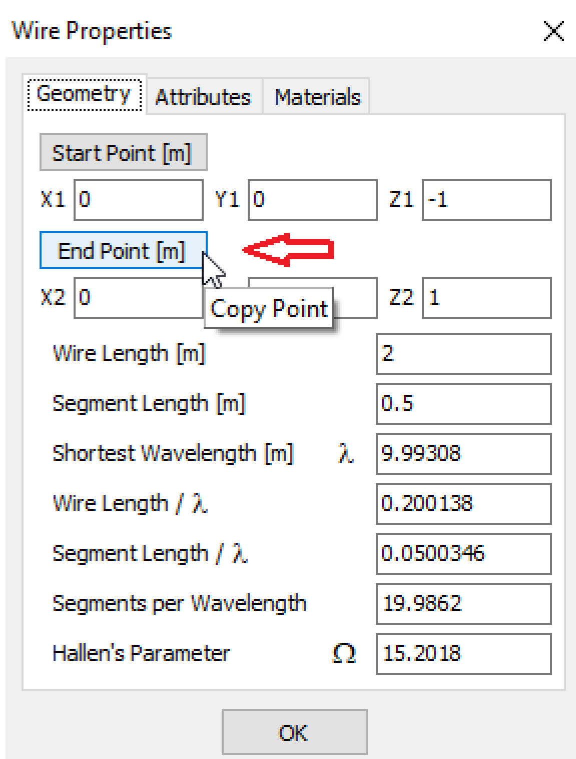


Fig. 3(b): *Wire Properties window for wire #1 – Click the End Point button to copy this wire end.*

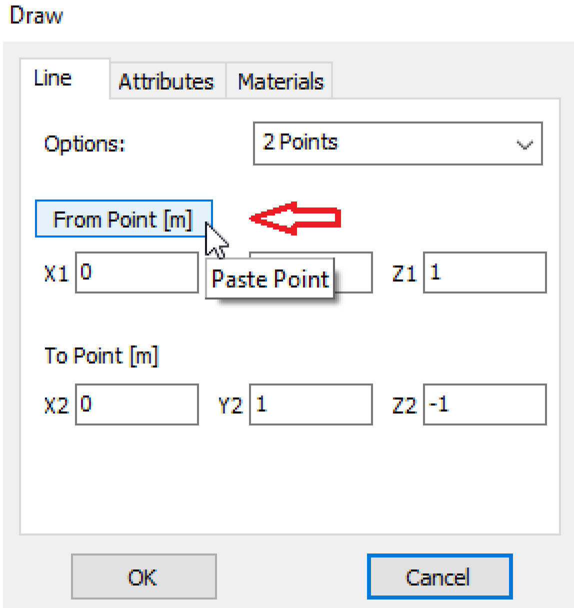


Fig. 4: *Draw dialog box for wire #2 – Click the From Point button to paste the copied end of wire #1.*

Moving, Rotating, and Scaling Wires

After drawing the wire structure, you may need to adjust the position, orientation, or size of individual wires or groups of wires. To modify wires, **you must first select them**.

Selecting Wires

1. **Using the Selection Box**:
   - Click the **Selection Box** button on the toolbar.
   - Drag a box using the mouse while holding the left button to enclose the wires you want to modify (see **Fig. 1**).
     - Dragging the box **from top to bottom** selects only fully enclosed wires.
     - Dragging the box **from bottom to top** selects partially enclosed wires as well.
     - Enclosing already selected wires with the selection box will **deselect** them.
2. **Selecting Wire by Wire**:
   - Click the **Select Wire** button (arrow icon) on the toolbar.
   - Hold the **Ctrl** key and left-click on individual wires to select them.
     To deselect a wire, hold the **Ctrl** key and click on it again.
   - **To deselect all wires**, **double-click** on the screen or press **ESC**.

You can also combine the **Selection Box** method with selecting wires individually.

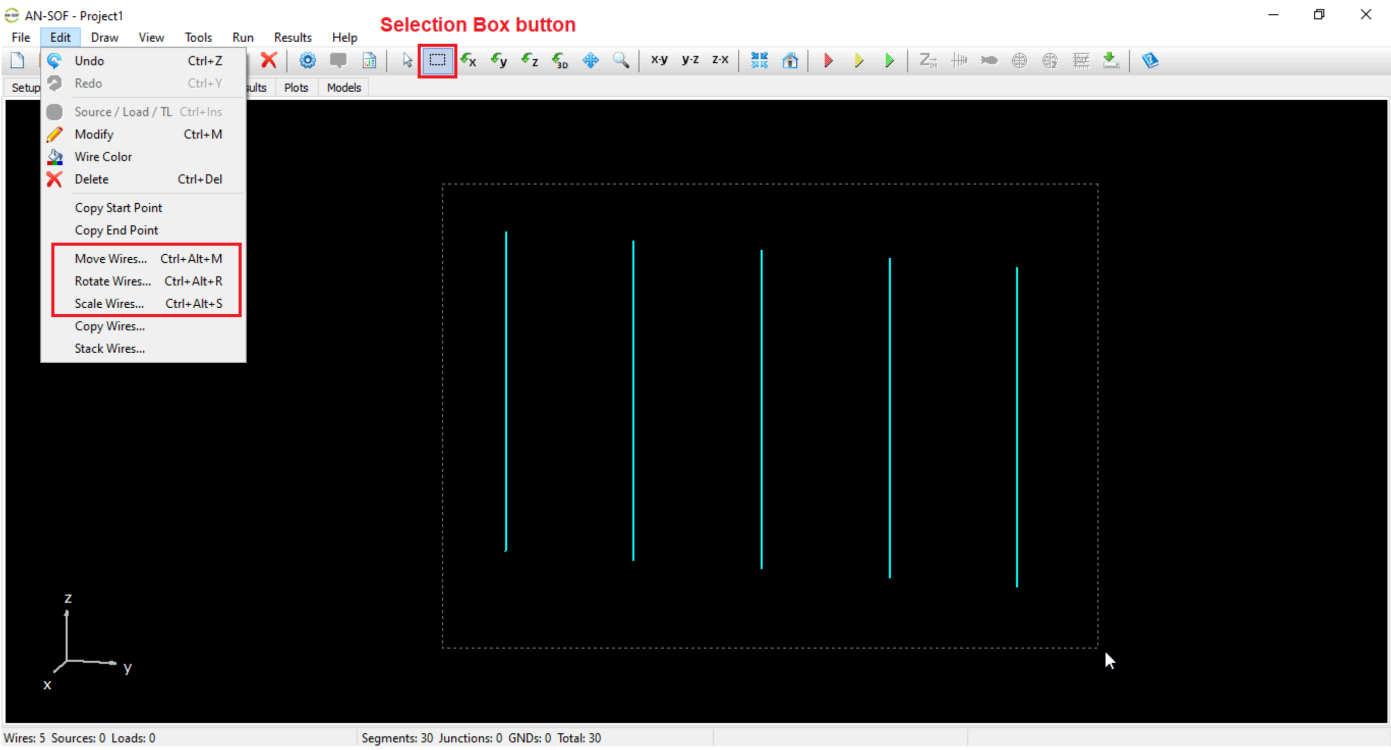


***Fig. 1:** "Selection Box" button on the toolbar to select a group of wires and commands in the Edit menu to move, rotate, and scale the selected wires.*

Transforming Selected Wires

Once the wires are selected, navigate to the **Edit** menu and choose one of the following commands:

Move Wires

Opens the **Move Wires** dialog box (see **Fig. 2**), allowing you to move the selected wire or group of wires to a new position. You can specify the shift along the **X**, **Y**, and **Z** coordinates.

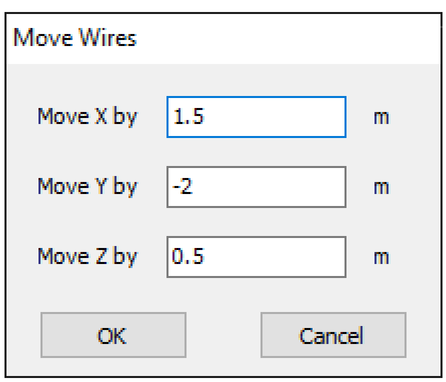


***Fig. 2:** Move Wires dialog box.*

Rotate Wires

Opens the **Rotate Wires** dialog box (see **Fig. 3**), enabling you to rotate the selected wire or group of wires around a chosen axis. In addition to the Cartesian axes (**X**, **Y**, and **Z**), the **Custom** option allows you to define a rotation axis using spherical coordinates (**Theta**, **Phi**). You can also set the **Rotation Center** to rotate around a point other than the origin.

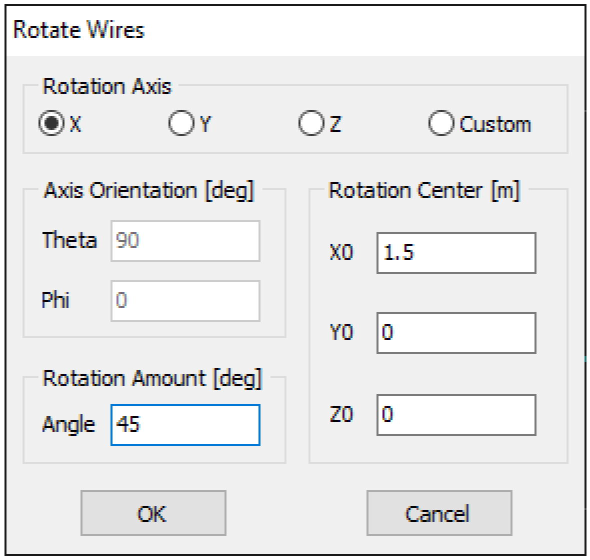


Fig. 3: *Rotate Wires dialog box.*

Scale Wires

Opens the **Scale Wires** dialog box (see **Fig. 4**), providing the following scaling options:

1. **Single Factor**:
    - Apply a single scale factor to all point coordinates of the selected wires.
    - Optionally, scale the wire cross-section and coating thickness by the same factor by checking the corresponding boxes.
2. **Line Length**:
    - Apply scaling only to **linear wires**.
    - Enter a scale factor and specify an anchored point: the line's start point (**P1**) or end point (**P2**). This allows you to lengthen or shorten the line while keeping one end fixed.
3. **Advanced**:
    - Apply different scale factors for each Cartesian coordinate (**X**, **Y**, and **Z**).
    - Stretch or contract the selected wires along the direction of one of the Cartesian axes.

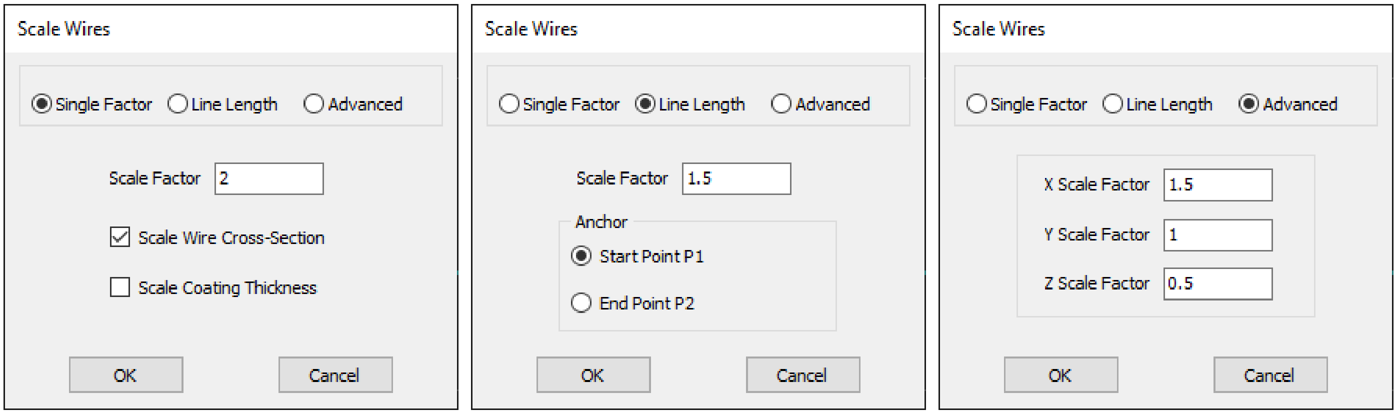


Fig. 4: *Scale Wires dialog box. (Left) "Single Factor" option. (Center) "Line Length" option. (Right) "Advanced" option.*

> **Note**
>
> Transmission lines fully enclosed by the selection box will be transformed along with the associated wires when using the **Move**, **Rotate**, or **Scale** transformations in the **Edit** menu. This ensures that transmission lines remain connected to wires selected via the **Selection Box**.

Copying and Stacking Wires

When drawing a wire structure, it's often necessary to copy wires from one position to another—for example, when creating an **antenna array**. To copy wires, **first select them** using one of the following methods:

1. Click the **Selection Box** button on the toolbar and drag a box with the mouse to enclose the wires you want to copy (see the **Moving, Rotating and Scaling Wires** section for details).
2. Hold the **Ctrl** key and **left-click** on wires to select them individually.
3. Combine methods 1 and 2 for more flexible selection.

For a full overview of wire selection techniques, see:

New Tools in AN-SOF: Selecting and Editing Wires in Bulk

In the **Edit** menu, you will find the following commands for copying the selected wires:

Copy Wires

Opens the **Copy Wires** dialog box (see **Fig. 1**), allowing you to copy the selected wire or group of wires.

- Specify the **number of copies** of the selected group of wires.
- The first copy will be offset from the original wire group based on the entered **X, Y, and Z offsets** and/or rotated around each axis according to the entered angles.
- Each subsequent copy will be offset and/or rotated relative to the previous copy.

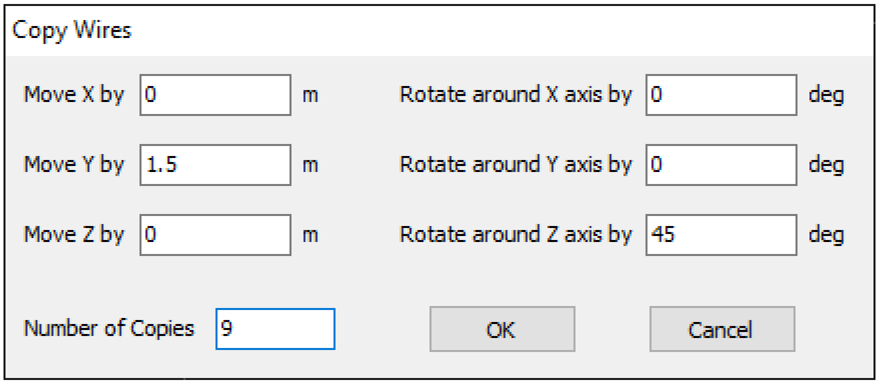


Fig. 1: *Copy Wires dialog box.*

Stack Wires

Opens the **Stack Wires** dialog box (see **Fig. 2**), allowing you to stack the selected wire or group of wires along a specified direction.

- Specify the **number of elements** in the stack. Each “element” consists of the selected wires, which could be a single wire or a group of wires.
- Define the **spacing** between the elements.

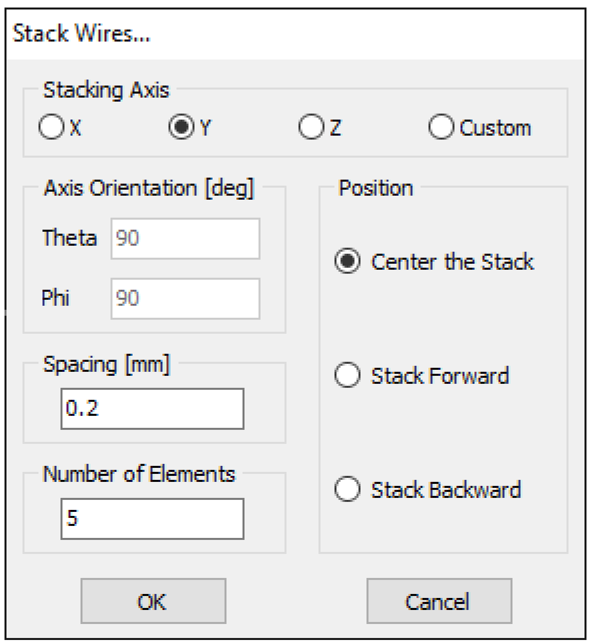


Fig. 2: *Stack Wires dialog box.*

Grids and Surfaces

Types of Grids and Surfaces

In AN-SOF, you can model metallic structures using either **wire frameworks** or **solid sheets**. Understanding the distinction between these two is key to accurate electromagnetic modeling:

- **Wire Grids:** These are frameworks composed of interconnected wires with gaps (holes) between them. The wires do not overlap; they meet at junctions to form a mesh.
- **Solid Surfaces:** These represent continuous, solid metal sheets without holes.

Both grids and surfaces can be constructed using straight or curved segments, allowing for highly complex geometries.

Available Geometries

To create these structures, navigate to **Draw > Wire Grid / Solid Surface** in the main menu, or use the **Drawing Panel** (**View > Drawing Panel**) for faster access.

Shape	Description
Patch	Creates a rectangular surface parallel to the XY-plane.
Plate	Creates a flat plate or a "bilinear surface" (a four-sided shape where corners may not be coplanar).
Disk	Creates a solid circular plate or a radial wire mesh.
Flat Ring	A disk with a central hole (annulus).
Cone	A standard conical structure ending in a single point.
Truncated Cone	A cone with the top portion removed, resulting in two circular faces of different radii.
Cylinder	A traditional cylindrical tube or solid rod representation.
Sphere	A full spherical shell modeled as a grid or surface.
Paraboloid	Essential for modeling parabolic reflectors and dish antennas.

Modeling Tips

- **The Drawing Panel:** We recommend keeping the Drawing Panel open if you are building complex reflectors. It saves several clicks when switching between different primitive shapes.
- **Grid Density:** When using wire grids to simulate solid surfaces, ensure the "hole" size is small relative to the wavelength (typically less than $\lambda/10$) to ensure the grid behaves like a solid shield.

Attributes of Grids and Surfaces

The **Attributes** tab in the **Draw** dialog box allows you to define how a geometric shape (like a Plate, Disc, or Sphere) is mathematically represented in the simulation. While the shapes differ, they all share the same attribute logic.

Defining the Structure: Wire Grid vs. Solid Surface

The most critical choice is made in the **Cross-Section** field (**Fig. 1**). This setting determines whether the object has “holes” or is a continuous sheet of metal.

- **Wire Grid:** Select **Circular** cross-section and set a radius (a>0). This creates a mesh of cylindrical wires with open gaps between them.
- **Solid Surface:** Select **Flat** or **Rectangular** cross-section. AN-SOF automatically adjusts the width of these “strip” wires so they touch edge-to-edge, covering the surface completely with no gaps.
  - *Flat:* Represents a surface with zero physical thickness.
  - *Rectangular:* Allows you to specify a physical **thickness** for the metal sheet.

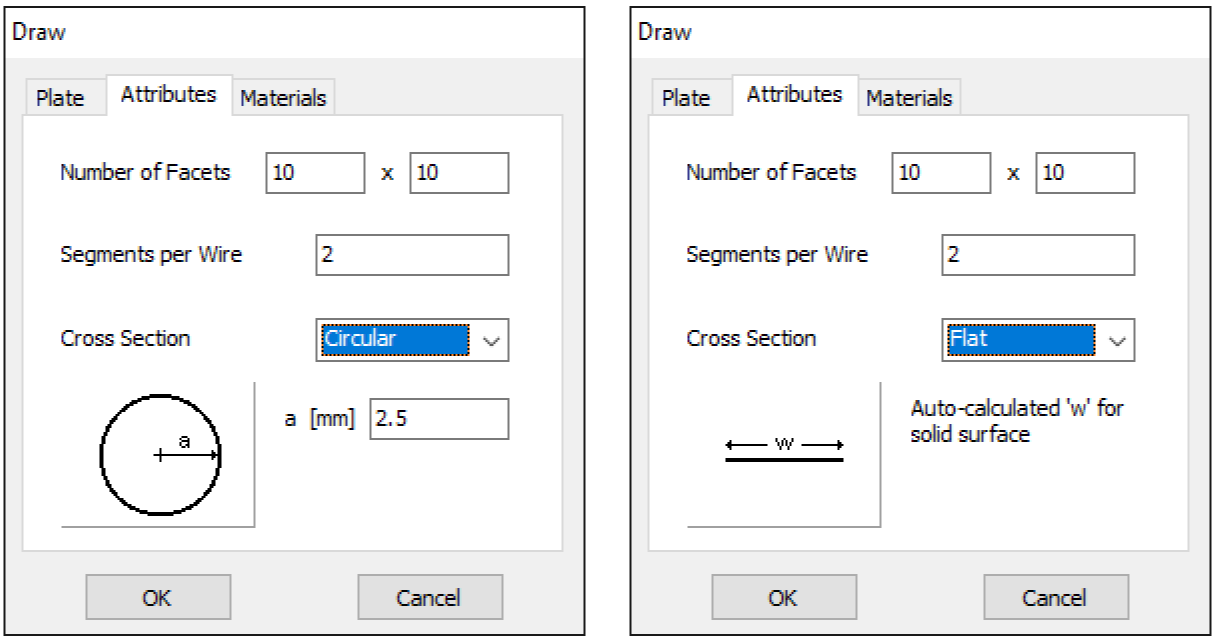


Fig. 1: Attributes page in the Plate Draw dialog box. Selection of Circular cross-section represents a wire grid, while Flat or Rectangular cross-section represents a solid surface.

Number of Facets

Facets are the “building blocks” of the grid or surface. They are typically quadrilaterals formed by four interconnected wires.

- **Grid Arrangement:** You define facets as a grid (e.g., 10×10 for a **Plate** or 6×12 for a **Disk**).
- **Automatic Calculation (Patch only):** Setting facets to **0x0** for a **Patch** allows AN-SOF to auto-calculate the density based on the highest frequency (targeting 10 segments per wavelength).

Segments per Wire

This defines how many computational segments exist on each individual wire that makes up the facets.

- **Manual Control:** Set a specific integer for custom precision.
- **Automatic:** Set to **zero** to let AN-SOF apply the 10-segments-per-wavelength rule.
- **Note on Patches:** The **Patch** tool does not have this option because its facets are always composed of single-segment wires.

Individual vs. Mass Editing

Once a grid or surface is created, you have two ways to modify it:

- **Mass Edit:** Refer to **Modifying a Grid/Surface** to change the attributes of the entire object at once (e.g., changing the radius of all wires in a mesh).
- **Individual Edit:** You can **right-click any single wire** within the mesh to change its specific properties or position without affecting the rest of the grid.

Modifying a Grid or Surface

In AN-SOF, wire grids and solid surfaces are treated as collections of individual wires or strips. Once created, they cannot be re-opened in the original "Draw" dialog box. Instead, you use **Massive Edition** tools to adjust the entire structure at once.

Selecting the Structure

To modify a grid or surface, you must first select the components you want to change:

- **The Selection Box:** Click the **Selection Box** button on the main toolbar.
- **Draw the Box:** Left-click and drag in the workspace to encompass the grid or surface. Selected wires will turn **light blue** (see **Fig. 1**).

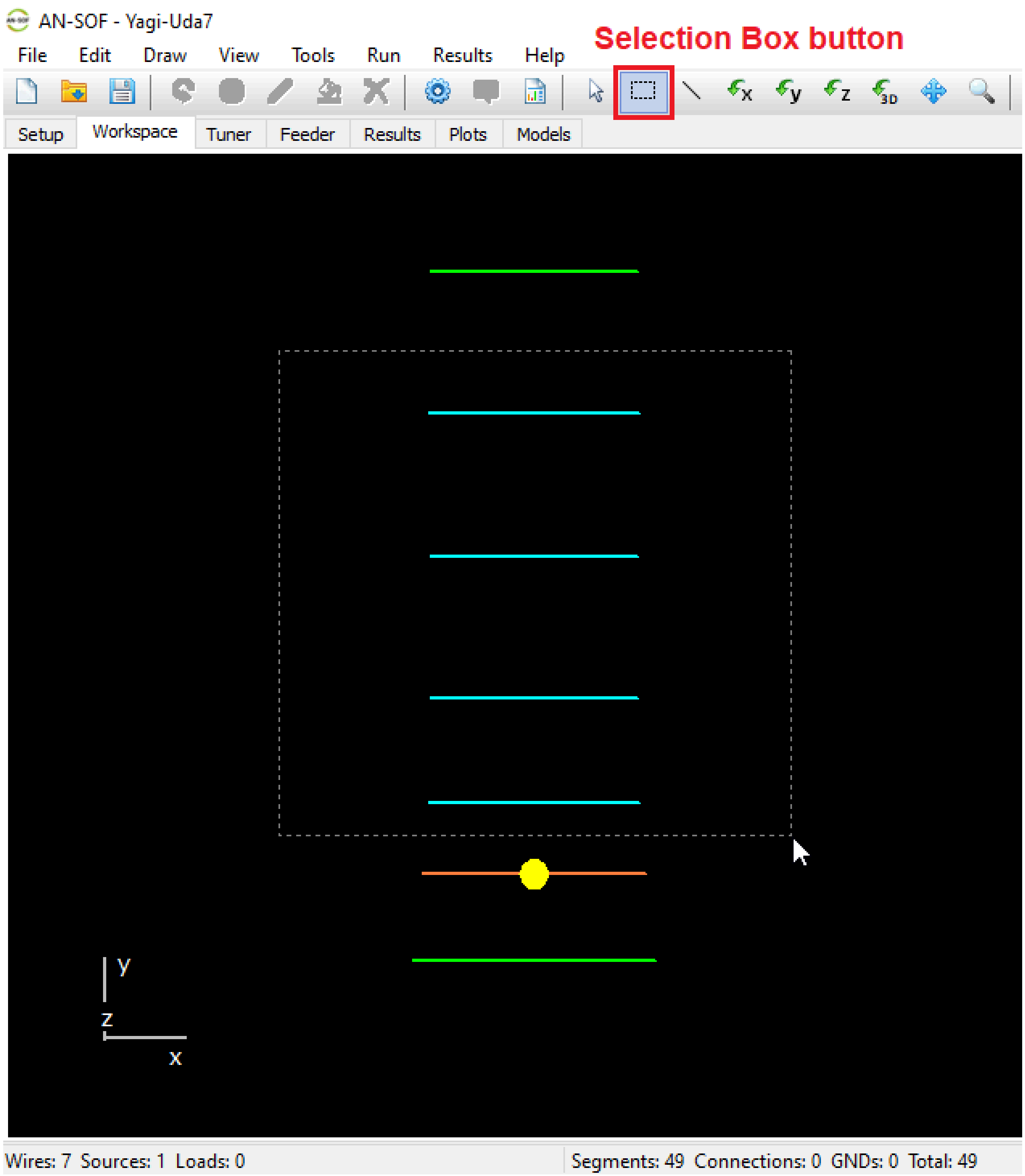


Fig. 1: *Selecting multiple wires using the Selection Box.*

Executing Modifications

Once your selection is highlighted, you can access the modification tools:

- **The Modify Command:** Go to **Edit > Modify**, use the shortcut **Ctrl + M**, or click the **Modify** button on the toolbar.
- **Scope of Changes:** This opens a dialog box where you can update the **Cross-Section**, **Resistivity**, or **Coating** for all selected wires simultaneously.

- **Refer to: [Selecting and Modifying Multiple Wires]**

Advanced Manipulations

If you need to change the physical orientation or duplicate the structure, use the following specialized commands:

- **Move, Rotate, & Scale:** Used to reposition a grid (e.g., moving a reflector behind a dipole) or resize it for a different frequency band.
- **Copy & Stack:** Useful for repeating a grid pattern across an array.

Deleting a Grid or Surface

Since wire grids and solid surfaces are composed of numerous individual segments, they must be **selected as a group** before they can be removed from your project (**Fig. 1**).

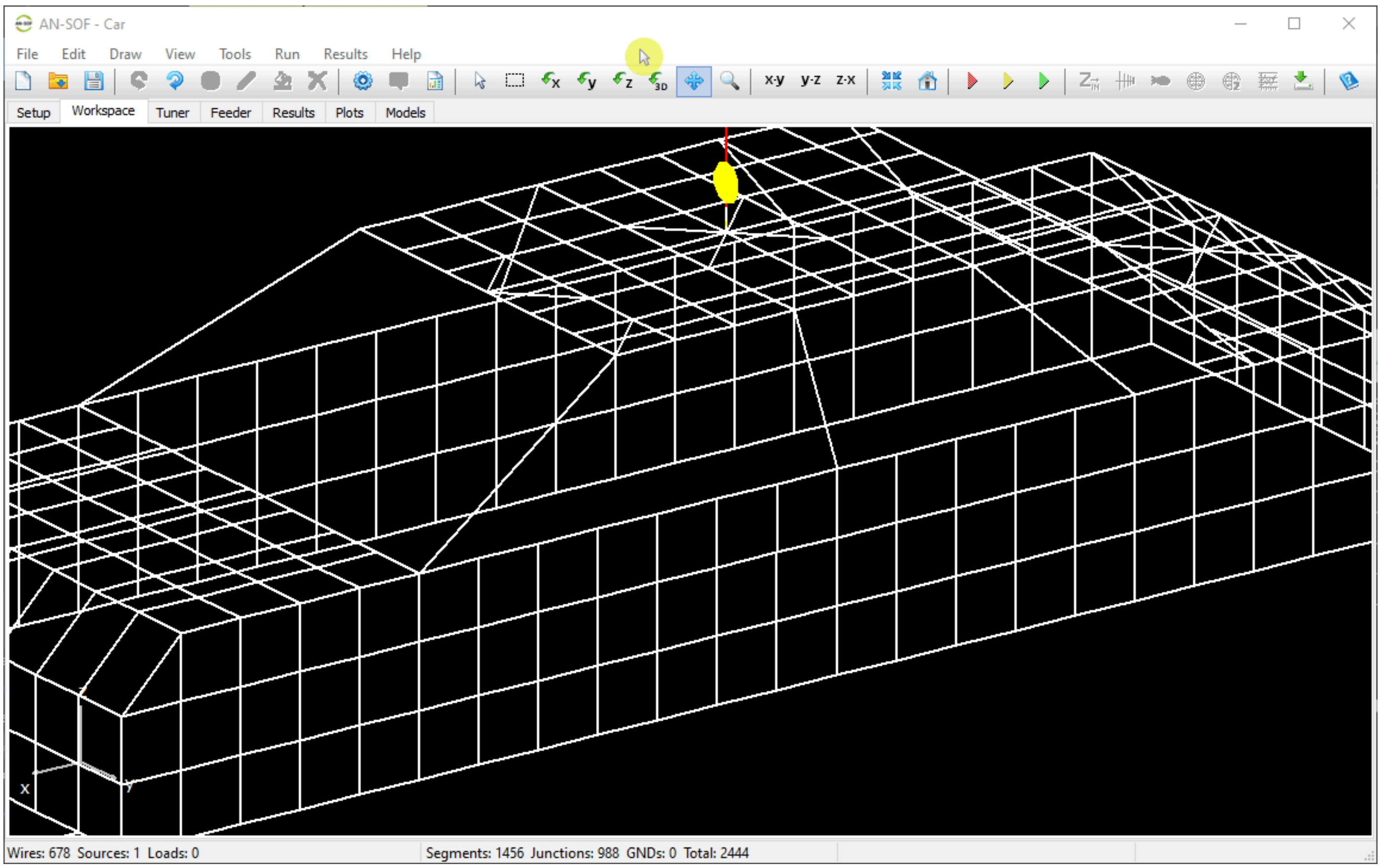


***Fig. 1:** Selecting multiple wires using Ctrl + left-click, followed by pressing the Delete button on the toolbar.*

Step-by-Step Deletion

1. **Activate Selection Mode:** Click the **Selection Box** button on the main toolbar. Refer to: **Selecting and Modifying Multiple Wires**.
2. **Define the Area:** Left-click in the workspace and drag a box around the grid or surface you wish to remove. All components within the box will be highlighted in **light blue**.

3. **Execute Delete:**
    - Go to **Edit > Delete** in the main menu.
    - Use the keyboard shortcut **Ctrl + Del**.
    - Click the **Delete** icon on the toolbar.

Important Considerations

- **Selective Deletion:** Because grids are collections of individual wires, you can choose to delete only a portion of a grid by drawing the selection box around specific segments rather than the whole structure.
- **Undo Support:** If you accidentally delete the wrong section, you can typically use **Ctrl + Z** to restore the wires.
- **Selection State:** The Delete command remains disabled until at least one wire is highlighted.

Color of Grids and Surfaces

Changing the color of a grid or surface is an effective way to visually organize complex models, especially when distinguishing between different materials, feed structures, or reflectors.

How to Change the Color

1. **Select the Structure:** Activate the **Selection Box** tool from the main toolbar. Click and drag to encompass the entire grid or surface. The selected elements will turn **light blue** (**Fig. 1**).
2. **Open the Color Menu:**
    - Navigate to **Edit > Wire Color** in the main menu.
    - Alternatively, click the **Wire Color** icon on the toolbar.
3. **Choose a Color:** A standard color picker dialog will appear. Select your desired color and click **OK**.

This command only becomes active when a selection has been made. If the icon is grayed out, ensure your wires are highlighted in light blue first. Refer to: **Selecting and Modifying Multiple Wires**.

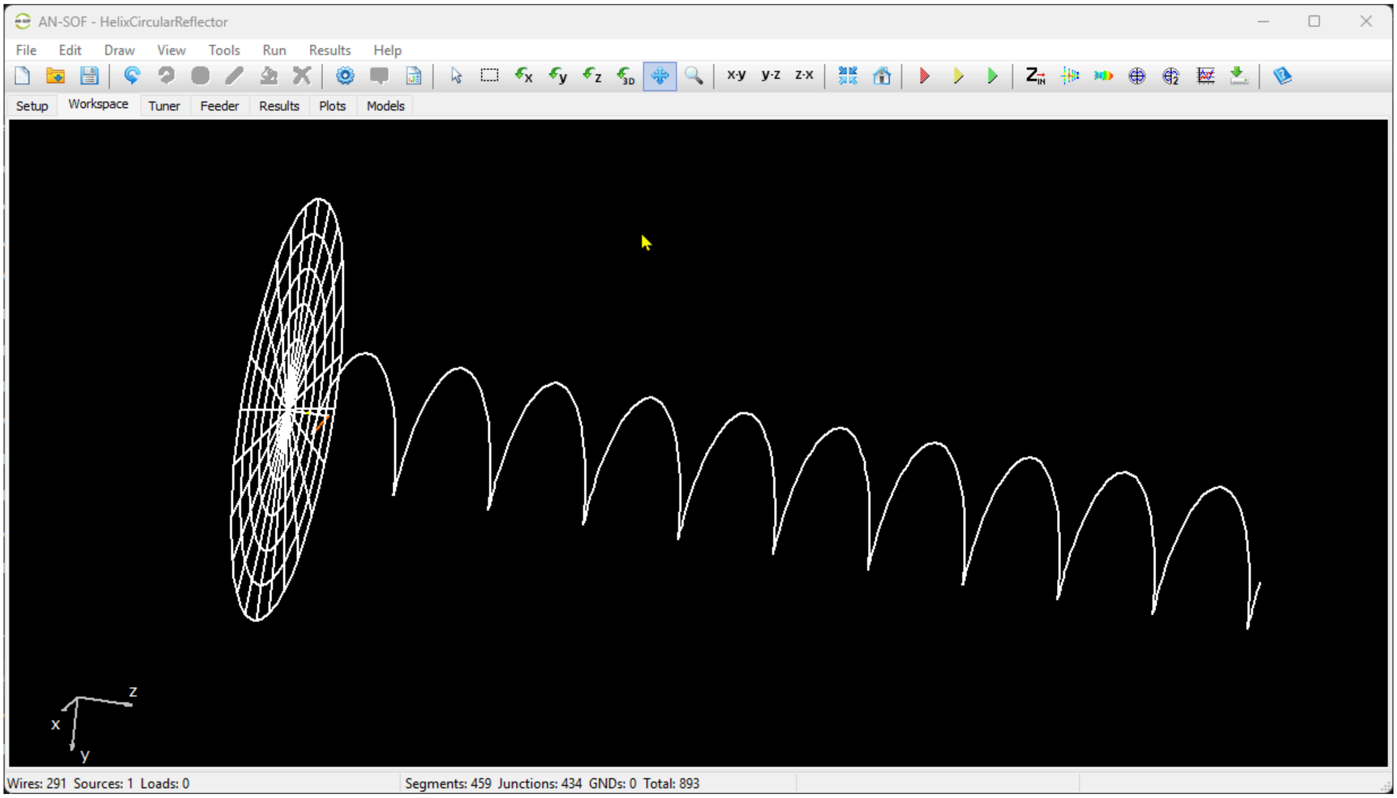


Fig. 1: *Demonstration of color editing for an individual wire (a helix) and groups of wires (a circular ground plane).*

Benefits of Color Coding

- **Material Identification:** Use different colors to represent different metals (e.g., Copper wires in orange, Aluminum in gray) to match your resistivity settings.
- **Structural Clarity:** Color-code your **Solid Surfaces** (flat strips) differently from your **Wire Grids** to quickly verify the modeling approach used for each part of the antenna.
- **Report Presentation:** When exporting screenshots for technical documentation, distinct colors make it much easier for readers to follow your design labels.

Adding Wire Grids/Solid Surfaces

Patch

The Patch Command

A **Patch** in AN-SOF is a specialized tool for creating **solid, rectangular conductive surfaces** that lie parallel to the **XY-plane** (z=constant). Unlike a general plate, the Patch is specifically optimized for applications like microstrip antenna design.

Key Characteristics

- **Structure:** It is composed of flat or rectangular strips that automatically adjust their width to form a continuous, gapless metal sheet.

- **Orientation:** Always horizontal (parallel to the XY-plane).
- **Primary Use Case:** Modeling **Patch Antennas** on a substrate.

  Requirement: To model a patch on a dielectric, you must set the **Ground Plane** to **Substrate** by navigating to the **Setup tab >** **Environment panel**.

> **Tip**
>
> **Patch vs. Plate:** Use **Patch** when your structure is parallel to the ground and you are using the **Substrate** ground option. Use **Plate** if you need to model a rectangular surface in free space, above a real ground, or if the surface needs to be tilted or vertical.

Configuration Steps

To open the tool, go to **Draw > Wire Grid / Solid Surface > Patch**. The dialog box features three tabs:

1. **Patch Tab:** Define the geometry by entering the coordinates of two **opposite corner points** (X1, Y1) and (X2, Y2) at a specific height (Z) (**Figs. 1 and 2**).
2. **Attributes Tab:** Specify the **Number of Facets**.

   If you enter **0x0**, AN-SOF automatically calculates the density based on the highest frequency (10 segments per λ).
3. **Materials Tab:** Assign the resistivity (e.g., Copper) or coating for the metallic sheet.

Automatic Facet Calculation

When the facets are set to **0x0**, AN-SOF ensures that the discretization is sufficient for electromagnetic convergence across your entire frequency sweep. This is particularly useful for broadband designs where the electrical size of the patch changes significantly.

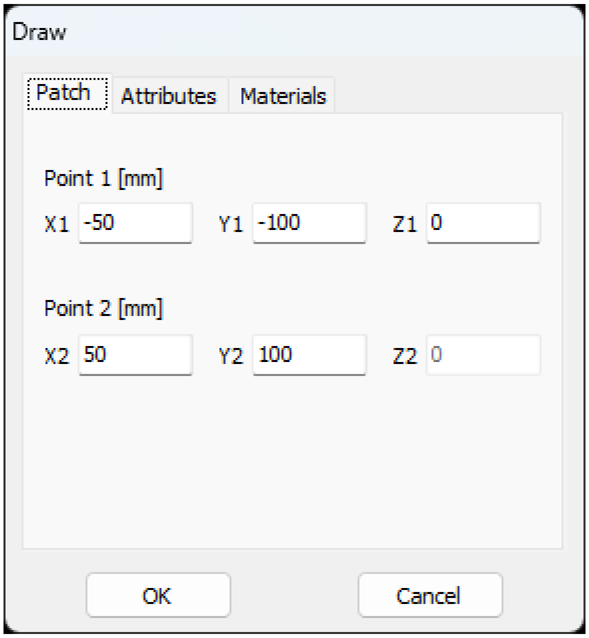


Fig. 1: *Patch tab in the Draw dialog box for the Patch.*

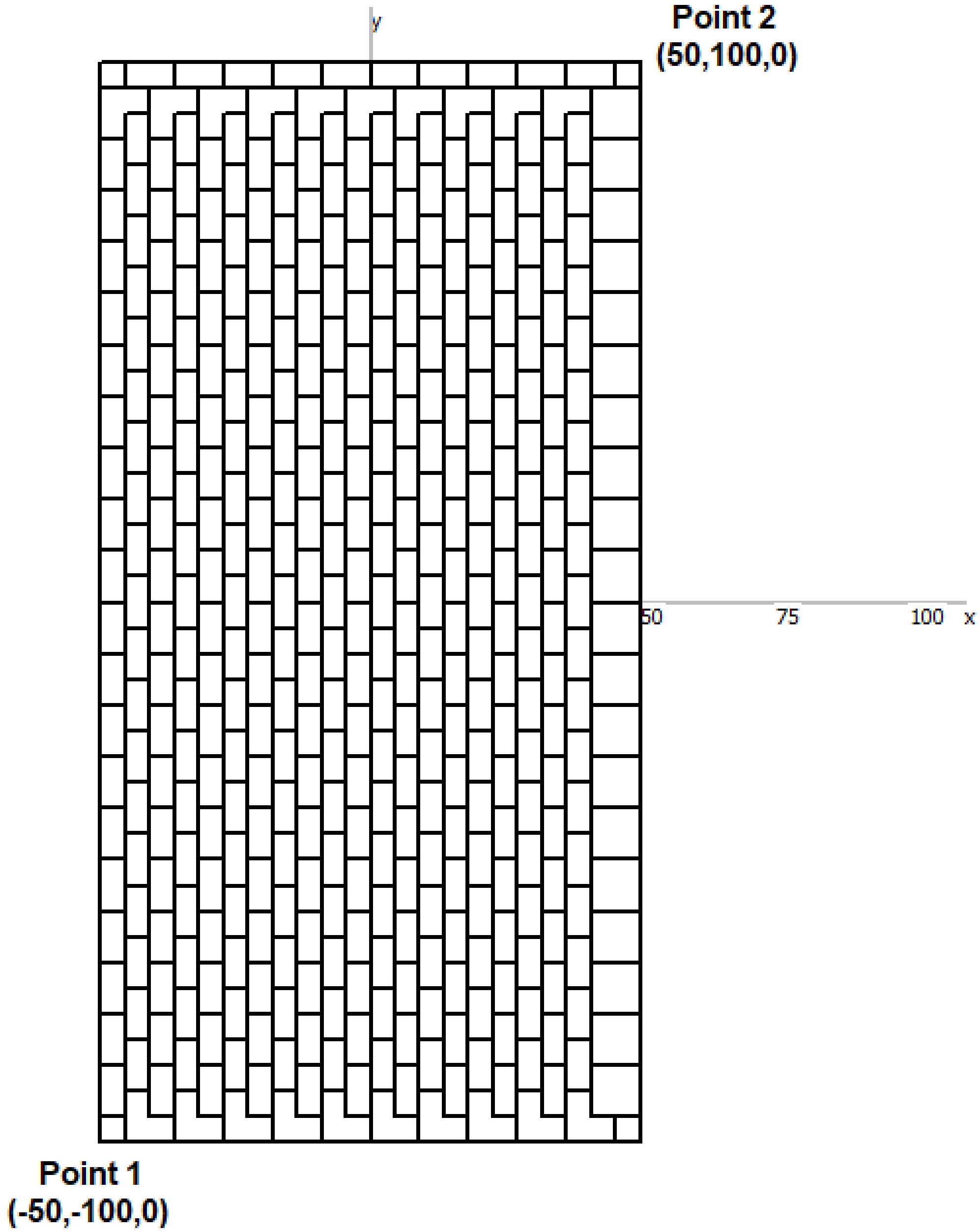


Fig. 2: *A Patch created using the input data from Fig. 1.*

Plate

The Plate Command

The **Plate** command is a versatile tool used to create four-sided metallic structures. Unlike the **Patch**, which is restricted to the horizontal plane, a Plate can be oriented in any direction or even twisted into a non-planar shape.

Key Characteristics

- **Geometry:** Defined by **four arbitrary vertices** in 3D space.

- **Bilinear Surface:** If the four corner points are not in the same plane, AN-SOF automatically generates a “bilinear surface.” This is a curved quadrilateral where the edges remain straight but the surface itself twists to connect the points.
- **Degenerate Case:** If all four points are coplanar, the command simply creates a standard **flat quadrilateral** (rectangle, trapezoid, or rhombus).
- **Flexibility:** Can be used in free space or above a real ground.

Configuration Steps

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Plate**.

1. **Plate Tab:** Enter the (X, Y, Z) coordinates for each of the four corner points (P1, P2, P3, and P4) (see **Fig. 1 and 2**).
2. **Attributes Tab:**
   - **Selection:** Choose **Circular** for a wire grid or **Flat/Rectangular** for a solid metal sheet.
   - **Facets:** Define the grid density (e.g., $N \times M$ facets).
3. **Materials Tab:** Specify the resistivity and any optional insulation or dielectric coating.

Common Use Cases

- **Reflectors:** Creating tilted or vertical ground planes and corner reflectors.
- **Complex Housings:** Modeling the metallic chassis of a device where surfaces are not perfectly aligned with the primary axes.
- **Aero-structures:** Modeling segments of aircraft wings or fuselage where the surface may have a subtle twist or non-planar geometry.

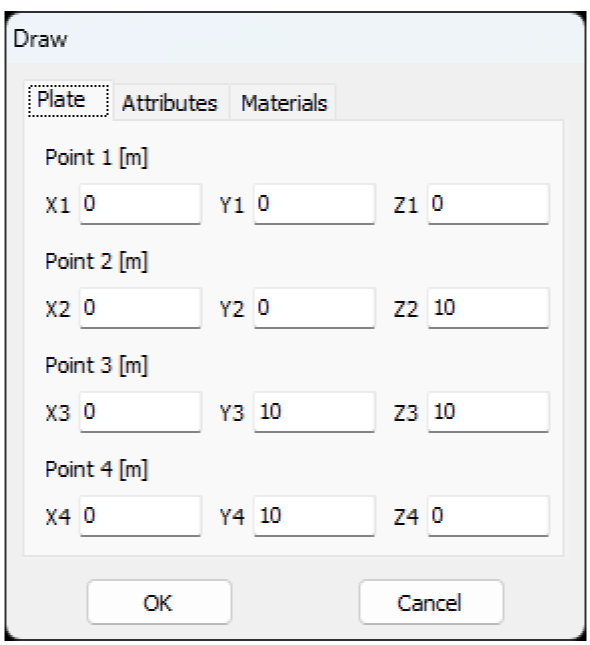


Fig. 1: *Plate tab in the Draw dialog box for the Plate.*

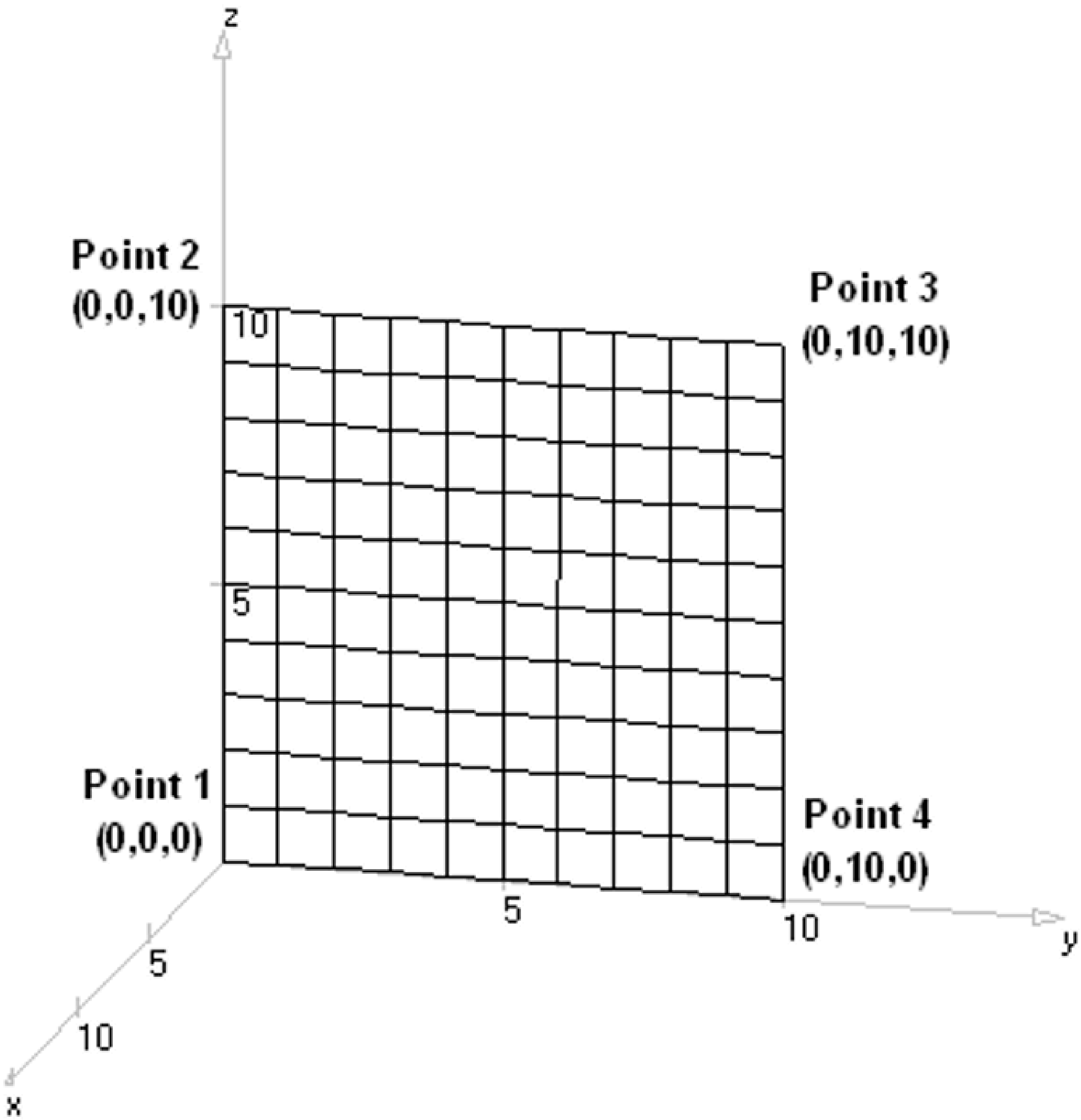


Fig. 2: *A Plate created using the input data from Fig. 1.*

Disk

The Disk Command

The **Disk** command allows you to create circular metallic structures, such as ground planes, circular patch antennas, or reflector elements. Like other surface tools in AN-SOF, it can be modeled as either a transparent wire mesh or a continuous solid sheet.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Disk**.

Geometric Precision: Curved vs. Straight

When setting up your disk, you have two options for how the circular edges are handled:

- **Curved Segments:** This provides an exact mathematical representation of the disk's curvature. It is generally recommended for high-frequency simulations where edge effects are critical.
- **Straight Segments:** This approximates the circle using a series of straight linear wires (creating a polygon). This can be useful for simplifying geometry when exporting to solvers that do not support curved entities.

Orientation and Positioning

To define the disk's location and "tilt" in 3D space, you use the following parameters:

- **Center Coordinates** (Cx, Cy, Cz$): The precise origin point of the disk (**Figs. 1 and 2**).
- **Radius:** The distance from the center to the outer edge.
- **Theta** and **Phi:** These angles define the **disk axis direction** (the vector perpendicular to the surface). For a disk lying flat on the XY-plane, you would set Theta = 0 deg.

Defining the Facets

In the **[Attributes]** tab, you define the density of the disk by specifying the number of **Radial** and **Azimuthal** (circular) divisions.

- **Radial Facets:** The number of divisions from the center to the edge.
- **Azimuthal Facets:** The number of divisions around the 360∘ circumference.

Grid vs. Solid Surface

- **Wire Grid:** Select **Circular** cross-section in the Attributes tab to create a "spoke and wheel" radial mesh.
- **Solid Surface:** Select **Flat** or **Rectangular** to create a solid circular sheet. AN-SOF will automatically use flat strips to fill the circular area completely.

Materials

In the **[Materials]** tab, you can assign the physical properties of the metal, such as resistivity (e.g., Copper or Aluminum), or apply a dielectric coating if the disk is insulated.

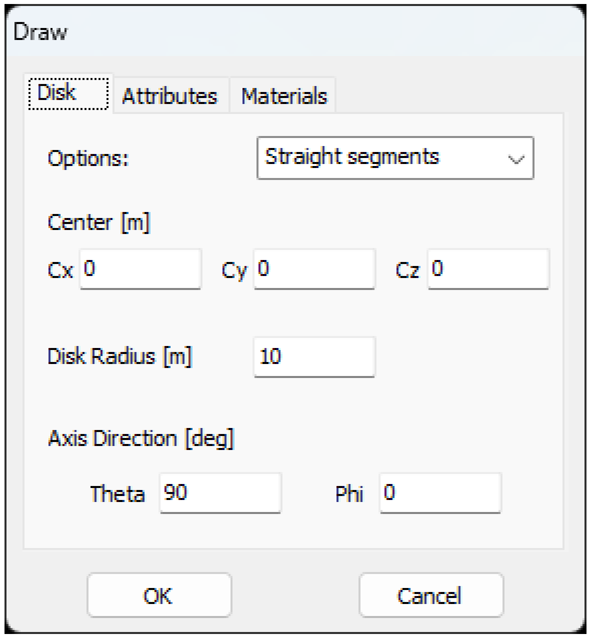


Fig. 1: *Disk tab in the Draw dialog box for the Disk.*

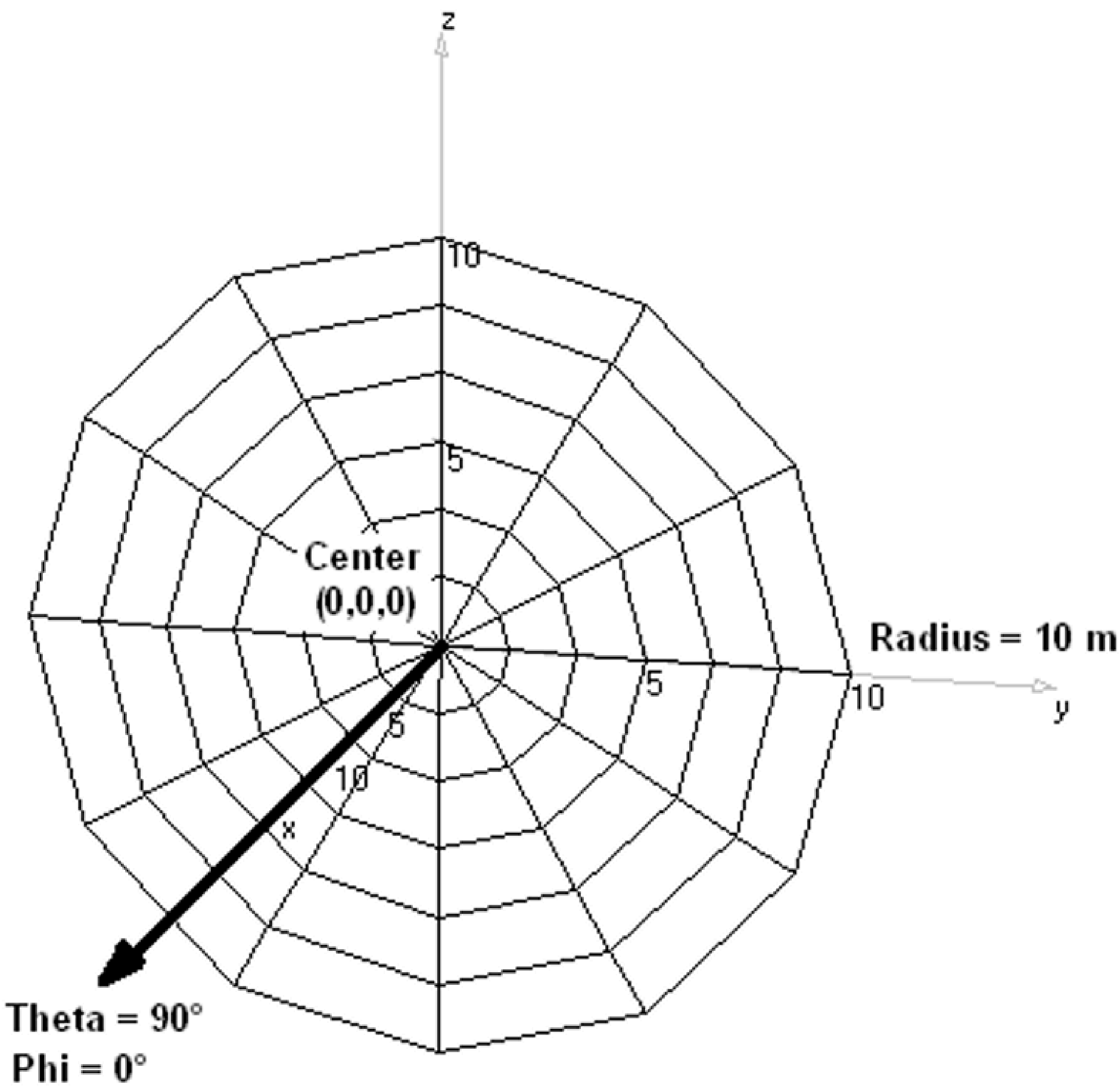


Fig. 2: *A Disk created using the input data from Fig. 1.*

Solid Disk Equivalent Radius Calculator

Actual Disk Radius R_0

Number of Radial Facets N_r

Flat Ring

The Flat Ring Command

The **Flat Ring** command is used to create an annular (ring-shaped) surface, essentially a disk with a circular hole at its center. This is a common geometry for specialized ground planes, circular loop-fed antennas, or parasitic ring elements.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Flat Ring**.

Geometry and Orientation

To define the physical placement and size of the ring, you provide the following parameters in the **Flat Ring** tab:

- **Center Coordinates** (Cx, Cy, Cz): The origin point of the ring's center (**Figs. 1 and 2**).
- **Radii:** You must define both the **Inner Radius** (the size of the central hole) and the **Outer Radius**.
- **Orientation** (Theta and Phi): These angles define the vector perpendicular to the ring's surface (the ring axis). For example, setting Theta = 0 places the ring on the horizontal XY-plane.

Curvature: Curved vs. Straight

Like the **Disk** command, you can choose how the software handles the circular edges:

- **Curved Segments:** Recommended for maximum electromagnetic accuracy, as it uses the exact mathematical curvature of the ring.
- **Straight Segments:** Approximates the ring circles using a series of straight wire segments (polygonal approximation).

Attributes and Faceting

In the **Attributes** tab, you determine how the ring is "meshed" for the simulation engine:

- **Facets:** Define the density of the mesh using **Radial** (from the inner edge to the outer edge) and **Azimuthal** (around the circle) divisions.
- **Wire Grid vs. Solid Surface:**
  - Select **Circular** cross-section to create a wire mesh.
  - Select **Flat** or **Rectangular** to create a continuous, solid metal sheet. AN-SOF will automatically calculate the strip widths to ensure there are no gaps between the facets.

Materials

In the **Materials** tab, you can assign the physical properties of the metal, such as resistivity (e.g., Copper or Aluminum), or apply a dielectric coating if the ring is insulated.

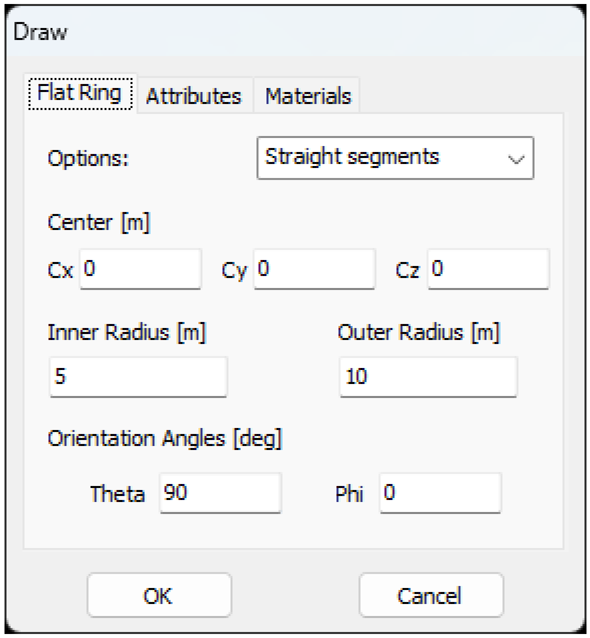


Fig. 1: *Flat Ring tab in the Draw dialog box for the Flat Ring.*

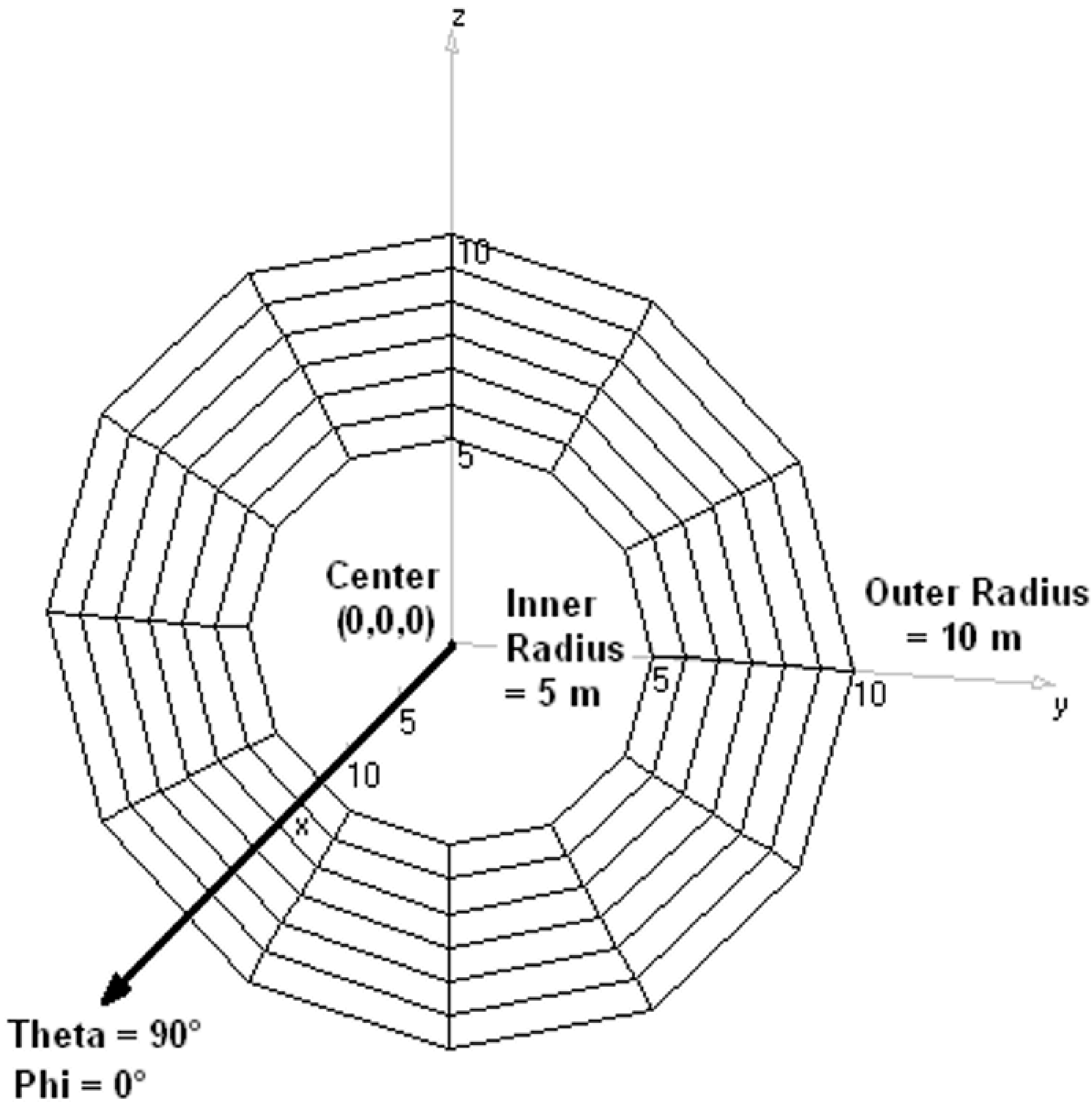


Fig. 2: *A Flat Ring created using the input data from Fig. 1.*

Solid Ring Equivalent Radii Calculator

Actual Inner Radius R_{10}

Actual Outer Radius R_{20}

Number of Radial Facets N_r

The Cone Command

The **Cone** command allows you to create conical metallic structures, which are frequently used in the design of wideband biconical antennas and horn antenna transitions.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Cone**.

Defining the Geometry

In the **Cone** tab, you configure the physical dimensions and orientation of the structure using the following parameters:

- **Vertex Coordinates** (Vx, Vy, Vz): The precise point where the cone originates (the tip) (see **Figs. 1 and 2**).
- **Aperture Angle:** The full angular opening of the cone.
- **Aperture Radius:** The radius of the circular base of the cone.
- **Orientation** (Theta and Phi): These angles define the direction of the cone's axis (the line passing through the vertex and the center of the base).

Curvature Options

You can choose how the software represents the circular base and the conical walls:

- **Curved Segments:** Uses exact mathematical curvature. This is preferred for high-precision simulations to avoid the artificial edge effects caused by polygonal approximations.
- **Straight Segments:** Models the cone as a series of flat triangular or trapezoidal facets using straight wires.

Faceting and Mesh Attributes

In the **Attributes** tab, you determine the density of the simulation mesh:

- **Facets:** Define the number of divisions along the **Axis** (from vertex to base) and **Azimuthally** (around the circumference).
- **Wire Grid:** Choosing a **Circular** cross-section creates a mesh of wires, often used to simulate "caged" biconical antennas.
- **Solid Surface:** Choosing **Flat** or **Rectangular** cross-sections creates a solid metal sheet. AN-SOF automatically adjusts the strip geometry to ensure the cone surface is completely covered without gaps.

Materials

The **Materials** tab allows you to set the resistivity (e.g., Copper, Aluminum, or Stainless Steel) and any optional dielectric coating for the cone.

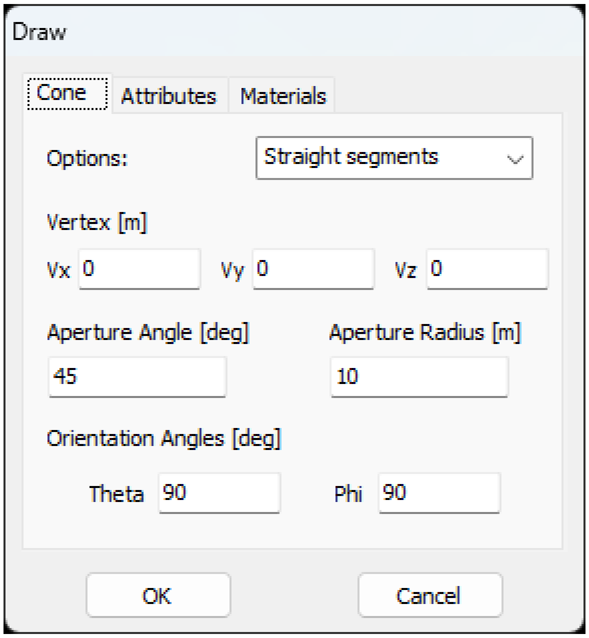


Fig. 1: *Cone tab in the Draw dialog box for the Cone.*

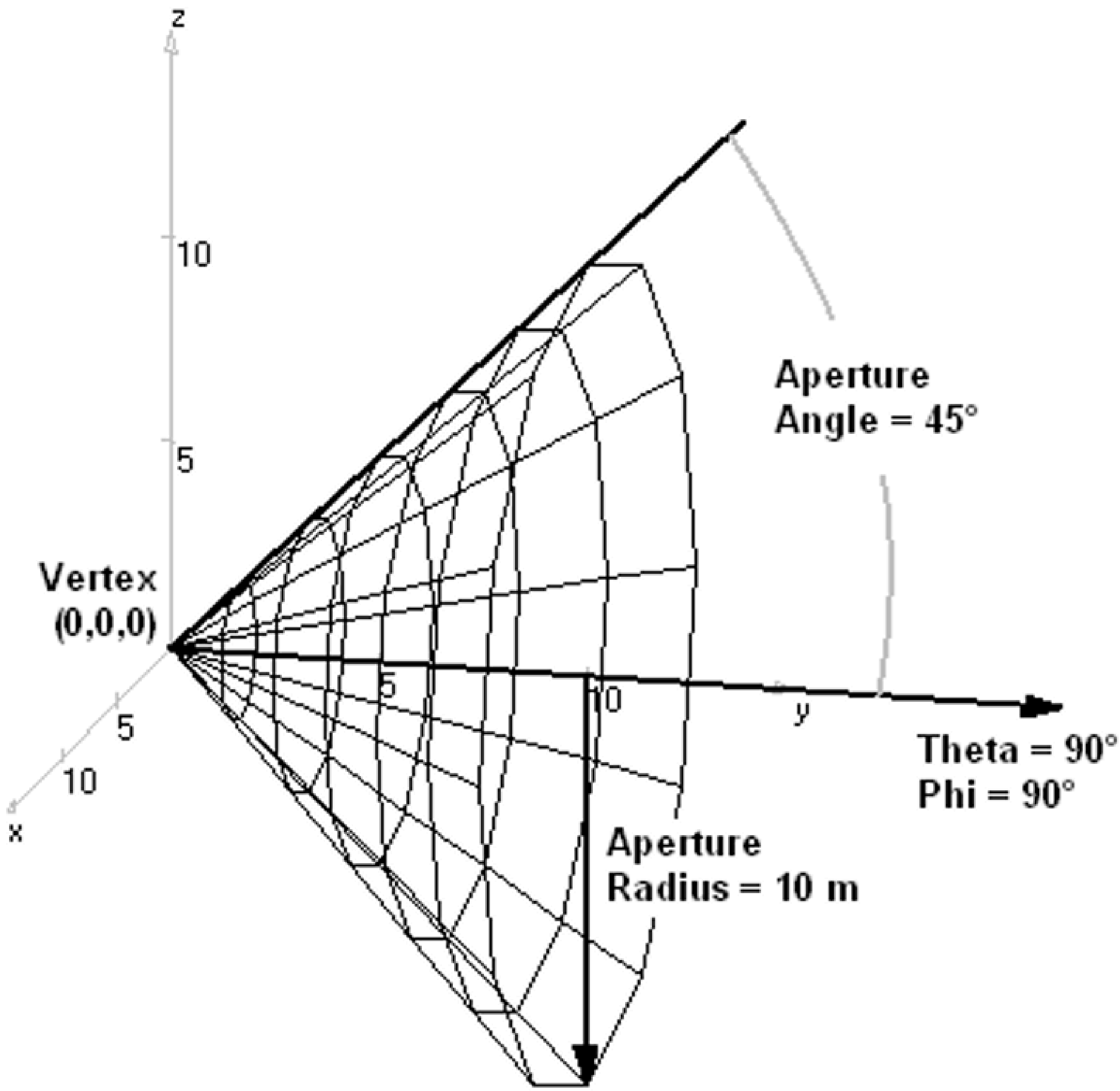


Fig. 2: *A Cone created using the input data from Fig. 1.*

Truncated Cone

The Truncated Cone Command

The **Truncated Cone** command allows you to create a “frustum”—a cone with the top portion removed. This versatile tool can be used to model anything from standard conical sections to transitions in horn antennas or even specialized cylindrical supports.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Truncated Cone**.

Defining the Geometry

In the **Truncated Cone** tab, you configure the physical dimensions and orientation of the structure using the following parameters:

- **Base Point Coordinates** (Px, Py, Pz): The center of the bottom circular face (**Figs. 1 and 2**).
- **Base Radius & Top Radius:** By adjusting these two values, you can create a wide variety of shapes:
  - **Top Radius ≈ 0:** Approaches a standard **Cone**.
  - **Top Radius = Base Radius:** Creates a **Cylinder**.
  - **Different Radii:** Creates a standard **Truncated Cone**.
- **Aperture Angle:** The angle of the conical slope.
- **Orientation** (Theta and Phi): These angles define the direction of the central axis connecting the base and top centers.

Curvature Options

- **Curved Segments:** This provides an exact mathematical representation of the circular edges and the sloped surface. It is recommended for precision modeling to ensure correct current distribution at high frequencies.
- **Straight Segments:** Approximates the structure using flat facets made of linear wires, forming a polygonal shape.

Faceting and Mesh Attributes

In the **Attributes** tab, you determine the density of the simulation mesh:

- **Facets:** Define the number of divisions along the **Axis** (from base to top) and **Azimuthally** (around the circumference).
- **Wire Grid:** Select **Circular** cross-section to create a “cage” or mesh structure.
- **Solid Surface:** Select **Flat** or **Rectangular** cross-sections. AN-SOF will automatically calculate the necessary strip widths to form a continuous, gapless metallic sheet.

Materials

The **Materials** tab allows you to set the resistivity (e.g., Aluminum, Brass, or Copper) and apply any optional dielectric insulation to the surface.

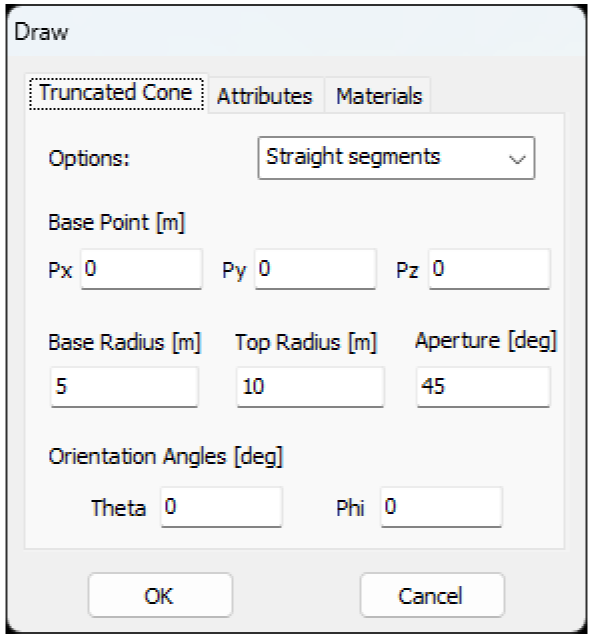


Fig. 1: *Truncated Cone tab in the Draw dialog box for the Truncated Cone.*

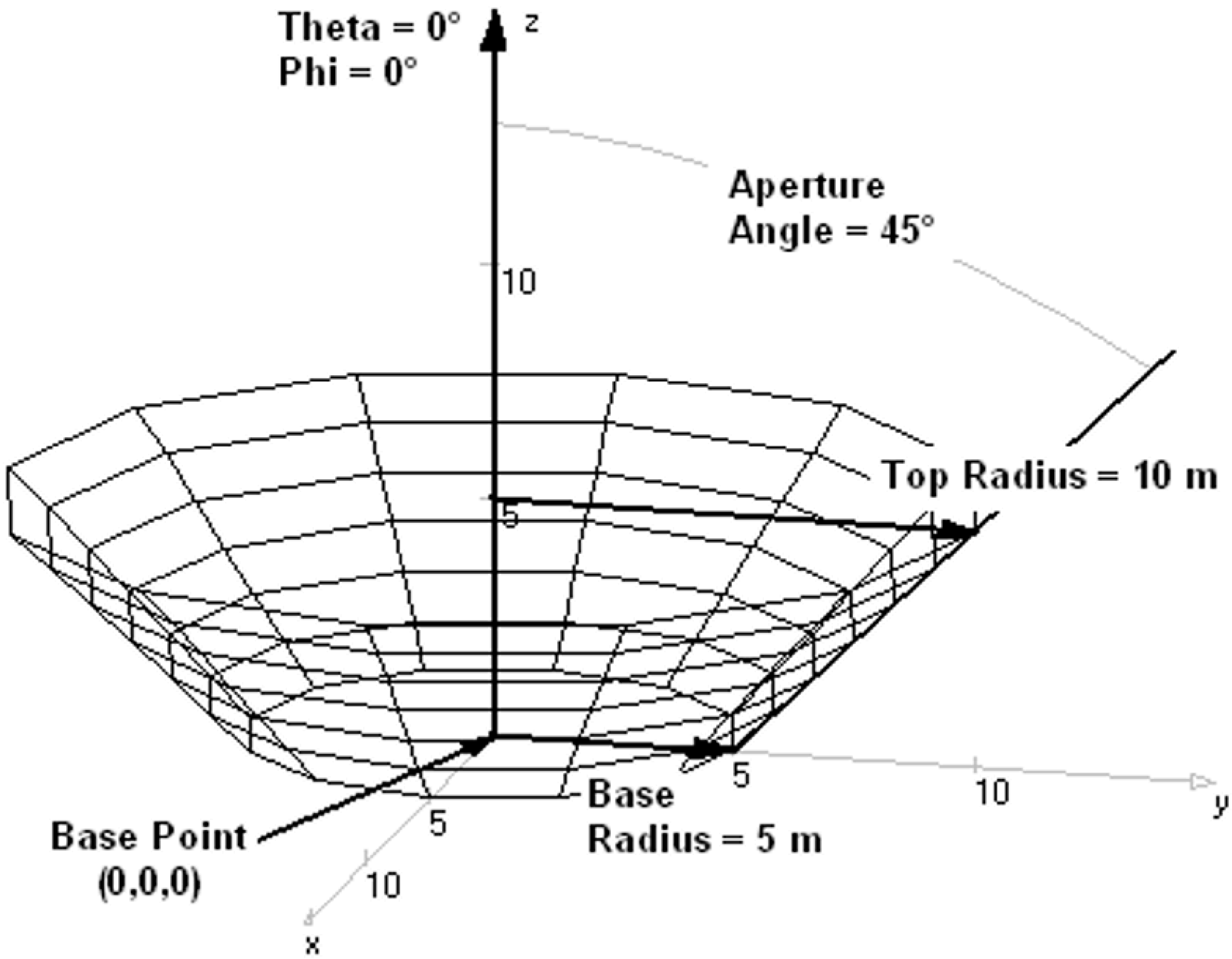


Fig. 2: *A Truncated Cone created using the input data from Fig. 1.*

Cylinder

The Cylinder Command

The **Cylinder** command allows you to create cylindrical conductive structures, which are essential for modeling thick antenna elements, tubular masts, coaxial outer conductors, or shielding cans.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Cylinder**.

Defining the Geometry

In the **Cylinder** tab, you configure the physical dimensions and spatial orientation of the structure using these parameters:

- **Base Point Coordinates** (Px, Py, Pz): The center point of the bottom circular face of the cylinder (**Figs. 1 and 2**).
- **Length:** The total height of the cylinder along its axis.

- **Radius:** The distance from the central axis to the outer surface.
- **Orientation** (Theta and Phi): These angles define the direction of the cylinder's axis. For a vertical cylinder, you would set Theta = 0 deg.

Curvature Options

You can choose how the software represents the circular cross-section:

- **Curved Segments:** Provides an exact mathematical representation of the cylinder's curve. This is highly recommended to ensure the simulation accurately captures surface current distributions.
- **Straight Segments:** Approximates the cylinder as a prism with a polygonal base.

Faceting and Mesh Attributes

In the **Attributes** tab, you define the computational mesh:

- **Facets:** Specify the number of divisions along the **Axis** (lengthwise) and **Azimuthally** (around the circumference).
- **Wire Grid:** Select **Circular** cross-section to create a "caged" cylinder made of a wire framework.
- **Solid Surface:** Select **Flat** or **Rectangular** cross-sections. AN-SOF will automatically adjust the strip widths to form a continuous, solid metal tube.

Materials

Use the **Materials** tab to define the resistivity of the cylinder (e.g., Aluminum for a mast or Copper for a high-Q resonator) and to add any dielectric coating or insulation.

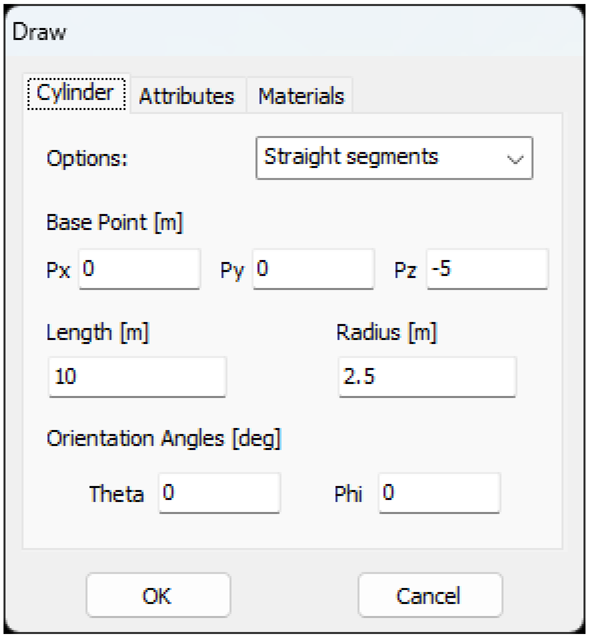


Fig. 1: *Cylinder tab in the Draw dialog box for the Cylinder.*

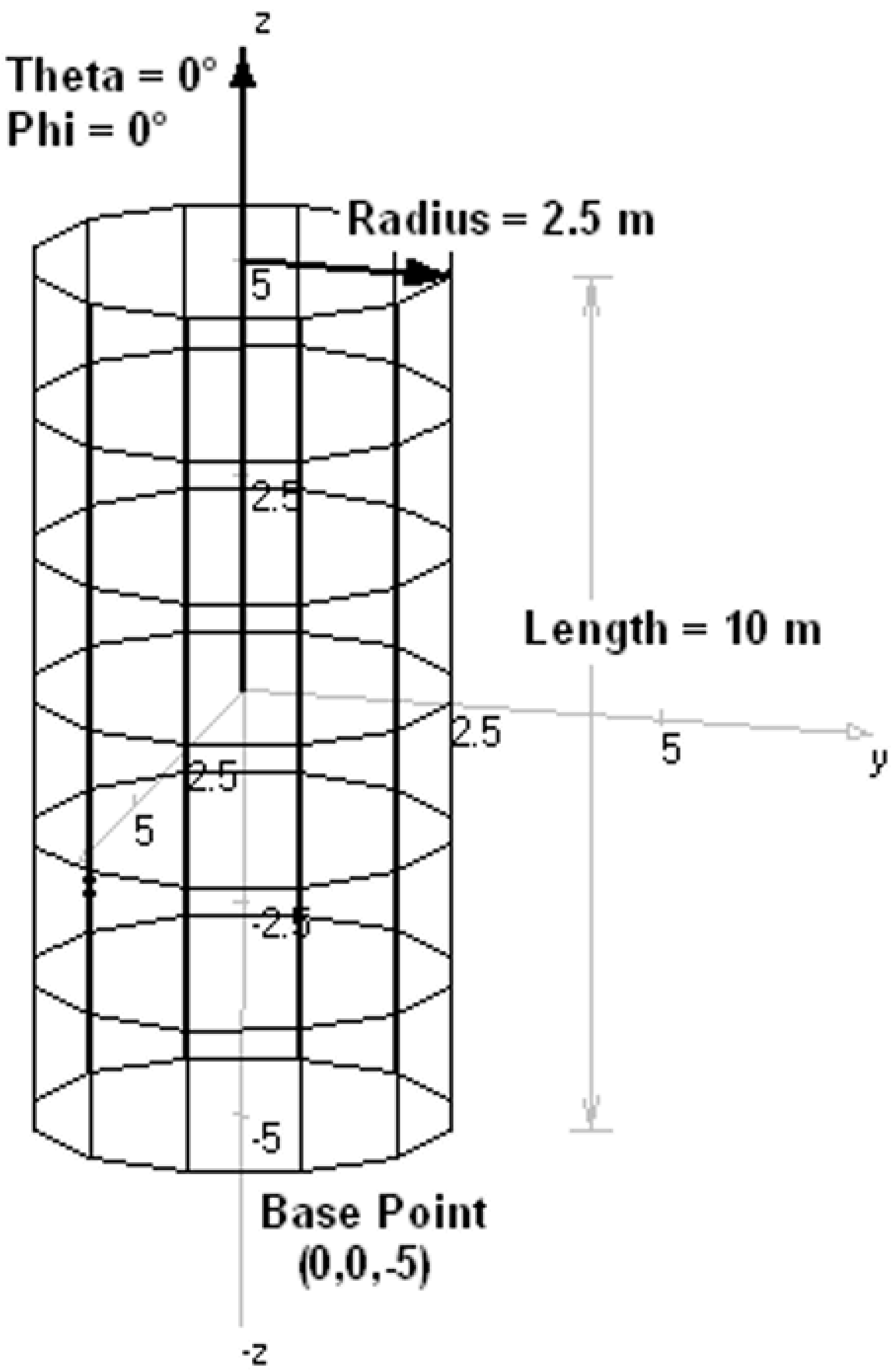


Fig. 2: *A Cylinder created using the input data from Fig. 1.*

Sphere

The Sphere Command

The **Sphere** command allows you to create spherical conductive structures, which are useful for modeling satellite bodies, spherical antennas, scattering targets (RCS studies), or spherical ground systems.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Sphere**.

Defining the Geometry

In the **Sphere** tab, you configure the physical dimensions and orientation of the sphere using these parameters:

- **Center Coordinates** (Cx, Cy, Cz): The exact origin point of the sphere.
- **Radius:** The distance from the center to the surface.
- **Orientation** (Theta and Phi): While a perfect sphere is symmetrical, these angles define the orientation of the poles (the axis of the azimuthal and polar facets). This is important when you need to align the sphere's mesh with other connected wires.

Curvature Options

- **Curved Segments:** This provides an exact mathematical representation of the sphere's curvature. It is the preferred setting for high-frequency simulations to ensure smooth current flow and accurate scattering results.
- **Straight Segments:** Approximates the sphere using a series of flat facets made of linear wires, resulting in a polyhedral shape (similar to a geodesic dome).

Faceting and Mesh Attributes

In the **Attributes** tab, you define the computational density of the sphere:

- **Facets:** Specify the number of divisions along the **Polar** (latitude) and **Azimuthal** (longitude) directions.
- **Wire Grid:** Select **Circular** cross-section to create a transparent mesh, often used for "caged" or wire-frame spherical reflectors.
- **Solid Surface:** Select **Flat** or **Rectangular** cross-sections. AN-SOF will automatically calculate the tapering strip widths to ensure the spherical surface is completely covered without gaps.

Materials

The **Materials** tab allows you to set the resistivity of the sphere (e.g., Silver, Copper, or Aluminum) and to add any optional dielectric coating or insulation.

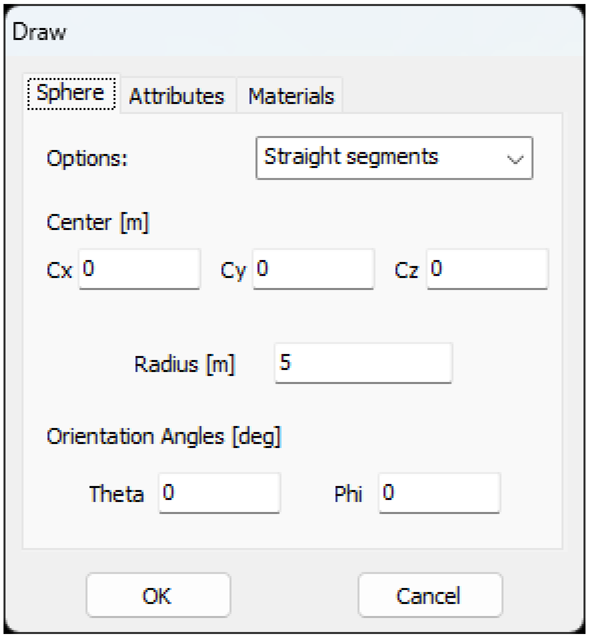


Fig. 1: *Sphere tab in the Draw dialog box for the Sphere.*

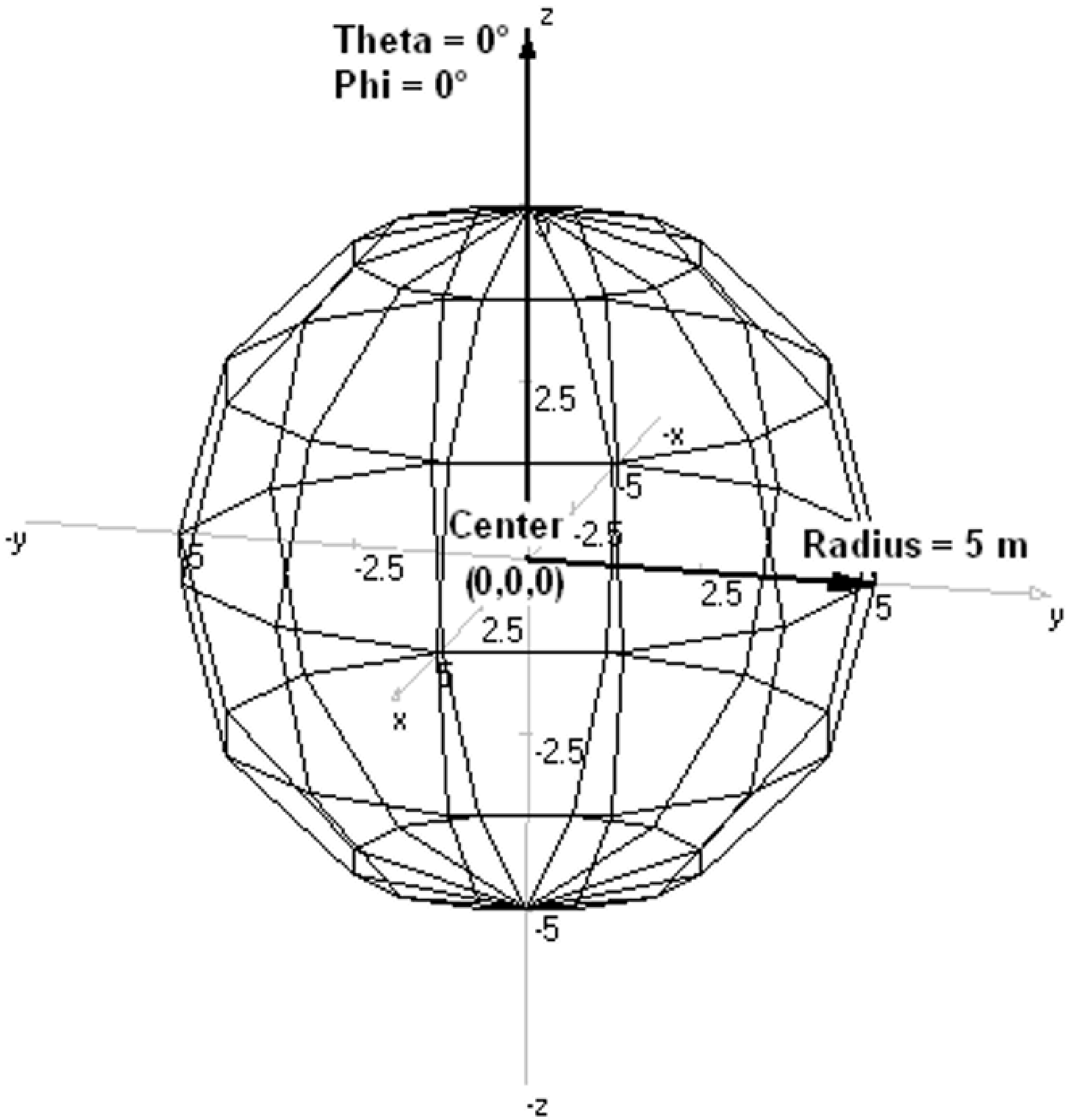


Fig. 2: *A Sphere created using the input data from Fig. 1.*

Paraboloid

The Paraboloid Command

The **Paraboloid** command is a specialized tool for creating parabolic reflectors, the most common geometry for high-gain dish antennas used in satellite communications, radar, and radio astronomy.

To open the tool, navigate to **Draw > Wire Grid / Solid Surface > Paraboloid**.

Defining the Geometry

In the **Paraboloid** tab, you configure the mathematical curve and physical size of the reflector:

- **Vertex Coordinates** (Vx, Vy, Vz): The coordinates of the bottom-most point (the center) of the dish (**Figs. 1 and 2**).
- **Focal Length:** The distance from the vertex to the focal point where the feed antenna is typically placed. This determines the "deepness" of the dish.
- **Aperture Radius:** The distance from the central axis to the outer rim of the dish.
- **Orientation** (Theta and Phi): These angles define the direction of the paraboloid's axis of symmetry. For a dish pointing directly upward, set Theta = 0 deg.

Curvature Options

- **Curved Segments:** This utilizes the exact mathematical parabolic curve. It is highly recommended for reflectors, as even small polygonal errors in the surface can lead to significant phase errors in the reflected wave at high frequencies.
- **Straight Segments:** Approximates the parabolic surface using a series of flat facets made of linear wires.

Faceting and Mesh Attributes

In the **Attributes** tab, you define the computational density of the reflector:

- **Facets:** Specify the number of divisions along the **Radial** (from vertex to rim) and **Azimuthal** (around the 360∘ circumference) directions.
- **Wire Grid:** Select **Circular** cross-section to create a "mesh dish." This is often used for lower-frequency antennas to reduce wind load and weight.
- **Solid Surface:** Select **Flat** or **Rectangular** cross-sections to model a solid metallic dish for maximum gain and front-to-back ratio.

Materials

The **Materials** tab allows you to set the resistivity of the reflector surface. While most reflectors are modeled as Perfect Electric Conductors (PEC), you can specify materials like Aluminum or Steel to account for ohmic losses in the reflector itself.

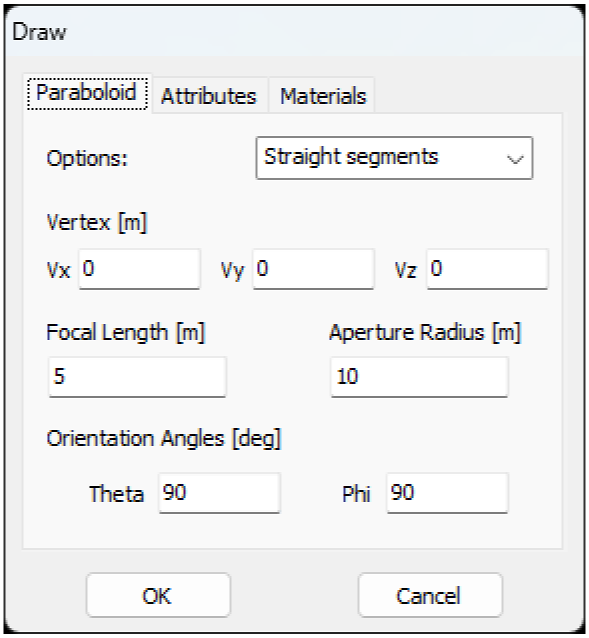


Fig. 1: *Paraboloid tab in the Draw dialog box for the Paraboloid.*

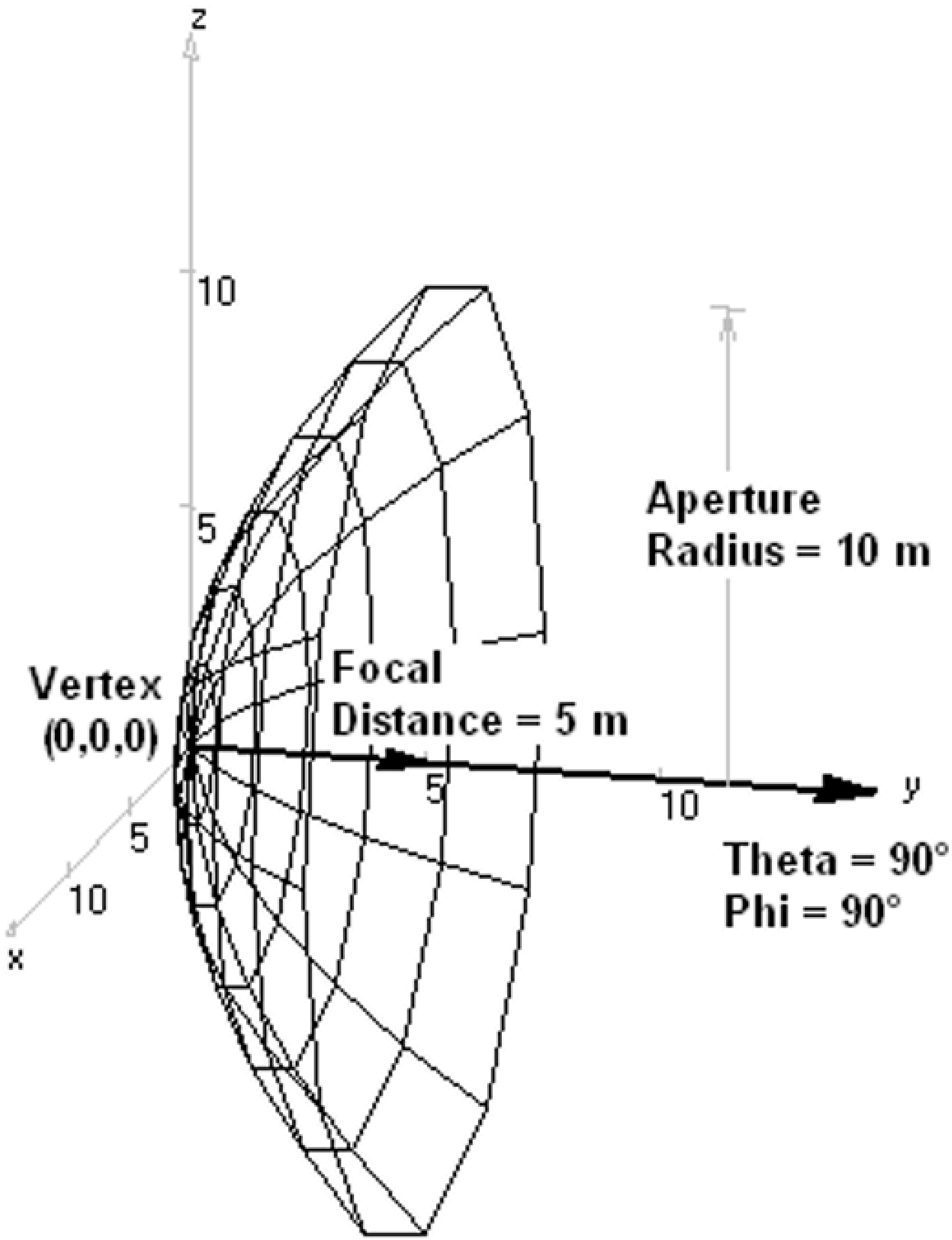


Fig. 2: *A Paraboloid created using the input data from Fig. 1.*

Sources and Loads

Types of Excitations and Loads

Discrete Sources, Incident Field, and Loads

A structure in AN-SOF can be excited either by **discrete sources** or by an **incident electromagnetic field**. For the latter, refer to **Excitation by an Incident Field**. Discrete **sources can be placed on any wire segment**, and multiple sources may be added—up to one per segment if needed.

A **source** represents the feed point of a transmitting antenna or an electrical generator. There are two types of sources:

- **Voltage sources**
- **Current sources** (used to model impressed currents)

For each source, you must define its **amplitude** and **phase**. Additionally, you can assign an **internal impedance** to simulate non-ideal sources. The impedance can take one of the following forms:

- Series RL (inductive)
- Series RC (capacitive)
- Fixed complex impedance (R + jX)

In addition to sources, **lumped loads** can be inserted into any wire segment to model passive components or matching circuits. Six types of loads are available:

- **Series RL** impedance (inductive)
- **Series RC** impedance (capacitive)
- **Fixed R + jX** impedance (reactance does *not* scale with frequency)
- **Series RLC** circuit
- **Parallel RLC** circuit
- **Trap RLC** circuit

To model a pure resistor, set an inductive load with **L = 0**.

> **Note:**
>
> You can set the units for inductance and capacitance in **Main Menu > Tools > Preferences**. Supported units include:
> • **Inductance**: pH, nH, μH, mH, H
> • **Capacitance**: pF, nF, μF, mF, F

In the 3D workspace:

- **Sources** are represented by a **yellow circle**
- **Loads** are indicated by a **green-highlighted wire segment** (see **Fig. 1**)

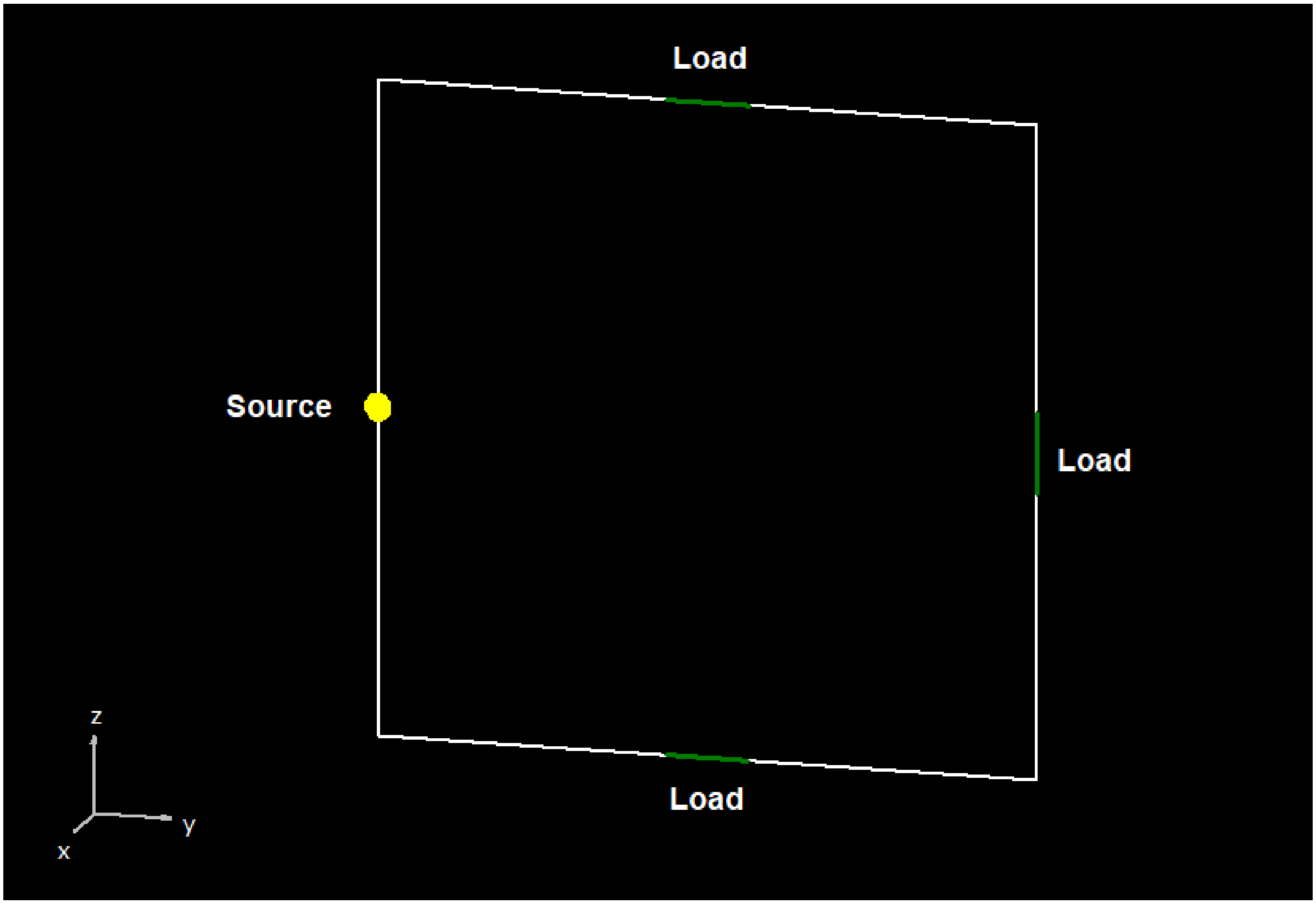


Fig. 1: Source and load indicators on wires in the workspace.

> **Tips**
>
> To customize the default colors of sources and loads, go to: **Main Menu > Tools > Preferences > Workspace** tab.
>
> **To change the size of the source** displayed in the workspace, go to **Main Menu > Tools > Preferences > Workspace** tab, and adjust the **Source Size %** setting.
>
> **Voltage sources** have their internal impedance **in series**.
> To model an ideal (perfect) voltage source, set the impedance to **zero**.
>
> **Current sources** have their internal impedance **in parallel**.
> To approximate an ideal current source, use a **very high impedance** (e.g., **1 MΩ**).

Excitation by Sources

To excite the wire structure using discrete sources, go to the **Setup tab**, open the **Excitation panel**, and select the **Discrete Sources** option (see **Fig. 2**).

If the **Set Input Power** option is enabled, you can specify the total input power delivered to the structure. In this case, the amplitudes of the voltage and current sources will be automatically adjusted to match the desired input power.

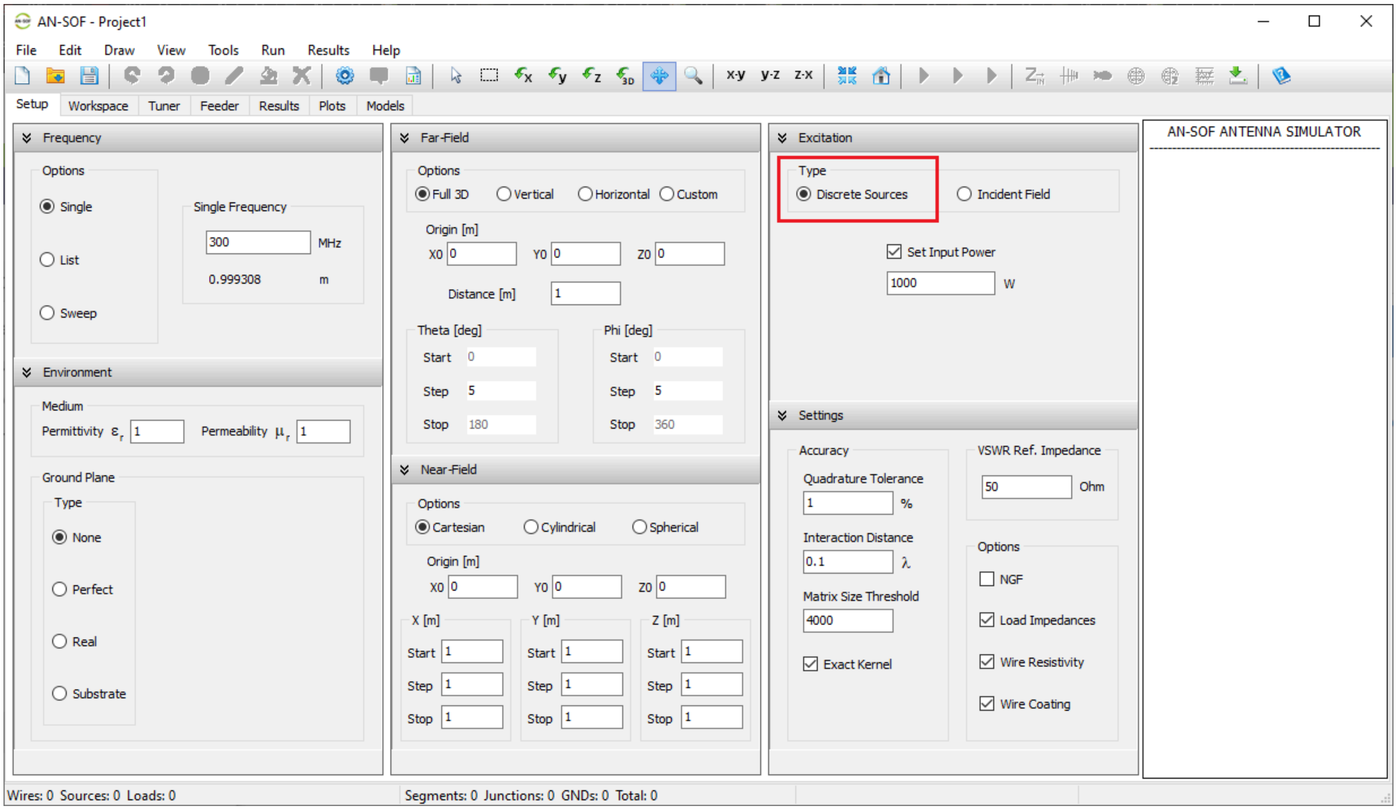


Fig. 2: The Discrete Sources option in the Excitation panel of the Setup tab.

To add a source to a wire segment, first right-click on the desired wire and select **"Source / Load / TL (Ctrl + Ins)"** from the [pop-up menu](). A toolbar will appear at the bottom of the screen. Use the slider on this toolbar to select the specific wire segment where you want to place the source.

For more details, see [The Source/Load/TL Toolbar]().
The Source/Load/TL Toolbar

The **Source/Load/TL toolbar** is used to connect a source, load, or transmission line to a specific segment of a wire. It also provides tools to edit or remove existing connections.

Accessing the Toolbar

To open this toolbar:

- **Right-click** on any part of a wire and select **Source / Load / TL (Ctrl + Ins)** from the [pop-up menu]() (**Fig. 1**).
- Alternatively, use the **main toolbar** or go to **Edit > Source / Load / TL (Ctrl + Ins)** from the main menu. Before using this command, click the **Select Wire** button (arrow icon on the main toolbar), then **left-click** the wire where you want to place the source or load.

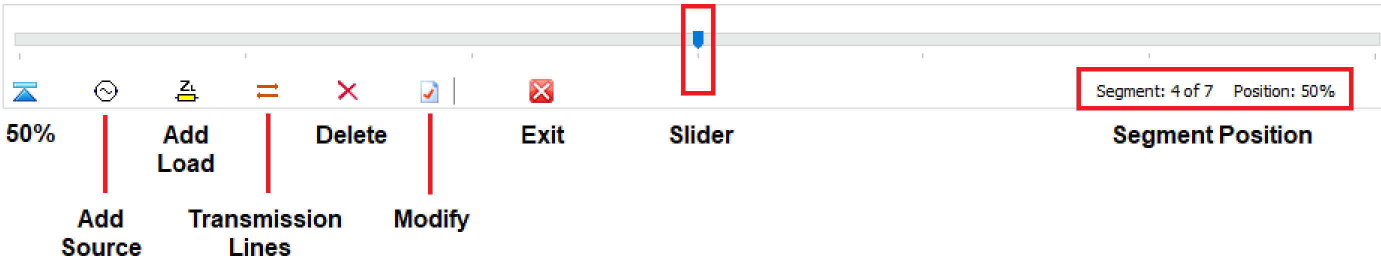


Fig. 1: The Source/Load/TL toolbar used to add and edit sources, loads, and transmission lines.

Toolbar Components

Slider

- The slider lets you choose a specific wire segment.
- Each slider position corresponds to a segment along the selected wire.
- On the right side of the toolbar, you'll see:
  - The segment number.
  - The segment's **relative position** (in %) along the wire, measured from the wire's starting point to the midpoint of the selected segment:

`Segment Position % = 100 × (segment position / wire length)`

50% Button

- Quickly positions the slider at the center of the wire.
- Useful for placing sources or loads at the midpoint.

> Note: The wire must have an **odd number of segments** for an exact center segment to exist.

Add Source Button

- Opens a dialog box to add a source to the selected segment (**Fig. 2**).
- You can specify:
  - Source type
  - Amplitude
  - Phase
  - Internal impedance Zs

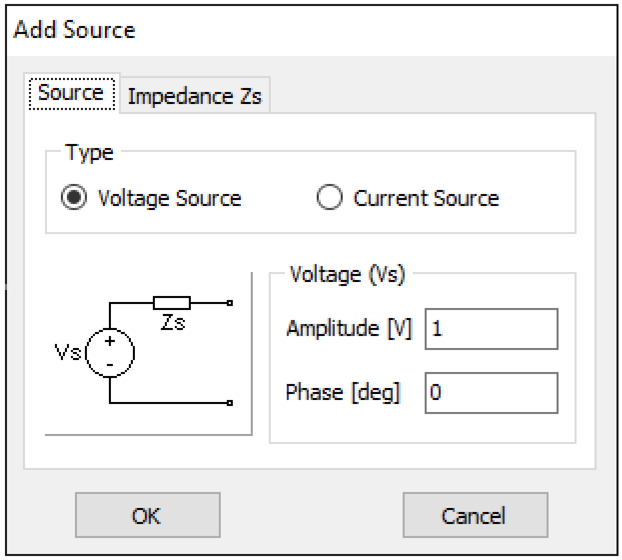


Fig. 2: Dialog box for adding a discrete source to a wire segment.

- **Voltage sources** have their internal impedance **in series**. To model an ideal (perfect) voltage source, set the impedance to **zero**.
- **Current sources** have their internal impedance **in parallel**. To approximate an ideal current source, use a **very high impedance** (e.g., **1 MΩ = 1E6 Ohm**).
- **To change the size of the source** displayed in the workspace, go to **Main Menu > Tools >** **Preferences** **> Workspace** tab, and adjust the **Source Size %** setting.
- To customize the default color of sources, go to: **Main Menu > Tools >** **Preferences** **> Workspace** tab.

Add Load Button

- Opens a dialog to add a load to the selected segment (**Fig. 3**).
- Load types include:
  - Series **RL** (resistor + inductor)
  - Series **RC** (resistor + capacitor)
  - Fixed **R + jX** impedance (non-frequency-dependent reactance)
  - Series or parallel **RLC**
  - **Trap** (series RL + parallel C)

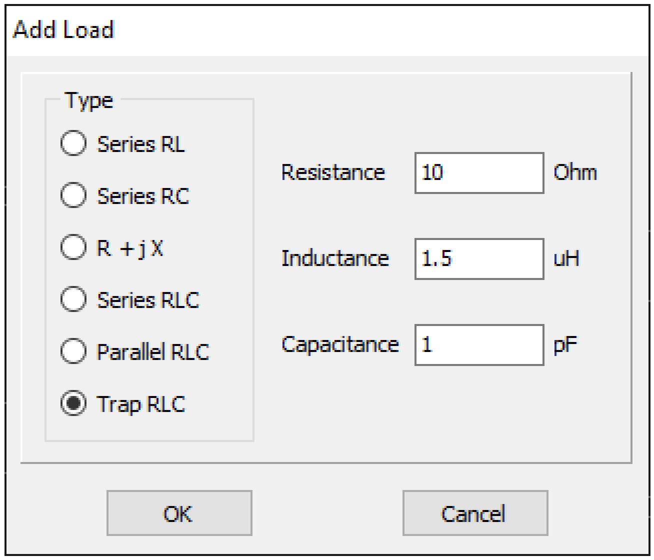


Fig. 3: Dialog box for adding a load to a wire segment.

To customize the default color of loads, go to: **Main Menu > Tools > Preferences > Workspace** tab.

Transmission Lines Button

- Opens a dialog to connect a transmission line to the selected segment.
- See the section **Adding Transmission Lines** for more information.

Delete Button

Removes any existing source or load from the selected segment.

Modify Button

Opens a dialog to edit the source or load on the selected segment.

Exit Button

Closes the Source/Load/TL toolbar.

Adding Sources

To add a source to a specific wire segment, follow these steps:

- **Right-click** on any part of the wire to open the **pop-up menu**.
- Select **Source / Load / TL (Ctrl + Ins)** to open the **Source / Load / TL toolbar** at the bottom of the screen (**Fig. 1**).
- Use the **slider** to choose the wire segment where the source will be placed.
- Click the **Add Source** button to open the *Add Source* dialog box (**Fig. 2**).

- Specify the **type of source**, its **amplitude** (rms value), **phase**, and **internal impedance**, then click **OK**.
- Click **Exit** to close the *Source / Load / TL* toolbar.

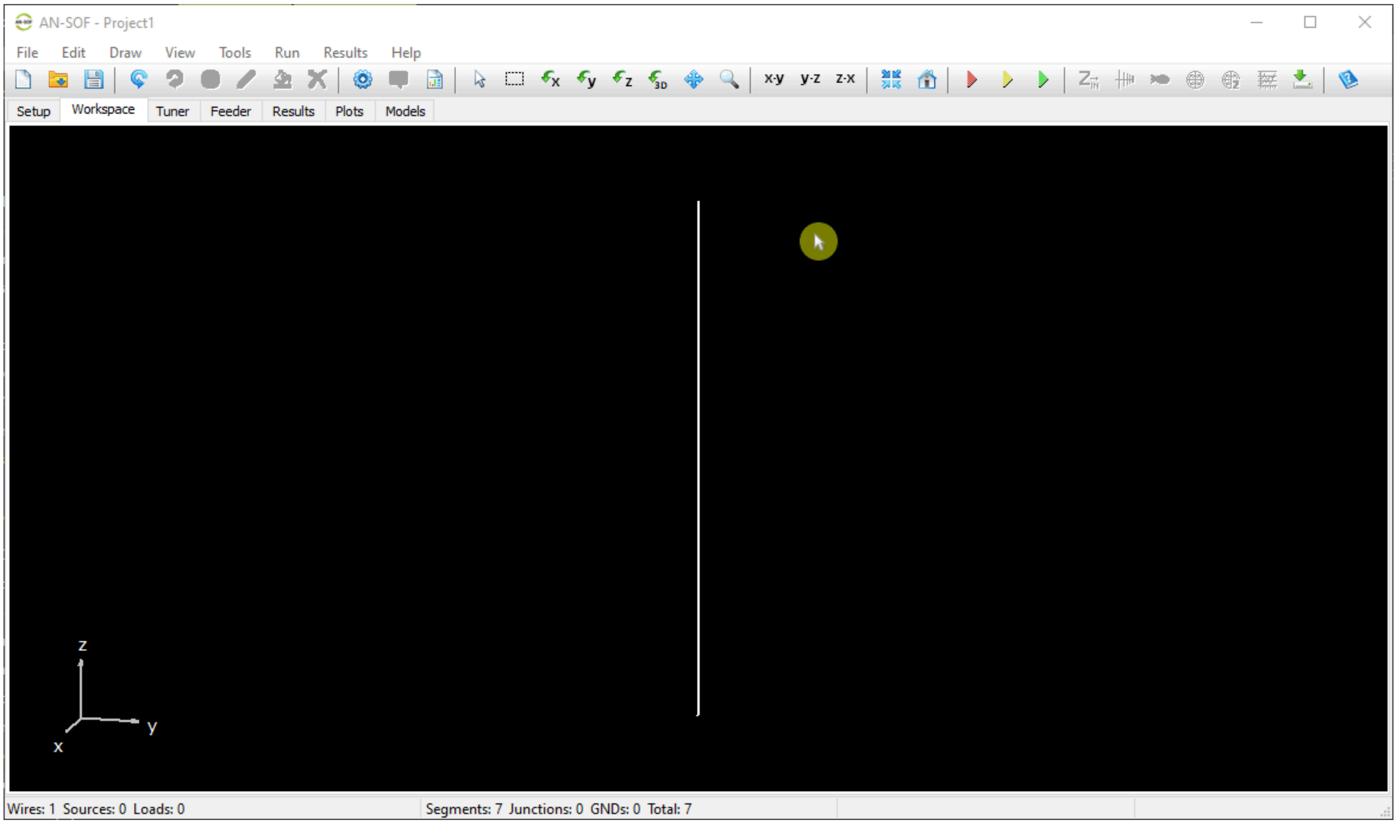


Fig. 1: Demonstration of how to add a source to a wire segment.

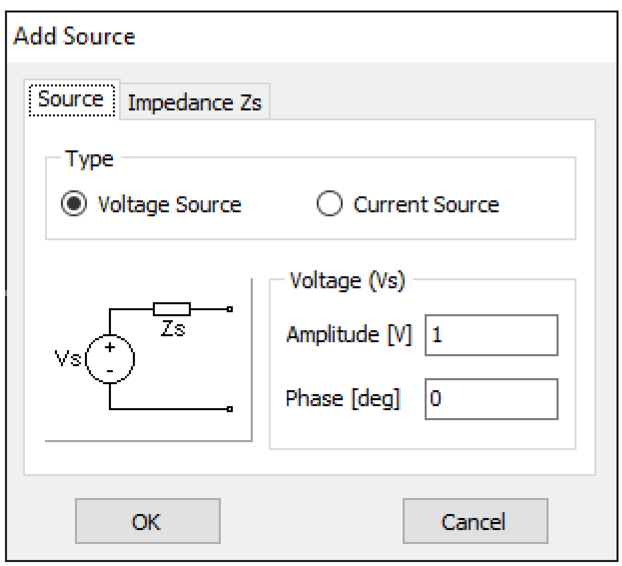


Fig. 2: Dialog box for adding a discrete source to a wire segment.

> **Tips**
>
> To customize the default color of sources, go to: **Main Menu > Tools > Preferences > Workspace** tab.
>
> **To change the size of the source** displayed in the workspace, go to **Main Menu > Tools > Preferences > Workspace** tab, and adjust the **Source Size %** setting.
>
> **Voltage sources** have their internal impedance **in series**.
> To model an ideal (perfect) voltage source, set the impedance to **zero**.
>
> **Current sources** have their internal impedance **in parallel**.
> To approximate an ideal current source, use a **very high impedance** (e.g., **1 MΩ = 1E6 Ohm**).

Editing Sources

To edit an existing source on a wire segment, follow these steps:

- **Right-click** on any part of the wire to open the pop-up menu.
- Select the **Source / Load / TL (Ctrl + Ins)** command to open the Source/Load/TL toolbar (**Fig. 1**).
- Use the **slider** to locate the segment where the source is placed.
- Click the **Modify** button to open a dialog box where you can edit the source parameters.
  To remove the source, click the **Delete** button.
- Click the **Exit** button to close the *Source / Load / TL* toolbar.

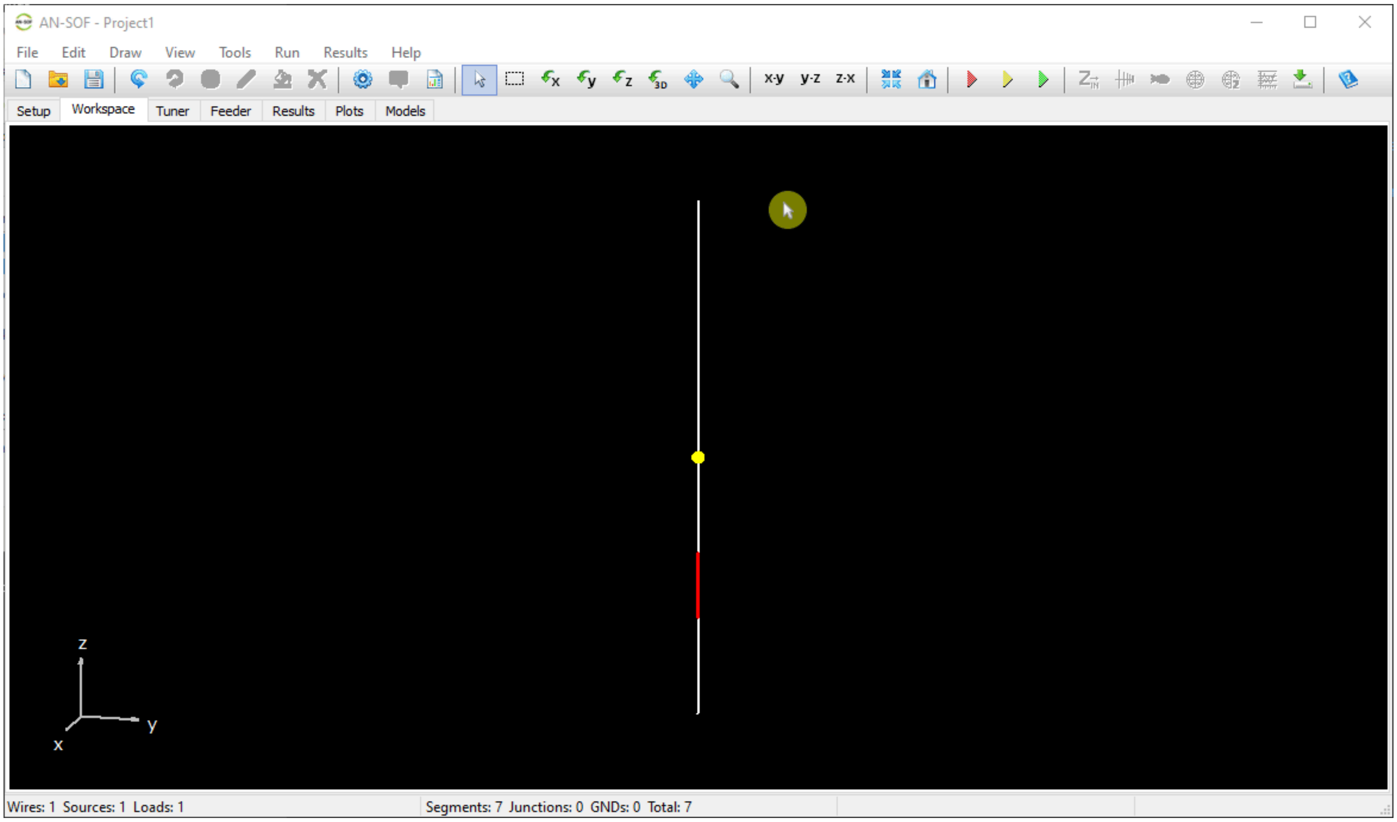


Fig. 1: Editing a source on a wire segment.

> **Tips**
>
> To customize the default color of sources, go to: **Main Menu > Tools > Preferences > Workspace** tab.
>
> **To change the size of the source** displayed in the workspace, go to **Main Menu > Tools > Preferences > Workspace** tab, and adjust the **Source Size %** setting.
>
> **Voltage sources** have their internal impedance **in series**.
> To model an ideal (perfect) voltage source, set the impedance to **zero**.
>
> **Current sources** have their internal impedance **in parallel**.
> To approximate an ideal current source, use a **very high impedance** (e.g., **1 MΩ = 1E6 Ohm**).

Adding Loads

To add a load to a selected wire segment, follow these steps:

- **Right-click** on any part of the wire to display the pop-up menu.
- Select the **Source / Load / TL (Ctrl + Ins)** command from the menu to open the Source / Load / TL toolbar (**Fig. 1**) at the bottom of the screen.
- Use the **slider** on the toolbar to select the desired wire segment.
- Click the **Add Load** button to open the *Add Load* dialog box (**Fig. 2**).

- Specify the **load type**, along with the required RLC values, then click **OK**.
- Click the **Exit** button to close the *Source/Load/TL* toolbar.

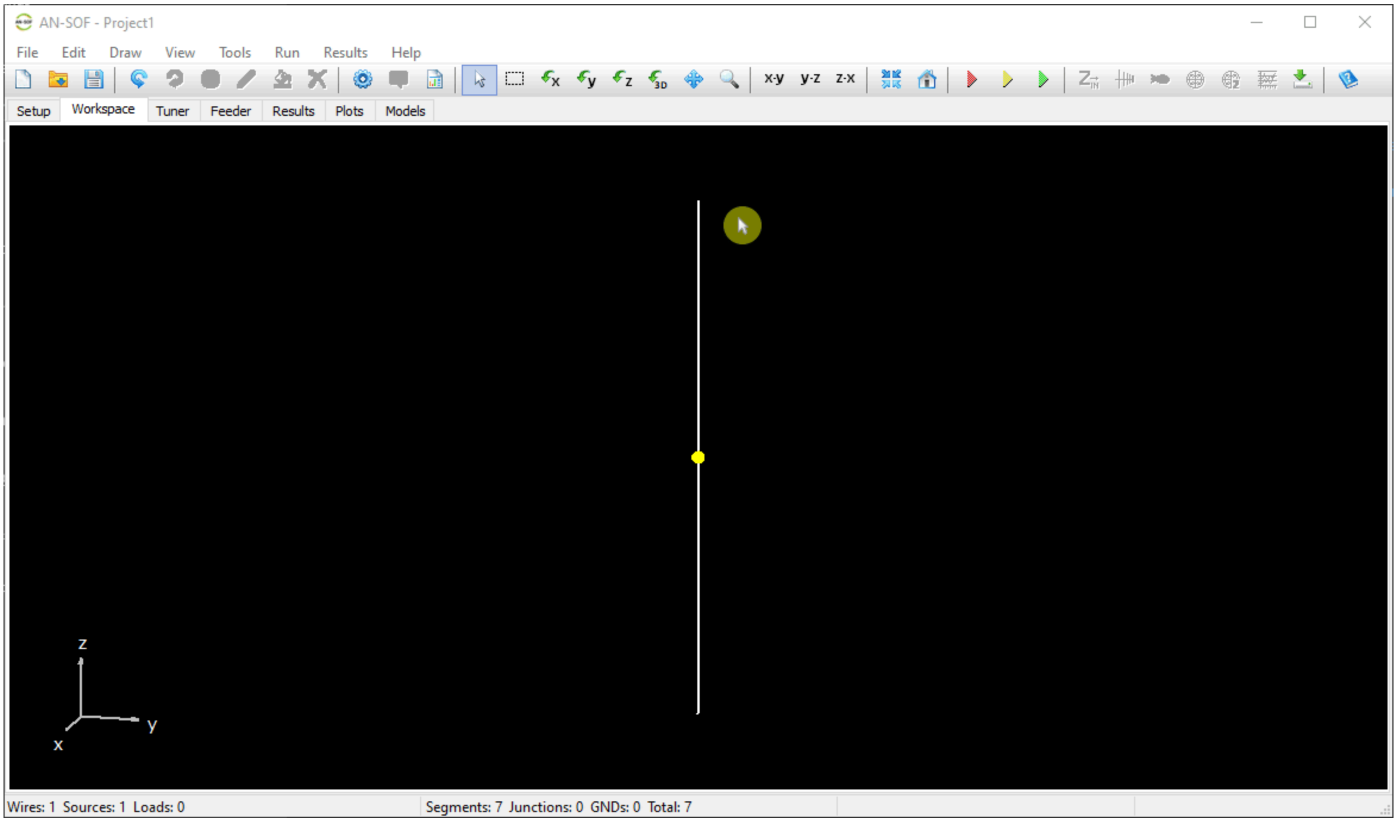


Fig. 1: Demonstration of how to add a load to a wire segment.

Load types include (**Fig. 2**):

- Series **RL** (resistor + inductor)
- Series **RC** (resistor + capacitor)
- Fixed **R + jX** impedance (non-frequency-dependent reactance)
- Series **RLC**
- Parallel **RLC**
- **Trap** (series RL + parallel C)

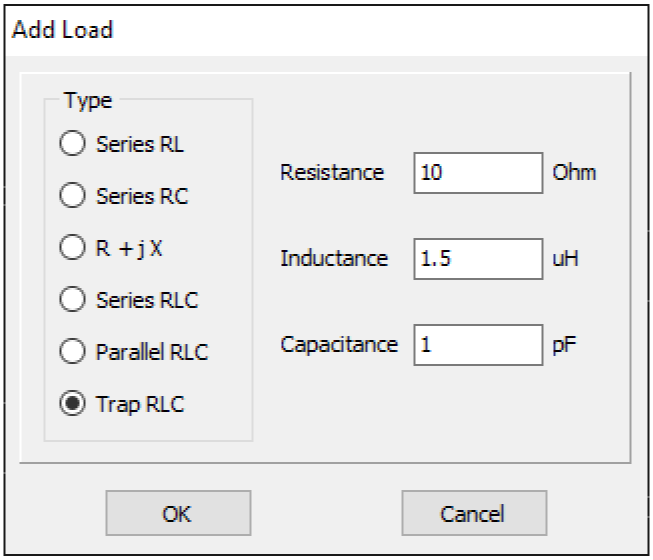


Fig. 2: Dialog box for adding a load impedance to a wire segment.

> **Tip**
>
> To customize the default color of loads, go to: **Main Menu > Tools > Preferences > Workspace** tab.

Editing Loads

To edit a load on a wire segment, follow these steps:

- **Right-click** on any part of the wire to open the pop-up menu.
- Select **Source / Load / TL (Ctrl +Ins)** from the menu to display the **Source / Load / TL toolbar**.
- Use the **slider** on the toolbar to select the segment where the load is located.
- Click the **Modify** button to open the **Edit Load** dialog box, where you can update the load's parameters.
- To remove the load, click the **Delete** button.
- Click the **Exit** button to close the toolbar.

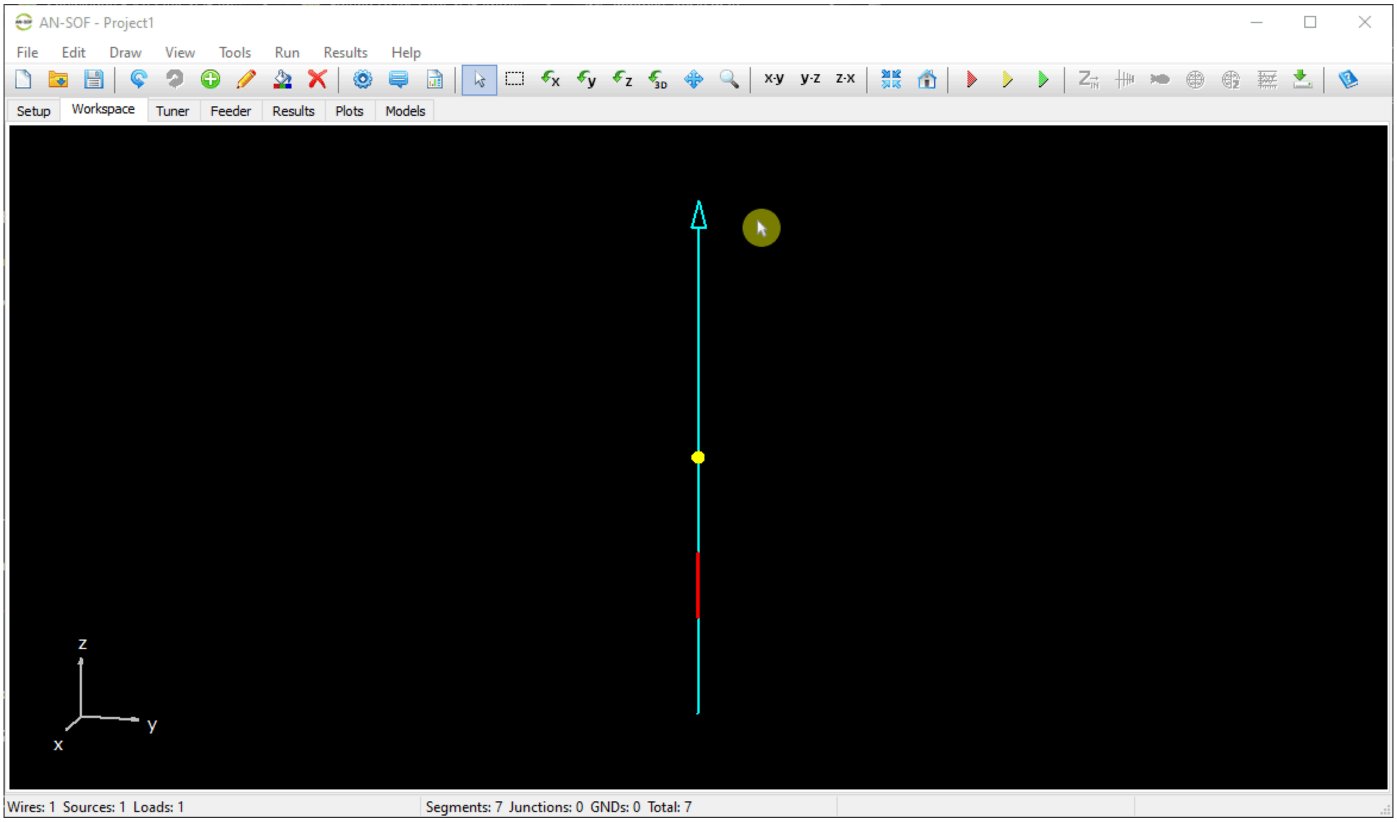


Fig. 1: Editing a load on a wire segment.

> **Tip**
>
> To customize the default color of loads, go to: **Main Menu > Tools > Preferences > Workspace** tab.

Enabling/Disabling Loads

You can enable or disable all loads in the simulation without deleting them. This is useful when you want to temporarily exclude load impedances from the analysis.

1. Go to the **Setup tab** and locate the **Settings panel** in the main window.
2. Check or uncheck the **Load Impedances** option:
    - **Checked**: Loads are enabled and included in the simulation.
    - **Unchecked**: Loads are disabled but remain defined in the model.

See **Fig. 1** for reference.

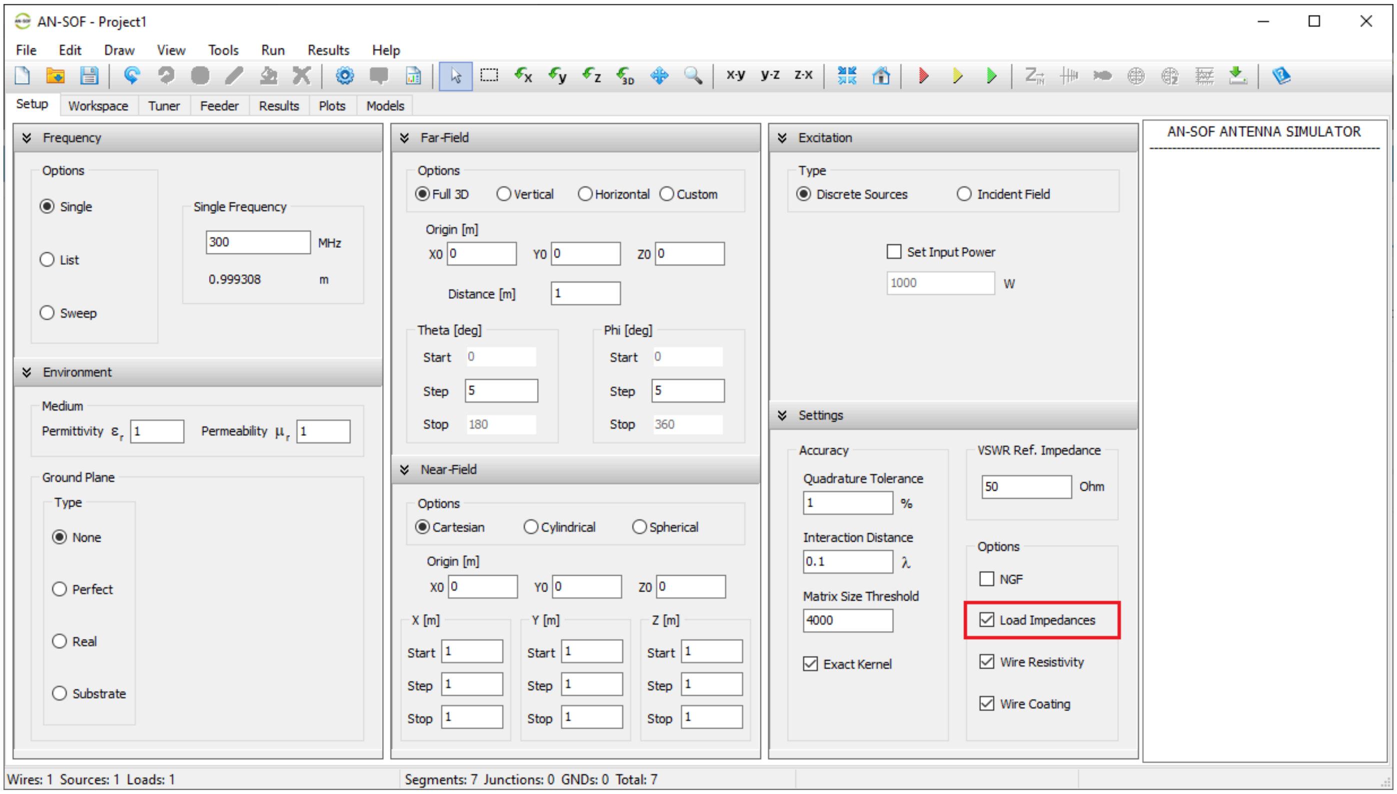


Fig. 1: "Load Impedances" option in the Settings panel of the Setup tab.

Incident Field

Excitation by an Incident Field

When you want to simulate how an antenna or a metallic structure (like an aircraft or a tower) reacts to external electromagnetic waves, such as for **Radar Cross Section (RCS)** or **scattered field analysis**, you use an **incident field**.

To configure this in AN-SOF:

1. Navigate to the **Setup** tab.
2. Locate the **Excitation** panel.
3. Select the **Incident Field** radio button (**Fig. 1**).

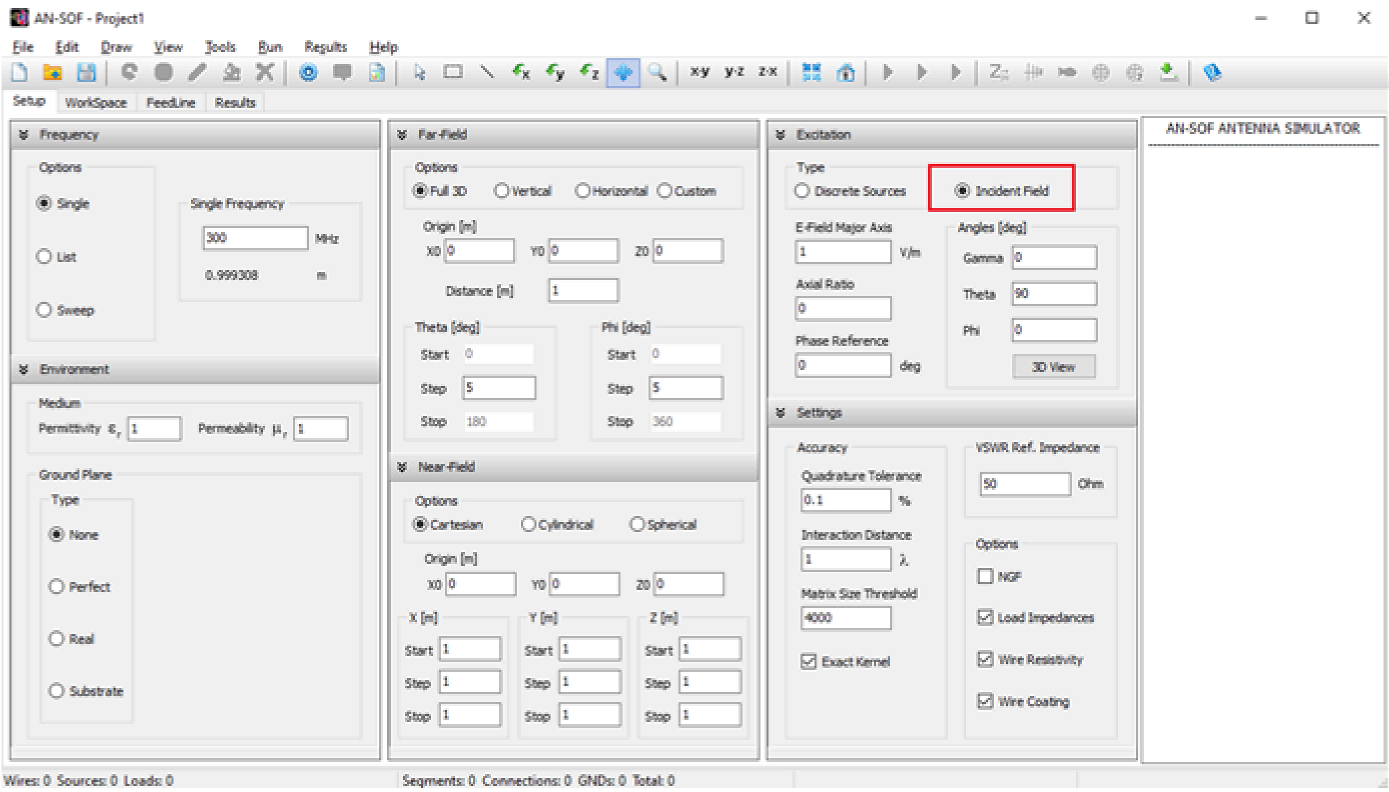


Fig. 1: *"Incident Field" option in the Excitation panel of the Setup tab.*

Important Note on Discrete Sources

If you have previously defined **discrete sources** (voltage or current sources) on specific wire segments, AN-SOF will **ignore** them once the **Incident Field** option is active. The structure will be treated as a passive scatterer reacting only to the external plane wave.

> **Note**
>
> When an incident plane wave is used as the excitation, the calculated **far-field** and **near-field** results represent the **scattered fields**. The resulting radiation pattern is the **scattered field pattern**, observed in the far-field region where the scattered field amplitude decays with the inverse of the distance (1/r), and the scattered power density decays with the inverse square of the distance ($1/r^2$) from the structure.

Incident Field Parameters

Once the **Incident Field** option is selected in the **Excitation** panel, you must define the characteristics of the incoming plane wave. These parameters determine the wave's strength, orientation, and polarization.

1. Electric Field & Polarization

- **E-Field Major Axis:** The amplitude of the incident electric field in **V/m (Volts RMS per meter)**. For linear polarization, this is the field strength. For elliptical polarization, it represents the length of the major axis of the ellipse.

- **Axial Ratio:** Defines the "roundness" of the polarization.
  - **0:** Linearly polarized.
  - **Positive value:** Right-handed elliptical polarization.
  - **Negative value:** Left-handed elliptical polarization.
- **Gamma** (γ): The polarization angle in degrees. For linear waves, this is the angle between the plane of incidence and the E-field vector. For elliptical waves, it marks the angle to the major axis (**Fig. 1**).

2. Direction of Arrival

The direction from which the plane wave originates is defined using spherical coordinates:

- **Theta** (θ): The zenith angle of the incident direction.
- **Phi** (ϕ): The azimuth angle of the incident direction.

3. Phase Control

Phase Reference: The phase of the wave (in degrees) at the coordinate origin (0,0,0). Changing this value simply shifts the phase of all induced currents and scattered fields across the entire structure by the specified amount.

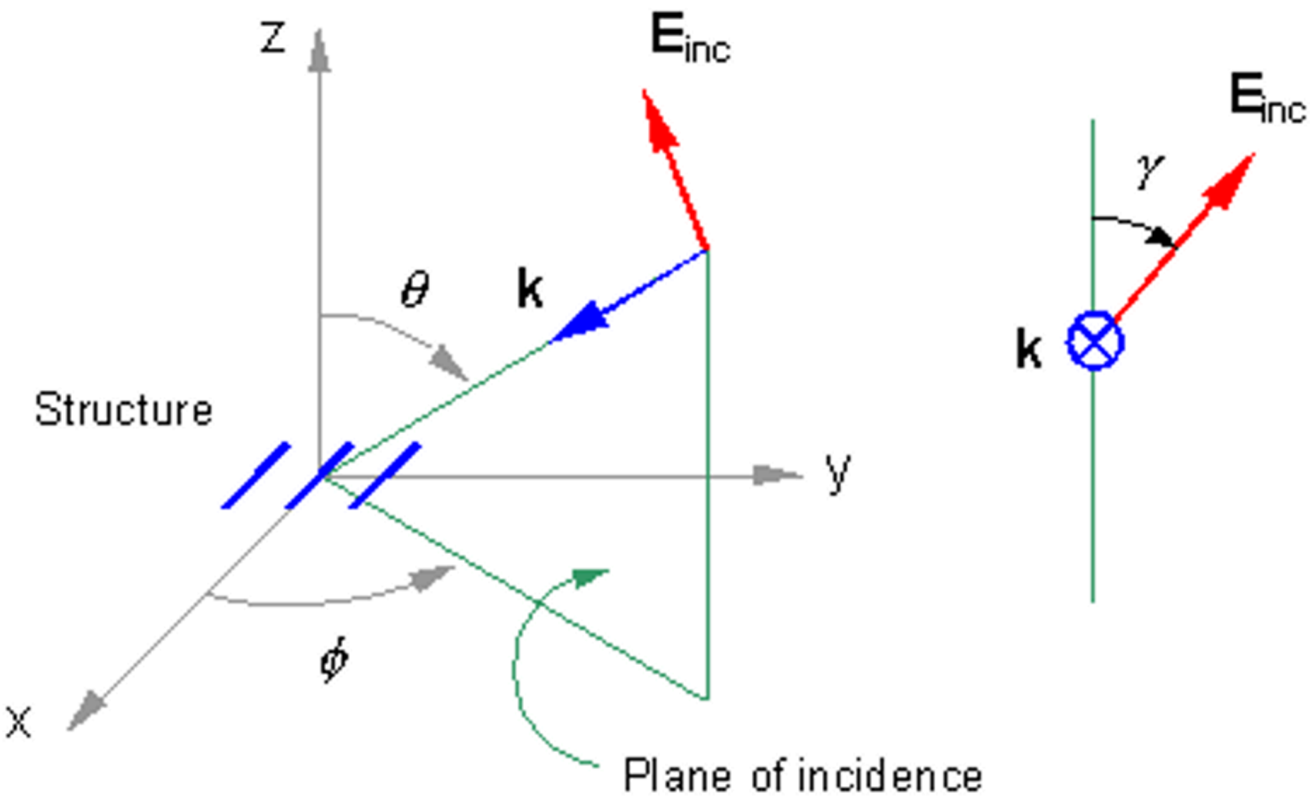


__Fig. 1:__ Parameters of an incident field, defining the direction, polarization, and orientation of the incoming plane wave.

> **Note**
>
> If the incident plane wave option is active, the simulation treats the structure as a **passive scatterer**. Any defined voltage or current sources are excluded from the calculation.

The 3D-View Interface

The **3D-View interface** provides an intuitive, graphical method for configuring incident field parameters, allowing you to visualize the direction and polarization of the incoming wave relative to your structure.

Step-by-Step Configuration

1. **Activate Incident Field:** Go to the **Setup** tab and select the **Incident Field** option within the **Excitation** panel.
2. **Launch the Interface:** Click the **3D View** button. This opens the 3D-View window along with the **Incident Wave** dialog box (**Fig. 1**).
3. **Adjust Angles:** Enter your desired values for **Gamma**, **Theta**, and **Phi**, then press **ENTER**. Alternatively, you can use the incremental arrows to fine-tune these angles while observing the visual representation in the 3D space.
4. **Confirm and Sync:** Close the **Incident Wave** dialog box. The values you set graphically are automatically synchronized and will now appear in the **Excitation** panel of the main interface (**Fig. 2**).

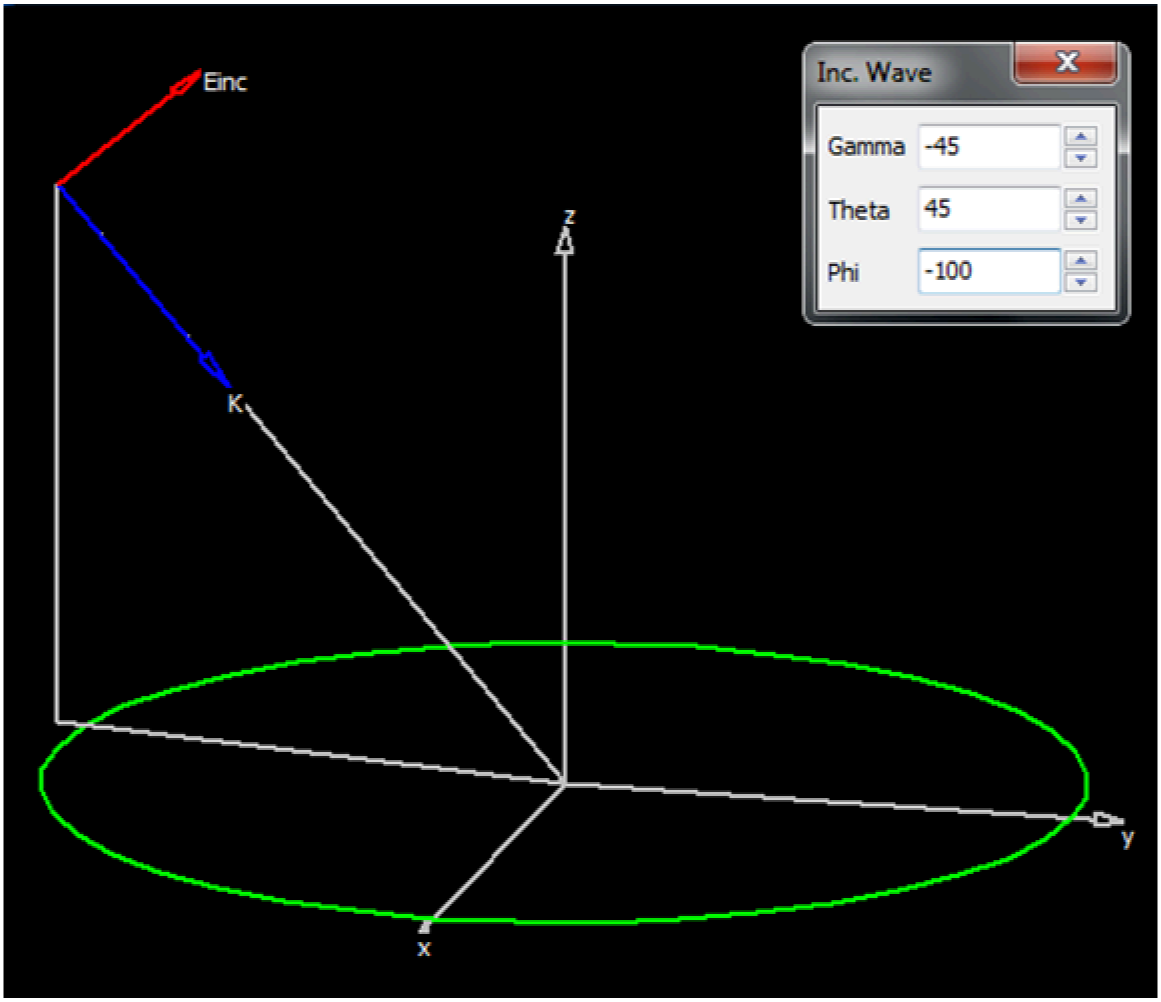


Fig. 1: *3D-View interface for the definition of the incident field, featuring the Incident Wave dialog box. In this configuration, the parameters are set to* $\gamma=-45^\circ$, $\theta=45^\circ$, *and* $\phi=-100^\circ$.

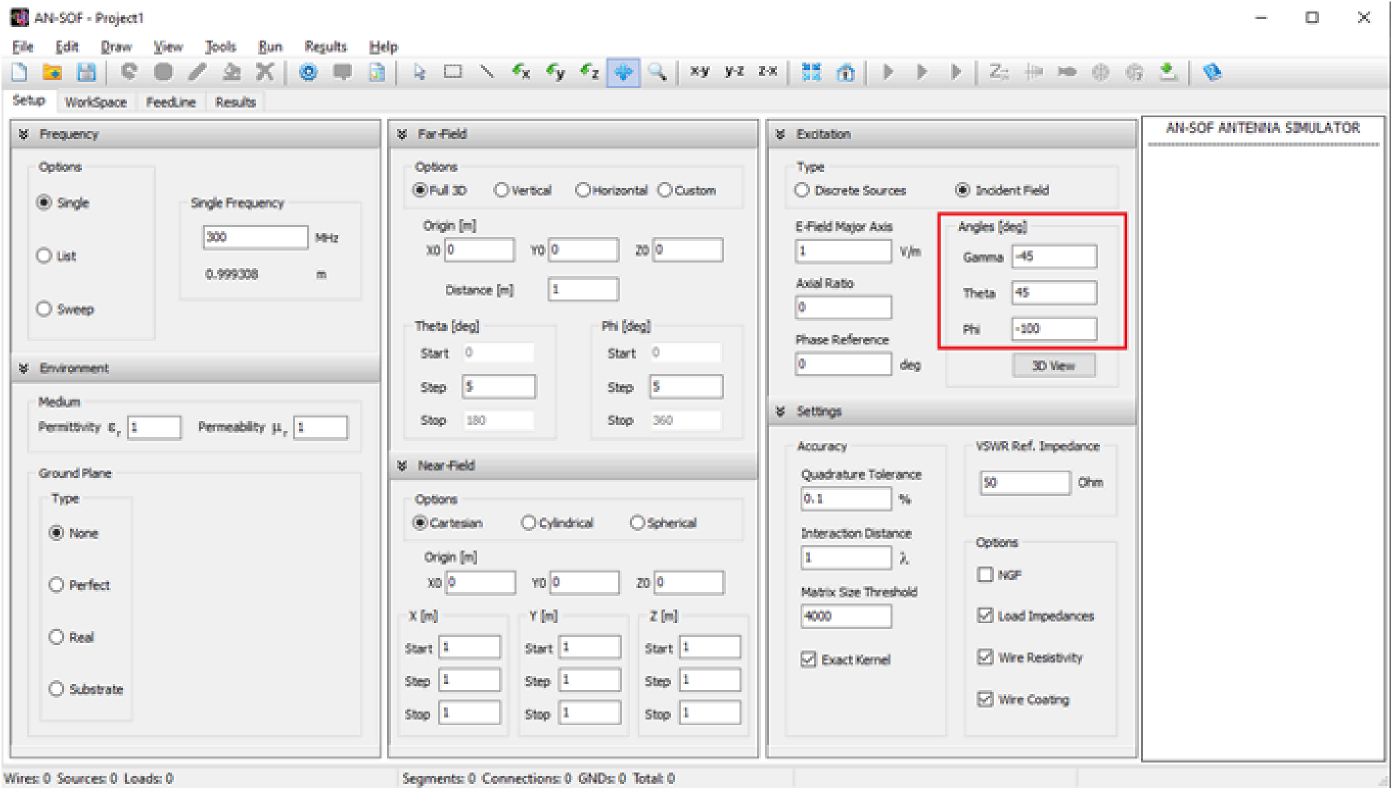


Fig. 2: *The Gamma (γ), Theta (θ), and Phi (ϕ) angles entered in the Incident Wave dialog box are automatically synchronized and will appear in the Excitation panel of the Setup tab.*

Why Use 3D-View?

Using the graphical interface is highly recommended for complex geometries where the "Plane of Incidence" might be difficult to visualize numerically. It ensures that the **Gamma** (polarization) angle is oriented exactly as intended relative to the physical orientation of your wires or surfaces.

Ground Planes

Adding a PEC Ground Plane

A perfectly electric conducting (PEC) ground plane, parallel to the xy-plane, can be added to the model using the following procedure:

1. Navigate to the **Setup tab > [Environment panel](#)**.
2. Select the **Perfect** option in the **Ground Plane** box (see **Fig. 1**).
3. Set the ground plane position under the **Position** label (Z-coordinate).

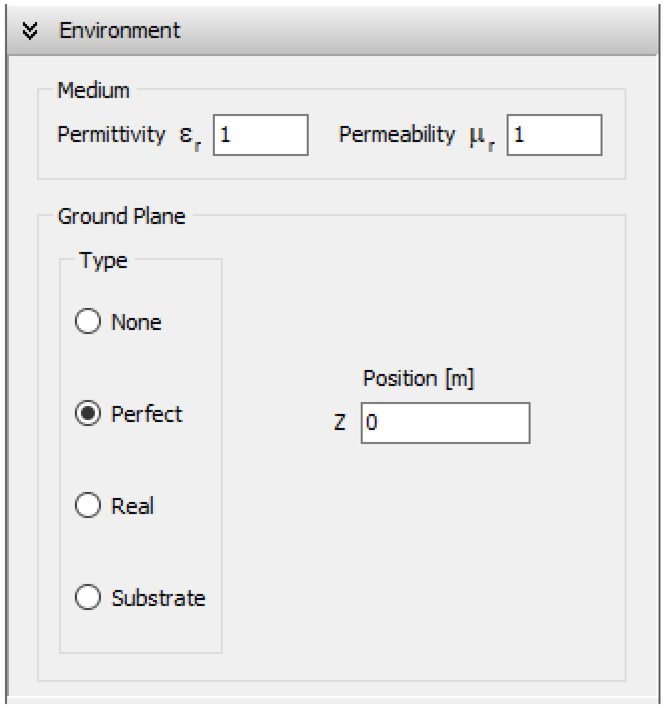


Fig. 1: Perfect option in the Ground Plane box of the Environment panel.

When the **Perfect** ground is selected, an infinite PEC ground plane will be placed at the specified Z-coordinate relative to the xy-plane:

- If **Z > 0**, the PEC ground plane will be **above** the xy-plane.
- If **Z = 0**, the PEC ground plane will be **on** the xy-plane.
- If **Z < 0**, the PEC ground plane will be **below** the xy-plane.

The ground plane is represented as a square with cross diagonals to visualize its position (see **Fig. 2**). Note that this is only a symbolic representation, as **the ground plane itself is infinite in extent**.

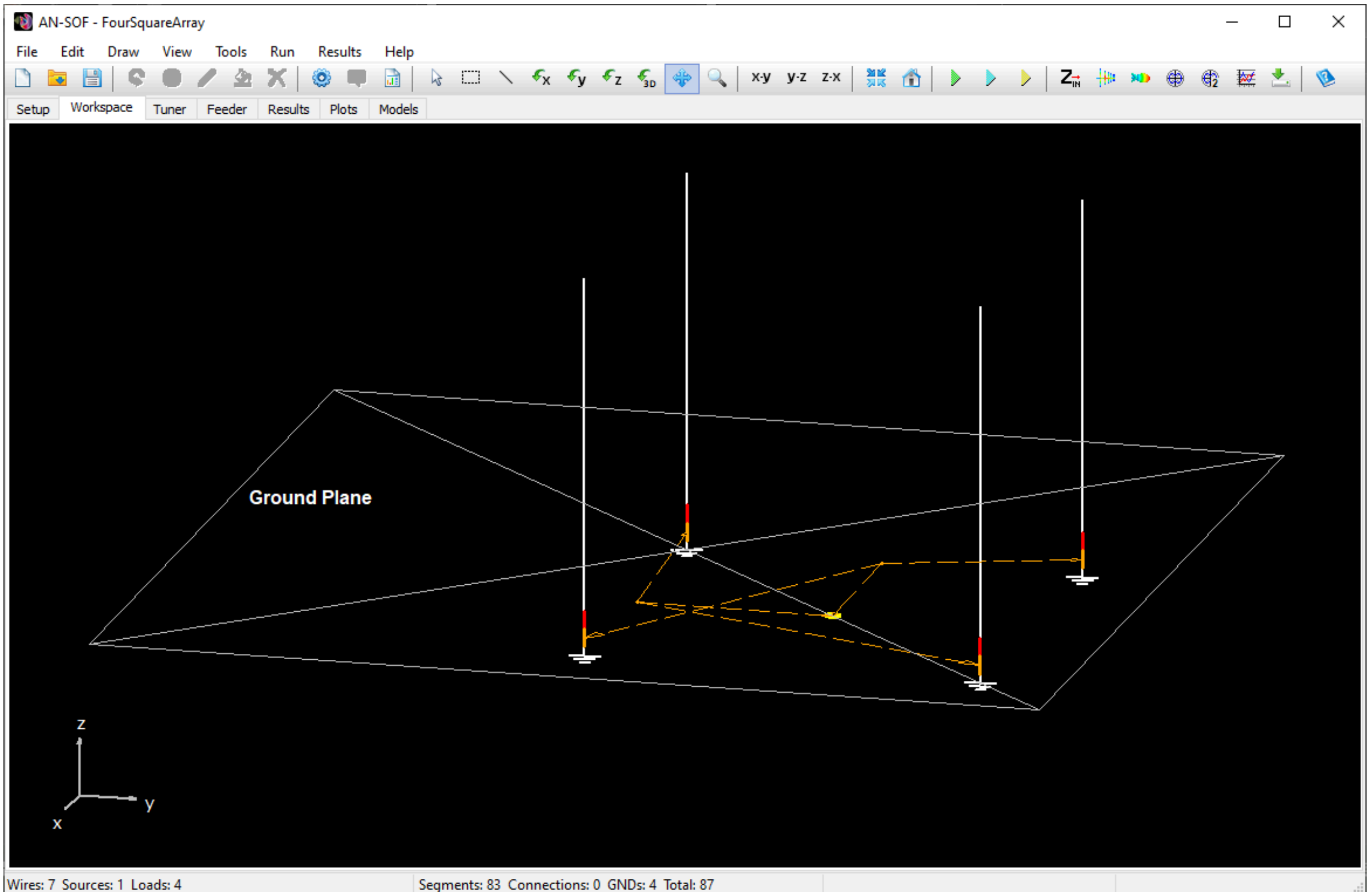


Fig. 2: Ground plane symbol in the workspace, indicating the position of the ground plane.

Adding a Real Ground Plane

A real (i.e., imperfect or lossy) ground plane, located on the xy-plane (Z = 0), can be added to the model using the following procedure:

1. Navigate to the **Setup tab > Environment panel**.
2. Select the **Real** option in the **Ground Plane** box (see **Fig. 1**).
3. Specify the **Real Ground Option**:
   - **Sommerfeld-Wait Asymptotic**
   - **Reflection Coefficients**
   - **Radial Wire Ground Screen**
   - **Sommerfeld-Norton**
4. Set the ground **Permittivity** and **Conductivity**. If a **Radial Wire Ground Screen** is selected, also specify the **radial length**, **number of radials**, and **wire radius**.

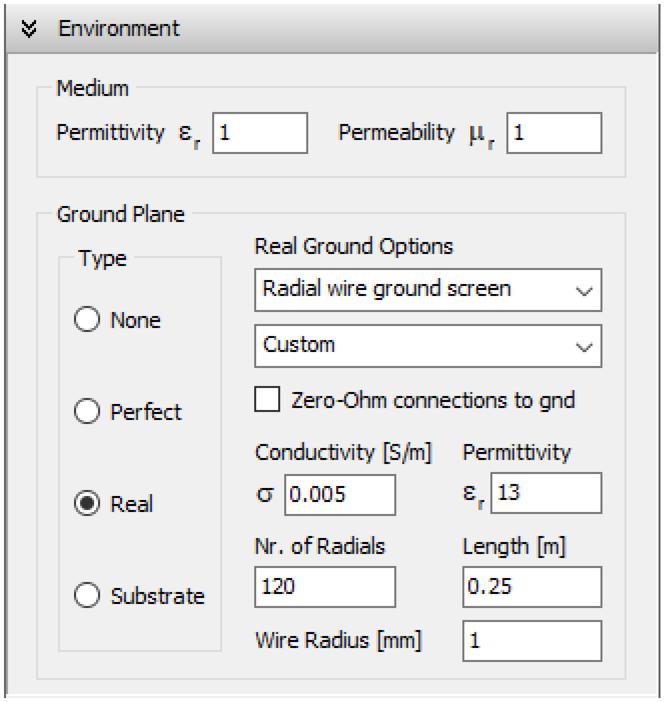


Fig. 1: Real option in the Ground Plane box of the Environment panel.

The ground plane is represented as a square with cross diagonals to indicate its position (see **Fig. 2**). Note that this is only a symbolic representation, as **the ground plane itself is infinite in extent**. When a **Radial Wire Ground Screen** is selected, the radial wires lying on the ground will be displayed instead of the ground plane symbol (see **Fig. 3**).

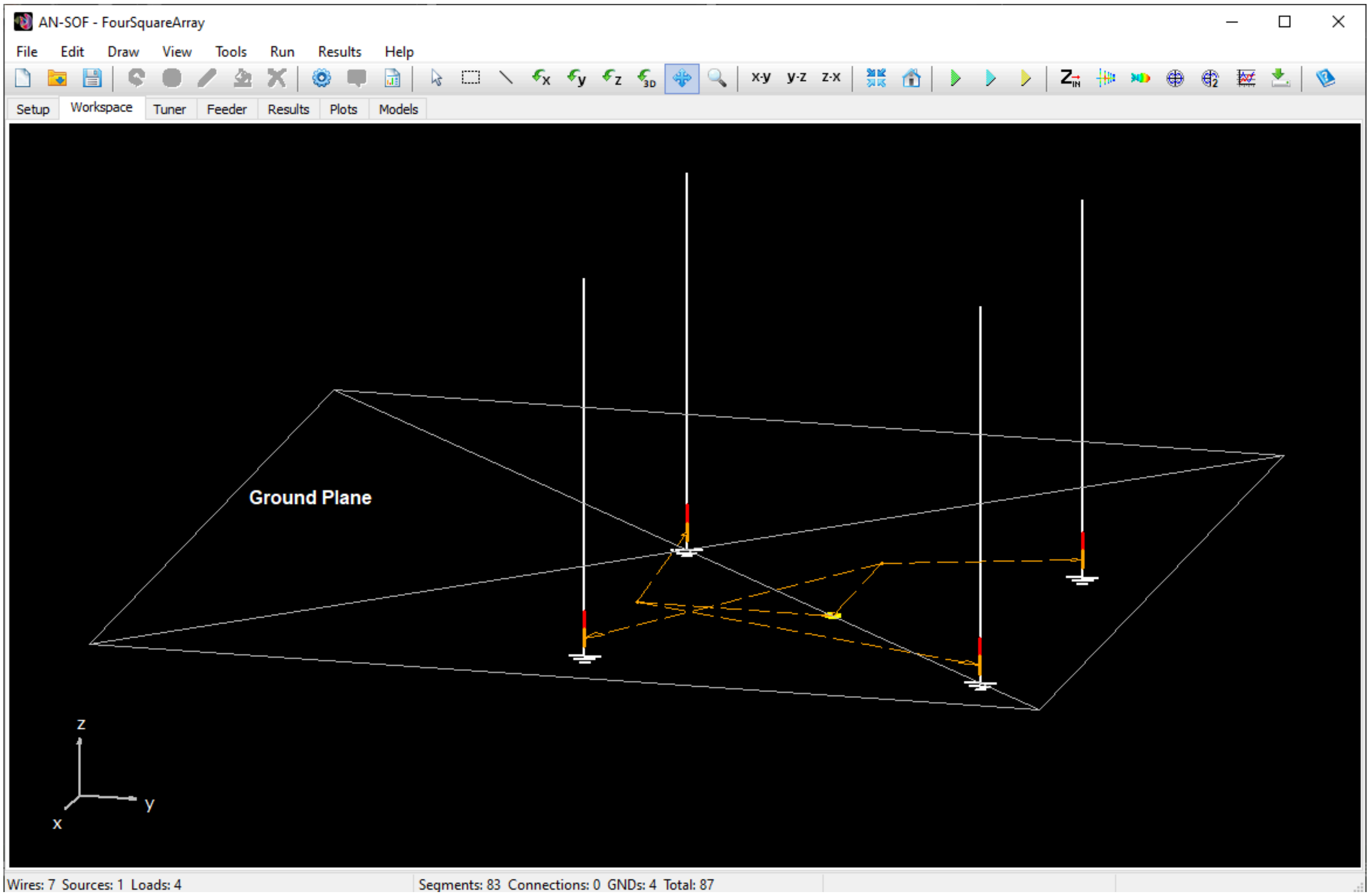


Fig. 2: Ground plane symbol in the workspace, indicating the position of the ground plane.

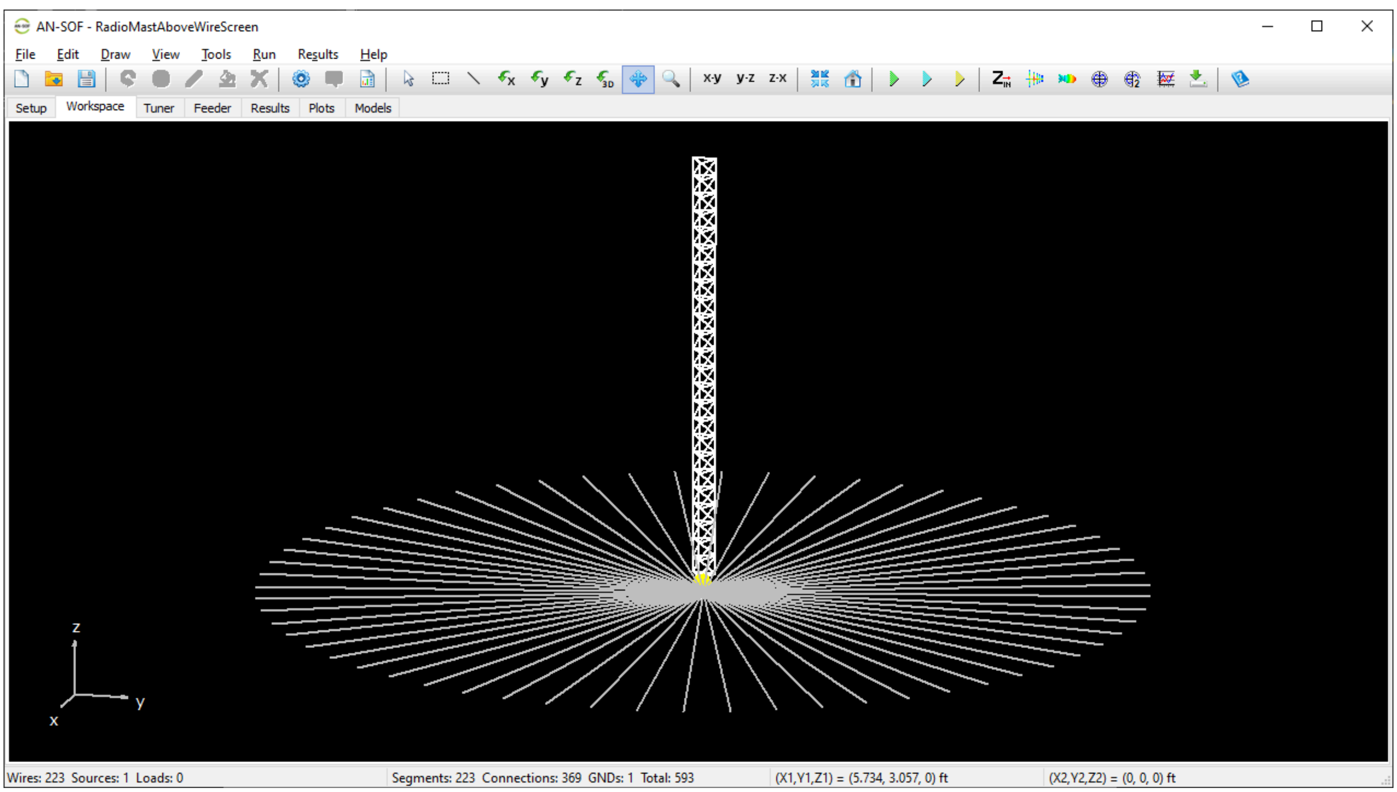


Fig. 3: Radial wire ground screen in the workspace, showing the position of the ground plane and ground screen.

Adding a Dielectric Substrate

To incorporate a **dielectric substrate** beneath the xy-plane (Z < 0) into the model, follow these steps:

1. Navigate to the **Setup tab > Environment panel**.
2. In the **Ground Plane** box, select the **Substrate** option (see **Fig. 1**).
3. Choose between an **infinite** or **finite** slab in the **Substrate Slab Options** box.
4. Select a substrate material from the provided list, or choose **Custom** to specify the substrate's **Permittivity**. Set the slab's **Thickness (h)** and, if a finite slab is selected, configure its dimensions along the **X** and **Y** axes.

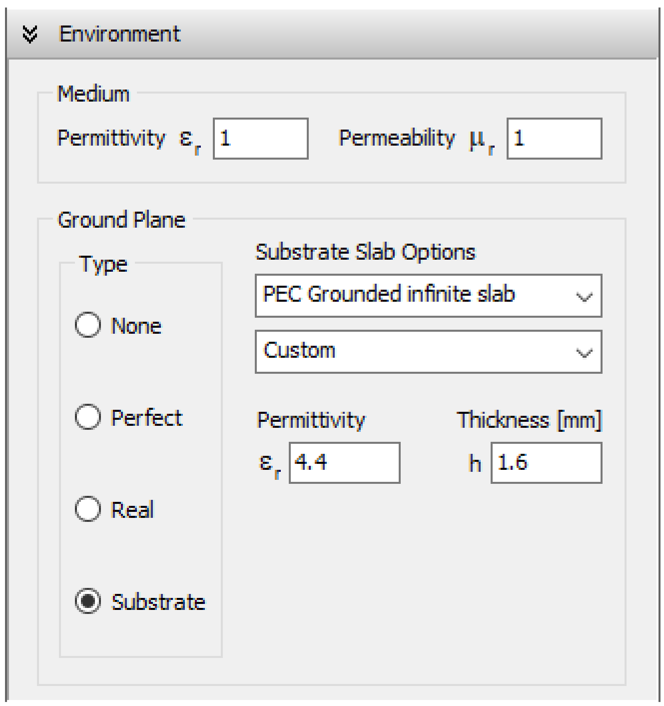


Fig. 1: Substrate option in the Ground Plane box of the Environment panel.

Note:

The substrate slab is backed by a **PEC ground plane**, which runs parallel to the xy-plane at **Z = -h. This ground plane cannot be removed from the simulation** (see **Fig. 2**).

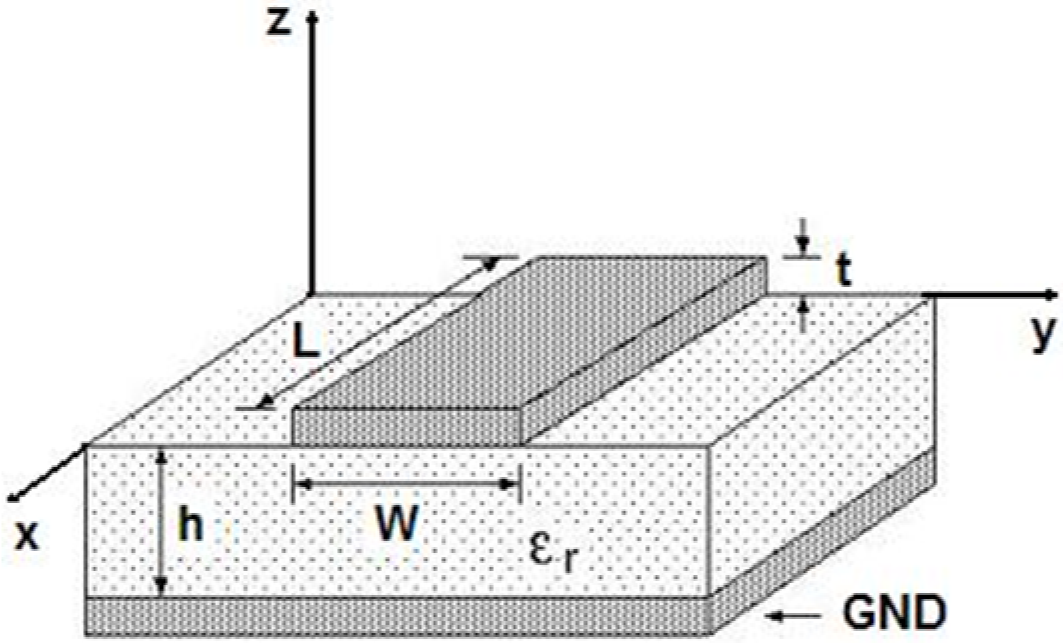


Fig. 2: Dielectric substrate positioned below the xy-plane, with a microstrip line placed above the xy-plane.

Connecting Wires to the Ground

A wire will automatically connect to the ground plane when the z-coordinate of one of its ends coincides with the position of the ground plane.

- When a **PEC ground plane** is selected, the ground position is specified by the **Z** value in the **Environment panel > Ground Plane** box.
- When a **real ground** is selected, the ground position is **Z = 0** (xy-plane).
- When a **substrate** is selected, a PEC ground plane is placed at **Z = -h**, where **h** is the substrate thickness.

Wire connections to the ground plane are indicated with **3D symbols** (see **Fig. 1**).

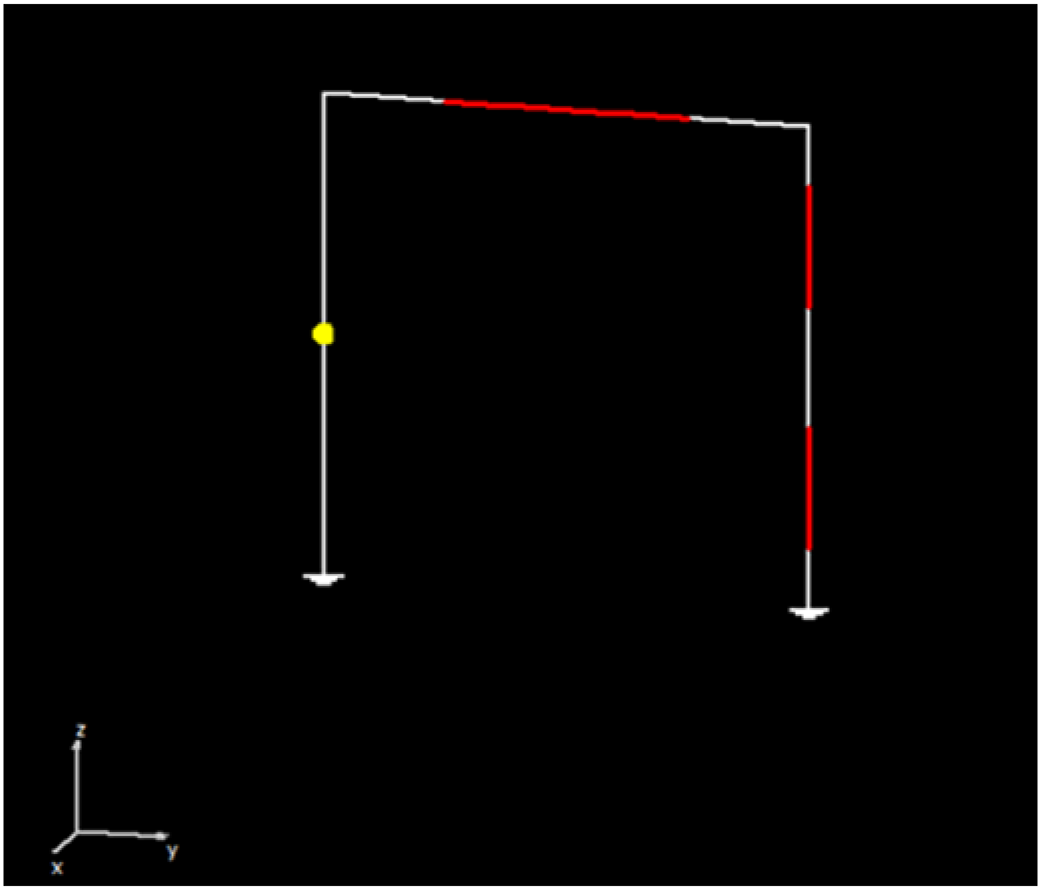


Fig. 1: 3D symbols indicating wire-to-ground connections.

> **WARNING!**
>
> - All wires must be **above** the ground plane.
> - Wires that cross the ground plane from one side to the other are **not allowed**.

Removing the Ground Plane

To remove the ground plane, follow these steps:

1. Navigate to the **Setup tab > Environment panel**.
2. Select the **None** (free space) option in the **Ground Plane** box (see **Fig. 1**).

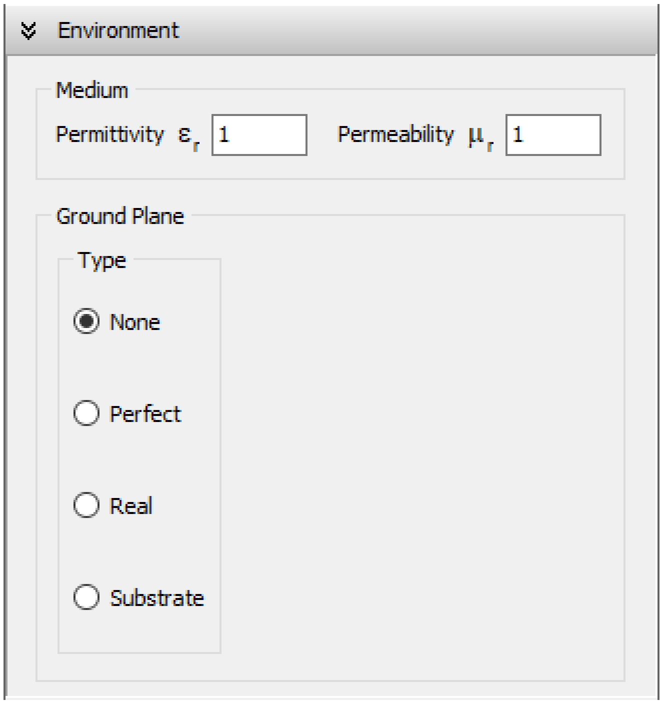


Fig. 1: "None" option in the Ground Plane box of the Environment panel.

Running Calculations

Calculating the Current Distribution

Once the **frequencies**, **environment**, **geometry**, and **excitation** are set, AN-SOF is ready to compute the currents flowing on the wire segments.

To calculate the current distribution:

Navigate to **Run > Run Currents (Ctrl + R)** in the main menu (see **Fig. 1**).

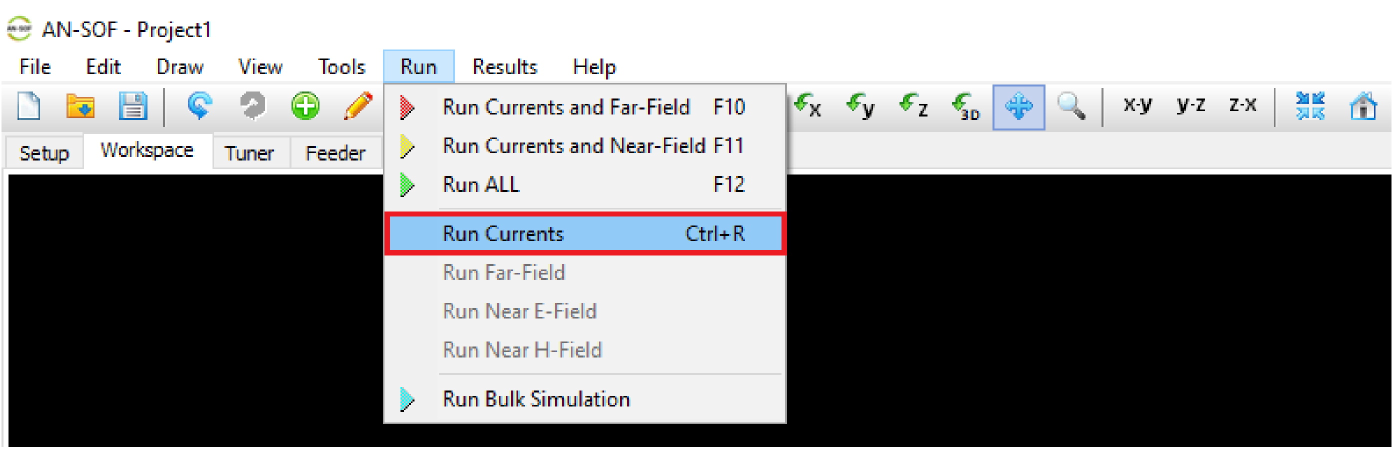


Fig. 1: The "Run Currents (Ctrl + R)" command in the main menu.

> **Tip**
>
> When modeling a transmitting antenna and only the **input impedance** and **VSWR/S_{11}** are needed, the **Run Currents (Ctrl + R)** command saves time by skipping the calculation of the radiated field.

Calculating the Far Field

Once the current distribution on the structure has been calculated, the far-field can be computed within the angular ranges specified in the **Far-Field panel** of the **Setup tab**.

To calculate the far-field:

Navigate to **Run > Run Far-Field** in the main menu (see **Fig. 1**).
This command is enabled only after the current distribution has been calculated.

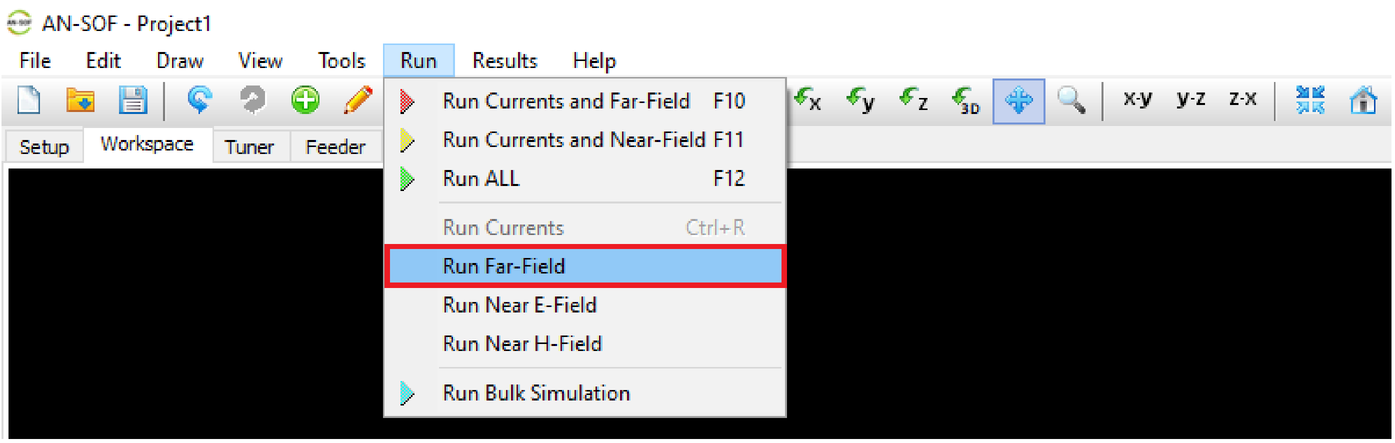


Fig. 1: The “Run Far-Field” command in the main menu.

> **Tip**
>
> To sequentially and automatically calculate both the **current distribution and the far field**, click the **Run Currents and Far-Field (F10)** button on the toolbar.

> **Note**
>
> When an **incident plane wave** is used as the excitation, the calculated **far-field** results represent the **scattered field**. The resulting radiation pattern is the **scattered field pattern**, observed in the far-field region where the scattered field amplitude decays with the inverse of the distance (1/r), and the scattered power density decays with the inverse square of the distance ($1/r^2$) from the structure.

Calculating the Near E-Field

Once the current distribution on the structure has been calculated, the near electric field can be computed at the points in space specified in the **Near-Field panel** of the **Setup tab**.

To calculate the near electric field:

> Navigate to **Run > Run Near E-Field** in the main menu (see **Fig. 1**).
>
> This command is enabled only after the current distribution has been calculated.

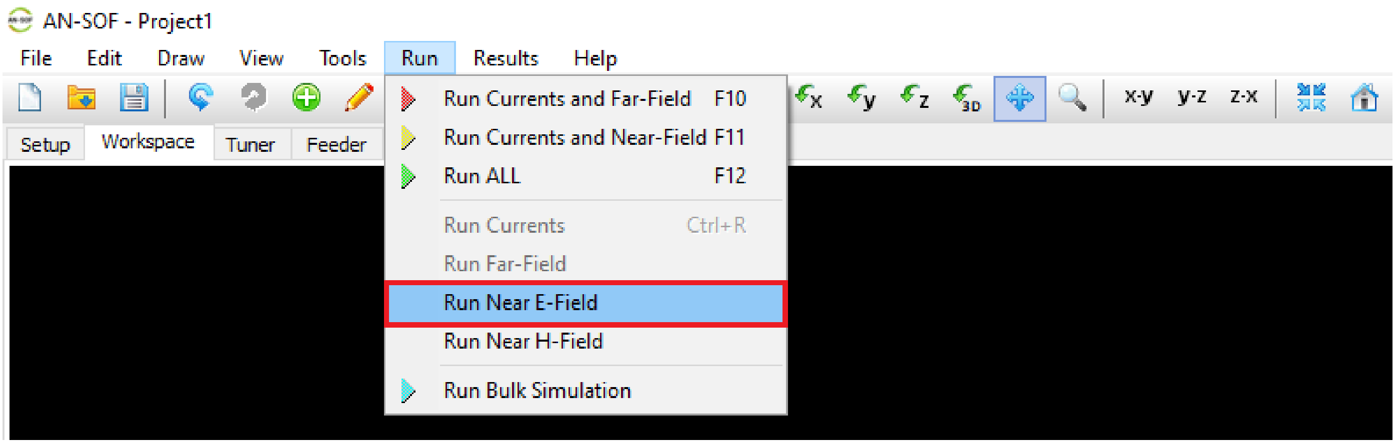


Fig. 1: The "Run Near E-Field" command in the main menu.

> **Tips**
>
> To sequentially and automatically calculate both **the current distribution and the near fields**, click the **Run Currents and Near-Field (F11)** button on the toolbar. This command also calculates the near **H-Field**.
>
> To avoid calculating the **H-Field**, go to **Main Menu > Tools > Preferences > Options** and uncheck the **"Run ALL" also calculates the H-Field** option.

> **Note**
>
> When an **incident plane wave** is used as the excitation, the calculated **near-field** results represent the **scattered fields**.

Calculating the Near H-Field

Once the current distribution on the structure has been calculated, the near magnetic field can be computed at the points in space specified in the **Near-Field panel** of the **Setup tab**.

To calculate the near magnetic field:

> Navigate to **Run > Run Near H-Field** in the main menu (see **Fig. 1**).
>
> This command is enabled only after the current distribution has been calculated.

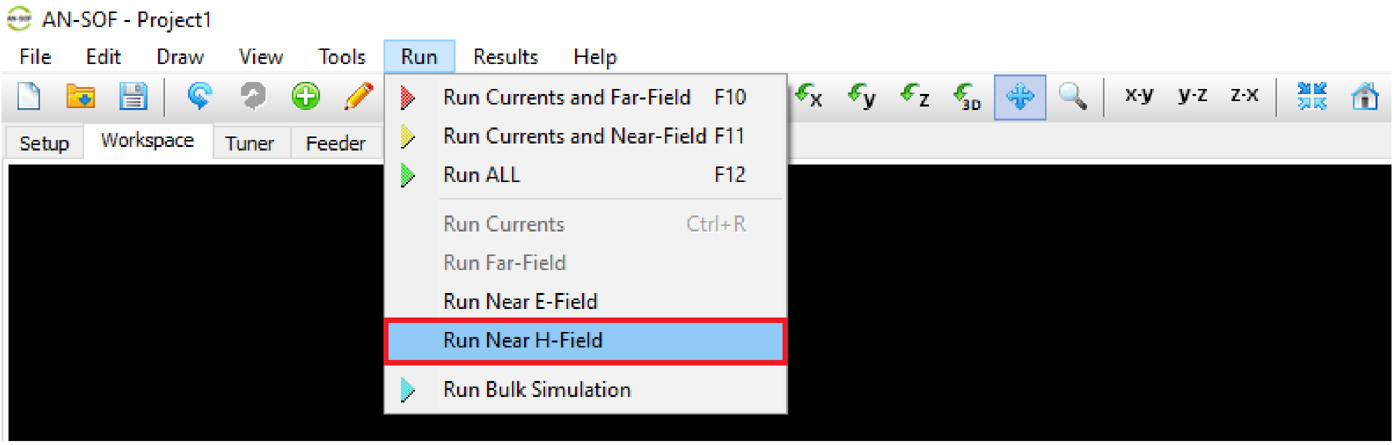


Fig. 1: The "Run Near H-Field" command in the main menu.

> **Tips**
>
> To sequentially and automatically calculate both **the current distribution and the near fields**, click the **Run Currents and Near-Field (F11)** button on the toolbar. This command also calculates the near **E-Field**.
>
> To enable the calculation of the **H-Field**, go to **Tools > Preferences > Options** in the main menu and check the **"Run ALL" also calculates the H-Field** option.

> **Note**
>
> When an **incident plane wave** is used as the excitation, the calculated **near-field** results represent the **scattered fields**.

The Run ALL Command

Once the **frequencies**, **environment**, **structure geometry**, **excitation**, and **far-field observation points** have been defined, AN-SOF is ready to perform the calculations. If **near-field** data is also required, the observation points must first be specified in the **Near Field panel**. The simulation begins by calculating the **current distribution** along the **wire segments**, which also yields the input impedance for a transmitting antenna. The near and far fields are then computed based on these segment currents.

Run ALL (F12) Command

The **Run ALL (F12)** command allows you to sequentially and automatically calculate the current distribution, far fields, and near fields. To use this command:

1. Navigate to **Main Menu > Run > Run ALL (F12)** (see **Fig. 1**).
2. Alternatively, click the **Run ALL** button on the toolbar.

Alternative Commands

- If the near field is not required, you can calculate only the currents and far fields by clicking **Run > Run Currents and Far-Field (F10)**. This command is also available on the toolbar.
- If the far field is not required, you can calculate only the currents and near fields by clicking **Run > Run Currents and Near-Field (F11)**. This command is also available on the toolbar.

Separate Calculations

The currents, far fields, and near fields can also be computed separately, as explained in the following articles:

- **Calculating the Current Distribution**
- **Calculating the Far Field**
- **Calculating the Near E-Field**
- **Calculating the Near H-Field**

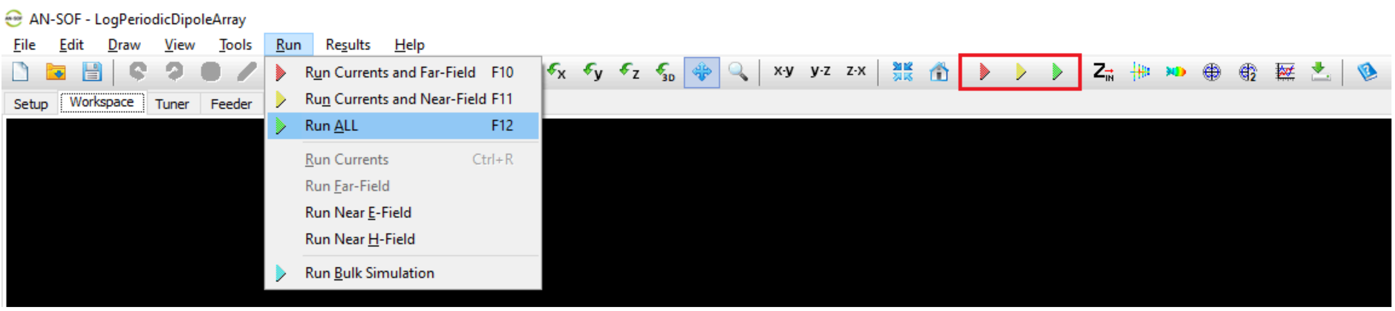


Fig. 1: The "Run ALL" command in the main menu. There are also buttons on the toolbar to run the calculations.

Aborting the Calculations

When a calculation is executed using the commands under the **Run menu**, the **Processing** window will be displayed (see **Fig. 1**). This window includes a button to abort the calculation at any time.

Note:

You will be prompted to save the project before aborting, as AN-SOF will restart after the process is terminated.

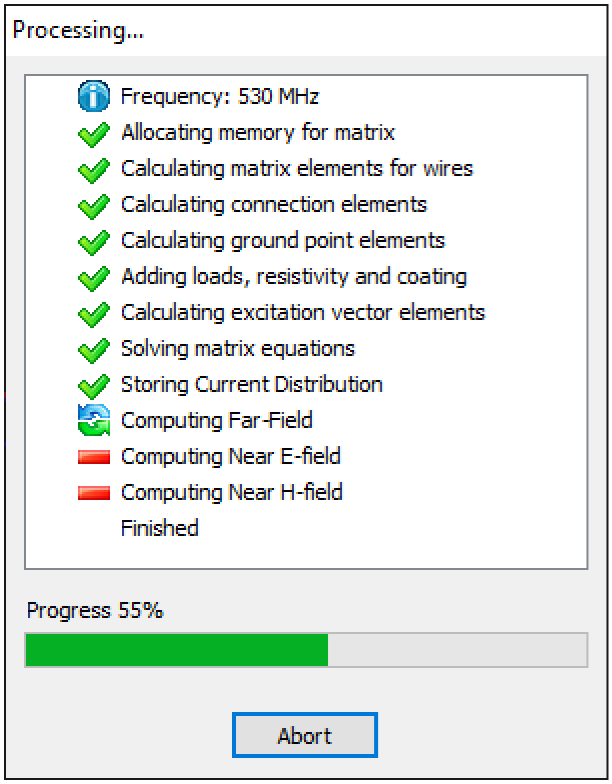


Fig. 1: The Processing window.

Numerical Green's Function

In simulations where the excitation of the structure needs to be changed frequently—such as adjusting the amplitudes of discrete sources or altering the direction of arrival of an incident field—the **Numerical Green's Function (NGF)** option can save significant computation time. To enable this option, navigate to the **Settings panel** of the **Setup tab**, as shown in **Fig. 1**.

How NGF Works

- During an NGF calculation, the **LU-decomposed matrix** of the system is stored in a file after the initial computation.
- Subsequent calculations reuse this stored matrix, allowing them to be performed much faster than the initial one.

Automatic NGF Activation

When **transmission lines** are included in the model, the **NGF** option is automatically enabled.

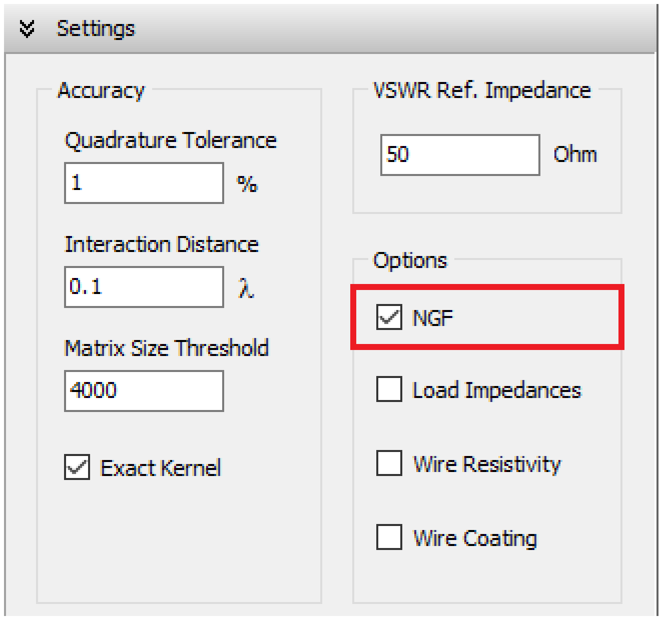


Fig. 1: NGF option in the Settings panel of the Setup tab.

Running a Bulk Simulation

AN-SOF can import a sequence of input files to generate a corresponding sequence of output files, all without requiring user intervention during the process. The input files must adhere to the **NEC format** and have a **.nec** extension. The supported NEC commands for importing wires are described in the section: **Importing Wires**.

Output Data

The output data includes:

- **Power Budget** or **RCS (Radar Cross Section)**.
- **Input Impedances**.
- **Far Field** and **Near Fields**.

All output data is provided in **CSV** format. For each NEC input file, AN-SOF generates an individual project containing **.emm** and **.wre** files (see **File Formats**). This allows each project to be opened separately after the bulk simulation is completed.

Initiating a Bulk Simulation

1. Navigate to the main menu and select **Run > Run Bulk Simulation**.
2. A prompt will appear, asking if you want to save changes in the current project, as the bulk simulation requires closing the currently open project.
3. A dialog box will be displayed, allowing you to select a directory and the input **.nec** files.
4. After selecting the desired files and clicking the **Open** button, the bulk simulation will begin. The input files will be imported and computed one after another in alphabetical order.

Generated Files

For an input file named **“InputFile.nec”**, the following files will be generated:

AN-SOF Project Files

- **InputFile.emm**: Main project file (can be opened with AN-SOF).
- **InputFile.wre**: Geometry data (wires, segments, connections).
- **InputFile.txt**: Comments.
- **InputFile.cur**: Current distribution.
- **InputFile.pwr**: Input and radiated powers, directivity, gain, etc.
- **InputFile.the**: Theta component of the far field.
- **InputFile.phi**: Phi component of the far field.
- **InputFile.nef**: Near electric field.
- **InputFile.nhf**: Near magnetic field.

Output CSV Files with Results

- **InputFile_PowerBudget.csv**: Input and radiated power, efficiency, gain, etc.
- **InputFile_Zin.csv**: Input impedances, VSWR, S_{11}, etc.
- **InputFile_FarFieldX.csv**: E-theta and E-phi far field components.
- **InputFile_EFieldX.csv**: Near electric field components.
- **InputFile_HFieldX.csv**: Near magnetic field components.

Note:

- **“X”** represents the frequency in Hz (e.g., **X = 300000000** for a frequency of **300 MHz**).
- A **FarField**, **EField**, and **HField** file will be generated for each frequency if a frequency sweep simulation has been configured.

Automating Parameter Variations

Bulk simulations automate the calculation process for multiple **NEC files**, even if they are unrelated, eliminating the need for manual calculations file by file. They are particularly useful for sequentially running calculations on **NEC files** generated with varying geometric parameters for an antenna. The results can then be analyzed by reading data from the generated **CSV files**.

For example, you can create a script to generate a sequence of **NEC files** for a **Yagi-Uda antenna**, where the spacing between its elements varies. To learn how to accomplish this and read the output data from the **CSV files**, refer to the following article: **Element Spacing Simulation Script for Yagi-Uda Antennas**.

Displaying Results

Types of Results

Commands to Display Results

The output data of a simulation can be listed in tables or displayed in graphs. All results are found under the **Results menu**, and are categorized into four groups:

Results related to current distribution

- Results > **Plot Current Distribution** command.
- Results > **Plot Currents** command.
- Results > **List Currents** command.
- Results > **Export Currents** command.
- Results > **List Input Impedances** command.

Results related to the far field

- Results > **Plot Far-Field Pattern** command.
- Results > **Plot Far-Field Spectrum** command.
- Results > **List Far-Field Pattern** command.
- Results > **List Far-Field Spectrum** command.
- Results > **Power Budget/RCS** command.

Results related to the near E-Field

- Results > **Plot Near E-Field Pattern** command.
- Results > **Plot Near E-Field Spectrum** command.
- Results > **List Near E-Field Pattern** command.
- Results > **List Near E-Field Spectrum** command.

Results related to the near H-Field

- Results > **Plot Near H-Field Pattern** command.
- Results > **Plot Near H-Field Spectrum** command.
- Results > **List Near H-Field Pattern** command.
- Results > **List Near H-Field Spectrum** command.

Results related to the Power Density

- Results > **Plot Power Density Pattern** command.
- Results > **Plot Power Density Spectrum** command.
- Results > **List Power Density Pattern** command.
- Results > **List Power Density Spectrum** command.

> **Tip**
>
> See the most relevant results for transmitting antennas in the **Results tab** of the main window.

Lists and Plots

Listing the currents or input impedances means tabulating them as a function of frequency.

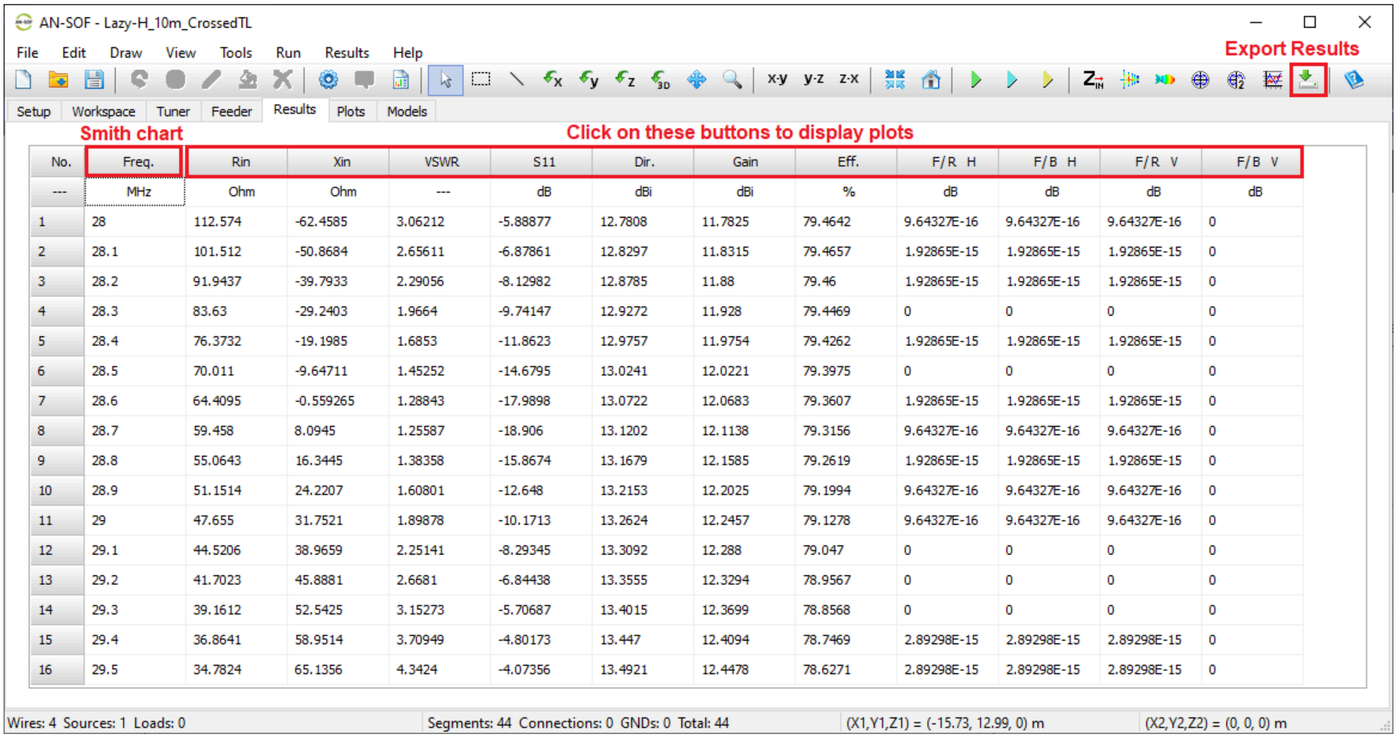
In the case of fields, they can be listed at a given point versus the frequency **(Spectrum)** or at a given frequency versus the observation point **(Pattern)**.

AN-SOF includes a suite of four tools for plotting results: **AN-XY Chart**, **AN-Smith**, **AN-Polar** and **AN-3D Pattern**.
The Results Tab

The **Results** tab in the AN-SOF main window (see Fig. 1) displays a table with the primary results for a transmitting antenna, including:

- **Input Impedance** ($Z_{in} = R_{in} + jX_{in}$)
- **VSWR**
- **S_{11}**
- **Directivity**
- **Gain**
- **Radiation Efficiency**
- **Horizontal (H)** and **Vertical (V) Front-to-Rear (F/R)** and **Front-to-Back (F/B)** Ratios

This table is automatically populated only when the wire structure is excited by a **discrete source**. It will not be filled if the excitation is an **incident wave**. The tabulated results persist until a new calculation is performed, allowing you to reference them at any time, even when making changes to the project. To export these results to a **CSV** file, click the **Export Results** button on the toolbar (see Fig. 1).



No.	Freq.	Rin	Xin	VSWR	S11	Dir.	Gain	Eff.	F/R H	F/B H	F/R V	F/B V
---	MHz	Ohm	Ohm	---	dB	dBi	dBi	%	dB	dB	dB	dB
1	28	112.574	-62.4585	3.06212	-5.88877	12.7808	11.7825	79.4642	9.64327E-16	9.64327E-16	9.64327E-16	0
2	28.1	101.512	-50.8684	2.65611	-6.87861	12.8297	11.8315	79.4657	1.92865E-15	1.92865E-15	1.92865E-15	0
3	28.2	91.9437	-39.7933	2.29056	-8.12982	12.8785	11.88	79.46	1.92865E-15	1.92865E-15	1.92865E-15	0
4	28.3	83.63	-29.2403	1.9664	-9.74147	12.9272	11.928	79.4469	0	0	0	0
5	28.4	76.3732	-19.1985	1.6853	-11.8623	12.9757	11.9754	79.4262	1.92865E-15	1.92865E-15	1.92865E-15	0
6	28.5	70.011	-9.64711	1.45252	-14.6795	13.0241	12.0221	79.3975	0	0	0	0
7	28.6	64.4095	-0.559265	1.28843	-17.9898	13.0722	12.0683	79.3607	1.92865E-15	1.92865E-15	1.92865E-15	0
8	28.7	59.458	8.0945	1.25587	-18.906	13.1202	12.1138	79.3156	9.64327E-16	9.64327E-16	9.64327E-16	0
9	28.8	55.0643	16.3445	1.38358	-15.8674	13.1679	12.1585	79.2619	1.92865E-15	1.92865E-15	1.92865E-15	0
10	28.9	51.1514	24.2207	1.60801	-12.648	13.2153	12.2025	79.1994	9.64327E-16	9.64327E-16	9.64327E-16	0
11	29	47.655	31.7521	1.89878	-10.1713	13.2624	12.2457	79.1278	9.64327E-16	9.64327E-16	9.64327E-16	0
12	29.1	44.5206	38.9659	2.25141	-8.29345	13.3092	12.288	79.047	0	0	0	0
13	29.2	41.7023	45.8881	2.6681	-6.84438	13.3555	12.3294	78.9567	0	0	0	0
14	29.3	39.1612	52.5425	3.15273	-5.70687	13.4015	12.3699	78.8568	0	0	0	0
15	29.4	36.8641	58.9514	3.70949	-4.80173	13.447	12.4094	78.7469	2.89298E-15	2.89298E-15	2.89298E-15	0
16	29.5	34.7824	65.1356	4.3424	-4.07356	13.4921	12.4478	78.6271	2.89298E-15	2.89298E-15	2.89298E-15	0

Fig. 1: Results tab in the main window, with the Export Results button highlighted in the toolbar.

Interactive Column Headers

The column headings, from **Rin** through **F/B V**, are interactive buttons. Clicking on them displays rectangular plots, where the data in the column is plotted as a function of frequency.

- Click the **"Freq."** column header to display the input impedance ($Z_{in} = R_{in} + jX_{in}$) in a **Smith Chart**. By default, this is **the input impedance at the antenna feedpoint**.
- If the antenna has a **feeder** and/or **tuner** connected to its terminals, the impedance seen at the feeder input or tuner input can also be tabulated in the **Results** tab. These can be plotted against frequency in rectangular or Smith charts by clicking the corresponding column headers.

Plotting Impedance at Different Points

1. Navigate to the **Plots tab > "Zin" box** (see Fig. 2) and choose between **Antenna**, **Feeder**, or **Tuner**.
2. Go to the **Results** tab, where $\mathbf{R_{in}}$, $\mathbf{X_{in}}$, **VSWR**, and $\mathbf{S_{11}}$ will be tabulated for the selected option (antenna, feeder, or tuner input).
3. Click the header buttons as indicated in Fig. 1 to plot these results against frequency.

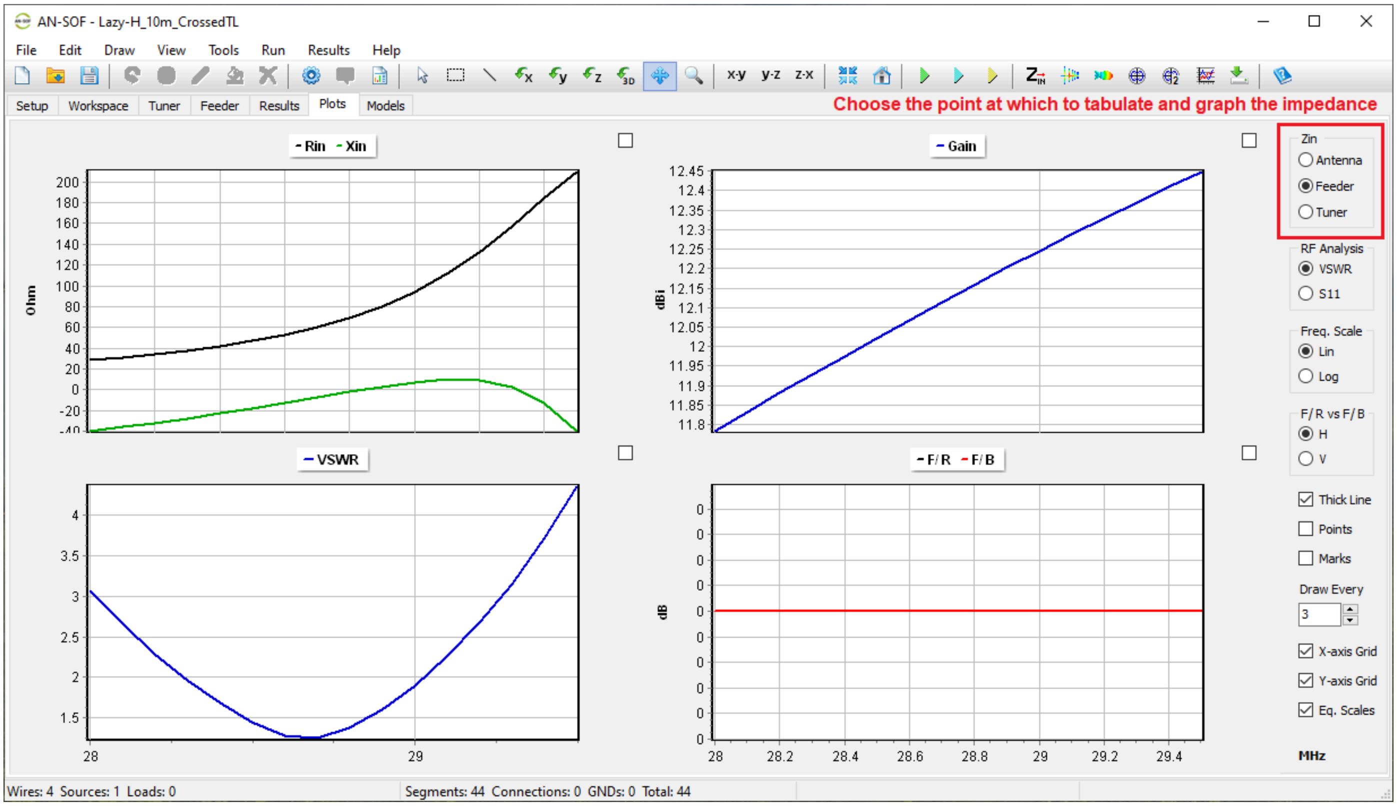


Fig. 2: Plots tab in the main window, showing the Antenna, Feeder, and Tuner options in the Zin box for selecting the point to tabulate and graph impedance.

The Plots Tab

Select the **Plots** tab in the AN-SOF main window to visualize the plots of the main results for a transmitting antenna as a function of frequency, as shown in Fig. 1. These results are derived from the table in the **Results** tab.

Plot Layout

- **Left Column**: Displays the real and imaginary parts of the **input impedance** (Z_{in} = R_{in} + jX_{in}) and the **VSWR**.
- **Right Column**: Shows the **antenna gain** in **dBi** and the **Front-to-Rear (F/R)** and **Front-to-Back (F/B)** ratios in **dB**.

The plots are aligned vertically to facilitate easy comparison.

Customizing Plots

Use the controls on the right side of the **Plots** tab to adjust various aspects of the graphics, including:

- Line thickness.
- Visualization of points and marks.
- Scales and axes.
- Selection between **VSWR** or **S_{11}**.

- Choice between **Horizontal (H)** or **Vertical (V)** F/R and F/B ratios.

Each plot can be maximized by clicking the **Maximize** checkbox located in its upper-right corner.

Input Impedance and VSWR/S_{11} Options

The **input impedance** and **VSWR/S_{11}** plots can represent:

- The antenna input impedance.
- The feeder + antenna input impedance.
- The tuner input impedance.

The **tuner** and **feeder** can be configured in their corresponding tabs next to the **Results** tab. Whenever a tuner or feeder parameter is changed, the recalculated results in the **Results** and **Plots** tabs can be refreshed by selecting the desired option under the **Zin** box (highlighted in Fig. 1).

Tuner Option

If the **Tuner** option is selected:

- The input impedance of the tuner will be displayed.
- If the tuner is connected to a combination of feeder + antenna, the input impedance and **VSWR/S_{11}** of the **tuner + feeder + antenna** system will be displayed.

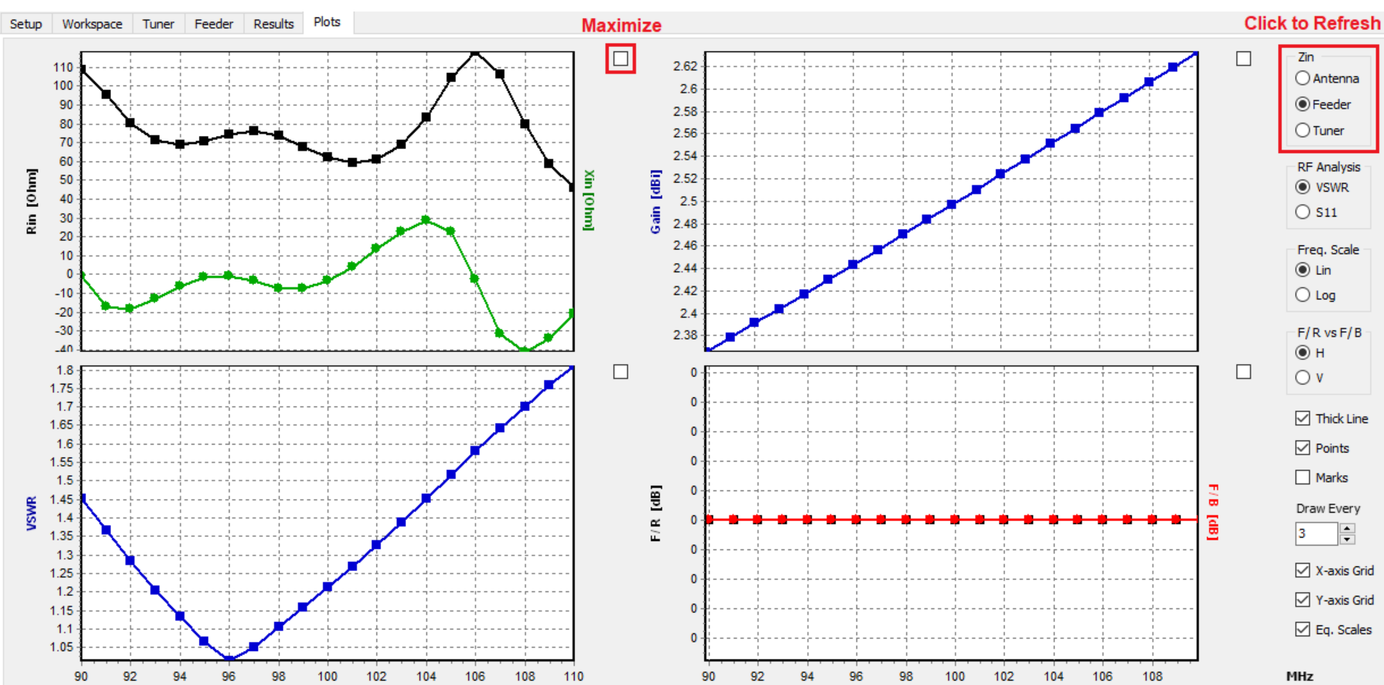


Fig. 1: Plots tab in the AN-SOF main window.

Current Distribution

Plotting the Current Distribution

3D Plots

To visualize how current flows across your entire model, navigate to **Results > Plot Current Distribution** in the main menu. This command launches the **AN-3D Pattern** application, which renders the current amplitude directly on the 3D structure using a dynamic color scale (**Fig. 1**).

Beyond amplitude, you can also analyze the complex components of the current. By selecting the **Plot** menu within AN-3D Pattern, you can display:

- Amplitude
- Phase
- Real part
- Imaginary part

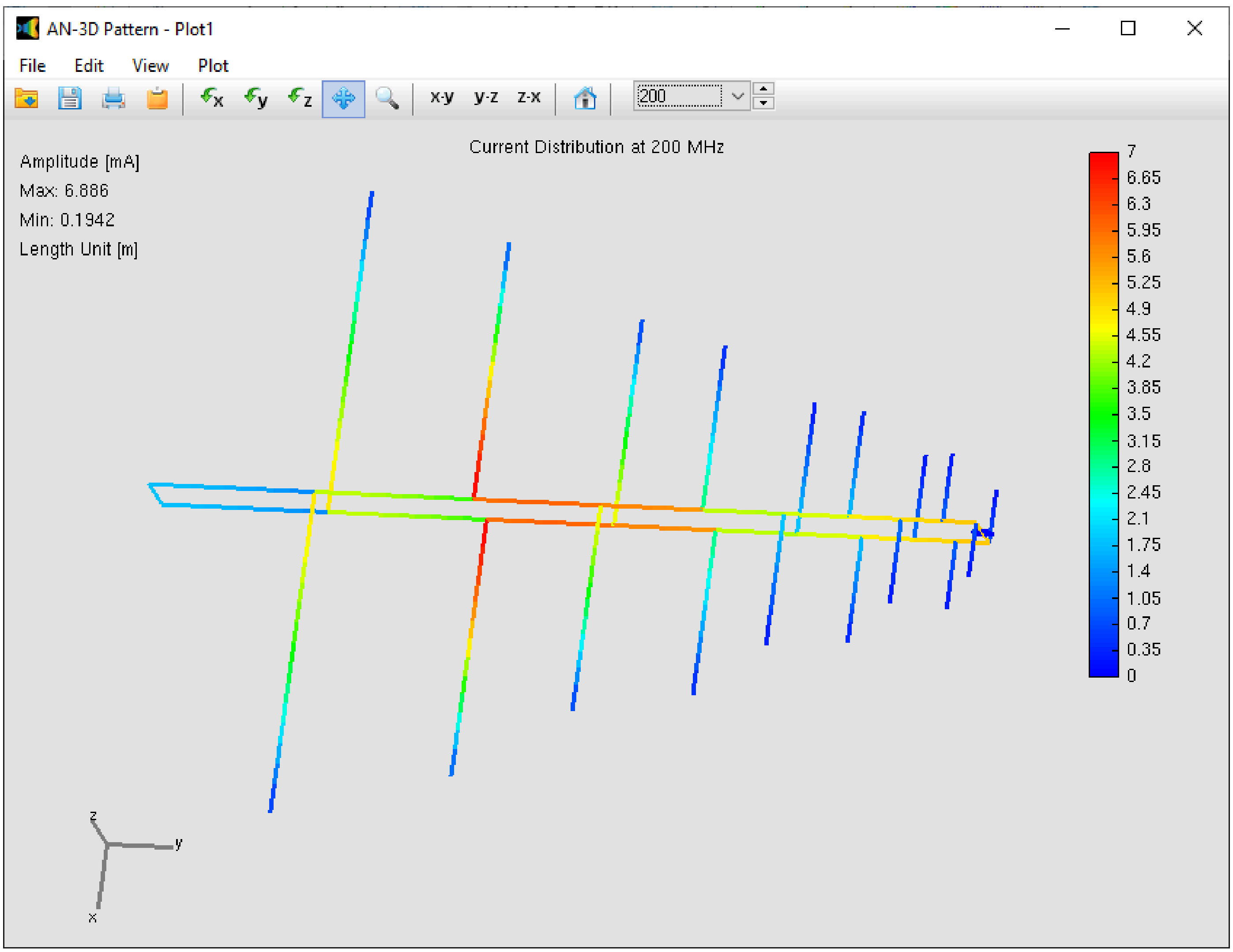


Fig. 1: Current distribution in amplitude as a color map on a Log-Periodic Dipole Array (LPDA) plotted in AN-3D Pattern.

Adjusting the Color Bar in AN-3D Pattern

For precise data interpretation and professional presentations, you can manually override the automatic scaling. This allows you to set specific increments (e.g., multiples of 5).

- Navigate to **Edit > Preferences** in the AN-3D Pattern menu (**Fig. 2**).
- Manually define the **Max** and **Min** values of the color scale to highlight specific ranges of interest.

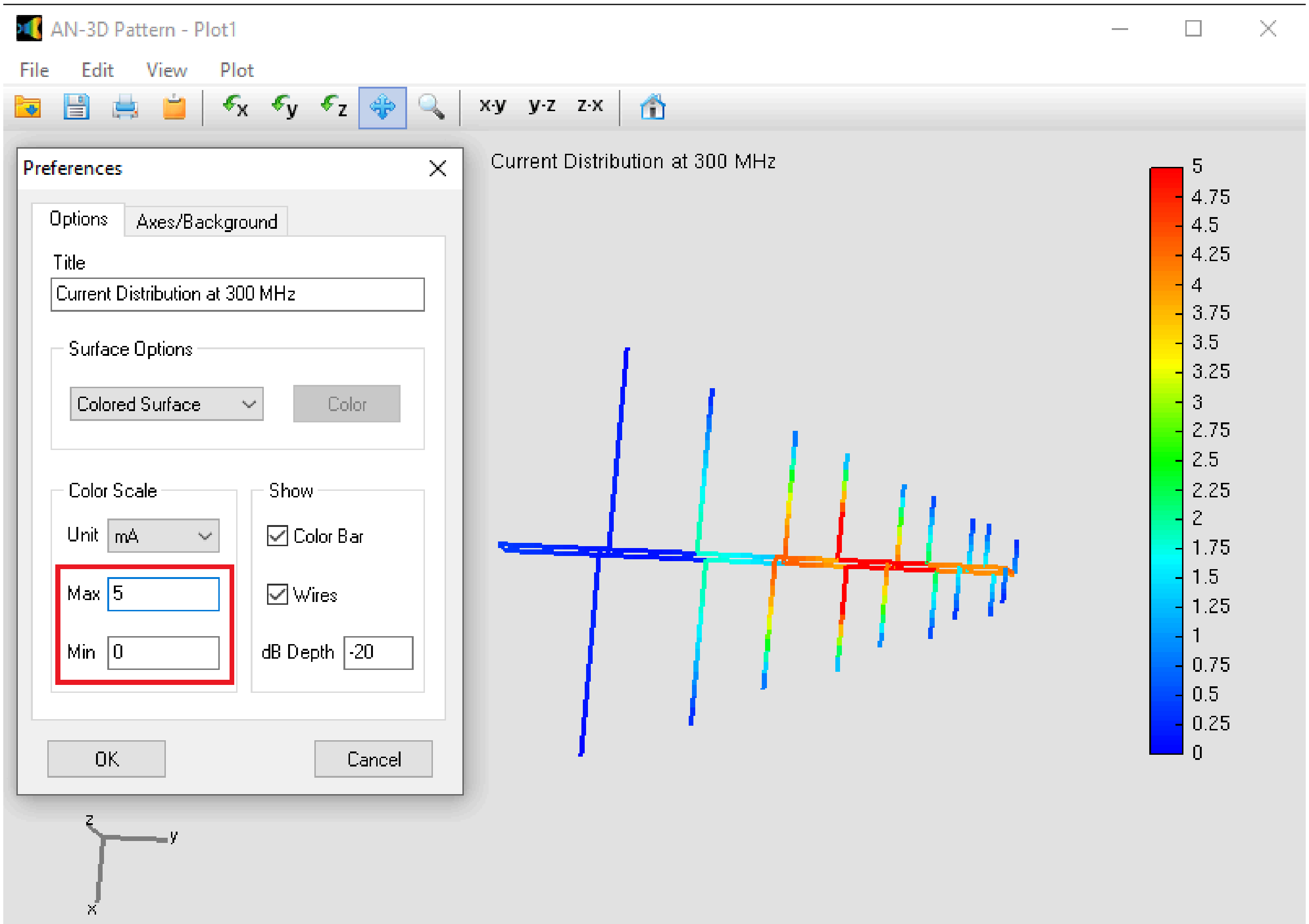


Fig. 2: Color bar scale can be adjusted in the Preferences window of AN-3D Pattern.

2D Plots

If you need to analyze the current along a specific wire in detail, you can generate a Cartesian plot using **AN-XY Chart**.

Method 1: Quick Access

Right-click directly on a wire in the workspace and select **Plot Currents** from the pop-up menu (**Fig. 3**).

Method 2: Toolbar Selection

1. Click the **Select Wire** button (arrow icon) on the toolbar.
2. Left-click the desired wire.
3. Go to **Results > Plot Currents** in the main menu.

Within the AN-XY Chart interface, you can toggle between amplitude, phase, and complex parts via the **View** menu.

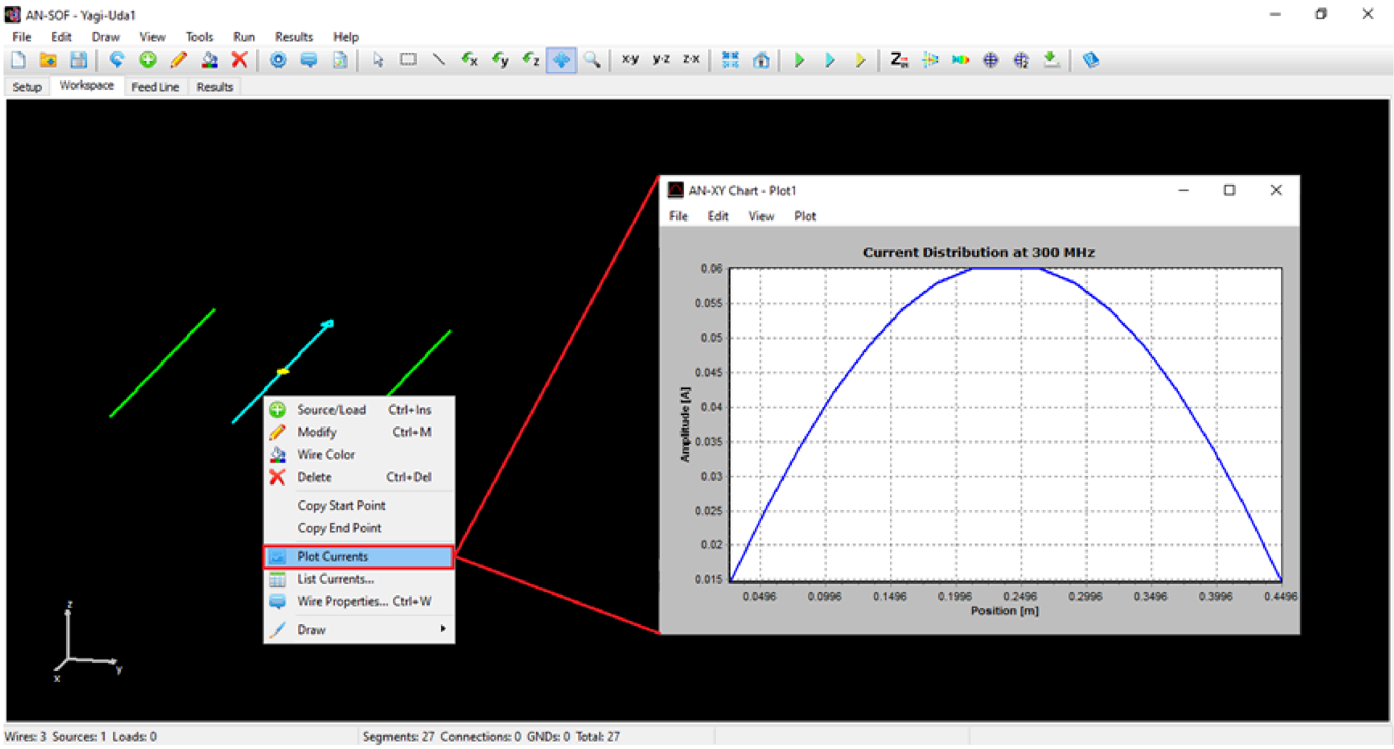


Fig. 3: *The "Plot Currents" command in the pop-up menu displayed after right-clicking a wire, and the current distribution in amplitude plotted in AN-XY Chart.*

> ## Tips
>
> To optimize your analysis of 2D current distributions, **AN-XY Chart** provides several interactive navigation and data management features:
>
> **Zooming:** To focus on a specific area of the plot, press and hold the left mouse button while expanding a box over the desired region.
>
> **Panning:** To move the view without changing the zoom level, **right-click** on the graph and drag the mouse.
>
> **Reset View:** To quickly return to the original full-scale view, **left-click** and expand a small rectangle in an **upward** direction anywhere on the plot.
>
> **Units and Export:** The main menu offers options to change the **units** of the plotted magnitudes (e.g., from linear to dB). You can also **export the data** to external formats for further processing in third-party software.

The List Currents Toolbar

The **List Currents** toolbar provides a localized view of electrical data for specific points on your model, allowing for detailed frequency-response analysis.

Accessing the Toolbar

There are two primary ways to access this tool:

- **Right-Click Menu:** Right-click directly on a wire in the workspace and select the **List Currents** command from the pop-up menu.
- **Main Menu:** Click the **Select Wire** button (arrow icon) on the toolbar, left-click the desired wire, and navigate to **Results > List Currents**. This option becomes available once the current distribution calculation is complete.

Key Functionalities

The toolbar enables you to pinpoint a specific segment within the selected wire to analyze the following data versus frequency:

- **Current Analysis:** View the current magnitude and phase for any individual segment.
- **Source and Load Data:** If the selected segment contains a source or load, the toolbar can display a comprehensive list of parameters:
  - Input Impedance
  - Voltages and Power levels
  - Reflection Coefficient and VSWR
  - Return and Transmission Losses

Components of the List Currents Toolbar

The **List Currents** toolbar provides specific controls to navigate wire segments and analyze their electrical properties across the frequency sweep (**Fig. 1**).

Fig. 1: *The List Currents toolbar.*

Navigation Controls

- **Slider:** Move the slider to select a specific segment along the wire. Each position corresponds to a segment index.
  - **Segment Number:** Displays the index of the currently selected segment.
  - **Segment Position %:** Shows the relative position from the wire's start to the segment midpoint. It is calculated as:

$$\text{SegmentPosition\%} = 100 \times (\text{segmentpositionwirelength})$$

- **50% Button:** Instantly moves the slider to the exact midpoint of the wire. This is particularly useful for verifying or placing center-fed sources and loads.

> Note: The wire must have an **odd number of segments** for an exact center segment to exist.

Data and Analysis Buttons

- **Current on Segment:** Opens a dialog box showing a frequency list of the current for the active segment. Click **Plot** to generate a frequency-domain graph of the current (**Fig. 2**).
- **Input List:** Enabled only if the selected segment contains a **source**. This dialog displays input impedance, currents, voltages, and powers (**Fig. 3**).
  - **Plot:** Graph the selected parameter vs. frequency.
  - **Smith:** Visualize input impedance on a **Smith Chart**.
  - **Export:** Save the data as a **CSV** file.
- **Source List:** Enabled when a **source** is present. It lists currents, voltages, and powers specifically within the **internal impedance** of the source. Data can be plotted or exported to CSV (**Fig. 4**).
- **Load List:** Enabled if the segment contains a **load**. It displays load impedances, currents, voltages, and powers. Like the other lists, these values can be plotted against frequency or exported (**Fig. 5**).
- **Exit Button:** Closes the toolbar and returns to the standard workspace view.

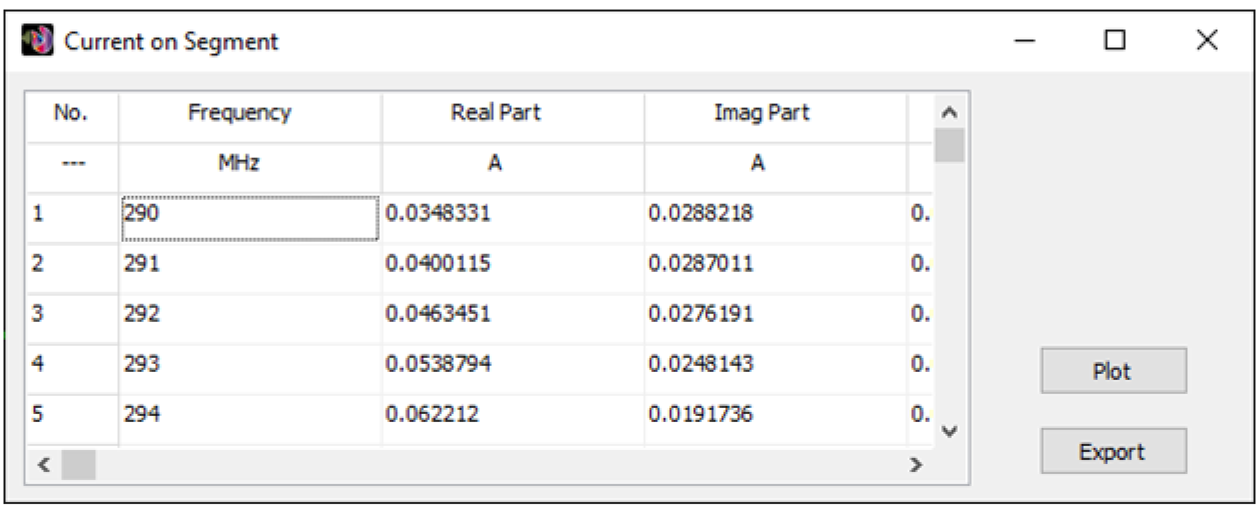


Fig. 2: *The “Current on Segment” dialog box.*

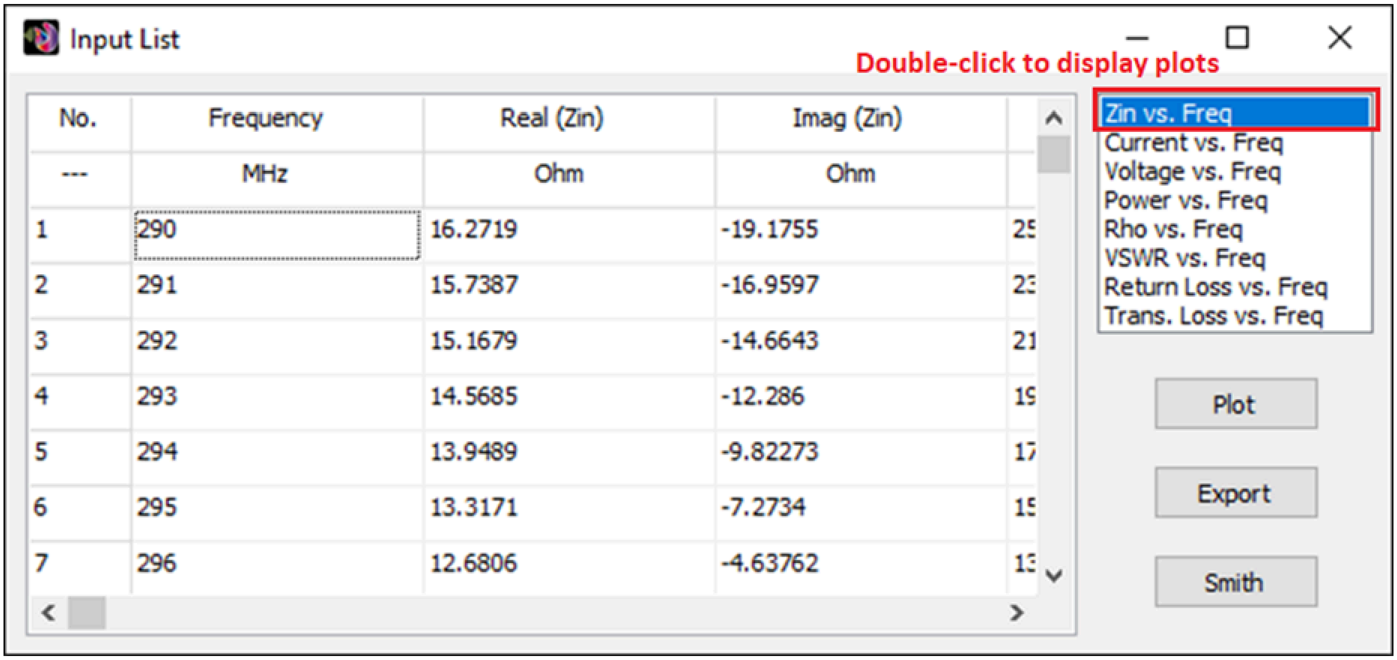


Fig. 3: *The “Input List” dialog box.*

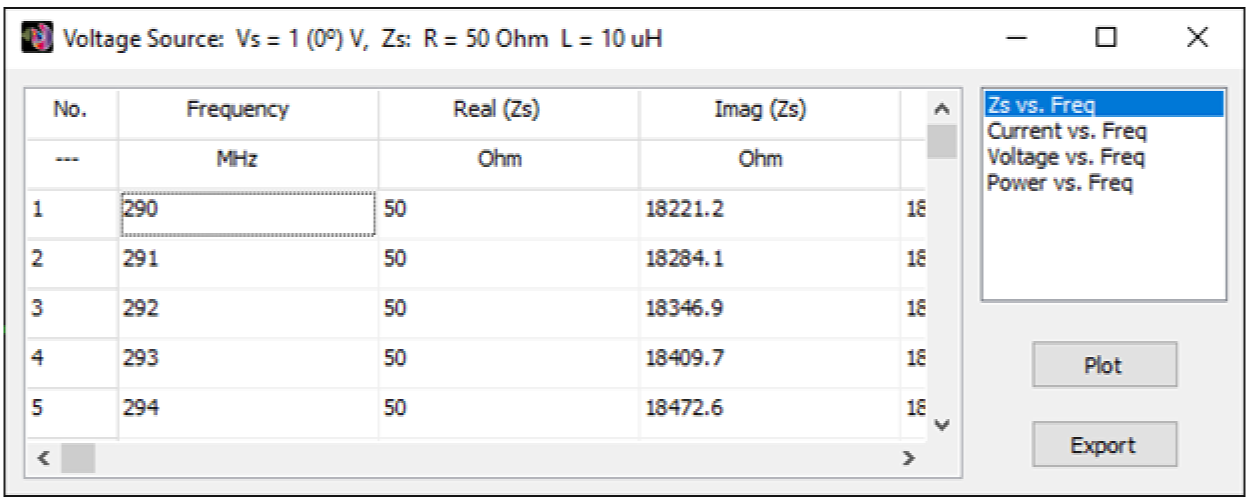


Fig. 4: *The “Source List” dialog box.*

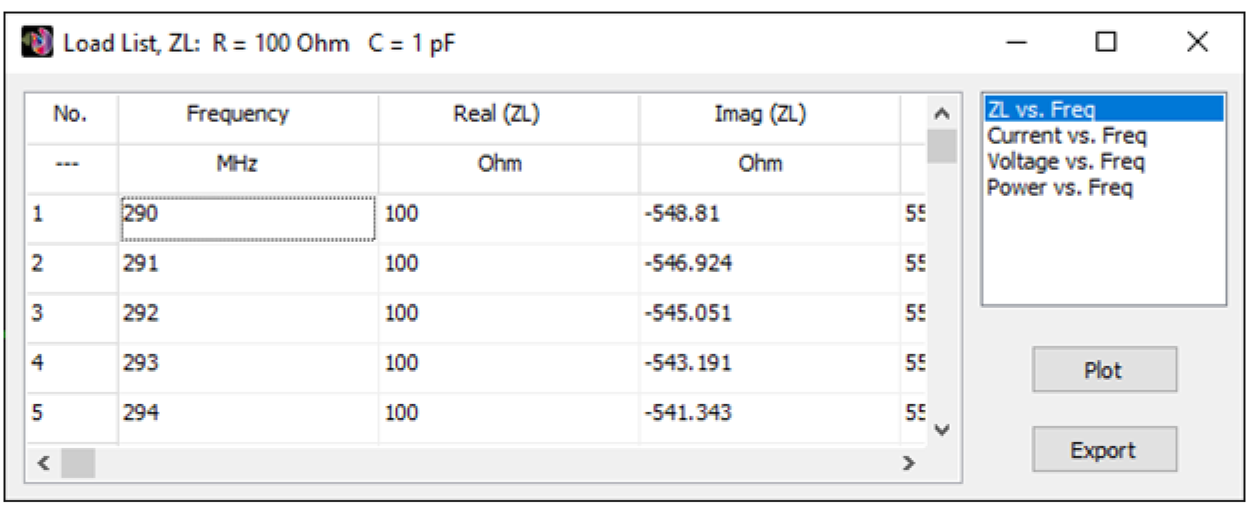


Fig. 5: *The “Load List” dialog box.*

Listing the Currents in a Segment

To analyze how the current at a specific point on your antenna behaves across a range of frequencies, follow this procedure:

1. **Select the Wire:** Right-click on the desired wire in the workspace to open the **pop-up menu**.
2. **Open the Toolbar:** Click the **List Currents** command. The **List Currents toolbar** will appear at the bottom of the interface (**Fig. 1**).
3. **Target the Segment:** Move the **slider** to navigate through the segments of the selected wire. You can use the **50% button** to jump to the center or watch the **Segment Position %** display to find a specific location.
4. **View Frequency Data:** Click the **Current on Segment** button. This opens a dialog box that tabulates the current data for every frequency step in your simulation. The table includes:
    - Amplitude
    - Phase
    - Real Part
    - Imaginary Part
5. **Generate a Plot:** Click the **Plot** button within the dialog box to launch **AN-XY Chart**, where you can visualize the current magnitude or phase as a function of frequency.

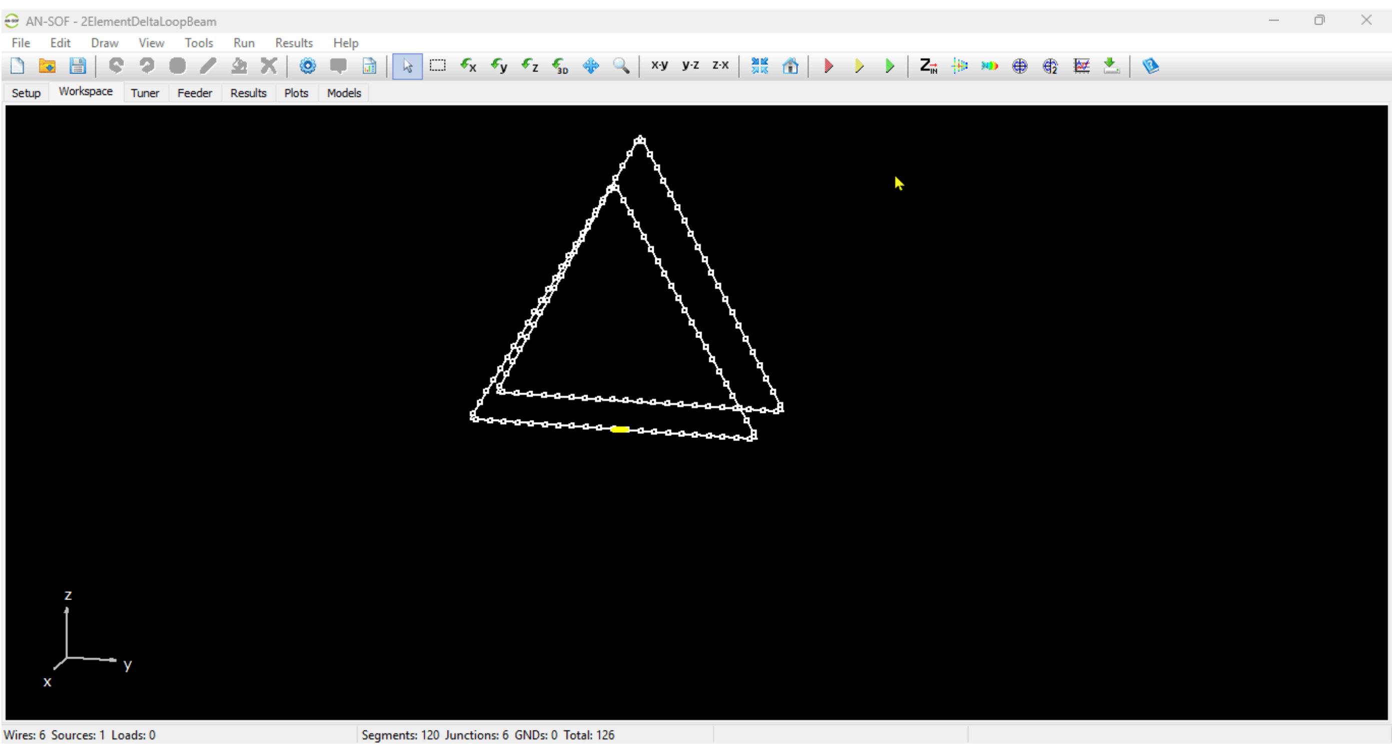


***Fig. 1:** Accessing the "List Currents" toolbar to tabulate and plot the current in a wire segment as a function of frequency.*

Exporting Currents on a Wire

The currents flowing on a wire can be exported to a **CSV (Comma-Separated Values)** file. Since the current is calculated at the midpoint of each wire segment, the current distribution is sampled at a finite set of points determined by the number of segments the wire is divided into. Additionally, as the current varies with frequency, there is a unique current distribution along the wire for each frequency.

To export the current distribution as a function of position along a selected wire and as a function of frequency, follow these steps:

1. Click the **Select Wire** button (arrow icon) in the toolbar.
2. Left-click on the wire to select it (it will be highlighted in light blue).
3. Navigate to the **Results menu > Export Currents** in the main menu (see **Fig. 1**).

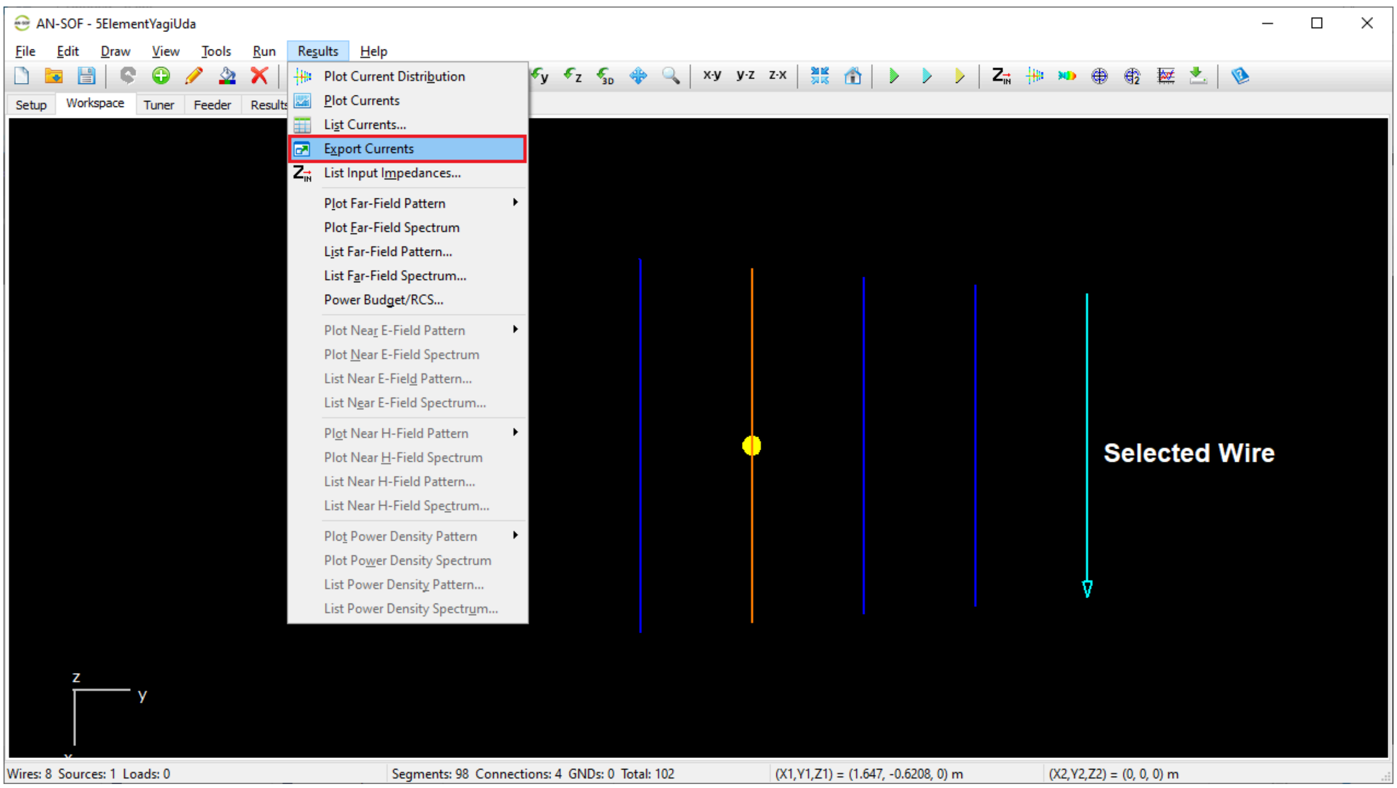


Fig. 1: *Export Currents option in the Results menu, enabled when an individual wire is selected.*

The complex current (real and imaginary parts) is tabulated based on the position along the wire. Position and frequency units can be configured in the **Preferences window**.

Note:

- Position is measured from the start point of the selected wire (the end without the arrow when selected).
- The exported CSV file contains the real and imaginary parts of the current as a function of position and frequency, as shown in **Fig. 2**.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	1st row of data: position along the wire in [m] (omit the first zero). 2nd row: real part of the current in [A]. 3rd row: imaginary part of the current in [A], and so on.														
2	1st column: corresponding frequencies in [MHz]														
3	0	0.8598	1.7196	2.5794	3.4392	4.299	5.1588	6.0186	6.8784	7.7382	Position				
4	12	0.000626	0.001141	0.00157	0.001909	0.002176	0.001909	0.00157	0.001141	0.000626	Real				
5	12	0.000267	0.000497	0.000701	0.000878	0.001041	0.000878	0.000701	0.000497	0.000267	Imag				
6	16	-0.00073	-0.00129	-0.00171	-0.00196	-0.00204	-0.00196	-0.00171	-0.00129	-0.00073	Real				
7	16	-0.00058	-0.0011	-0.00154	-0.0019	-0.00217	-0.0019	-0.00154	-0.0011	-0.00058	Imag				
8	20	3.73E-05	6.55E-05	8.85E-05	0.00011	0.000133	0.00011	8.85E-05	6.55E-05	3.73E-05	Real				
9	20	0.000134	0.000249	0.000341	0.000403	0.000432	0.000403	0.000341	0.000249	0.000134	Imag				
10	Freq.														

Fig. 2: *Exported current distribution in a CSV file, showing current vs. position along the wire and vs. frequency.*

Input Impedances

Listing the Input Impedances, VSWR, and S11

To analyze the impedance matching and power efficiency of your antenna across the simulated frequency range, use the following procedure:

1. **Select the Source Wire:** Right-click on the wire containing the source to open the **pop-up menu**.
2. **Activate the Toolbar:** Click the **List Currents** command to bring up the **List Currents toolbar** (**Fig. 1**).
3. **Target the Source Segment:** Use the **slider** to select the specific segment where the source is located. If you positioned the source at the exact center, the **50% button** will jump there instantly.
4. **Access the Input List:** Click the **Input List** button. This opens the **Input List dialog box** containing:
   - **Input Impedance:** Real and imaginary parts.
   - **Source Data:** Input current, voltage, and power.
   - **Matching Parameters:** Reflection coefficient, VSWR, and $S11$ (Return Loss) in decibels.
5. **Visualize the Data:**
   - **XY Plot:** Select a parameter (like VSWR or $S11$) from the list in the upper right corner and press **Plot** to see its frequency response.
   - **Smith Chart:** Press the **Smith** button to visualize how the input impedance moves across the complex plane relative to the characteristic impedance.

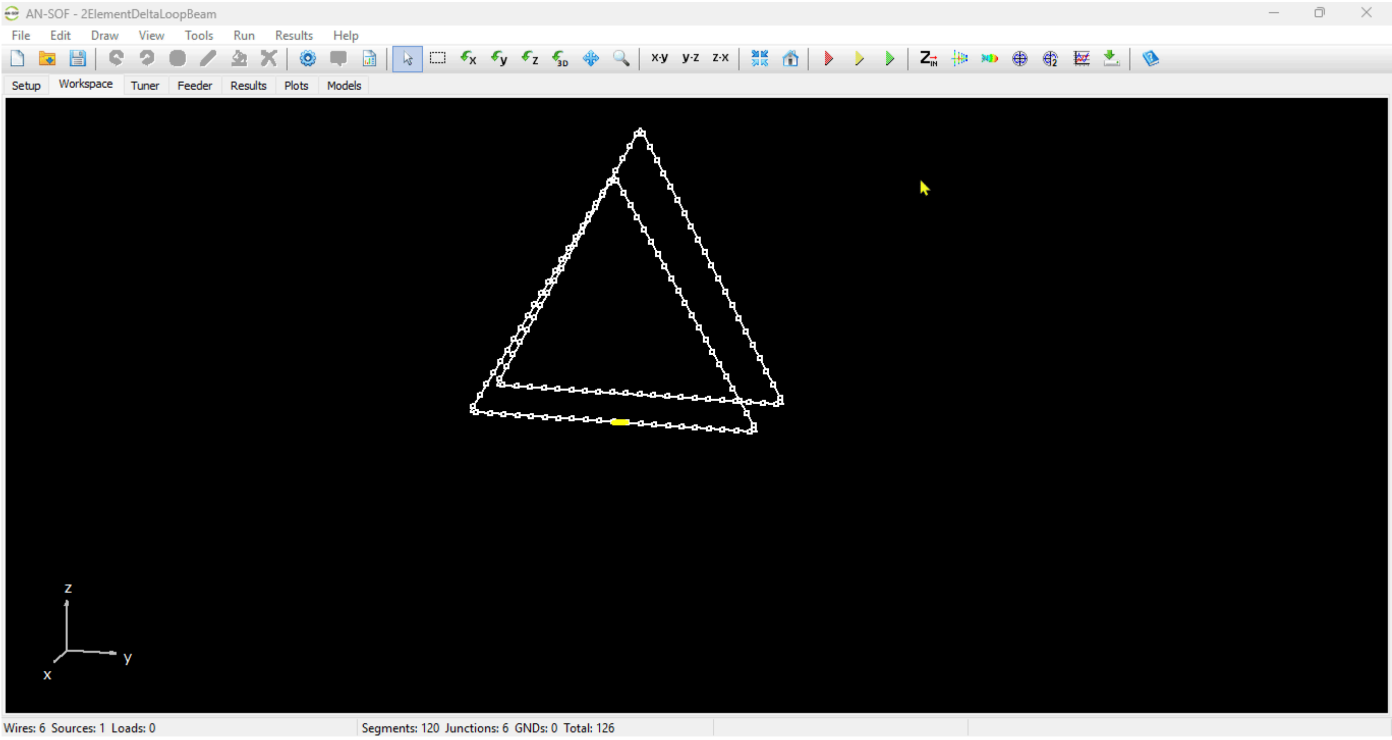


Fig. 1: *Accessing the "List Currents" toolbar to tabulate and plot the input impedance, VSWR, $S11$, etc., versus frequency at an antenna feed point.*

> **Tips**
>
> To ensure your reflection data (VSWR, $S11$, and Return Loss) is accurate for your specific system, you can customize the reference parameters and use shortcuts for single-source models.
>
> **Setting the Reference Impedance**
>
> By default, calculations are often referenced to 50Ω, but you can adjust this to match your specific feedline or connector:
>
> 1. Navigate to the **Setup** tab in the main workspace.
> 2. Locate the **Settings** panel.
> 3. Enter your desired value in the **VSWR Ref. Impedance** field.
>
> **Quick Access for Single-Source Models**
>
> If your antenna has only one source, you can bypass the segment selection slider and access the input data directly:
>
> - **Main Menu:** Go to **Results > List Input Impedances**.
> - **Toolbar:** Click the **List Input Impedances** icon (Z_{IN}).

Tuner for Impedance Matching

The Tuner Calculator

AN-SOF features a **tuner calculator** that enables **impedance matching** of an antenna input impedance, an antenna with a feeder already connected to its terminals, or a given custom load.

To access the tuner calculator, choose the **Tuner** tab in the AN-SOF main window (Fig. 1). Here, you can set the tuner parameters on the left side of the window and view the results on the right side. The tuner consists of three components, each of which will be described in the following sections:

> **Impedance Matching Network:** This component allows the synthesis of an impedance matching network based on the impedance seen at the network output and the desired impedance at the network input. The **quality factors** of the **network**, **inductors**, and **capacitors** can be adjusted to model real-world scenarios.
>
> **Stray Capacitance:** Some networks, particularly high-pass Tee networks, exhibit a parallel stray capacitance at the network output. This capacitance can be specified to account for this effect.
>
> **Impedance Transformer:** An impedance transformer can be specified at the network output to transform the input impedance of an antenna, the input impedance of a feeder connected to an antenna, or a custom load entered by the user. It can represent devices such as **baluns** and **ununs** — for example, a **1:1 balun** for balanced-to-unbalanced conversion without impedance change, a **1:4 balun** to match a 200-ohm antenna to a 50-ohm feedline, or a **1:9 unun** for matching high-impedance long-wire antennas. This feature is useful for optimizing power transfer and minimizing VSWR in multiband and off-resonance antenna configurations.

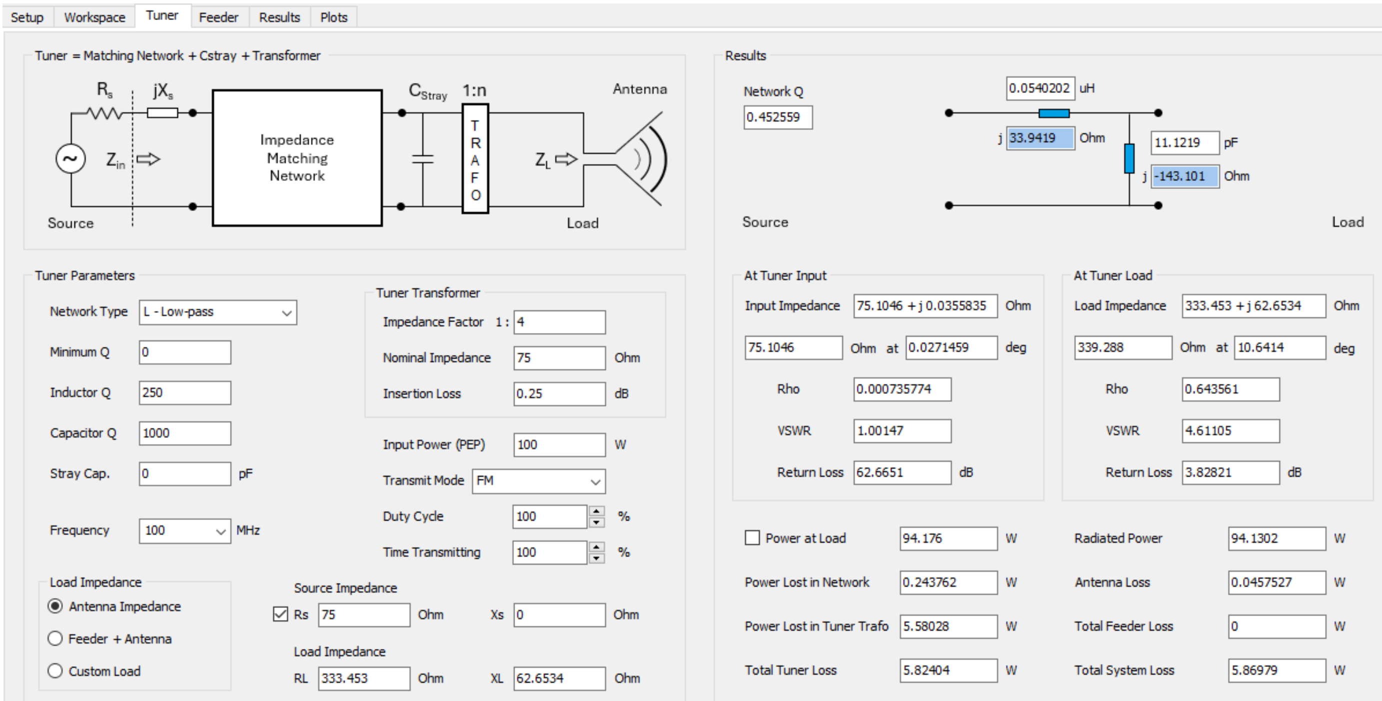


Fig. 1: Tuner tab in the AN-SOF main window where an antenna tuner can be configured.

Impedance Matching Network

In the **Tuner Parameters** box, you can configure the impedance matching network, as shown in Fig. 2.

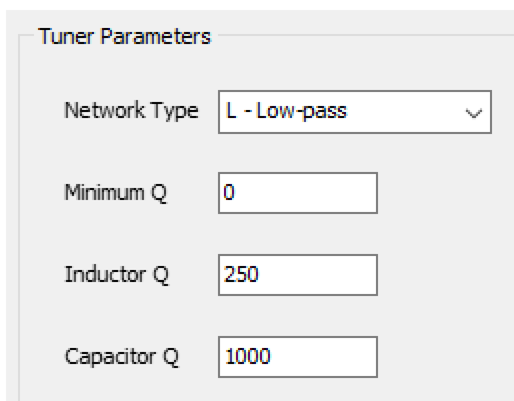


Fig. 2: Tuner Parameters box in the Tuner tab, showing options to specify the network type, minimum Q, and inductor and capacitor Q values.

By expanding the **Network Type** dropdown menu, you have the following options:

> **No Network**: Select this option to bypass the matching network, making the network input impedance equal to the impedance at the network output.

Based on the impedance seen at the network output and the source impedance connected to the network input side, AN-SOF can synthesize the following networks:

- **L – Low-pass**
- **L – High-pass**
- **PI – Low-pass**
- **PI – High-pass**
- **T – Low-pass**
- **T – High-pass**

The network components will be automatically calculated to match the **source impedance** (R_s + jX_s) connected to the network input side. If the source impedance has a reactance component, jX_s, the network will "absorb" this reactance so that the input impedance of the network plus jX_s will match the real part, R_s, of the source impedance. The same principle applies to the load impedance seen at the network output side. If the network load impedance has an imaginary part, it will be absorbed by the network to synthesize the network components (inductors and capacitors).

Note that a low-pass network could include series capacitors instead of inductors or parallel inductors instead of capacitors, depending on the **complex impedances** (with real and imaginary parts) being matched. Similarly, a high-pass network might involve series inductors instead of capacitors or parallel capacitors instead of inductors.

You can specify a **minimum Q** for the network synthesis calculations, as well as the **Q for the inductors and capacitors**. This allows you to account for component losses to represent real-world components. To model ideal zero-loss components, enter high Q values, such as 1E8.

Stray Capacitance

Stray capacitance, also known as **parasitic capacitance**, refers to unintended capacitance between two conductors separated by a dielectric or free space. This effect is particularly noticeable at the network output side when a **transmission line** is connected. AN-SOF allows for the configuration of a **feeder** composed of a transmission line to feed an antenna, enabling modeling of stray capacitance to accommodate this scenario. While stray capacitance is commonly observed in Tee high-pass networks, it can be added in any case. Typical values range from around 10 pF in HF bands.

Impedance Transformer

In the **Tuner Parameters** box, an impedance transformer, also known as a "trafo" in RF jargon, can be specified, as shown in Fig. 3.

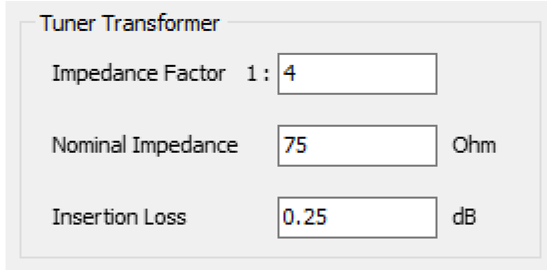


Fig. 3: Specification of a tuner transformer giving its impedance transformation factor, nominal impedance, and insertion loss.

The transformer allows us **to divide a load impedance by a factor, n**, making it a **1:n transformer**. It's important to note that this is the **impedance transformation factor**, not the voltage transformation factor, which is $n^{-1/2}$ and is determined by the primary-to-secondary winding relationship of a transformer. A transformer can be used to reduce a high impedance to approach the standard 50 or 75 Ohms used in transmission lines and RF devices. Both the real and imaginary parts of the load impedance will be divided by n.

If n is in the range **$0 < n < 1$**, the transformed impedance will be **higher** than the load impedance connected to the output side of the transformer. A factor **$n = 1$** can be used to model a **1:1 transformer**, also known as an **isolation transformer**, which is used to transfer voltage from one electrical circuit to another and to isolate a powered device from the power source. The 1:1 ratio transformer has the same input and output voltage and current. It is used to protect secondary circuits and individuals from electrical shocks between energized conductors and earth ground. It also reduces voltage spikes in the power supply line caused by rapid changes in lighting, static electricity, or voltage.

Real-life transformers are manufactured for a specified **nominal impedance** transformation. The nominal impedance can be entered in the **Tuner Transformer** box, as well as the transformer **insertion loss** in decibels. Manufacturers specify a transformer insertion loss relative to a nominal impedance, so it is important to specify the nominal impedance as well. The insertion loss is defined as the power lost inside the transformer, measured in dB relative to the input power. Thus, the output power delivered by the transformer to the load impedance will be lower than the input power due to losses inside the transformer materials (coil conductor losses, magnetic core losses, etc.).

Tuner Frequency and Input Power

The components synthesized in the impedance matching network of the tuner will be automatically calculated for a **specified frequency**, which can be chosen from a dropdown menu in the Tuner Parameters box, as shown in Fig. 4.

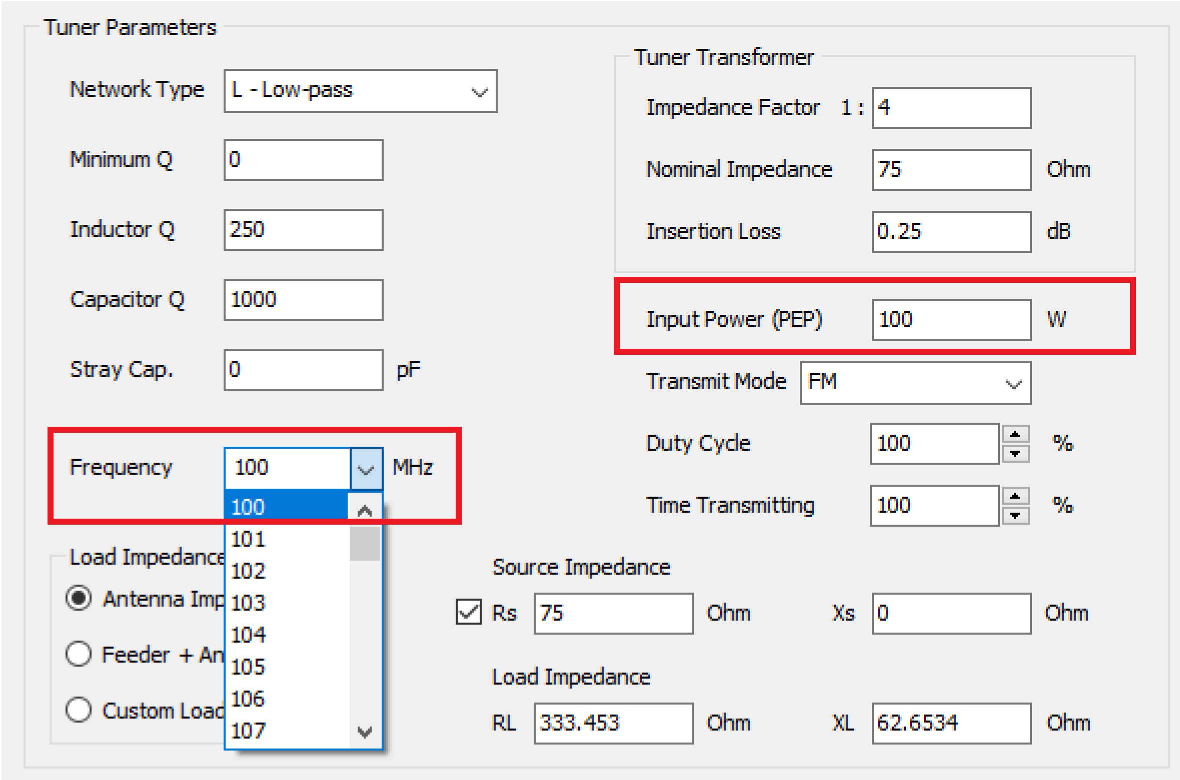


Fig. 4: The tuner design frequency and input power can be set in the Tuner Parameters box.

This list of frequencies is taken from the **Frequency panel** in the **Setup tab**, where a single frequency, a list of frequencies, or a frequency sweep can be configured. Therefore, to change the list of frequencies available in the **Tuner tab**, go to the Setup tab and enter the desired frequencies in the Frequency panel. Note that the frequency chosen for the tuner will be its **design frequency**; thus, the tuner components, inductors, and capacitors will be recalculated if the design frequency changes.

The **Input Power** to the tuner can also be specified in the Tuner Parameters box. This is the power delivered by the **source** connected to the input side of the impedance matching network of the tuner. This input power affects the powers calculated in the **Results** box on the right side of the Tuner tab, as explained below. It is worth mentioning that the tuner input power is not the power delivered to the antenna terminals, which can be set in the **Excitation panel** of the Setup tab. However, if the tuner is connected to an antenna, we can specify that the tuner output power be delivered to the antenna terminals, as detailed below.

Transmit Mode, Duty Cycle, and Time Transmitting

The input power specified is the transmitter's **Peak Envelope Power (PEP)**. However, when performing **RF exposure evaluations**, the **average power** supplied by the transmitter over time is the critical factor. The average power is a fraction of the PEP, determined by the **duty cycle** (or duty factor) of the selected **transmit mode**. The transmit mode can be chosen, and the corresponding percentage duty cycle will be displayed, as shown in Fig. 5. To enter a custom duty cycle, select "Custom" as the transmit mode.

It is also important to account for the percentage of time the transmitter remains active within a specific period, such as 6 minutes. For example, if the telegraph mode transmits for only 3 minutes in every 6-minute period, the power considered for RF exposure calculations is reduced by 50%. Therefore, the **Time Transmitting** parameter can be set as a percentage. Both the duty cycle and the time transmitting percentage will affect the PEP, and an **average input power** will be calculated accordingly.

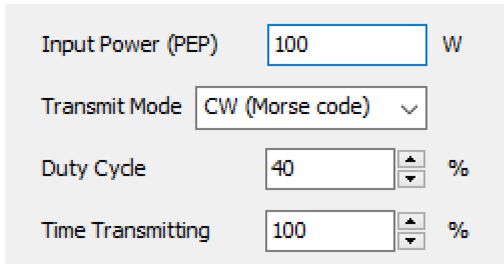


Fig. 5: Transmit Mode, Duty Cycle, and Time Transmitting settings will affect the entered Input Power (PEP).

Tuner Source and Load Impedances

The **source impedance** connected to the tuner input side can be set in real (R_S) and imaginary (X_S) parts, as shown in Fig. 6.

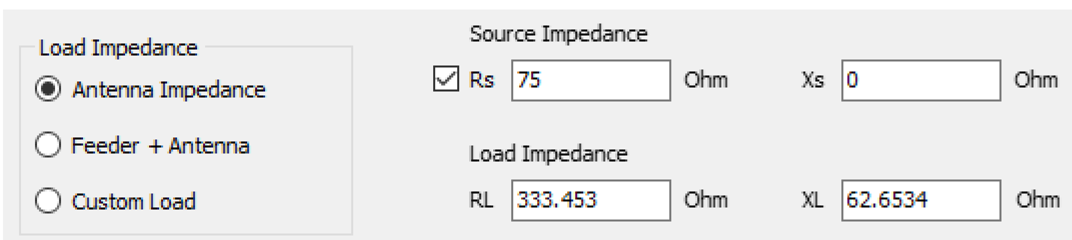


Fig. 6: Specification of the tuner source and load impedances.

When a non-null source reactance, X_S, is entered, it will be absorbed by the impedance matching network calculations. Thus, the **net input impedance of the network**, after adding jX_S, **will be matched to the real part of the source impedance**, R_S. Click on the checkbox next to the "R_S" label to set this resistance as the **reference impedance for VSWR calculations**. This same resistance will be automatically set in the **Settings panel** as the "VSWR Ref. Impedance".

There are three options for the tuner load impedance ($R_L + jX_L$):

> **Antenna Impedance:** Select this option to set the **antenna input impedance** as the **tuner load**. Note that the antenna impedance varies with frequency, so changing the design frequency for the tuner will trigger a recalculation of the impedance matching network.

Feeder + Antenna: This option allows us to set the combination of **feeder + antenna** as the **tuner load**. In this case, the feeder parameters will be taken from the [Feeder] tab at the chosen design frequency. Therefore, the load impedance connected at the tuner output is a function of frequency since it is the input impedance to the feeder connected to the antenna.

Custom Load: This option allows setting a tuner load impedance **manually** by specifying its real (R_L) and imaginary (X_L) parts. The Tuner tab can be used as an independent impedance matching calculator in this case.

Tuner Results

The results of the calculations based on the configured tuner parameters are displayed in the **Results** box on the right side of the Tuner tab, as shown in Fig. 7.

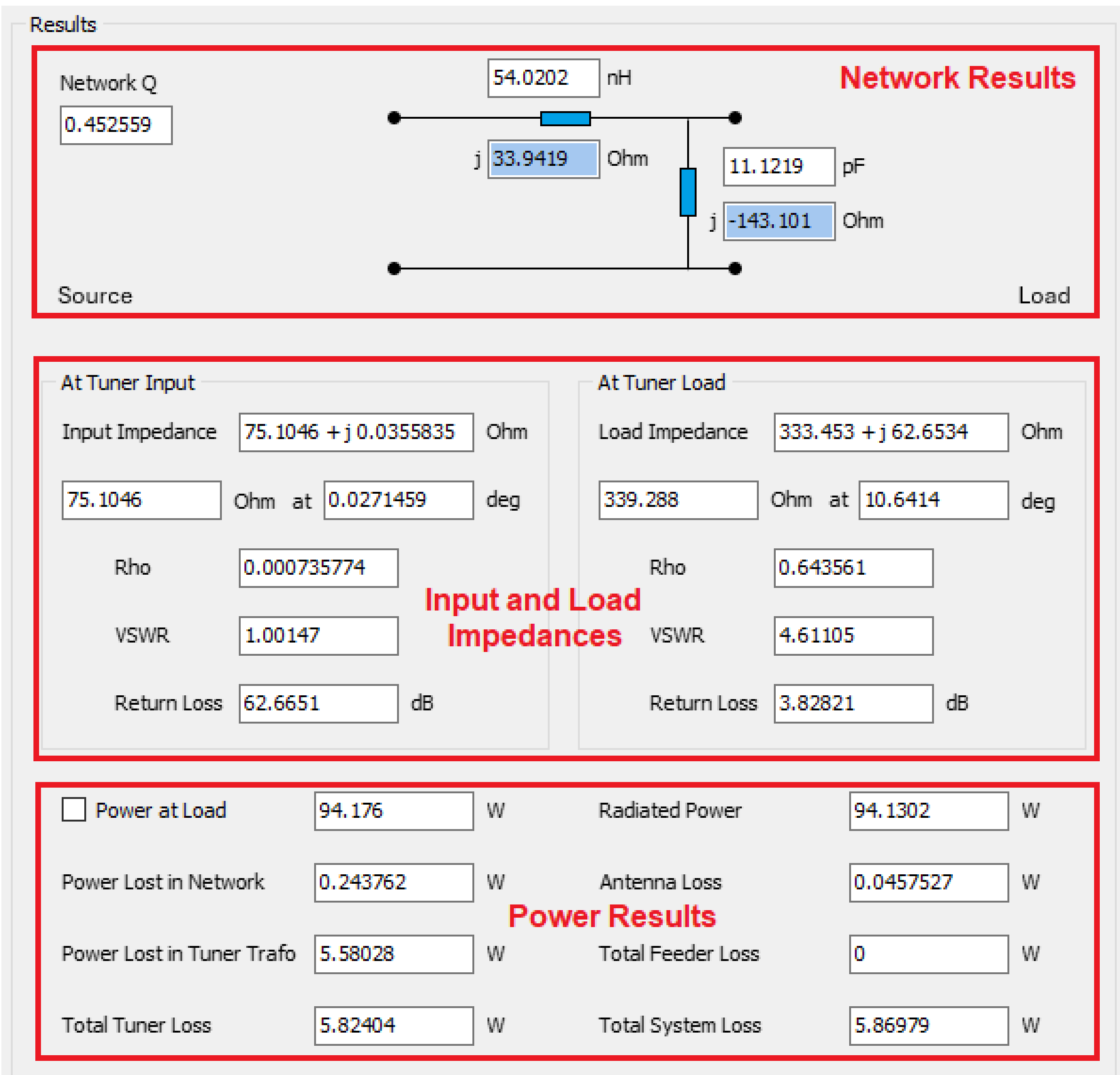


Fig. 7: Tuner Results box on the right side of the Tuner tab.

The results are categorized into three sections: **Network results**, **input and load impedances**, and **power results**.

Network Results

The network results shown include the resulting **network Q** and a diagram illustrating the **network components**, including **inductors** and **capacitors**. For inductors, their inductance in Henry and reactance in Ohms will be displayed, while for capacitors, their capacitance in Farads and reactance in Ohms will be shown. The units of inductance and capacitance displayed can be changed to pH, nH, uH, mH, H, or pF, nF, uF, mF, F, respectively, by navigating to the AN-SOF main menu > Tools > [Preferences](#) > Units tab.

It's worth mentioning that the resulting network Q for **L-type networks** is determined only by the impedances connected to the load and source side of the network. Therefore, the minimum Q specified in the Tuner Parameters box has no effect for L networks.

Tuner Input and Load Impedances

The resulting **input impedance to the tuner** will be displayed in both real and imaginary parts, along with a polar representation showing its magnitude in Ohms and phase in degrees. If the **source impedance**, $R_S + jX_S$, connected to the tuner has a non-null reactance, jXs, this will be absorbed by the impedance matching network. Consequently, the displayed tuner input impedance represents **the impedance seen towards the tuner just after R_S**, as illustrated in the diagram on the left side of the Tuner tab (Fig. 8).

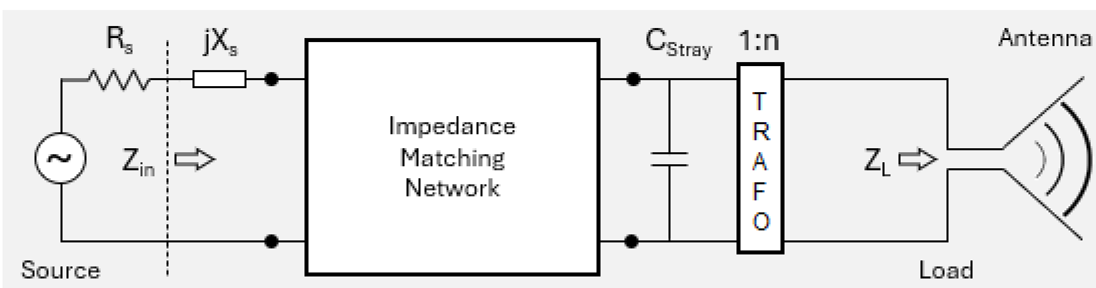


Fig. 8: Tuner diagram showing that the input impedance, Z_{in}, is seen towards the tuner after the source resistance, R_S, since the source reactance, jX_S, is absorbed by the impedance matching network.

The **load impedance** connected to the tuner output terminals will also be shown, which can be the **antenna input impedance**, a [feeder + antenna](#) combination, or a **user-entered impedance** in the Tuner Parameters box on the left side of the Tuner tab.

For both the tuner input and load impedances, the **reflection coefficient (Rho)**, **VSWR**, and **return loss in dB** will be displayed. These values are referred to the **reference impedance for VSWR**, which has been configured in the [Settings panel](#) of the Setup tab.

Powers Delivered and Lost

At the bottom of the Results box, the following powers are calculated:

Power at Load: This is the power effectively delivered to the **tuner load impedance**. Note that the tuner consists of the **impedance matching network + stray capacitance + transformer** sequence. Therefore, the power at the tuner load represents the power delivered at the transformer output terminals. If an **antenna impedance** is chosen as the tuner load, the “Power at Load” is the power delivered to the **antenna terminals**. If a **feeder + antenna** is chosen as the tuner load, the “Power at Load” is the power delivered to the **feeder terminals**. To apply this power to the antenna model in the Workspace tab, check the checkbox next to the “Power at Load” label.

Power Lost in Network: This is the total power lost in the network components, including **inductors** and **capacitors**, due to the losses related to the specified **quality factors**, Q. In the impedance matching network, a resistance, R = X/Q, representing component losses, is added in series to the inductor and capacitor reactance, X.

Power Lost in Tuner Trafo: This is the power lost in the impedance transformer due to the specified insertion loss.

Total Tuner Loss: This is the sum of the network and transformer losses.

Radiated Power: If an **antenna impedance** is set as the **tuner load**, this is the power effectively radiated by the antenna after discounting losses in the antenna system. If a **feeder + antenna** is set as the **tuner load**, this is the power radiated by the antenna after discounting losses in the feeder and the antenna system.

Antenna Loss: This is the power lost in the antenna structure, considering **conductor losses**, **transmission line losses**, if any, and **ground plane losses**.

Total Feeder Loss: If a **feeder + antenna** is chosen as the **tuner load**, this is the power lost in the feeder system.

Total System Loss: This is the sum of the power lost in the **tuner** (network + transformer), **antenna** (conductors, transmission lines, and ground plane), and **feeder** (feeding line + transformer), if specified.

Displaying Smith Charts

The **Smith Chart** is an essential tool for visualizing the input impedance of your antenna as it varies with frequency. It allows you to see how “matched” your antenna is to a reference system (typically 50Ω) at a glance.

How to Generate the Chart

1. Follow the procedure for **Listing the Input Impedances**: Right-click source wire > List Currents > Input List (**Fig. 1**).
2. In the **Input List** dialog box, click the **Smith** button. This launches the **AN-Smith** application.

Interactive Features in AN-Smith

- **Data Hints:** Left-click anywhere on the impedance curve. A hint message will appear displaying the **Frequency**, **Input Impedance (Zin)**, **Reflection Coefficient (Rho)**, **VSWR**, and **S11** for that specific point.
- **Admittance Plotting:** To switch from impedance coordinates to admittance, go to the main menu and select **Plot > Admittance**.
- **Customization:** Navigate to **Edit > Preferences** to adjust the visual appearance, such as line thickness and colors.

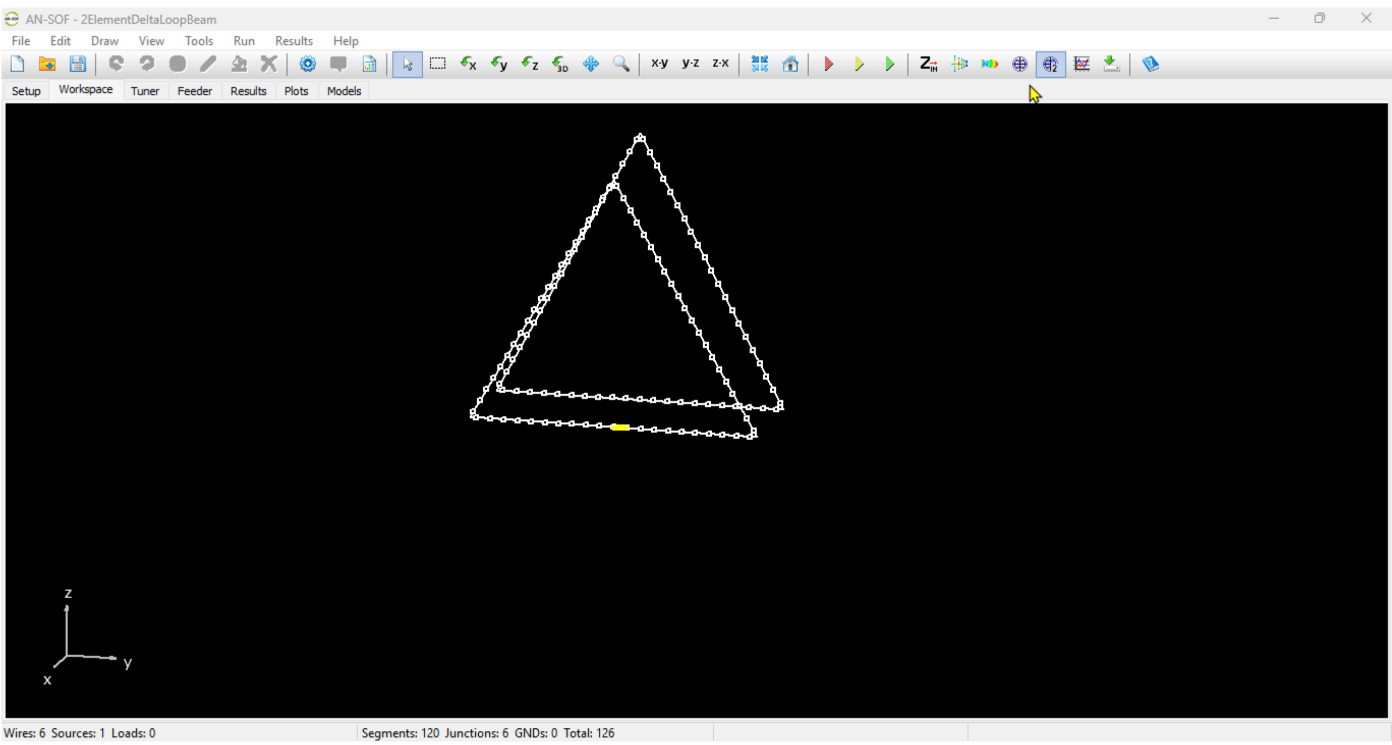


Fig. 1: *Accessing the "List Currents" toolbar to tabulate the input impedance versus frequency and plot it in a Smith chart.*

Antenna Feeder Calculator

Adding a Feed Line and Transformer

In this article, you will learn how to add a feed line and transformer to your AN-SOF project. These components are essential for connecting your antenna structure to the external circuitry and **impedance matching**.

In the case of a transmitting antenna with a single feed port, the feeder used to connect the transmitter to the antenna terminals can be modeled in the **Feeder** tab, as shown in Fig. 1. The feeder consists of a transmission line, or **feed line**, and an impedance **transformer**.

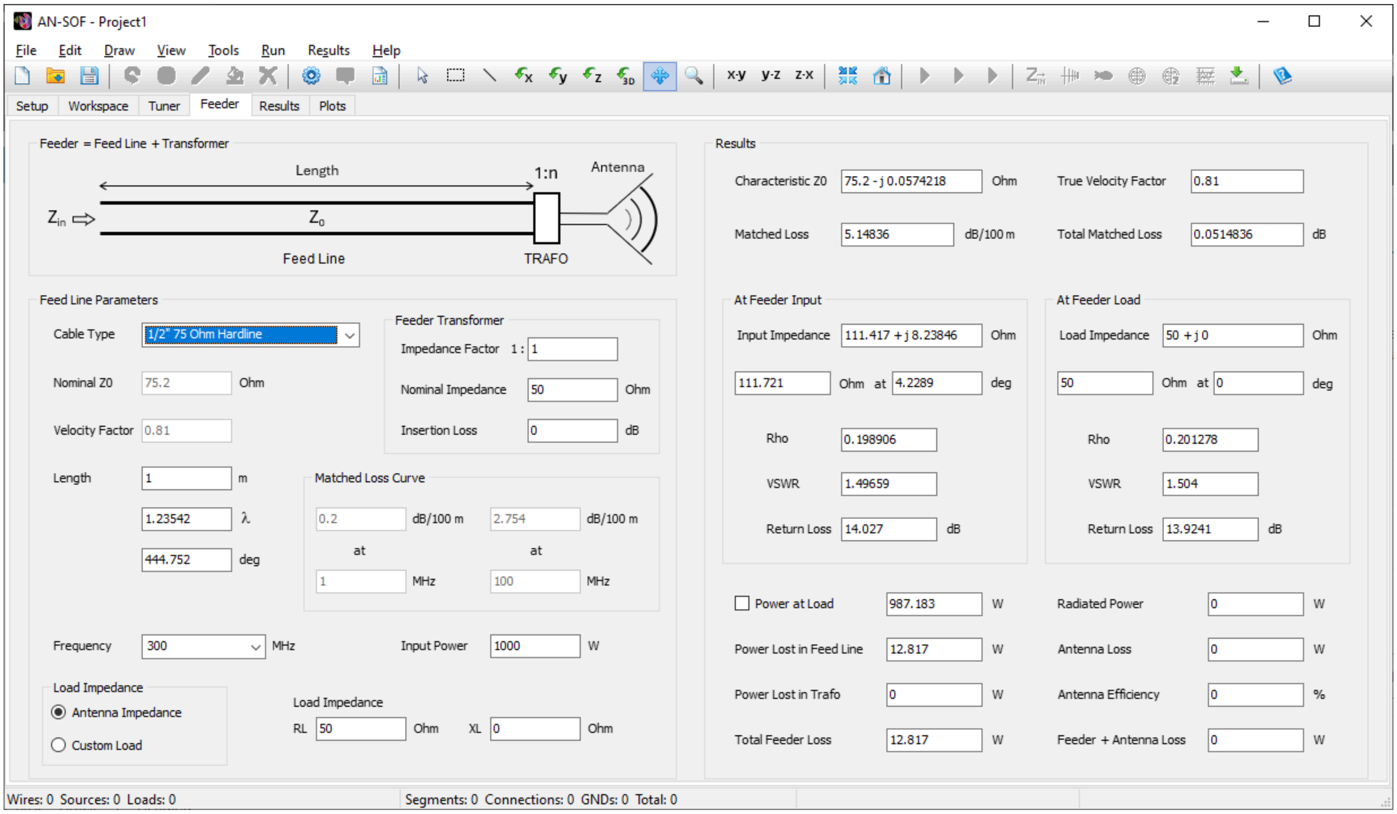


Fig. 1: Feeder tab where the feed line and transformer used to feed a transmitting antenna can be configured.

Setting the Impedance Transformer

The transformer, also known as **trafo**, can represent a **balun** or **unun** that connects directly to the antenna terminals to divide its input impedance by a factor, **n**. In the **Feeder Transformer** box, three parameters can be specified:

Impedance Factor 1:n

Here, "n" is the factor by which the antenna input impedance will be **divided**. For example, if we have a folded half-wave dipole, which typically has an input impedance on the order of 300 Ohms, we can set n = 4 to get 300/4 = 75 Ohms of input impedance after the transformer (i.e., a 1:4 balun). If the input impedance is complex, both its real and imaginary parts will be divided by n.

If the transformation factor is in the range **0 < n < 1**, the transformer input impedance will be greater than the antenna impedance. By setting **n = 1**, we can represent a **1:1 transformer**, also known as a common-mode choke or line isolator, used to transform a balanced or symmetrical antenna to an unbalanced feed line.

Note that “n” is the **impedance transformer factor**, not the **voltage transformation factor**. In a transformer, which is composed of a primary winding (inductor or coil) and a secondary winding, the voltage transformation factor is $n^{-1/2}$.

Nominal Impedance

All actual impedance transformers, whether **baluns** or **ununs**, are fabricated for a **nominal impedance**, for which the manufacturer warranties the transformer performance in terms of **bandwidth** and **insertion loss**. So, if a lossy transformer is going to be modeled, we should set its nominal impedance according to the manufacturer’s datasheet.

Insertion Loss

The insertion loss of the transformer can be set in **decibels** to represent the actual loss given in its datasheet. The insertion loss is defined as the power lost, in decibels, inside the transformer, so that its output power will be lower than its input power due to losses in the transformer materials (coil resistivity, magnetic core losses, etc.).

Note: If no transformer is needed, just set **n = 1** and an insertion loss of **0 dB**.

Setting Feed Line Parameters

In AN-SOF, various real-life transmission line types are available, each with matched loss parameters adjusted according to the cable datasheets. These cable types are organized by part numbers and include the manufacturer’s name.

For example, entering “RG-8” in the **Cable Type** option will display this part number for different manufacturers, as shown in Fig. 2. Selecting **RG-8 Belden 8237** will reveal a set of **K0**, **K1**, and **K2** parameters. These constants have been adjusted to match the **loss curve** as a function of frequency, based on the matched loss vs. frequency table published in the cable datasheet. **K0** relates to the **DC losses** in the transmission line conductors, **K1** to the **skin effect losses** dependent on the square root of frequency, and **K2** to **dielectric losses** increasing linearly with frequency. These losses are then considered in the standard **RLGC model** of a lossy transmission line.

The **nominal** values of the cable **characteristic impedance Z0** and **velocity factor** will also be shown for the chosen part number and manufacturer. After selecting the cable type, you can set the **operating frequency** and **input power** to the feed line. The frequency can be chosen from a list that displays the frequencies set in the **Setup tab**.

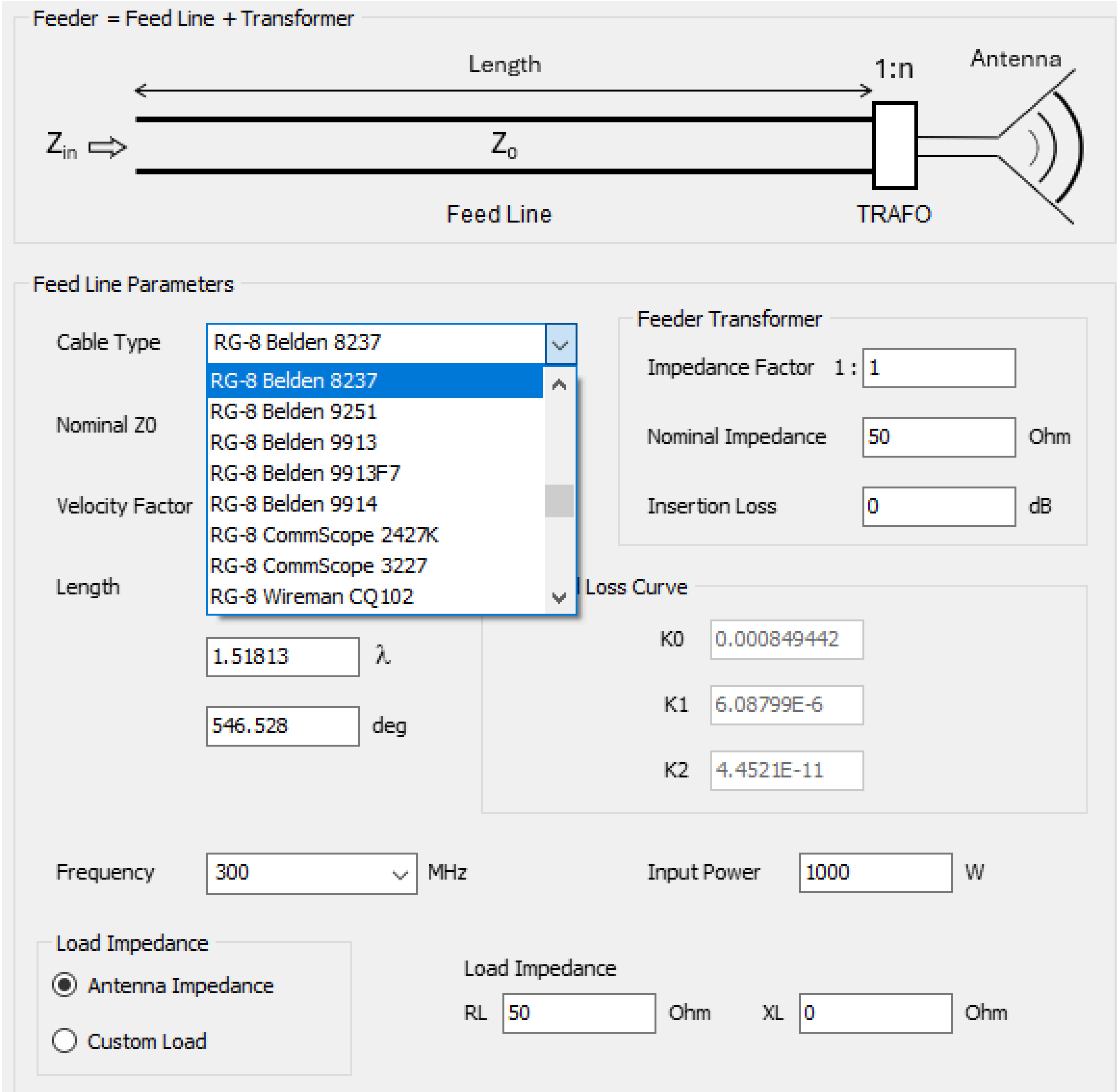


Fig. 2: Cable Type option where the type of transmission line can be chosen.

Next, you can set the **length** of the cable, entered according to the length unit used for drawing wires in the workspace. To change the length unit, go to **Tools > Preferences** in the main menu. As you type the cable length, the length measured in wavelengths (λ) and electrical degrees will be automatically displayed. In fact, all feed line results are calculated automatically by modifying any of the feed line parameters.

You can then choose the **load impedance** of the feed line. The default option considers the **Antenna Impedance** as the load impedance of the transmission line, automatically displaying the antenna input impedance at the chosen frequency as the load for the line. However, you can enter any value for the line load impedance by selecting the **Custom Load** option. **This allows you to use the Feed Line tabsheet as an independent calculator for transmission lines.**

Feeder Results: Input Impedance and Losses

After specifying the **feeder parameters** in the left side of the **Feeder** tab, the following results will be obtained in its right side (Fig. 1):

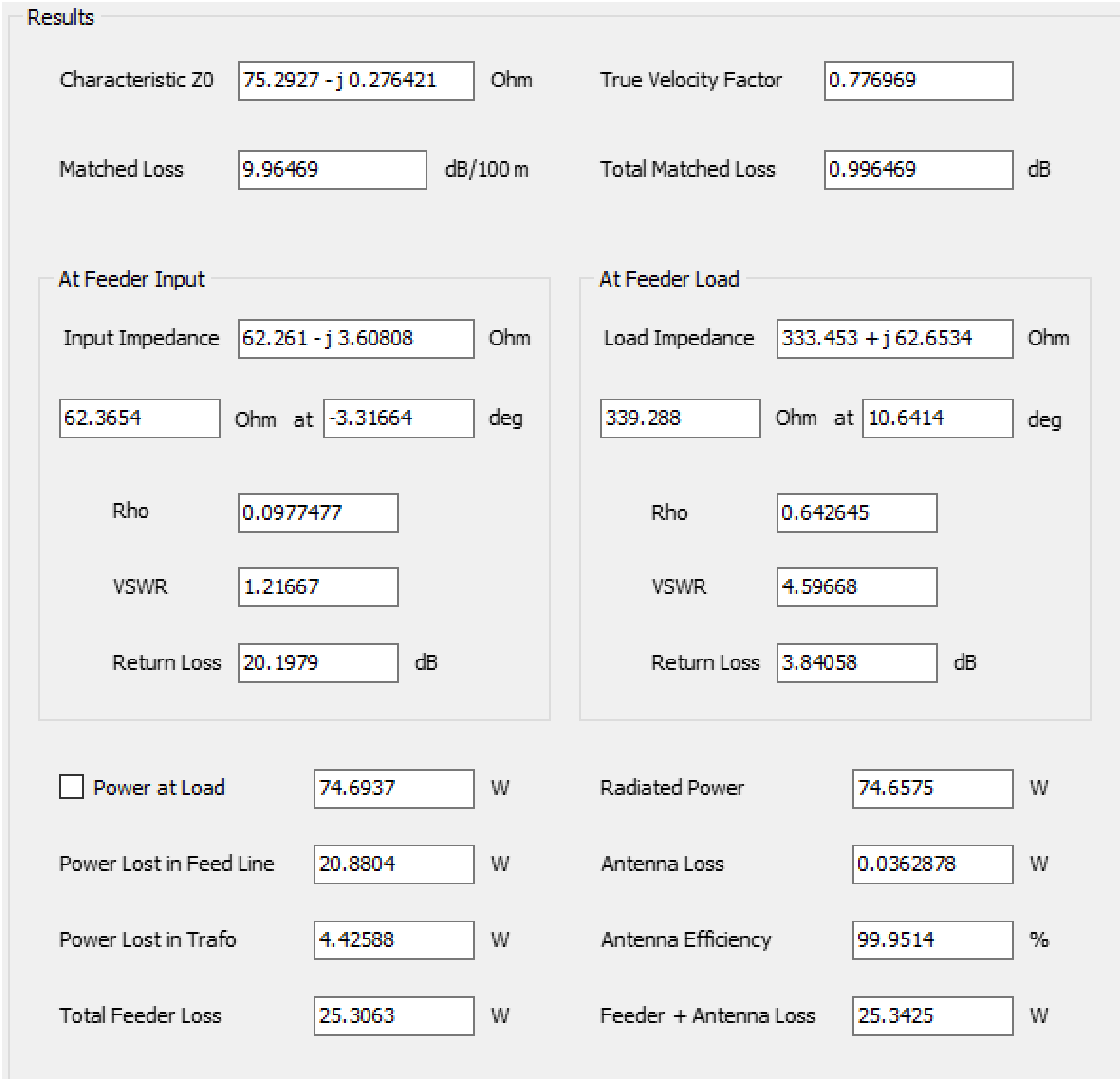


Fig. 1: Results panel on the right side of the Feeder tab. All results are automatically calculated as parameters in the left side of the Feeder tab are modified.

Characteristic Z0

This is the **"true" characteristic impedance** of the feed line obtained from the **RLGC model** via the K0, K1, and K2 constants. The real part of Z0 may differ somewhat from the nominal Z0 depending on frequency and losses in the transmission line. An imaginary part will always appear in Z0 due to non-zero losses. So, note that **the true characteristic Z0 will generally differ from the "Nominal Z0"** (Z0 in the cable datasheet).

True Velocity Factor

This is the **"true" velocity factor** obtained from the **RLGC model** of the transmission line, where the **wavenumber** (and wavelength inside the line) is affected by **losses**. The velocity factor will be modified relative to its nominal value accordingly. Therefore, the true velocity factor is a function of frequency and losses in the line.

Matched Loss

Any cable datasheet contains a table of **matched loss** values expressed in dB/100 feet or dB/100 m as a function of frequency. These values correspond to the **attenuation** of the line when it is matched (the line has a load impedance equal to Z0). So, the Matched Loss value shown in the Results panel is the attenuation of the line corresponding to the selected frequency.

Total Matched Loss

This is the matched loss that would be obtained for the specified length of the cable. Therefore, the Total Matched Loss equals the **Matched Loss** (dB/100 feet or dB/100 m) **multiplied by the cable length**.

At Feeder Input

The **input impedance of the feeder** (feed line + transformer) will be shown as well as the reflection coefficient (Rho), VSWR, and return loss, **all referred to the true characteristic impedance of the feed line**. This is the impedance at the feed line end opposite to the end where the load or antenna is connected.

At Feeder Load

The **load impedance connected to the feeder** (feed line + transformer) will be shown as well as the reflection coefficient (Rho), VSWR, and return loss, **all referred to the true characteristic impedance of the feed line**. The load impedance will be the antenna input impedance if the **Antenna Impedance** option was selected as a parameter for the feed line in the left side of the Feeder tab. If a “Custom Load” was selected, then the load impedance will be that entered by the user.

Power at Load

This is the power in Watts consumed at the **feeder load impedance** or **effectively delivered to the antenna terminals**. This power will be less than the input power specified as an input parameter for the feed line if the transmission line has losses. The power at the load will be equal to the input power in the case of a lossless transmission line. Check the **Power at Load** option to automatically set this power as the input power delivered to the antenna terminals. Otherwise, the antenna input power will be that set manually in the **Excitation** panel of the **Setup** tab.

Power Lost in Feed Line

This is the power lost along the transmission line in Watts.

Power Lost in Trafo

This is the power lost in the feeder transformer in Watts.

Total Feeder Loss

This is the sum of the powers lost in the feed line and in the transformer.

Radiated Power

This is the power in Watts radiated by the antenna when it is fed using the **Power at Load**, which is **the power effectively delivered to the load impedance of the feeder**. The radiated power will be different from the power delivered by the feeder if the antenna itself has its own losses. The radiated power will be shown if the option **Antenna Impedance** was selected as a load impedance for the feeder in the left side of the Feeder tab.

Antenna Loss

This is the power lost in the antenna structure. It will be shown if the option **Antenna Impedance** was selected as a load impedance for the feeder in the left side of the Feeder tab.

Antenna Efficiency

This is the ratio of the antenna radiated power to the antenna input power (the power delivered by the feeder). It is expressed as a percentage as it is usual. It will be shown if the option **Antenna Impedance** was selected as a load impedance for the feeder in the left side of the Feeder tab.

Feeder + Antenna Loss

This is the sum of the powers lost in the feeder (feed line + transformer) and antenna.
Custom Feed Line Options

In addition to the manufactured cables listed in the **Cable Type** option, the following custom line options can be chosen, as shown in Fig. 1:

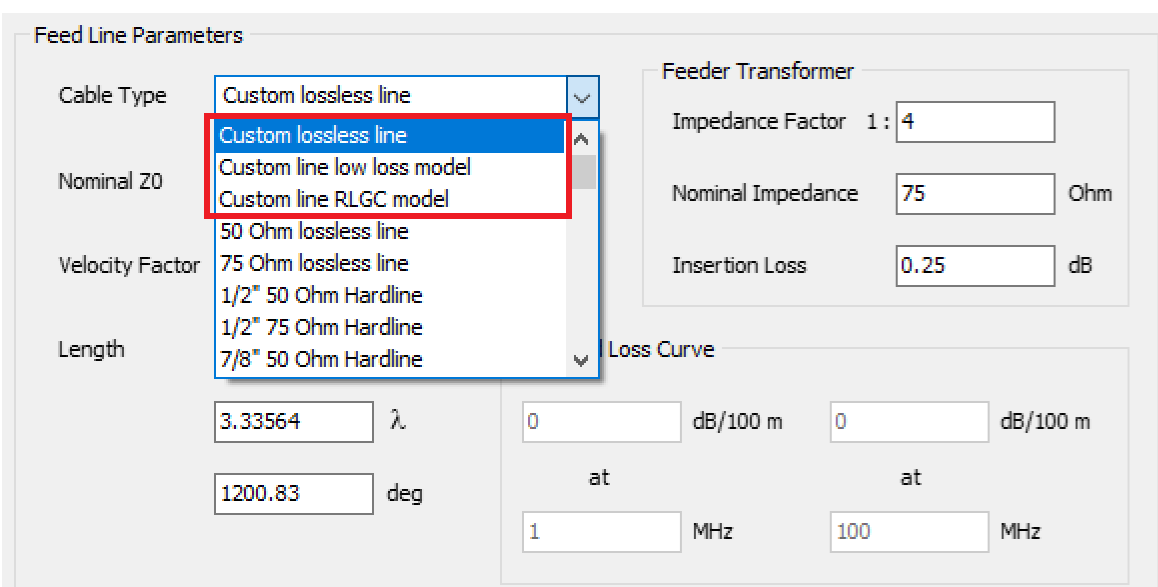


Fig. 1: Custom line options.

Custom lossless line

This option represents an **ideal transmission line** with **zero losses**. Only the **nominal Z0** and **velocity factor** can be specified in this case.

Custom line low-loss model

This option allows the specification of the **nominal Z0**, **velocity factor**, and **matched loss curve**. To define the matched loss curve, two values of attenuation must be entered at two different frequencies, with the second frequency being greater than the first one. AN-SOF will adjust a low-loss model to obtain a curve of attenuation vs. frequency for subsequent calculations. While the real part of the characteristic Z0 will be equal to the nominal Z0 in the low-loss model, which is a good approximation in many cases, especially for higher frequencies, the characteristic impedance will have an imaginary part that depends on the line losses and frequency. The “true” velocity factor is also assumed to be equal to the nominal velocity factor.

Custom line RLGC model

This option represents a transmission line model where losses are accurately considered by adjusting a **matched loss curve** to the table of attenuation vs. frequency in the **cable datasheet**. The K0, K1, and K2 constants must be entered in this case. The definition of K0, K1, and K2 considers that the frequency is in Hz and lengths are in meters (SI metric units). This option allows the entry of K0, K1, and K2 obtained from other transmission line calculators.

Load Impedances

Listing Load Impedances

Analyzing how power is dissipated or transferred through **lumped loads** (such as resistors, inductors, or capacitors) is vital for understanding matching networks or loaded antenna performance. Use the following steps to tabulate and plot these values:

1. **Select the Loaded Wire:** Right-click on the wire where you have placed a load to open the **pop-up menu**.
2. **Open the “List Currents” Toolbar:** Select the **List Currents** command (**Fig. 1**).
3. **Navigate to the Load:** Use the **slider** to move to the specific segment containing the load.
4. **Access the Load List:** Click the **Load List** button (this button remains disabled unless a load is detected on the selected segment).

Data Interpretation and Plotting

The **Load List** dialog box provides a detailed breakdown of the following parameters across your frequency range:

- **Load Current and Voltage:** The actual values present at the load’s terminals.
- **Power:** The real and reactive power consumed by the load.

- **Impedance:** The calculated impedance of the load at each frequency step.

To visualize this data, select the desired parameter from the menu in the upper-right corner and click the **Plot** button to open the **AN-XY Chart**.

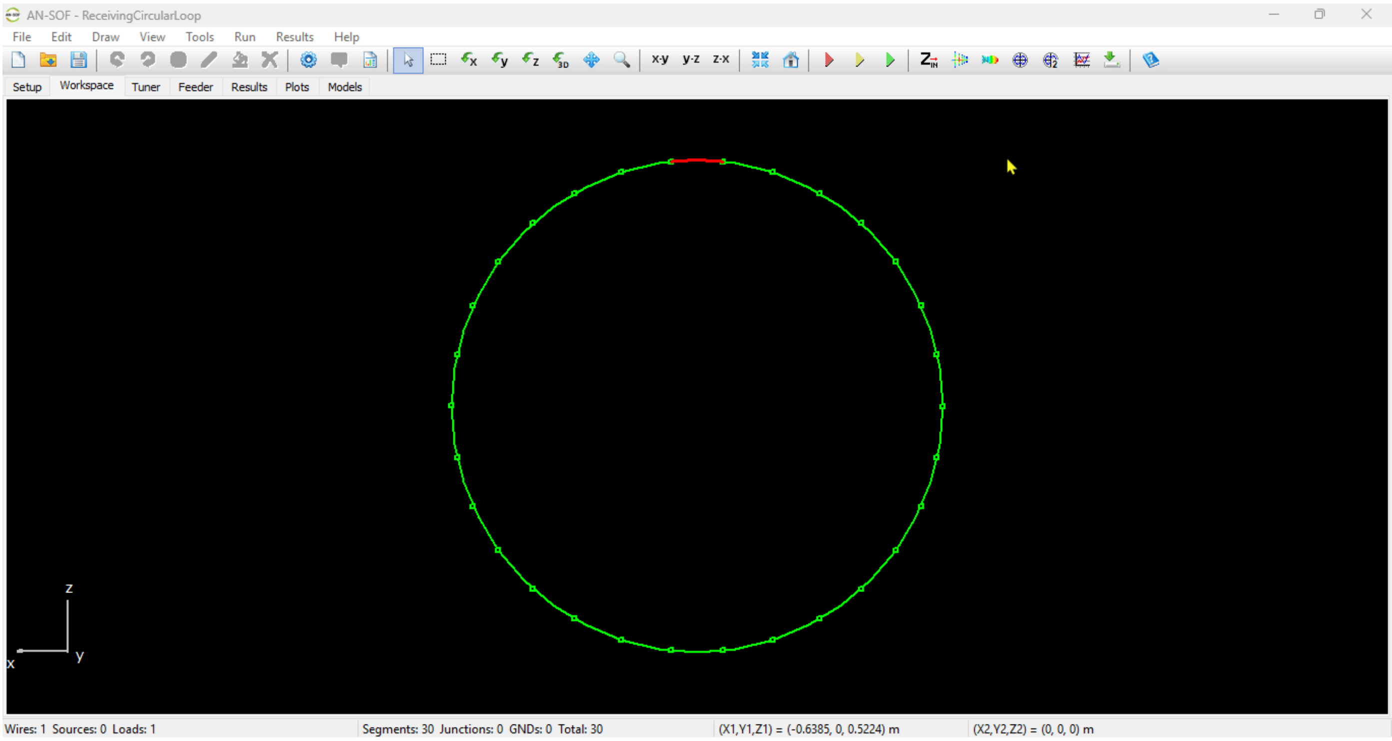


Fig. 1: *Accessing the "List Currents" toolbar to tabulate and plot the impedance, voltage, current, or power at a lumped load in a wire segment.*

Analyzing the Internal Impedance of a Source

In many simulations, a source isn't just an ideal voltage; it often includes an **internal impedance** that accounts for the physical characteristics of the feedline or generator. You can analyze how much power is lost internally versus how much is delivered to the antenna by following these steps:

1. **Locate the Source:** Right-click on the wire where the source is positioned to open the **pop-up menu**.
2. **Open the "List Currents" Toolbar:** Select **List Currents** to activate the corresponding toolbar.
3. **Select the Source Segment:** Use the **slider** (or the **50% button** if centered) to highlight the segment containing the source.
4. **Open the Source List:** Click the **Source List** button. This opens a dialog box that focuses specifically on the internal behavior of the source (**Fig. 1**).

Evaluating Internal Parameters

The **Source List** dialog provides a frequency-by-frequency breakdown of:

- **Internal Current & Voltage:** The electrical values specifically across the source's internal impedance.
- **Internal Power:** Useful for determining if the generator is properly matched to the load (the antenna).

To visualize these trends, select an item from the list in the upper-right corner and click **Plot** to view it in **AN-XY Chart**.

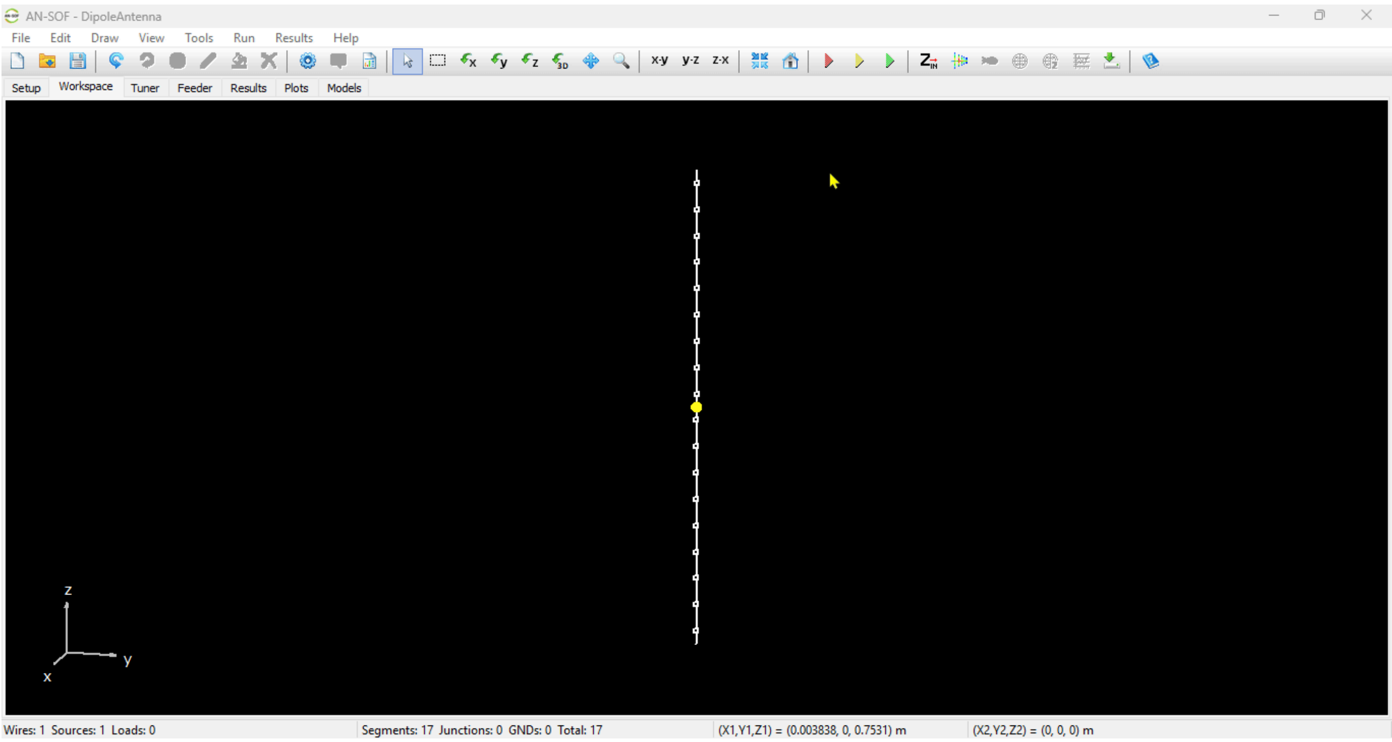


Fig. 1: *Accessing the "List Currents" toolbar to tabulate and plot the impedance, current, voltage, or power at the internal impedance of a source.*

Far Field

Plotting 2D Far Field Patterns

The radiation pattern can be visualized as a 2D rectangular plot by selecting **Results > Plot Far-Field Pattern > 2D Rectangular Plot** from the main menu. This action will open the **Radiation Pattern Cut** dialog box (Fig. 1), where two plot types are available:

- **Conical Plots:** Generated with a fixed Theta and variable Phi.
- **Vertical Plots:** Created with a fixed Phi and variable Theta.

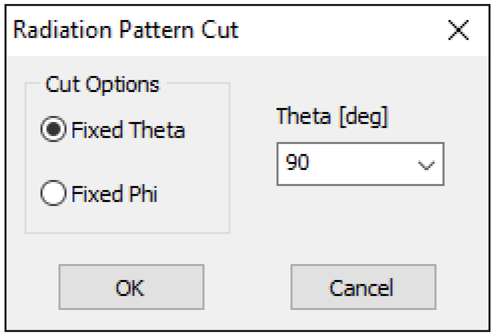


Fig. 1: The Radiation Pattern Cut dialog box.

Select a radiation pattern cut and click **OK** to launch the **AN-XY Chart** application (Fig. 2), where the radiation pattern is plotted against Phi for conical plots (fixed Theta) or against Theta for vertical plots (fixed Phi).

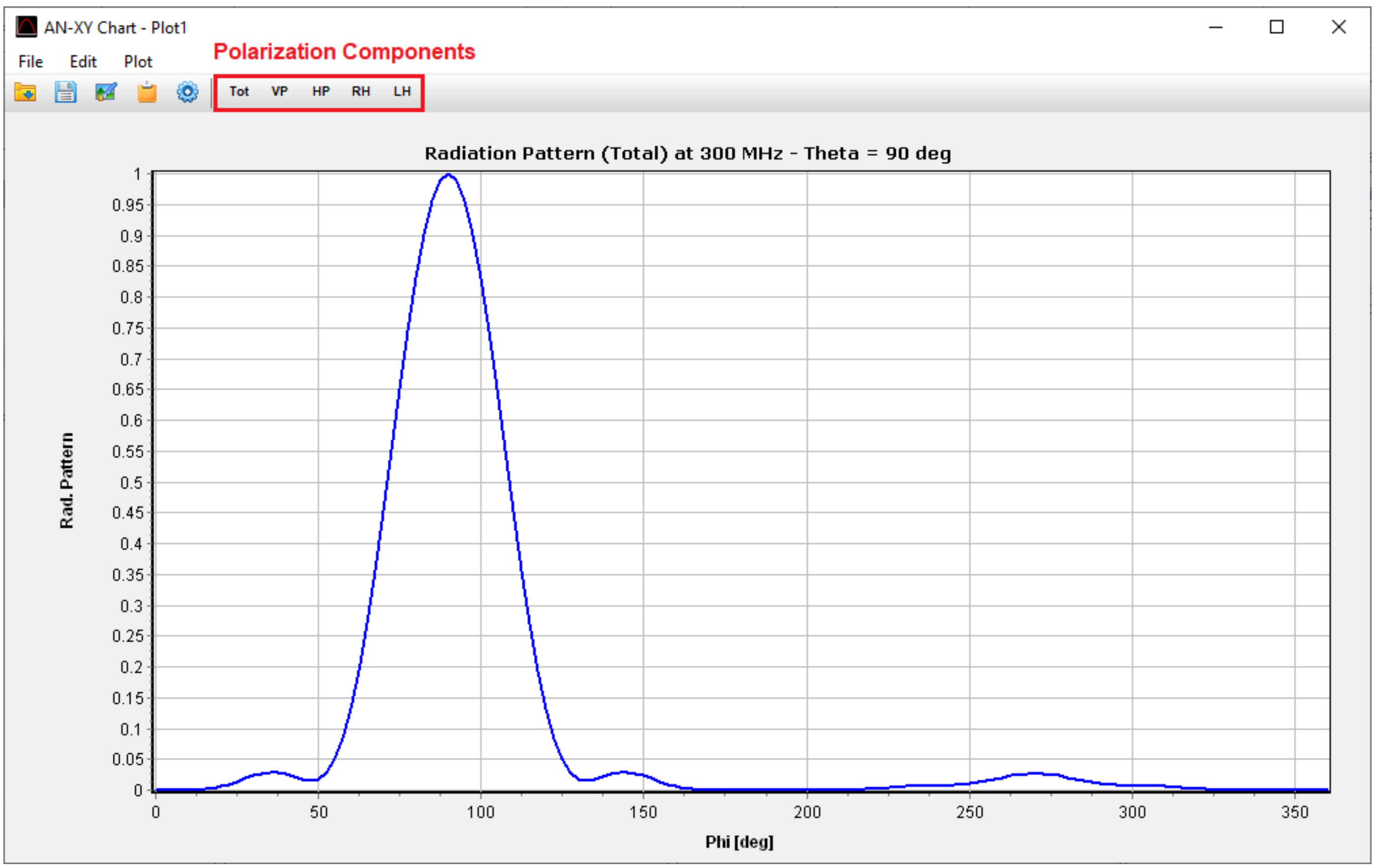


Fig. 2: A Radiation Pattern Cut plotted in AN-XY Chart in a rectangular chart.

Within the **AN-XY Chart** app, access the **Plot** menu to graph various parameters, including **Power Density**, **Directivity**, **Gain**, **E-field**, and **Axial Ratio**. This menu also allows you to represent these metrics in decibels (dBi for directivity and gain) and decompose them into linearly polarized components: **Theta** (VP: Vertically Polarized) and **Phi** (HP: Horizontally Polarized), as well as circularly polarized components: **Right** (RHCP: Right-Handed Circularly Polarized) and **Left** (LHCP: Left-Handed Circularly Polarized). The app's toolbar features buttons: **Tot**, **VP**, **HP**,

RH, and **LH** for quick switching between the total field metric and its corresponding polarization components. For instance, you can plot the total gain in dBi or decompose it into its Theta (VP), Phi (HP), Right (RHCP), or Left (LHCP) components to analyze antenna polarization characteristics.

In the case of **plane wave excitation**, where the antenna is receiving or the metallic structure is scattering electromagnetic waves, the **Radar Cross Section (RCS)** will be plotted instead of directivity and gain. The resulting radiation pattern is the **scattered field pattern**, observed in the far-field region where the scattered field amplitude decays with the inverse of the distance (1/r), and the scattered power density decays with the inverse square of the distance ($1/r^2$) from the structure.

The **Axial Ratio** is defined as the ratio of the minor axis to the major axis of the polarization ellipse. It ranges from 0 to 1 in absolute value and can also be plotted in decibels. A circularly polarized field exhibits an axial ratio of ±1 (or 0 dB), while a linearly polarized field has an axial ratio of zero. A positive (negative) axial ratio indicates a right-handed (left-handed) polarized field.

The far-field pattern can also be visualized in a 2D polar chart by selecting **Results > Plot Far-Field Pattern > Polar Plot 1 Slice** from the AN-SOF main menu (refer to Fig. 3). This action will launch the **AN-Polar** app, which displays information such as maximum radiation, beamwidth, and front-to-rear/back ratios.

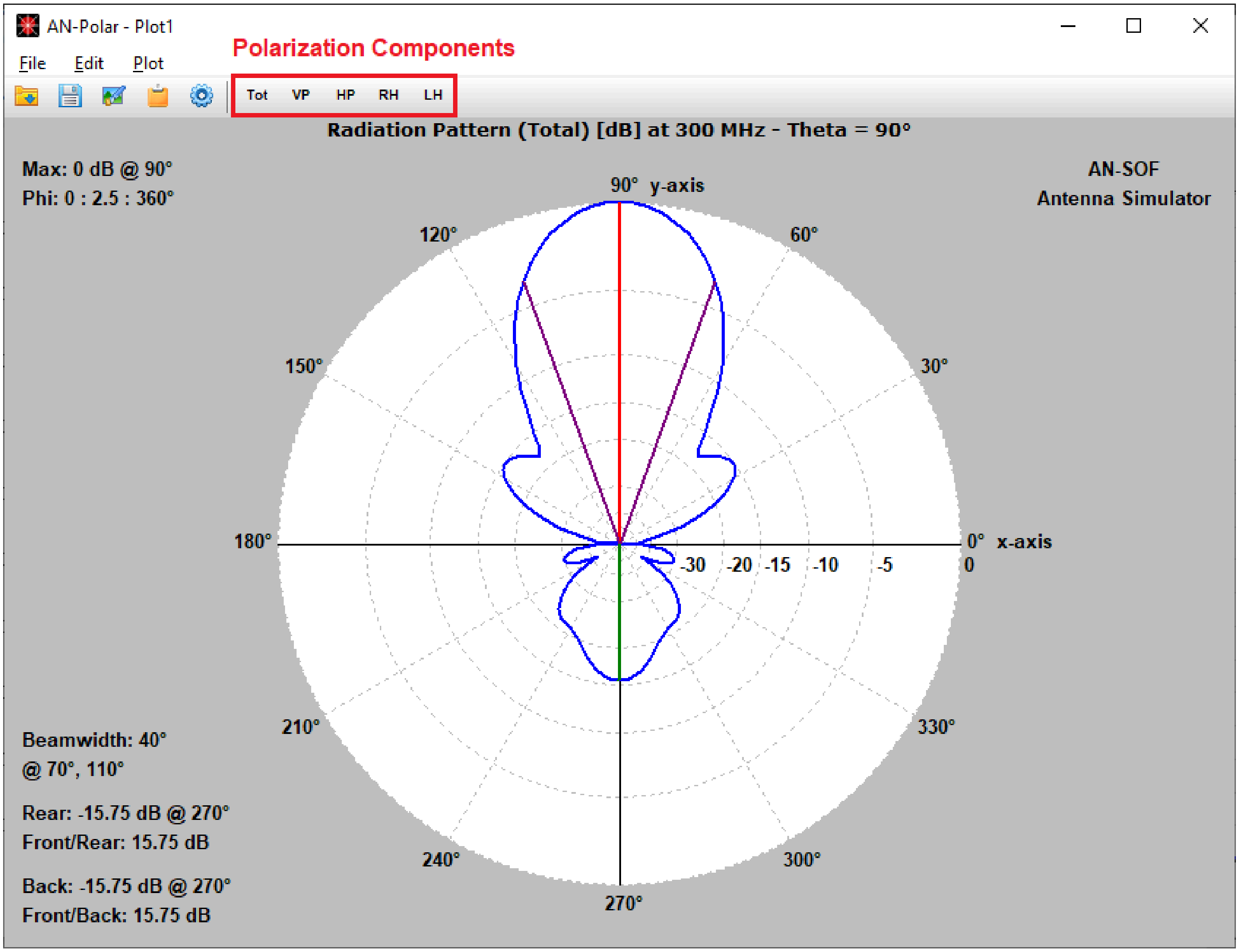


Fig. 3: A radiation pattern cut plotted in AN-Polar.

The **AN-Polar** app also features a toolbar with buttons: **Tot**, **VP**, **HP**, **RH**, and **LH** that enable the decomposition of the plotted metric into its polarization components.

To plot two slices of a 3D far-field pattern on the same polar chart, navigate to **Results > Plot Far-Field Pattern > Polar Plot 2 Slices** in the AN-SOF main menu. A dialog box will appear, allowing you to select the two slices. You can choose from two vertical slices, two conical slices, or vertical-conical combinations (see Fig. 4).

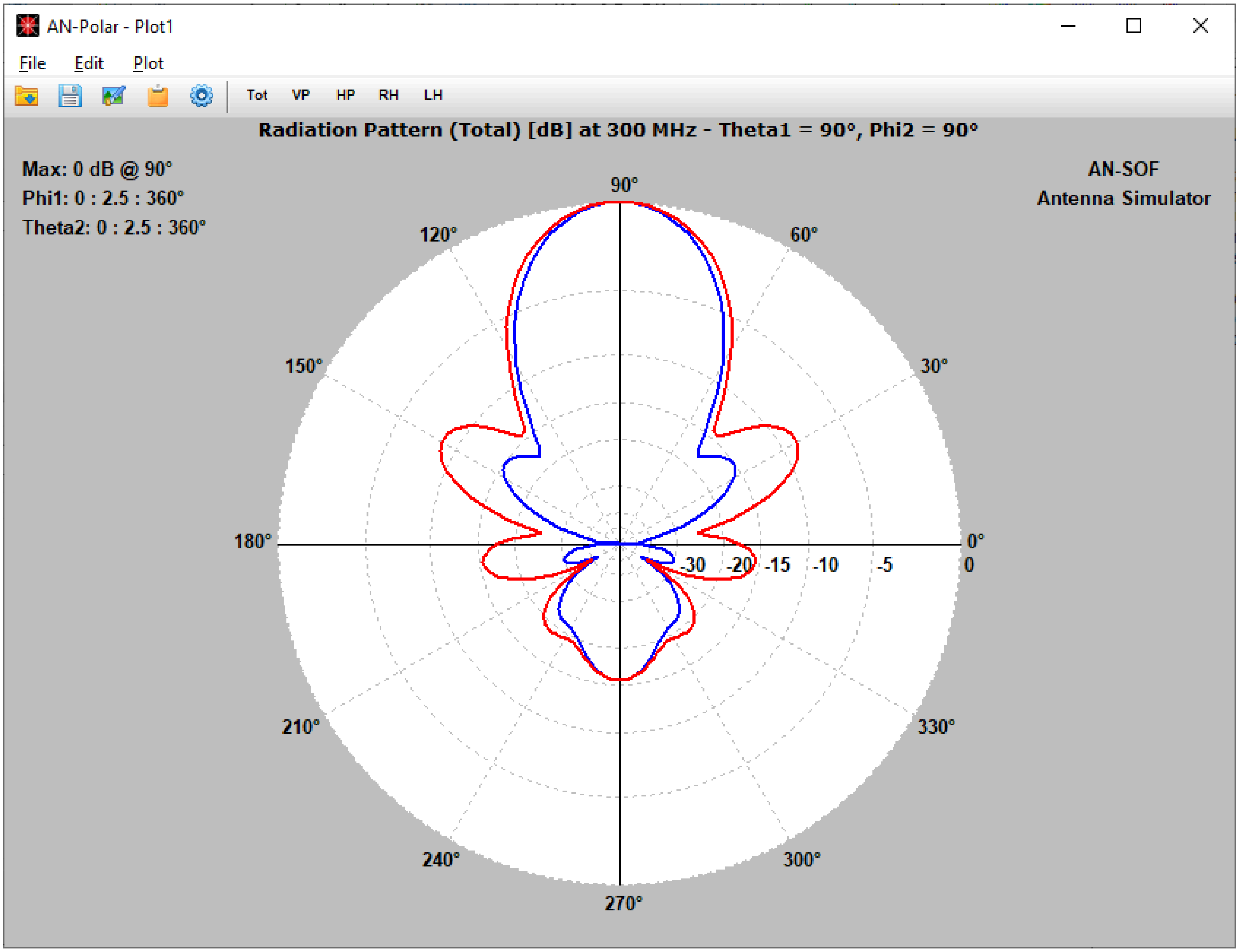


Fig. 4: Two slices of the radiation pattern plotted in AN-Polar.

Clicking on a point in the polar curve will display the corresponding value of the represented metric and the polar angle.

When plotting a **single slice** of the radiation pattern, the **AN-Polar** application automatically displays key performance metrics—**beamwidth**, **Front-to-Back ratio**, **Front-to-Rear ratio**, and their corresponding angles—in the lower-left corner of the window. For definitions and practical explanations of these metrics, refer to the article:

[Front-to-Rear and Front-to-Back Ratios: Applying Key Antenna Directivity Metrics]()

Plotting 3D Far Field Patterns

The far-field can be visualized as a 3D plot by selecting **Results > Plot Far-Field Pattern > 3D Plot** from the AN-SOF main menu. This action will open the **AN-3D Pattern** application, where the radiation pattern is displayed in a 3D view, showcasing the radiation lobes with their intensities represented by a color scale.

Within the **AN-3D Pattern** application, access the **Plot** menu to select the **Power Density**, **Directivity** (numerical and in dBi), **Gain** (numerical and in dBi), **Radiation Pattern** (normalized to unity and to 0 dB), **E-field**, and **Axial Ratio** (dimensionless and in dB) (see Fig. 1). Each field metric can be decomposed into its linearly polarized components **Theta** (VP: Vertical Polarization) and **Phi** (HP: Horizontal Polarization), as well as its circularly polarized components **Right** (RHCP: Right-Handed Circular Polarization) and **Left** (LHCP: Left-Handed Circular Polarization).

If the simulation involves **plane wave excitation**, the **Radar Cross Section (RCS)** can be plotted instead of directivity and gain. The resulting radiation pattern is the **scattered field pattern**, observed in the far-field region where the scattered field amplitude decays with the inverse of the distance (1/r), and the scattered power density decays with the inverse square of the distance ($1/r^2$) from the structure.

The **Axial Ratio** pattern is defined as the ratio of the minor to major axis of the polarization ellipse. It equals 0 for a linearly polarized field and 1 for a circularly polarized field. While lobes in a 3D polar plot can only represent absolute values, the sign of the axial ratio, which determines whether the field is RHCP or LHCP, cannot be directly visualized here but can be observed in a **2D rectangular plot**. However, the toolbar in the **AN-3D Pattern** application features buttons: **Tot**, **VP**, **HP**, **RH**, and **LH** for quick switching between the total field and its polarization components, facilitating polarization analysis.

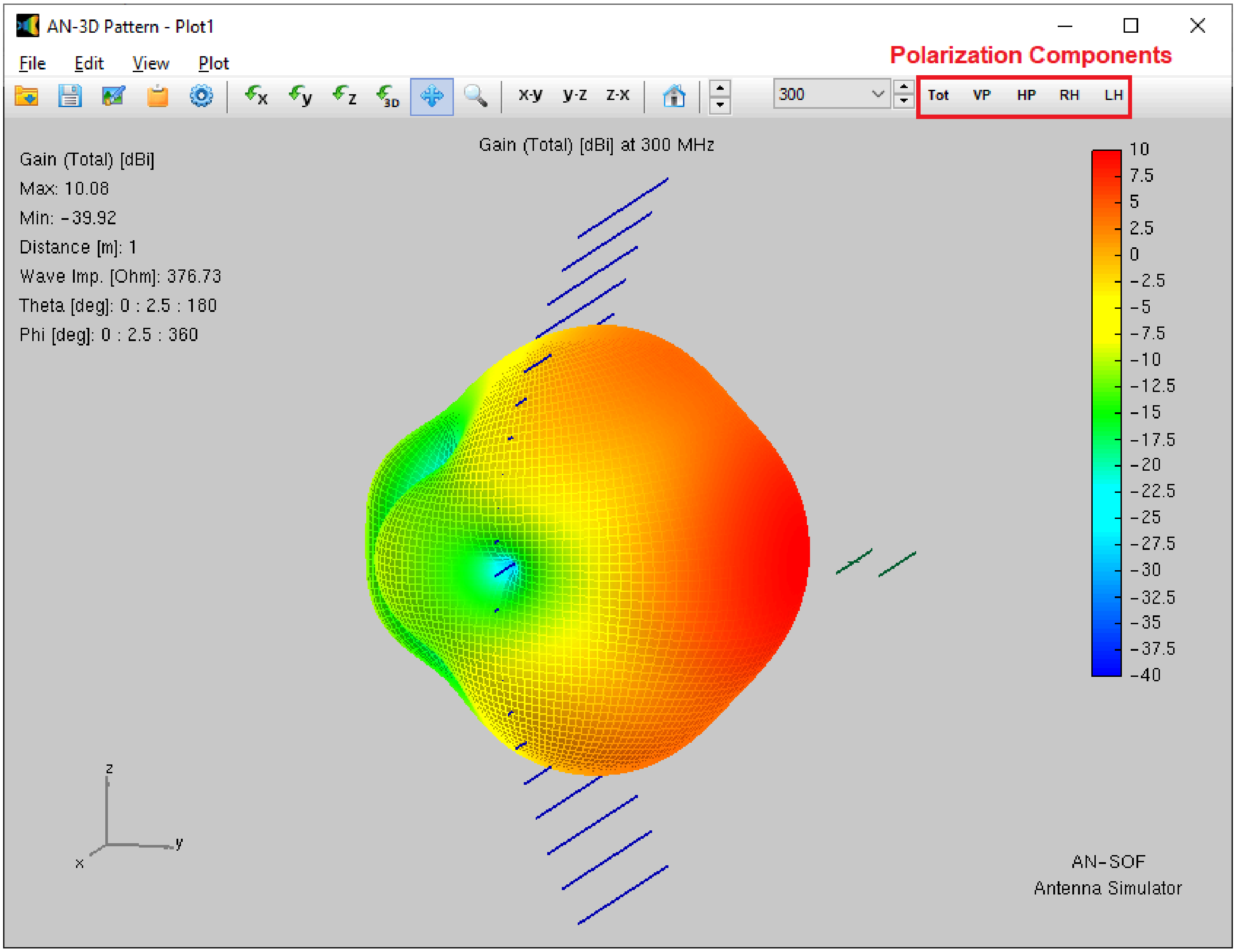


Fig. 1: 3D far-field pattern (Gain in dBi) plotted in AN-3D Pattern.

The 3D graph can be rotated and moved by clicking the "3D Rotation" or "Move" buttons on the toolbar and then dragging the mouse with the left button pressed. Use the mouse wheel to zoom in or out. The **AN-3D Pattern** toolbar also includes an option to change the frequency and dynamically observe the changes in the radiation pattern lobes as a kind of animation (use the up-down arrow buttons next to the displayed frequency value).

> **Note**
>
> - If **discrete sources** were used as the excitation of the structure, the plotted far-field represents the **total field**.
> - If **an incident plane wave** was used as the excitation, the plotted far-field represents the **scattered field**.

To access the **Preferences** dialog box in the **AN-3D Pattern** main menu, click on **Edit > Preferences** (refer to Fig. 2). This dialog box allows you to customize various options for the colored surface and mesh of the radiation lobes (see Fig. 3). Additionally, you can superimpose the wire structure onto the radiation pattern by selecting the **Wires** option in the “Show” box. You also have control over the graph’s scale and can display the main axes.

The radiation pattern cannot be directly exported from the **AN-3D Pattern** application. However, the far-field pattern for a specific frequency can be tabulated by navigating to the **AN-SOF** main menu > **Results > List Far-Field Pattern** and then pressing the “Export” button next to the displayed table to export the data to a CSV (Comma Separated Values) file.

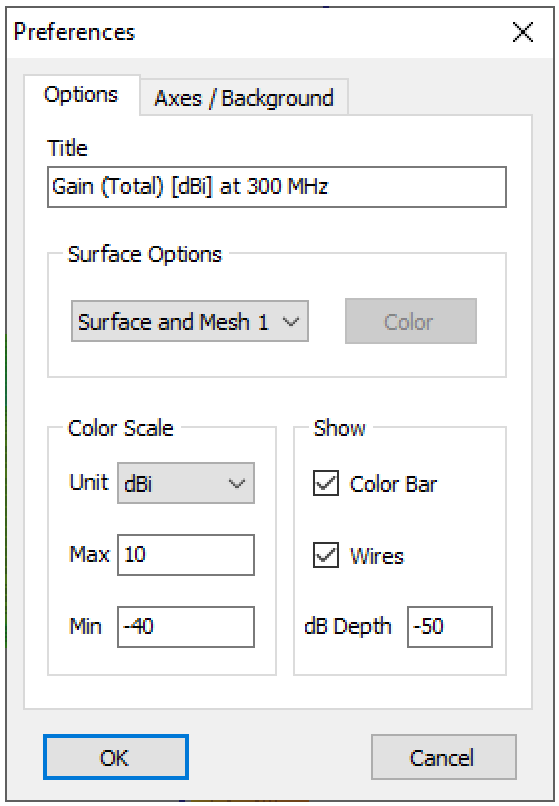


Fig. 2: Preferences dialog box of the AN-3D Pattern application.

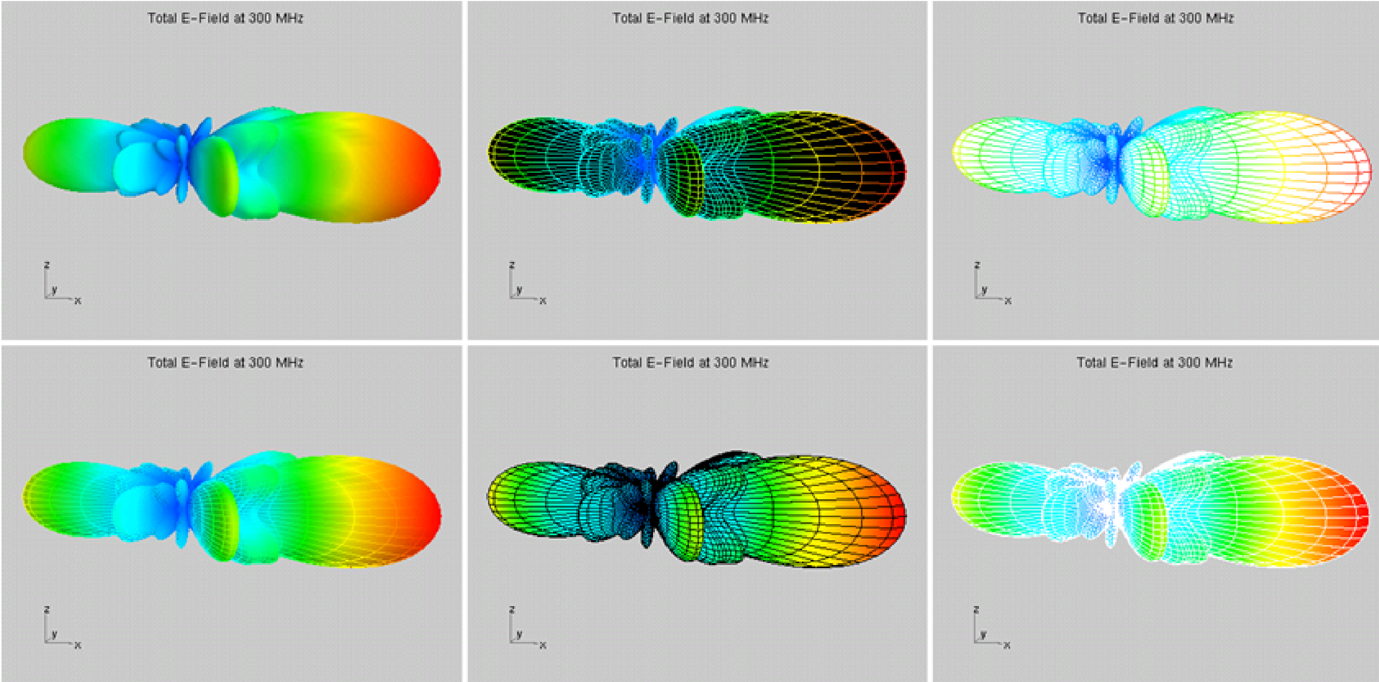


Fig. 3: Different options available for plotting radiation lobes.

Plotting the Far Field Spectrum

Far-field frequency spectra are generated when a simulation is run using either **a specified list of frequencies** or **a frequency sweep**. At each frequency, the far field is calculated over a range of directions defined by the **zenith (Theta)** and **azimuth (Phi)** angles, as well as the **observation distance** set in the **Far-Field panel** of the **Setup tab**. To plot the far field as a function of frequency, you must first choose a fixed direction—i.e., specific **Theta** and **Phi** angles.

To generate the far-field spectrum plot, go to **Results > Plot Far-Field Spectrum** in the main menu. This will open the *Select Far-Field Point* dialog box (see **Fig. 1**), where you can specify the desired Theta and Phi angles. After clicking **OK**, the **AN-XY Chart** application will display the far-field spectrum versus frequency (see **Fig. 2**).

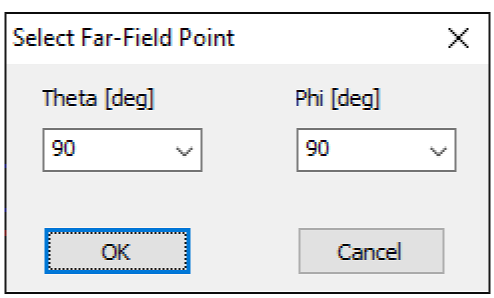


Fig. 1: “Select Far-Field Point” dialog box for choosing a fixed observation direction (Theta, Phi).

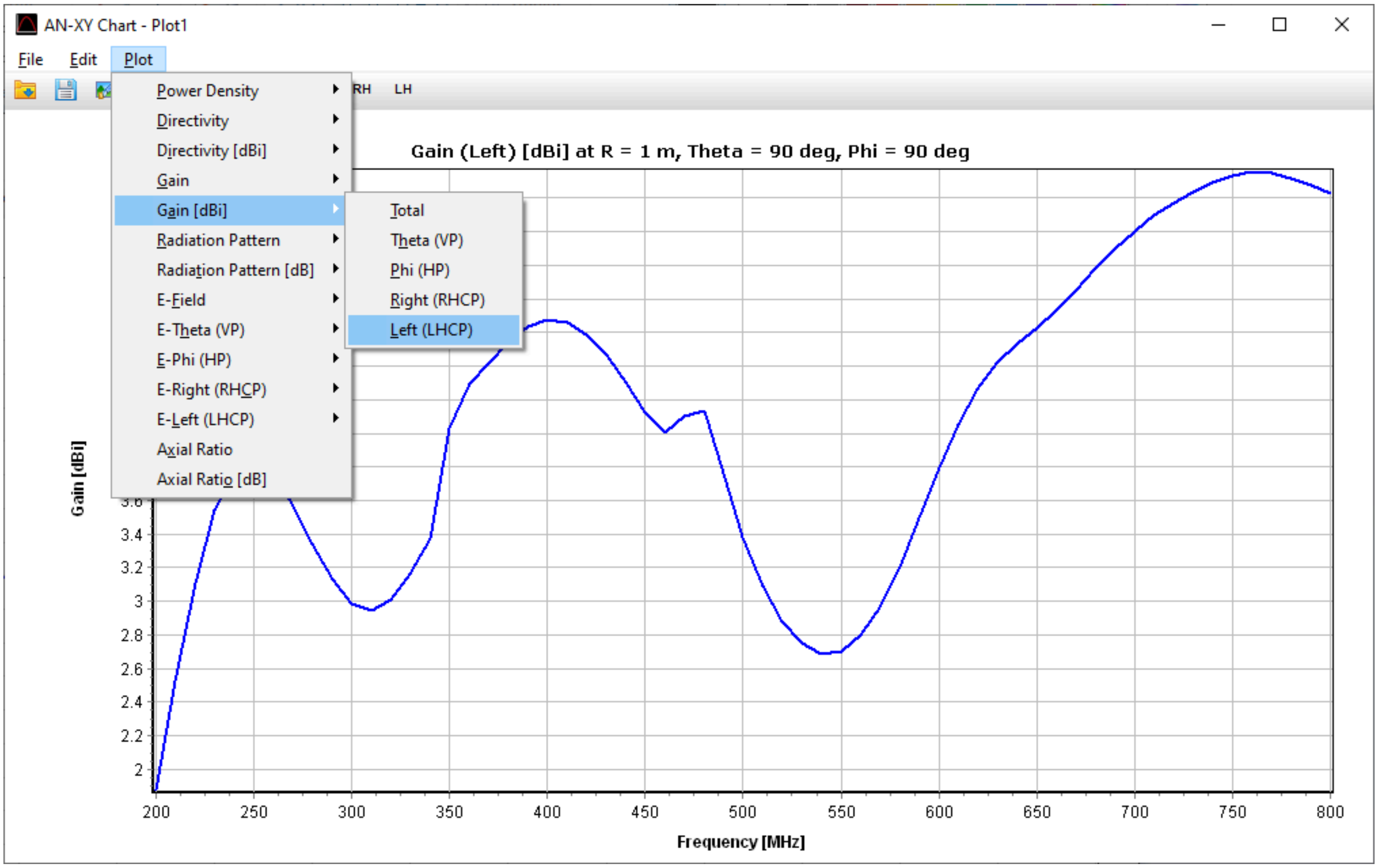


Fig. 2: Far-field frequency spectrum plotted in AN-XY Chart, showing the LHCP component of the Gain in dBi.

In the **Plot** menu of the *AN-XY Chart* application, various far-field metrics are available, including **Power Density, Directivity, Gain, Radiation Pattern, E-Field, and Axial Ratio**. Most of these metrics can be displayed in decibels and support polarization decomposition into **VP** (Vertically Polarized), **HP** (Horizontally Polarized), **RHCP** (Right-Hand Circularly Polarized), and **LHCP** (Left-Hand Circularly Polarized) components. The toolbar offers quick access to toggle between these polarization components (see **Fig. 3**).

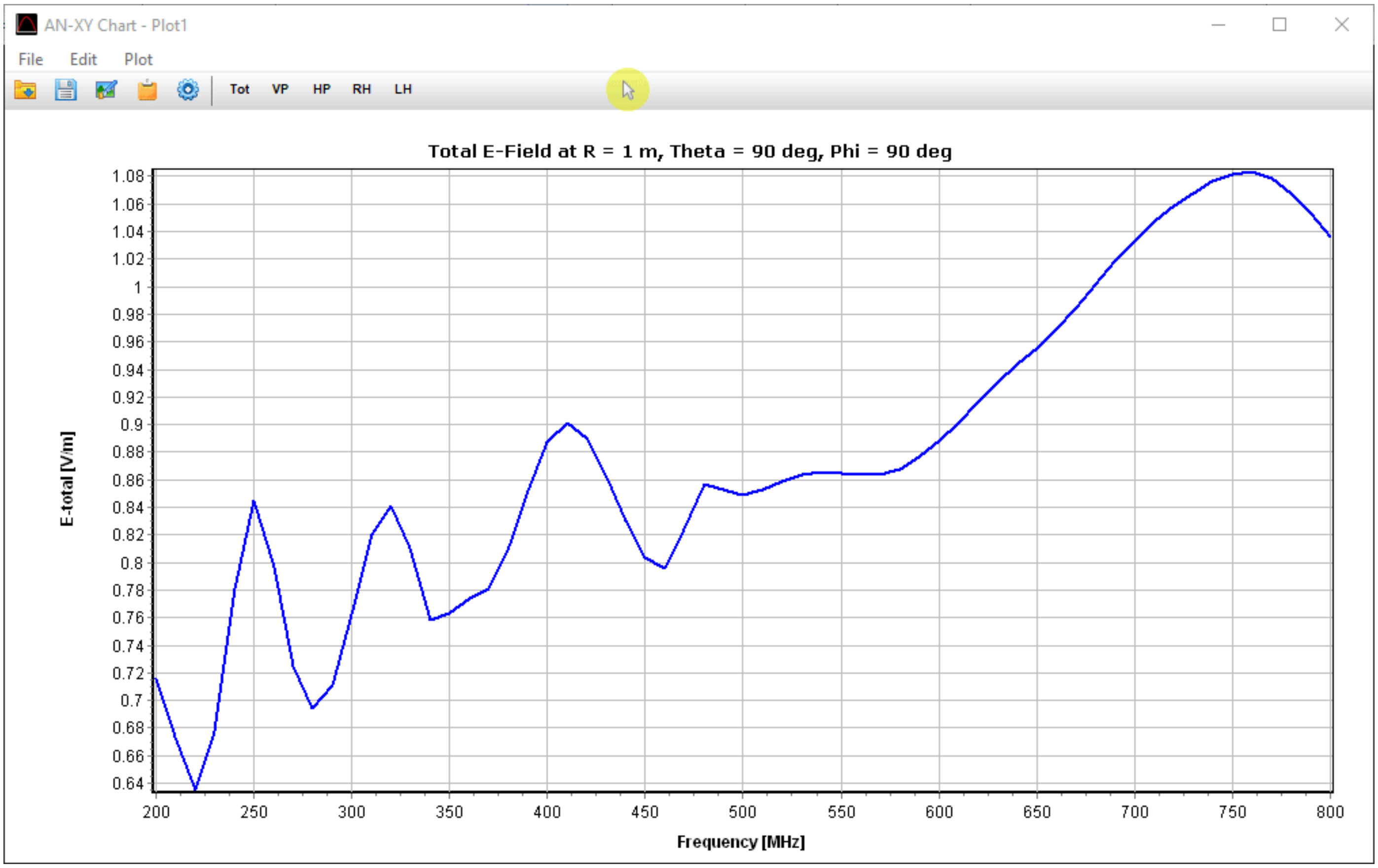
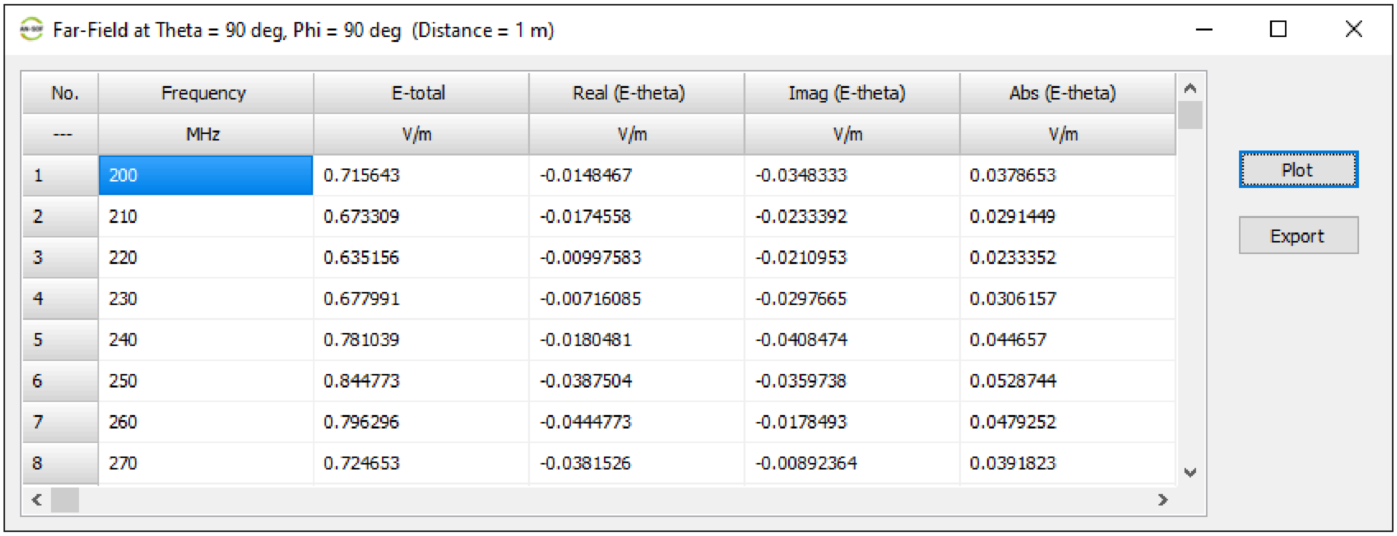


Fig. 3: Selecting VP, HP, RHCP, and LHCP polarization components using the toolbar buttons in the AN-XY Chart application.

You can also plot the linearly polarized electric field components (**E-Theta** and **E-Phi**) and the circularly polarized components (**E-Right** and **E-Left**) versus frequency. Each component can be displayed in terms of **amplitude**, **phase**, or **real and imaginary parts**. Additionally, the **Axial Ratio**—defined as the ratio of the minor to major axes of the polarization ellipse—can be plotted as a function of frequency, in both linear scale and decibels.

The far-field spectrum at a selected observation point can also be displayed in **tabular form**. To do this, go to **Results > List Far-Field Spectrum** in the AN-SOF main menu. This will open the **Select Far-Field Point** dialog box, where you can specify fixed values for **Theta** and **Phi**. Once selected, a table listing the far-field components versus frequency will be displayed. You can then click the **Plot** button to visualize the data (see **Fig. 4**). This set of tools offers valuable capabilities for analyzing far-field behavior across frequencies.



No.	Frequency	E-total	Real (E-theta)	Imag (E-theta)	Abs (E-theta)
---	MHz	V/m	V/m	V/m	V/m
1	200	0.715643	-0.0148467	-0.0348333	0.0378653
2	210	0.673309	-0.0174558	-0.0233392	0.0291449
3	220	0.635156	-0.00997583	-0.0210953	0.0233352
4	230	0.677991	-0.00716085	-0.0297665	0.0306157
5	240	0.781039	-0.0180481	-0.0408474	0.044657
6	250	0.844773	-0.0387504	-0.0359738	0.0528744
7	260	0.796296	-0.0444773	-0.0178493	0.0479252
8	270	0.724653	-0.0381526	-0.00892364	0.0391823

Fig. 4: Table listing far-field components as a function of frequency.

Power Budget

To access the **Power Budget** dialog box (see Fig. 1), navigate to **Results > Power Budget/RCS** in the main menu. When **discrete sources** are used for excitation, the following list of parameters versus frequency is displayed:

Parameters

Input Power: Total input power provided by the discrete sources in the structure.

Radiated Power: Total radiated power from the structure.

Structure Loss: Total consumed power, representing ohmic losses in the structure.

Efficiency: Radiated power-to-input power ratio, representing the **radiation efficiency**. For a lossless structure, this value is **100%**.

Directivity: Peak directivity, displayed both as a dimensionless value and in decibels (**dBi**) relative to an isotropic source.

Gain: Peak gain, displayed both as a dimensionless value and in decibels (**dBi**) relative to an isotropic source.

Av. EIRP (Effective Isotropic Radiated Power): Time-averaged EIRP in **Watts** and **dBW**. This value accounts for the duty cycle of the selected transmit mode in the **Tuner tab** and the **Time Transmitting** percentage.

Peak EIRP (Effective Isotropic Radiated Power): Peak EIRP in **Watts** and **dBW**, calculated directly from the **Peak Envelope Power (PEP)** without considering the duty cycle or time transmitting percentage.

Av. Power Density: Average power density, calculated by averaging the power density over all directions in space.

Peak Power Density: Maximum value of the radiated power density.

Theta (max) and **Phi (max)**: The zenith and azimuth angles, respectively, in the direction of maximum radiation, corresponding to the **peak power density**.

F/R H and **F/B H**: Front-to-rear and front-to-back ratios, respectively, in a horizontal slice of the radiation pattern given by **Theta = Theta (max)**.

F/R V and **F/B V**: Front-to-rear and front-to-back ratios, respectively, in a vertical slice of the radiation pattern given by **Phi = Phi (max)**.

Error and Average Gain Test (AGT)

Error: Represents the error in the power balance of the system. A necessary (but not sufficient) condition for a valid model is that the input power must equal the sum of the radiated and lost powers. The error is calculated as:

$$Error=100\times\frac{InputPower-LostPower-RadiatedPower}{InputPower-LostPower}$$

Average Gain Test (AGT): A validation metric that should be close to **1** for a valid model. It is calculated as:

$$AGT=\frac{RadiatedPower+LostPower}{InputPower}$$

Plotting and Exporting Data

Select an item from the list in the upper-right corner of the window and click the **Plot** button to plot the selected item versus frequency.

Click the **Export** button to export the list to a **CSV** file.

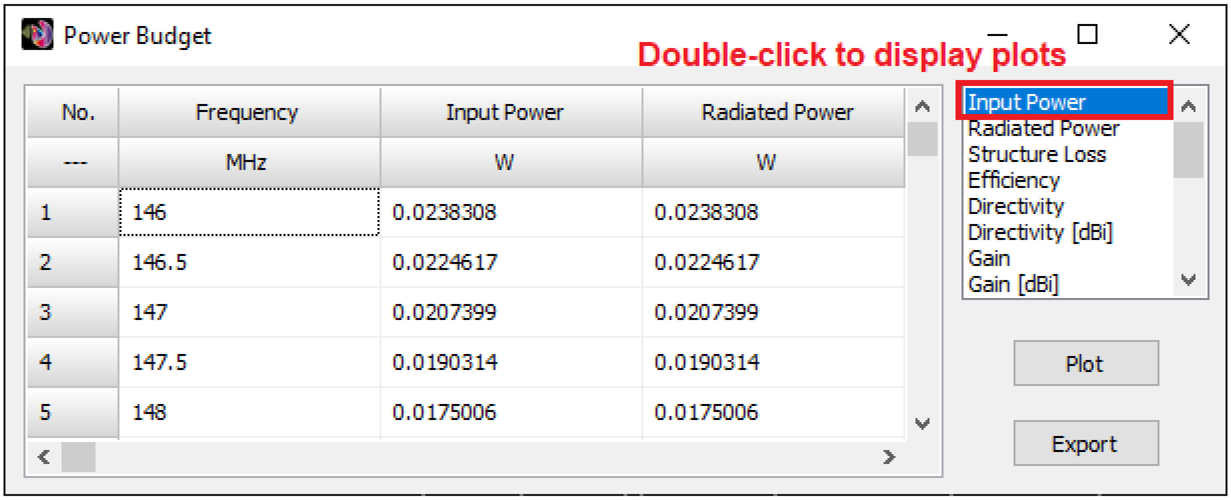


Fig. 1: The Power Budget dialog box.

> **Notes**
>
> A power budget **error** of about **±10%** is permissible from the engineering point of view.
>
> When a **real ground plane** is used, the Error column shows the percentage of power lost in the ground due to its finite conductivity.
>
> When a **substrate slab** is used, this column shows the percentage of power transferred to the dielectric material in the substrate (surface wave loss).
>
> **AGT = 1** means that the power balance is exact. An AGT between **0.99** and **1.01** is comparable to achieving an error of ±1%.

Radar Cross Section (RCS)

AN-SOF calculates the **bistatic RCS** based on the fixed plane wave incident direction defined in the **Excitation** panel of the **Setup** tab.

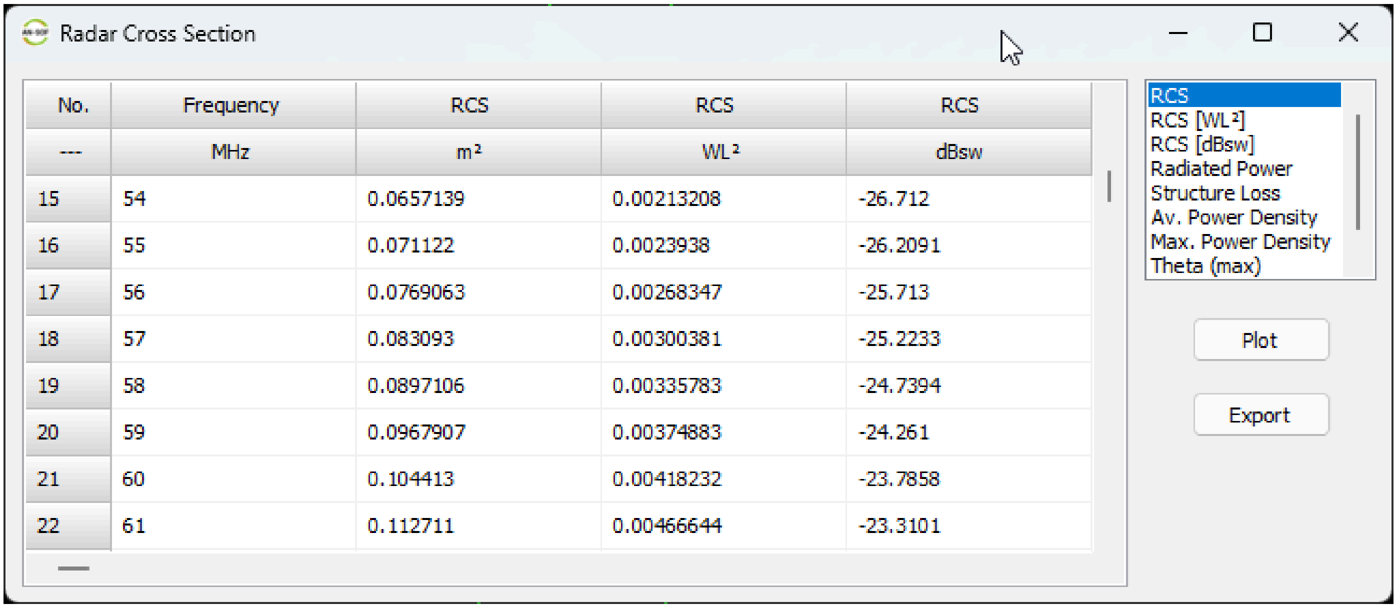
To calculate **monostatic RCS** (where the incident direction varies to match the observation point), you must use the **AN-SOF Engine** console application.

Accessing RCS Data

To open the **Power Budget/RCS** dialog box, navigate to **Results > Power Budget/RCS** in the main menu (**Fig. 1**). When an **incident field** is used for excitation, the following frequency-dependent parameters are available:

- **RCS (m^2):** The Radar Cross Section expressed in square meters.
- **RCS (WL^2):** The Radar Cross Section expressed in square wavelengths.
- **RCS (dBsw):** The Radar Cross Section in decibels relative to a square wavelength.
- **Radiated Power:** The total scattered power originating from the structure.
- **Structure Loss:** The total power consumed by ohmic losses within the structure itself.

- **Av. Power Density:** The scattered power density averaged over all directions in space.
- **Max. Power Density:** The maximum value of the scattered power density.
- **Theta (max) and Phi (max):** The zenith and azimuth angles identifying the direction of maximum scattered radiation.



No.	Frequency	RCS	RCS	RCS
---	MHz	m²	WL²	dBsw
15	54	0.0657139	0.00213208	-26.712
16	55	0.071122	0.0023938	-26.2091
17	56	0.0769063	0.00268347	-25.713
18	57	0.083093	0.00300381	-25.2233
19	58	0.0897106	0.00335783	-24.7394
20	59	0.0967907	0.00374883	-24.261
21	60	0.104413	0.00418232	-23.7858
22	61	0.112711	0.00466644	-23.3101

***Fig. 1:** The Radar Cross Section dialog box.*

Far-Field Physics

These metrics are derived from the **scattered field pattern** in the far-field region. In this zone:

1. The scattered field amplitude decays at a rate of $1/r$.
2. The scattered power density decays at a rate of $1/r$2.

Plotting Results

To visualize the frequency response of a specific parameter:

1. Select the desired item from the list in the **upper-right corner** of the dialog box.
2. Press the **Plot** button to display the data in **AN-XY Chart**.

Exporting the Far Field

To perform post-processing in external software or for detailed documentation, AN-SOF allows you to tabulate and export both spatial patterns and frequency spectra.

Available Export Commands

Navigate to the **Results** menu to access the following options:

- **List Far-Field Pattern:** Displays the spatial distribution of the radiation (gain, directivity, or field strength) at a specific frequency for a range of angles (Theta, Phi).

- **List Far-Field Spectrum:** Displays the far-field results across the entire simulated frequency range for a fixed observation direction (Theta, Phi).

The Results Table

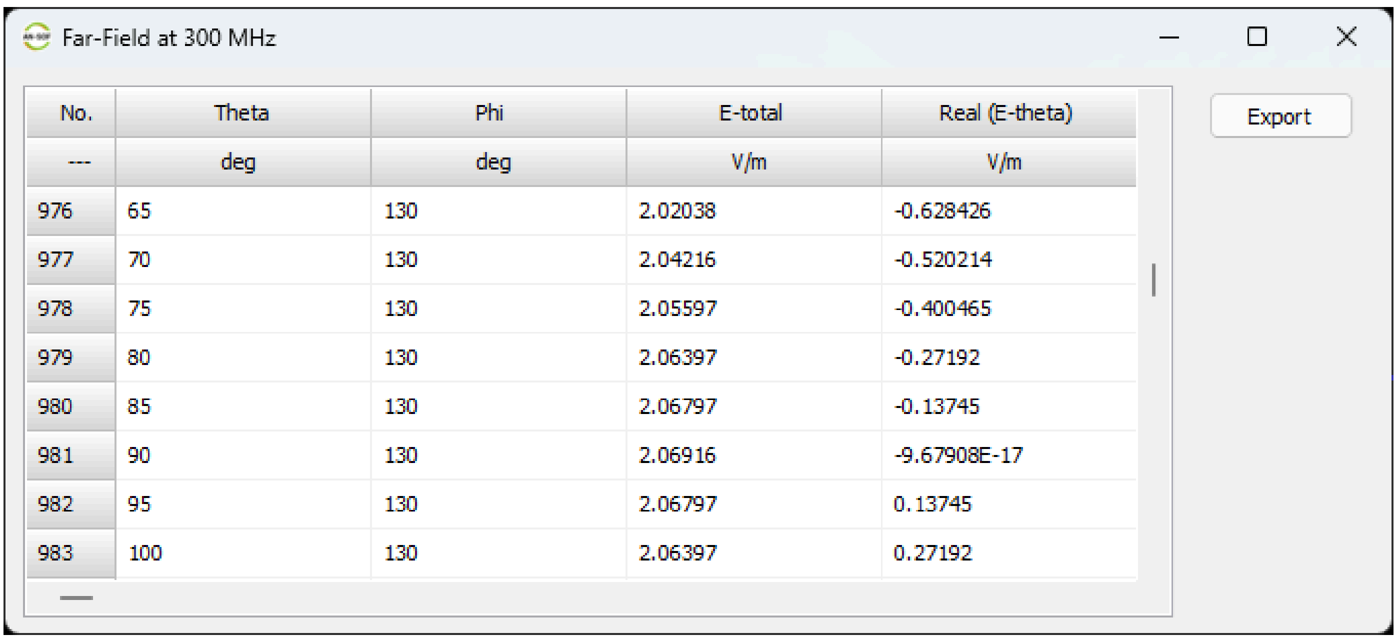
Once you execute a command, a results window will appear (**Fig. 1**). This table includes:

- **Angles:** Theta and Phi coordinates.
- **Electric Field Components:** E-total, E-theta, E-phi, E-right, E-left (real and imaginary parts as well as magnitudes and phases).
- **Axial Ratio:** The axial ratio of the field polarization ellipse (dimensionless and in decibels).

Saving Data

To save this information for use in Excel, MATLAB, or Python:

1. Click the **Export** button of the results window (**Fig. 1**).
2. Choose a destination and save the file in **CSV (Comma Separated Values)** format.



No.	Theta	Phi	E-total	Real (E-theta)
---	deg	deg	V/m	V/m
976	65	130	2.02038	-0.628426
977	70	130	2.04216	-0.520214
978	75	130	2.05597	-0.400465
979	80	130	2.06397	-0.27192
980	85	130	2.06797	-0.13745
981	90	130	2.06916	-9.67908E-17
982	95	130	2.06797	0.13745
983	100	130	2.06397	0.27192

Fig. 1: *Tabulated values of the far-field pattern at a selected frequency. Click on the Export button to export the list to a CSV file.*

Near Field

Plotting Near Field Patterns

In AN-SOF, near-field analysis allows you to evaluate the electric ($\mathbf{E}$) and magnetic ($\mathbf{H}$) fields in the immediate vicinity of an antenna or at any specified distance. This is essential for studying antenna-to-antenna coupling or electromagnetic compatibility (EMC).

Defining the Calculation Grid

Before plotting, you must define the observation points in the **Near-Field panel** of the **Setup tab**. You can specify the grid using:

- **Cartesian Coordinates** (X, Y, Z)
- **Cylindrical Coordinates** (R, Phi, Z)
- **Spherical Coordinates** (R, Theta, Phi)

Verification in the Far-Field

While these are "near-field" tools, they can be used to verify far-field behavior. As you move further from the antenna, you should observe:

1. $\mathbf{E}$ and $\mathbf{H}$ becoming perpendicular to each other and the radial direction.
2. Fields oscillating in phase.
3. The intrinsic impedance of free space: $|\mathbf{E}|/|\mathbf{H}| \approx 120\pi\Omega \approx 377\Omega$.

Power Density and Compliance

If both $\mathbf{E}$ and $\mathbf{H}$ fields are calculated, AN-SOF provides the **RMS Power Density** (S), calculated as:

$$S = |\mathbf{E} \times \mathbf{H}^*|$$

This metric is critical for ensuring that your design **complies with human exposure limits** (e.g., ICNIRP or FCC standards).

3D Visualization

To visualize the fields as a 3D color-coded map, use the following paths in the main menu:

- **Electric Field:** Results > Plot Near E-Field Pattern > 3D Plot
- **Magnetic Field:** Results > Plot Near H-Field Pattern > 3D Plot
- **Power Density:** Results > Plot Power Density Pattern > 3D Plot

These commands launch **AN-3D Pattern** (**Fig. 1**). If your simulation includes multiple frequencies, you will be prompted to select a specific frequency before the plot is rendered.

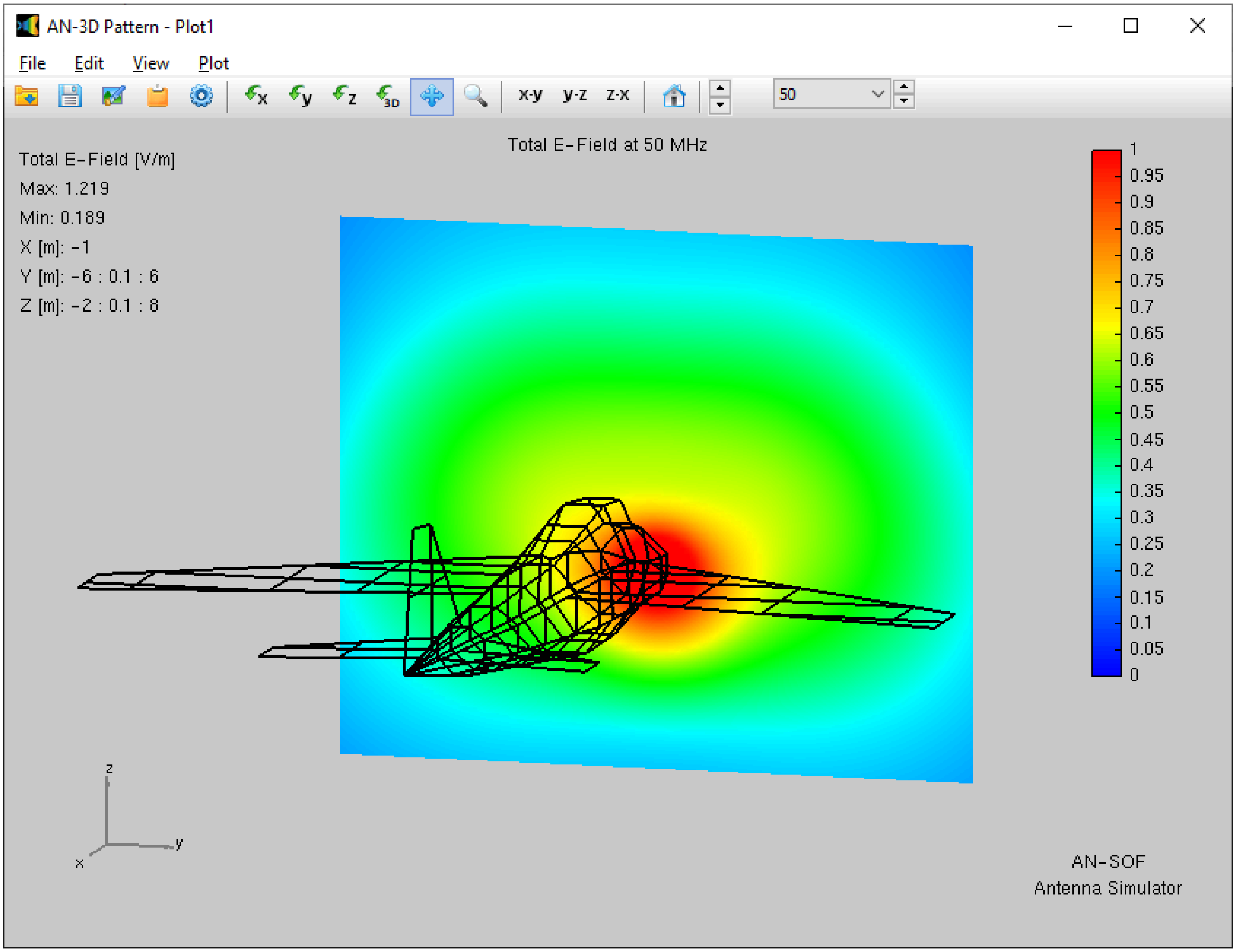


Fig. 1: *3D plot of the near E-field in the AN-3D Pattern application, showing the field distribution just in front of an aircraft receiving a vertically polarized plane wave from behind.*

2D Visualization and Individual Components

For a more detailed look at field gradients along a specific axis or path, use the **2D Plot** options under the same Results menus. This launches **AN-XY Chart**.

- **Total RMS:** The default view shows the total magnitude.
- **Individual Components:** You can isolate specific vectors (e.g., Ex, Ey, Ez) by navigating to the **Plot** menu within AN-XY Chart (**Fig. 2**).

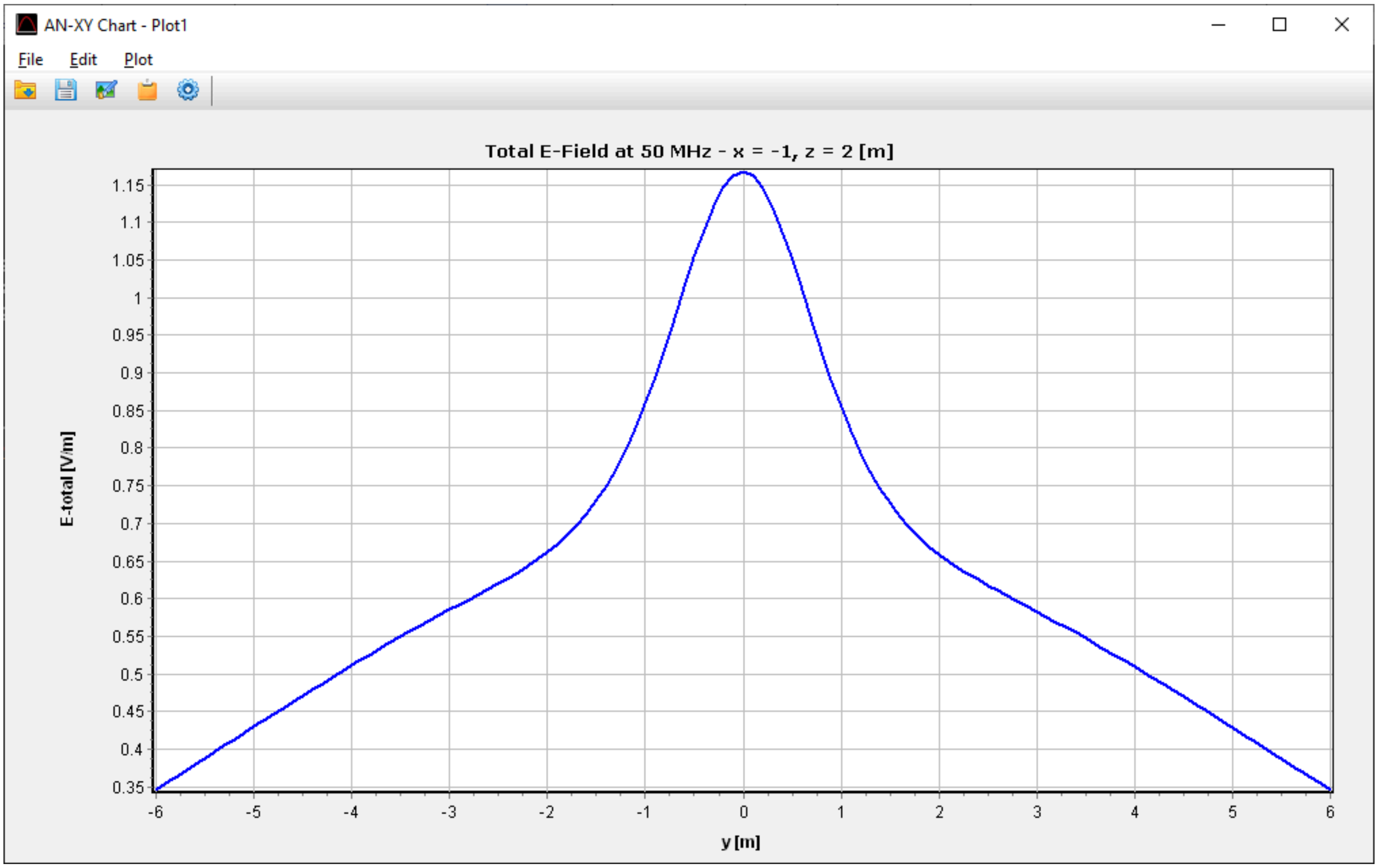


Fig. 2: *Near E-field plotted in AN-XY Chart as a function of the y-coordinate. This plot corresponds to the horizontal line passing directly in front of the aircraft's nose shown in Fig. 1.*

Tabulating and Exporting Data

To see the exact numerical values at every grid point, use:

- **Results > List Near E-Field Pattern**
- **Results > List Near H-Field Pattern**
- **Results > List Power Density Pattern**

These tables can be exported for further analysis in external mathematical tools.

Understanding Field Components by Coordinate System

The components of the electric and magnetic fields calculated by AN-SOF depend entirely on the coordinate system you select in the **Near-Field** panel of the **Setup** tab. Choosing the right system is crucial for matching the grid to the physical symmetry of your antenna.

1. Cartesian Coordinates (x,y,z)

When you select a rectangular grid, the fields are decomposed into their standard orthogonal components. This is ideal for planar structures or when evaluating fields along a straight line or over a flat surface.

- **Electric Field Components:** Ex, Ey, Ez
- **Magnetic Field Components:** Hx, Hy, Hz

2. Cylindrical Coordinates (ϱ, ϕ, z)

This system is best suited for structures with axial symmetry, such as vertical monopoles or circular arrays. The components are calculated relative to the radial distance from the z-axis and the rotation around it.

- **Electric Field Components:** $E\varrho$ (radial), $E\phi$ (azimuthal), Ez (axial)
- **Magnetic Field Components:** $H\varrho, H\phi, Hz$

3. Spherical Coordinates (r, θ, ϕ)

Spherical coordinates are typically used to analyze how fields radiate outward into space. This setup is perfect for visualizing how near-fields begin to transition into far-field patterns.

- **Electric Field Components:** Er (radial), $E\theta$ (zenith/elevation), $E\phi$ (azimuth)
- **Magnetic Field Components:** $Hr, H\theta, H\phi$

> **Tip**
>
> In the **AN-XY Chart** application, you can isolate these individual components via the **Plot** menu to see which specific orientation contributes most to the total field magnitude at a given point.

Plotting the Near Field Spectrum

While spatial plots show you how fields vary across a region at a single frequency, the **Near Field Spectrum** allows you to observe how the field intensity changes at a **single, fixed point** across a range of frequencies. This is essential for identifying resonant frequencies or ensuring field limits are not exceeded over a wide bandwidth.

Step 1: Selecting the Observation Point

Since a spectrum requires a stationary point, executing the plot command will first prompt the **Select Near-Field Point** dialog box (**Fig. 1**).

- To start, navigate to **Results > Plot Near [E-Field/H-Field/Power Density] Spectrum**.
- In the dialog box, select one of the points from the grid you previously defined in the **Near-Field panel** of the **Setup** tab.

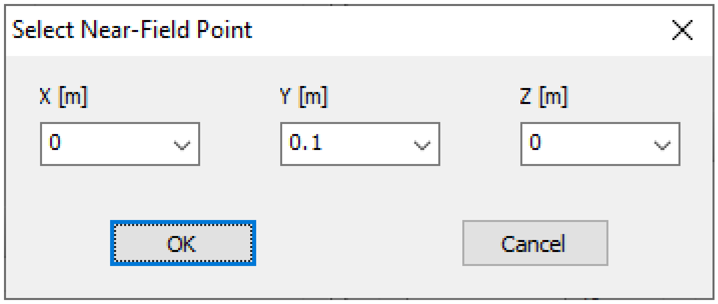


***Fig. 1:** "Select Near-Field Point" dialog box for choosing a fixed observation point from the grid.*

Step 2: Analyzing the Spectrum in AN-XY Chart

Once a point is selected, **AN-XY Chart** launches to display the magnitude of the field vs. frequency (**Fig. 2**).

To perform a deeper analysis of the vector components, you can use the **Plot** menu in the AN-XY Chart main menu to view:

- **Components:** Isolate Ex, Ey, Ez (or the corresponding components for your chosen coordinate system).
- **Data Types:** Switch between **Amplitude**, **Phase**, **Real**, and **Imaginary** parts.

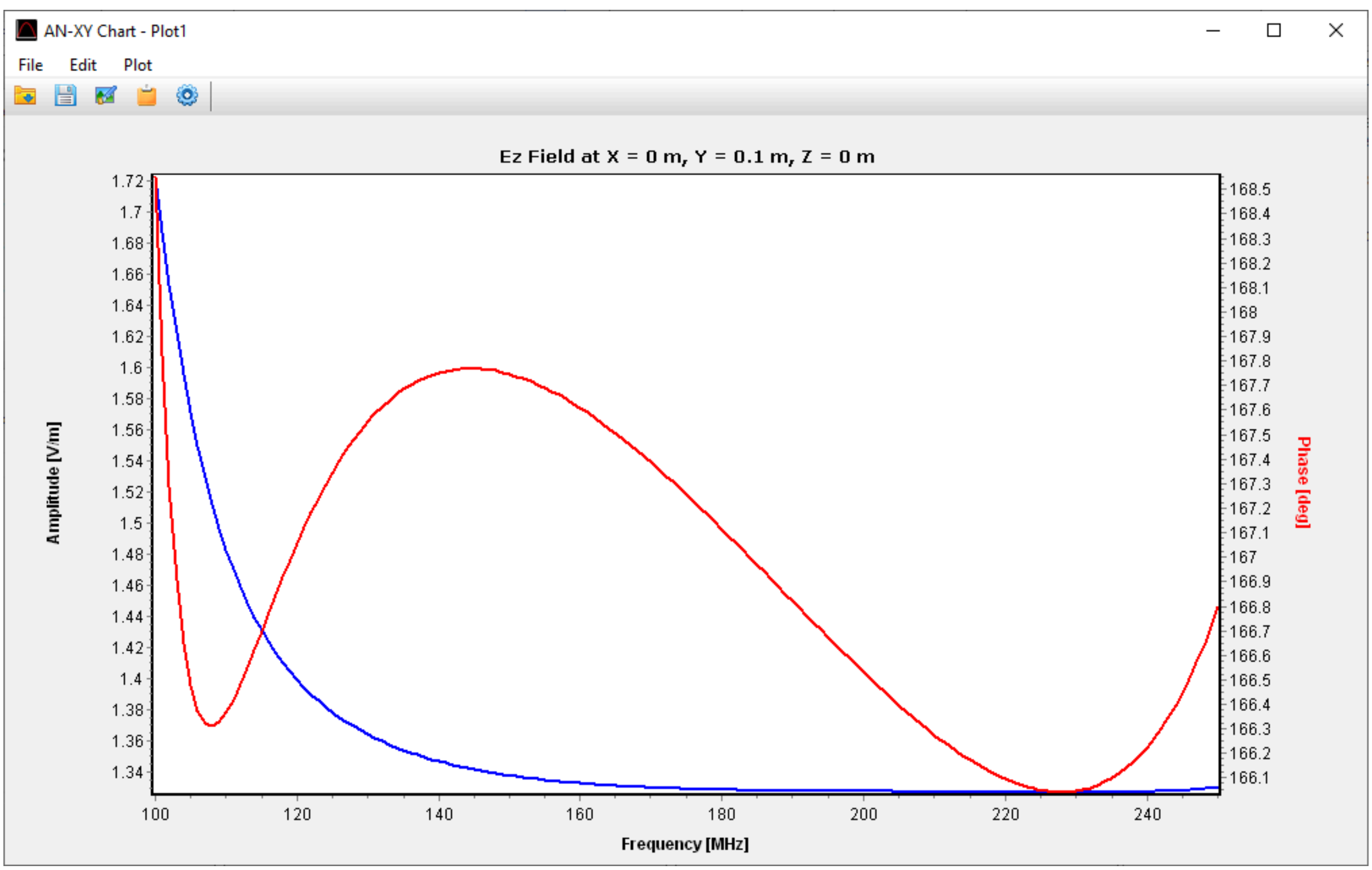


***Fig. 2:** Near E-field spectrum in amplitude and phase plotted in AN-XY Chart.*

Practical Application

This feature is particularly useful when checking for **Electromagnetic Interference (EMI)**. For instance, if you have sensitive electronics located at a specific distance from an antenna, you can plot the Power Density Spectrum at that exact coordinate to ensure it stays below a safe threshold across the entire operating frequency range.
Exporting the Near Field

For users who need to perform external numerical analysis or generate reports, AN-SOF provides a straightforward way to move near-field data into tools like Excel, MATLAB, or Python.

Available Export Commands

Access these options via the **Results** menu. Depending on whether you need spatial data (at a specific frequency) or frequency data (at a specific point), choose from the following:

- **List Near E-Field/H-Field/Power Density Pattern:** Displays a table of the electric field ($\mathbf{E}$), magnetic field ($\mathbf{H}$), or power density ($S = |\mathbf{E} \times \mathbf{H}^*|$) across your entire grid for a single frequency.
- **List Near E-Field/H-Field/Power Density Spectrum:** Displays the field behavior across all simulated frequencies for a single, fixed coordinate point.

The Results Table

Executing these commands opens a data window containing the raw numerical results. The table format adapts to your chosen coordinate system:

Column Type	Description
Coordinates	The position (x, y, z), (ϱ, ϕ, z), or (r, θ, ϕ).
Total Field	The total RMS magnitude of the field at that location/frequency.
Field Components	Magnitude and Phase for each vector component (e.g., Ex, Ey, Ez).

Saving to CSV

To export this data:

1. Click the **Expor**t button on the right side of the table.
2. Specify your file name and save it as a **.csv** file.

This format is universally compatible and allows you to recreate high-resolution heat maps or perform safety compliance calculations in third-party software.

Transmission Lines

Adding Transmission Lines

Adding a transmission line to a model has an impact on the entire calculation, affecting current distribution, input impedance, and near and far fields. AN-SOF allows for the addition of **lossy or lossless transmission lines** and has a list of preloaded lines with parameters adjusted to the attenuation curves published in the data sheets of real cables. This list of cables includes both **two-wire and coaxial transmission lines.**

After drawing and segmenting the wire structure that will represent an antenna or an object that will scatter electromagnetic waves, the recommended first step is to create a list of the transmission lines that will be connected to the structure. This is described below.

The ends of a transmission line in AN-SOF are called **Port 1** and **Port 2** since a line can be considered as a **two-port network**. Each end or port of a transmission line can be connected to a segment of the wire structure, as Fig. 1 shows. A transmission line is defined by its characteristic impedance, **Z0**, velocity factor, **VF**, a **loss model or attenuation curve**, and shunt admittances, **Y1** and **Y2**, connected across each port. Each transmission line must be connected between two different wire segments (the i-th and j-th segments in Fig. 1 should not be the same segment). In the calculation engine model, a gap is opened in the center of each segment to allow a transmission line to be connected there.

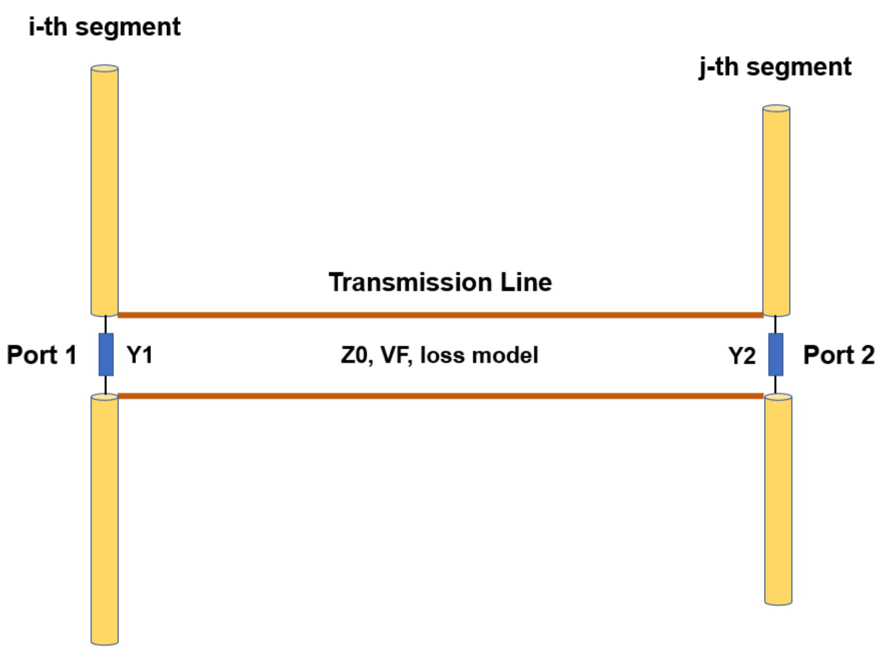


Fig. 1: A transmission line connected between two wire segments. It is defined by its characteristic impedance, Z0, velocity factor, VF, a loss model, and shunt admittances Y1 and Y2.

Transmission lines are modeled in an **implicit way**, meaning that the lines don't scatter electromagnetic waves in space, but rather interact with the wire structure by establishing boundary conditions on the voltages and currents at the connected segments. Implicit modeling is adequate when the disturbance in the electromagnetic field caused by the physical presence of the transmission line can be neglected, e.g., for twisted-pair lines in most cases. On the other hand, **explicit modeling** involves drawing the two parallel wires of a two-wire line in the workspace and dividing them into segments, like the rest of the structure. For coaxial lines, a "hybrid" modeling approach can be used, which is explained in **Modeling Coaxial Cables**.

To add transmission lines, go to the AN-SOF main menu > Draw > **Transmission Lines (Ctrl + L)**. A table will be displayed where a transmission line can be entered on each row. Follow the procedure below to enter the lines:

1. Select a row by clicking on the row number of your choice in the first column labeled 'No.', Fig. 2.

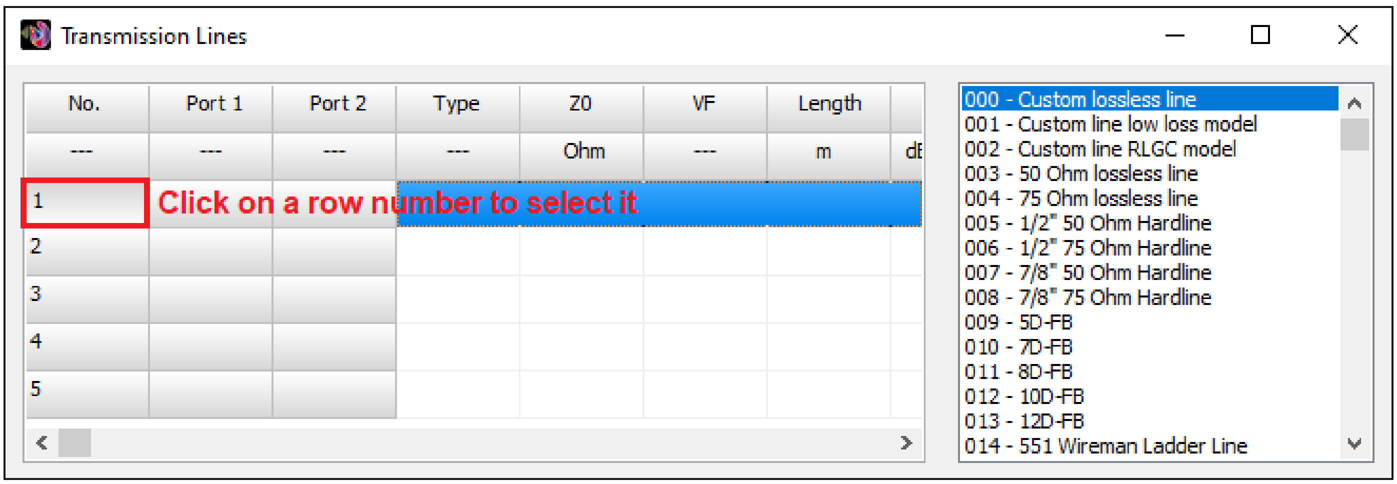


Fig. 2: Table for entering transmission lines. Rows are numbered. Click on a row number to select the entire row.

2. On the right-hand panel, choose a type of transmission line and double-click on your chosen type. The selected row will be automatically completed, Fig. 3.

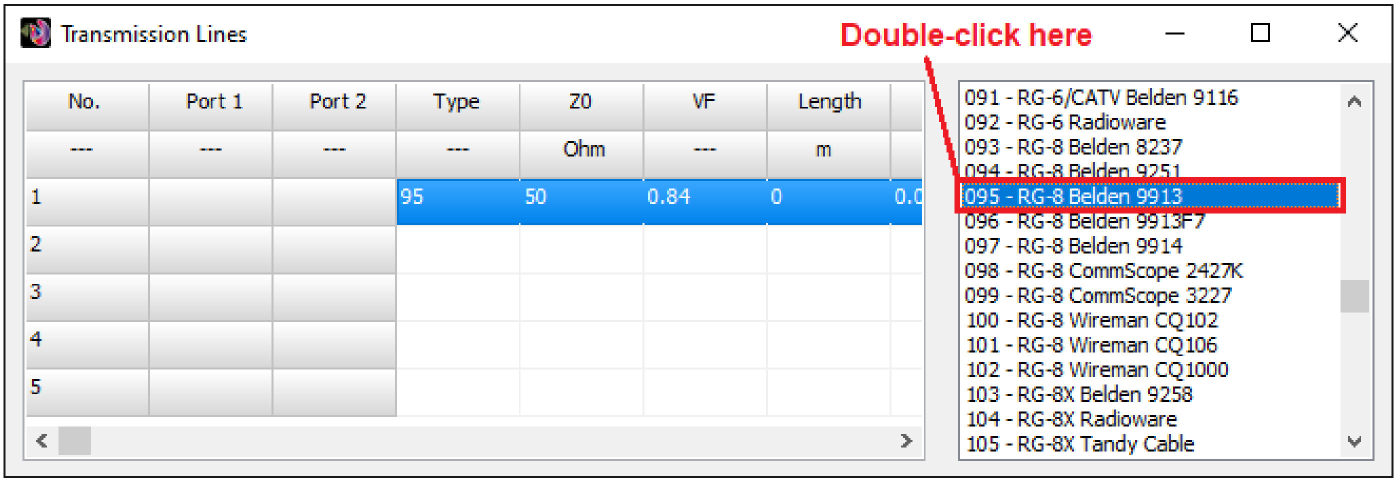


Fig. 3: On the right panel, double-click on the chosen line type to automatically complete the selected row.

3. From type 3 onwards, the parameters correspond to real cable datasheets. If you wish to enter your own parameters, choose types 0, 1, or 2. To edit the value in a cell, double-click on the cell.

Note that in this procedure, the ports of the transmission lines have not been connected to the wire segments yet. This is explained in [Connecting Transmission Lines].

The parameters that define a transmission line are:

1) **Type:** On the right-hand panel of the Transmission Lines window, there is a list of lines with the cable part number and the manufacturer in some cases. The first three types are used to input user-customized lines. The line type simply refers to its position in this list.

2) **Z0:** Nominal characteristic impedance, in Ohms. If a negative value is entered, the transmission line will be "crossed" with a 180° phase reversal with respect to the reference directions of the segments (the characteristic impedance of the line will of course be |Z0|).

3) **VF:** Velocity factor (dimensionless). The allowed range is 0 < VF <= 1.

4) **Length:** Length of the line, in the unit selected in the Preferences window (see Section "3.3 Preferences"). If a length of zero is entered, the length of the transmission line will be equal to the linear distance between the two wire segments connected at the ends of the line.

5) The **K0**, **K1**, **K2**, and **K3** columns define the line losses for the so-called **RLGC model**. These four columns will change to **Att. 1**, **Freq. 1**, **Att. 2**, **Freq. 2** when the chosen line model is that of **low losses**. These cells allow entering the attenuation curve of a real transmission line from its datasheet.

6) **Real(Y1)** and **Imag(Y1)** are the real and imaginary parts of the shunt admittance through **Port 1** of the transmission line, in Siemens [S].

7) **Real(Y2)** and **Imag(Y2)** are the real and imaginary parts of the shunt admittance through **Port 2** of the transmission line, in Siemens [S].

A transmission line without shunt admittances (Y1 = Y2 = 0) will always be symmetrical in the sense that if it is connected in reverse, i.e., by swapping ports 1 and 2, the same results will be obtained in a simulation. Ports 1 and 2 are identified so that the locations of the shunt admittances can be distinguished when they are not zero.

If you enlarge or maximize the Transmission Lines window, you will be able to see the columns corresponding to the loss model parameters and shunt admittances, Figs. 4 and 5. Initially, this window only displays cells up to the 'Length' column so that the user does not have to worry about the loss parameter values since these are automatically loaded when selecting a line type from the list. Adding an attenuation curve when modeling a cable that is not on the list is explained in **[Adding a Custom Lossy Line](#)**.

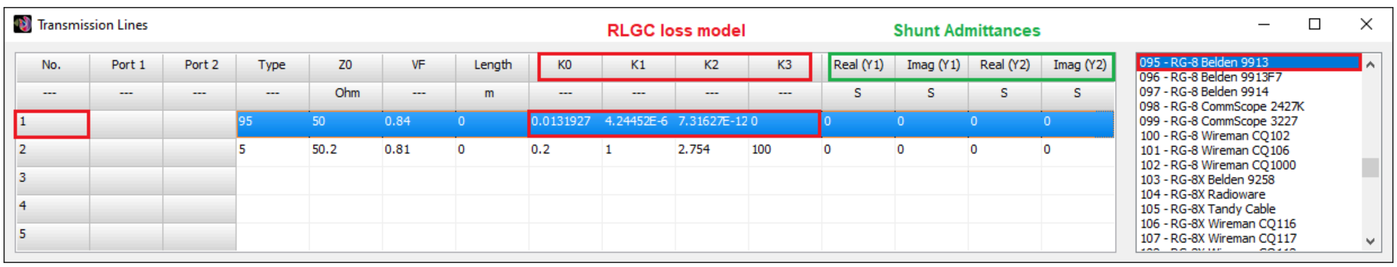


Fig. 4: Enlarge the Transmission Lines window to view the loss model parameters and shunt admittances. In this example, the K0, K1, K2, and K3 columns of the RLGC model are displayed since a line has been chosen whose attenuation curve is adjusted to this model.

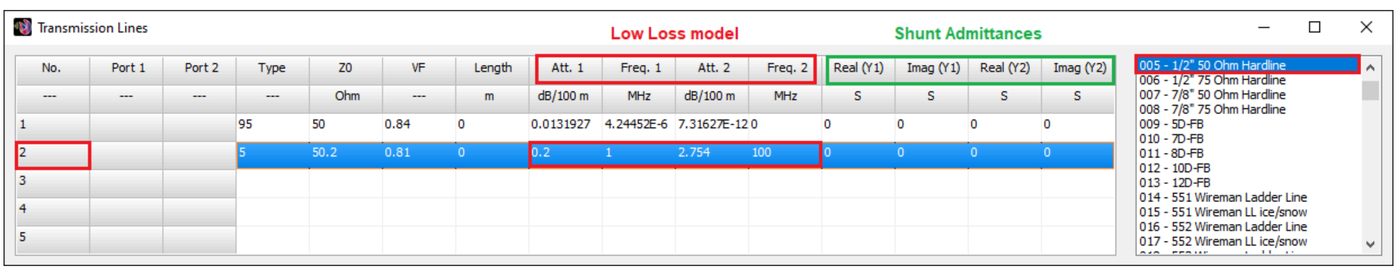


Fig. 5: Enlarge the Transmission Lines window to view the loss model parameters and shunt admittances. In this example, the Att. 1, Freq. 1, Att. 2, and Freq. 2 columns of the low loss model are displayed since a line has been chosen whose attenuation curve is adjusted to this model.

Custom Transmission Lines

If you want to add "custom" transmission lines with your own parameters, you have types 0, 1, and 2 available, Fig. 1, which are explained below.

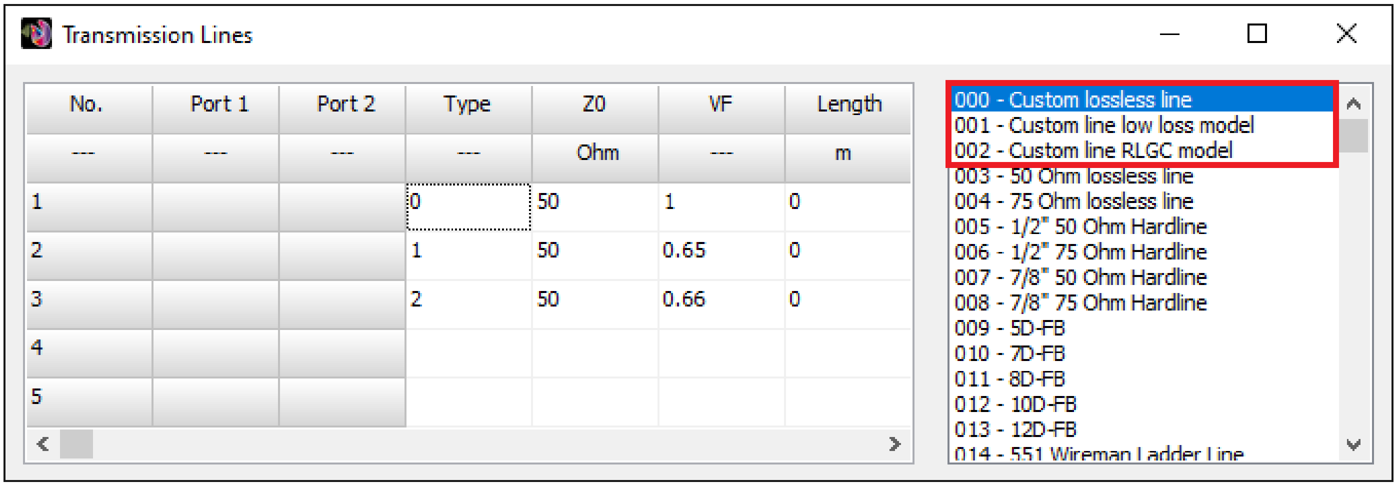


Fig. 1: The first three types of transmission lines, types 0, 1, and 2, are customizable lines.

Type 0: Custom Lossless Line

This is an ideal transmission line with zero losses, so only the nominal Z0 and velocity factor must be specified.

Type 1: Custom line – low loss model

This is a transmission line where the nominal Z0, velocity factor, and matched loss curve can be specified. To define the matched loss curve, two attenuation values must be entered at two different frequencies, with the second frequency being greater than the first. AN-SOF will then adjust a low-loss model to obtain an attenuation vs. frequency curve for subsequent calculations. **This is the simplest way to enter parameters from the datasheet of a manufactured real transmission line.** Refer to **Adding a Custom Lossy Line** where it explains how to add the parameters from an attenuation curve published in a datasheet of a real cable.

Type 2: Custom line – RLGC model

This is a transmission line model that considers losses by adjusting a matched loss curve to the table of attenuation vs. frequency in the cable datasheet. The K0, K1, and K2 constants must be entered in this case. The definition of K0, K1, and K2 assumes that the frequency is in Hz and the lengths are in meters (SI metric units). This option allows for the entry of K0, K1, and K2 obtained from third-party transmission line calculators (K3 is an additional constant that is zero for all available cables).

Connecting Transmission Lines

Any transmission lines added through the **Transmission Lines** command **(Ctrl + L)** under the Draw menu will remain in the table until the user decides to remove or modify them. During calculations, only transmission lines with both ports connected to respective wire segments will be considered for simulation. **Any lines with a single port connected or both ports disconnected will be omitted in the calculations.**

To connect a transmission line between two wire segments, follow these steps:

1. Right-click on the first wire to select it and choose the **Source / Load / TL (Ctrl + Ins)** command from the pop-up menu. This will open a horizontal toolbar with a slider control, Fig. 1.

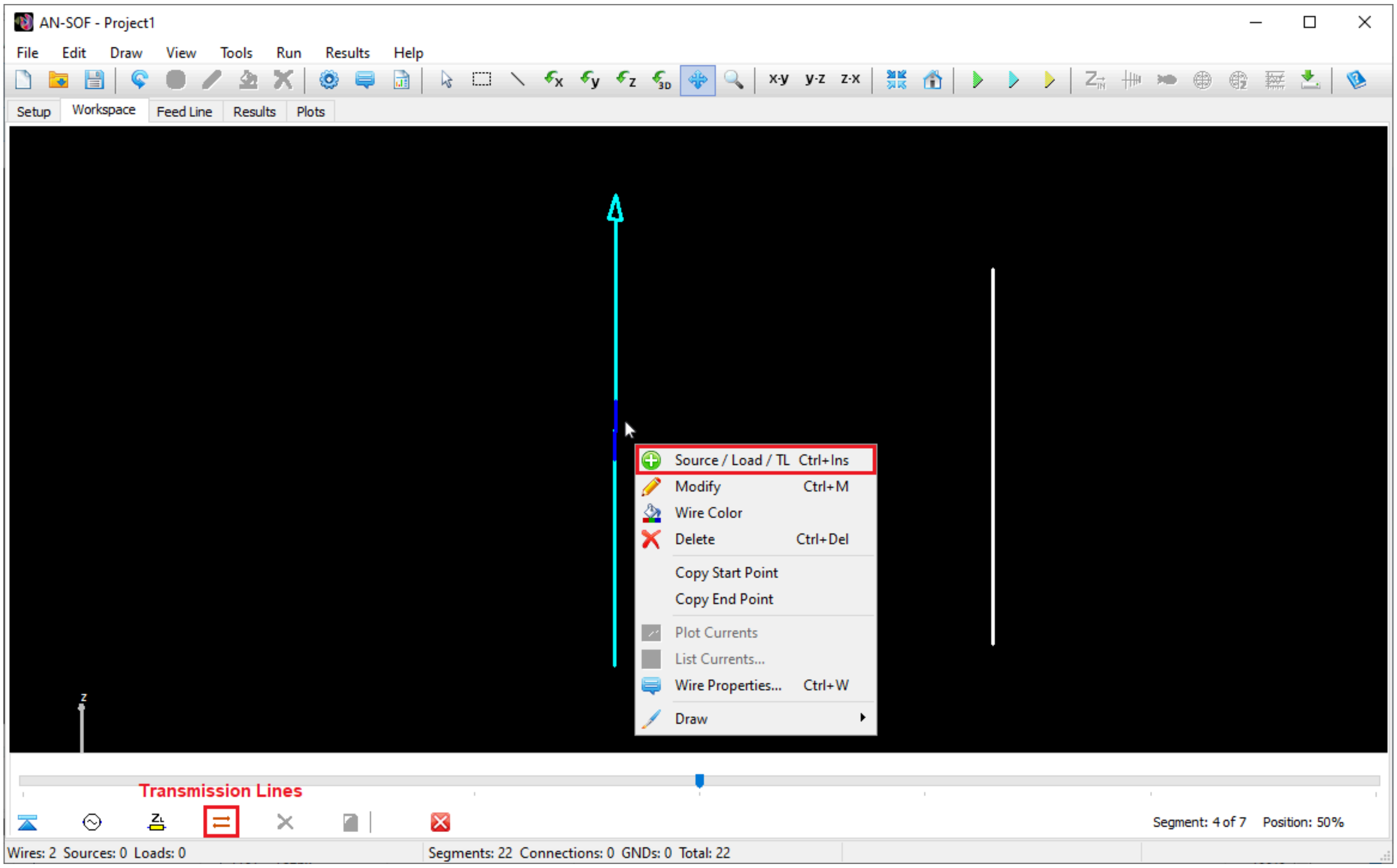


Fig. 1: To display the horizontal toolbar at the bottom of the workspace window, right-click on a wire and choose the "Source / Load / TL" command from the pop-up menu that appears. Then, move the slider to select a segment.

2. Use the slider to select the specific segment of the first wire to which you want to connect a port of the transmission line.

3. Once you've chosen the segment, click on the **Transmission Lines** button on the horizontal toolbar to open the Transmission Lines table, Fig. 2.

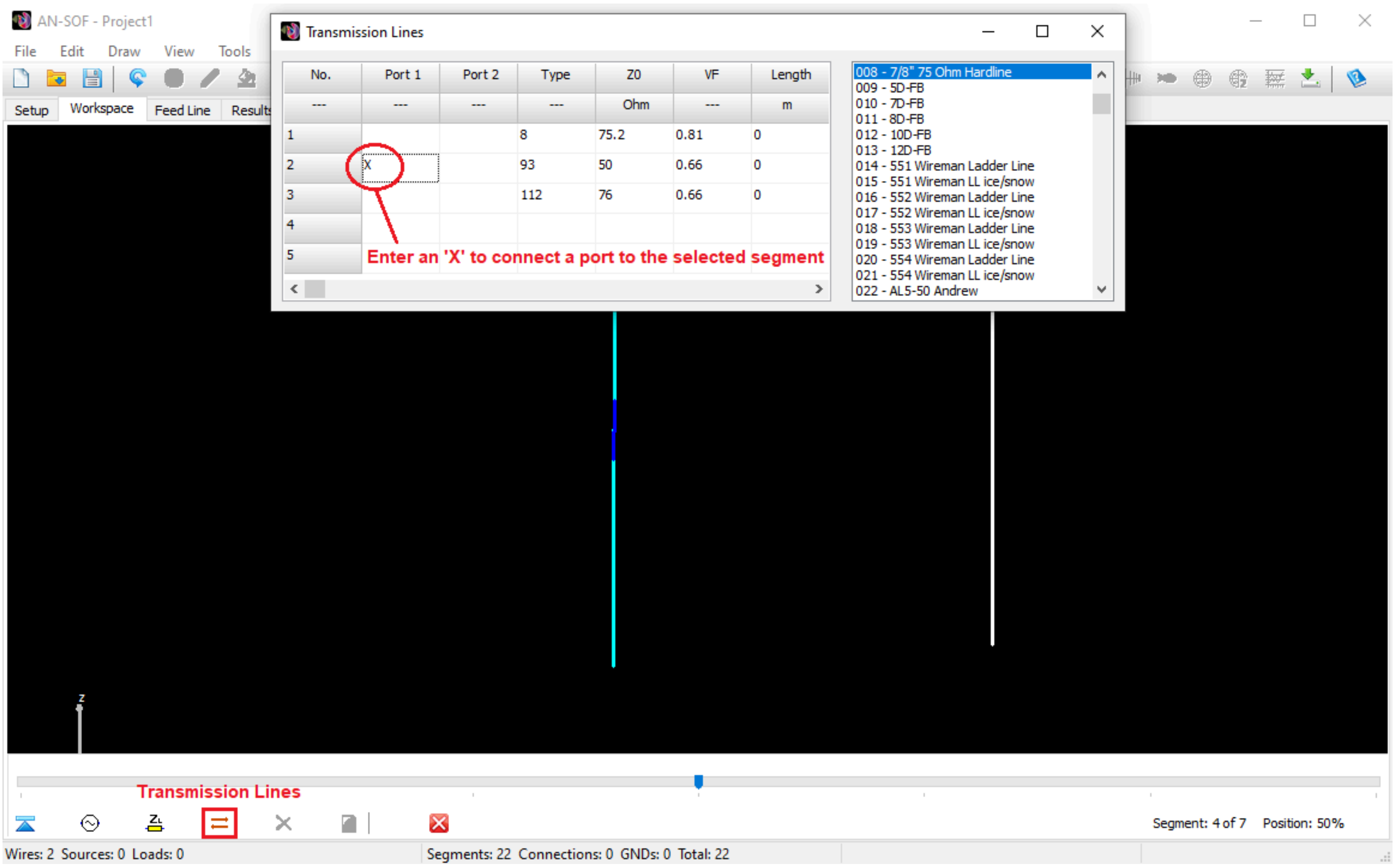


Fig. 2: To display the Transmission Lines window, click on the "Transmission Lines" button located on the horizontal toolbar. Then, enter the letter "X" (in either lowercase or uppercase, without quotes) in the cells of the ports that you wish to connect to the selected segment.

4. Enter an **"x"** or **"X"** (without quotes) in the corresponding cell for the port you want to connect to the selected segment (the cells located below the "Port 1" and "Port 2" columns), Fig. 2. You can enter an "X" for all the ports that need to be connected to the same segment as multiple transmission lines can be connected to it. Finally, close the Transmission Lines window.

5. Select the second wire and repeat steps 1-4 to connect the second port of the transmission line to another segment, Fig. 3. The transmission lines with both ports connected will be graphically displayed as shown in Fig. 4.

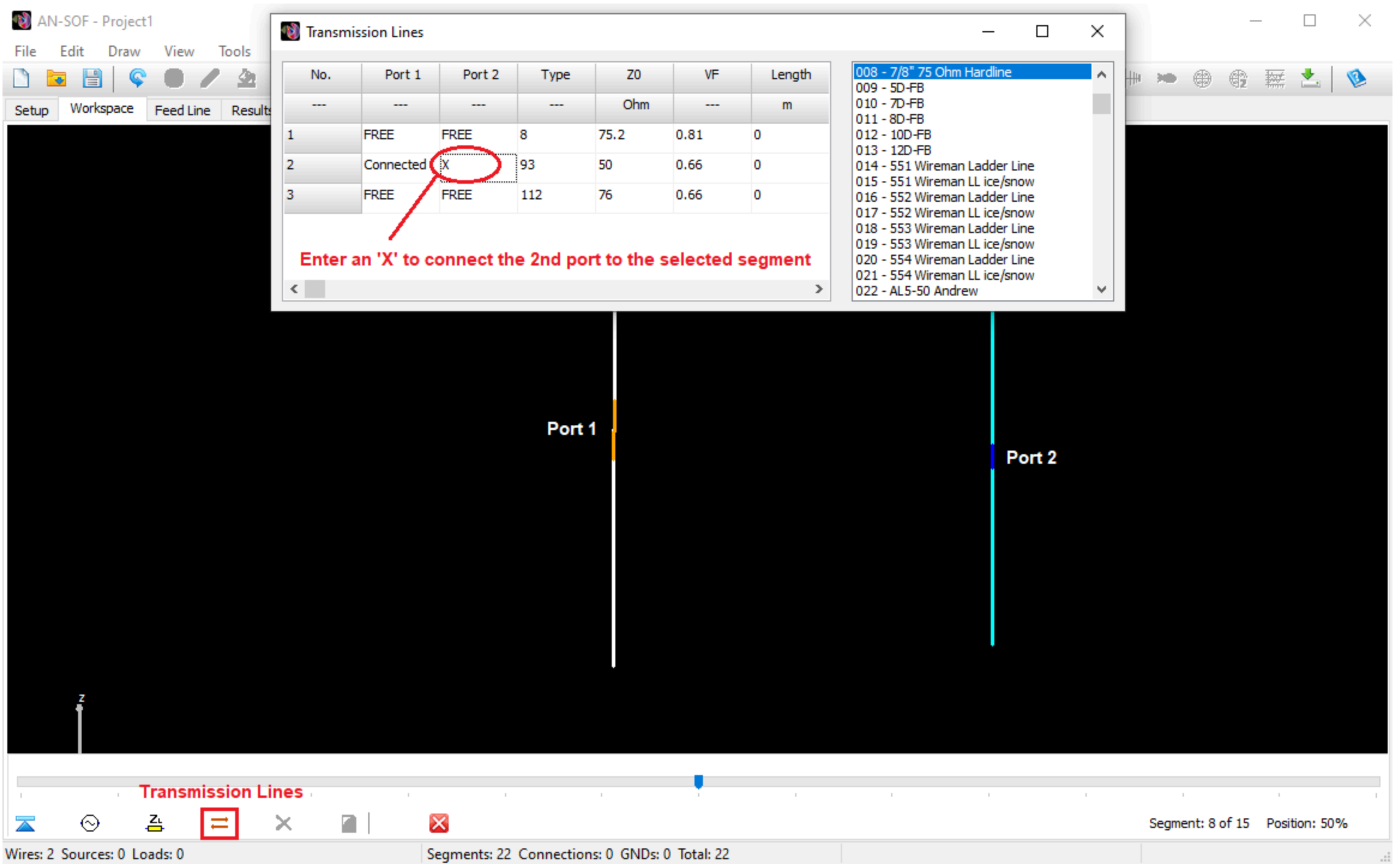


Fig. 3: Select the second segment and enter an 'X' in the ports that you want to connect there.

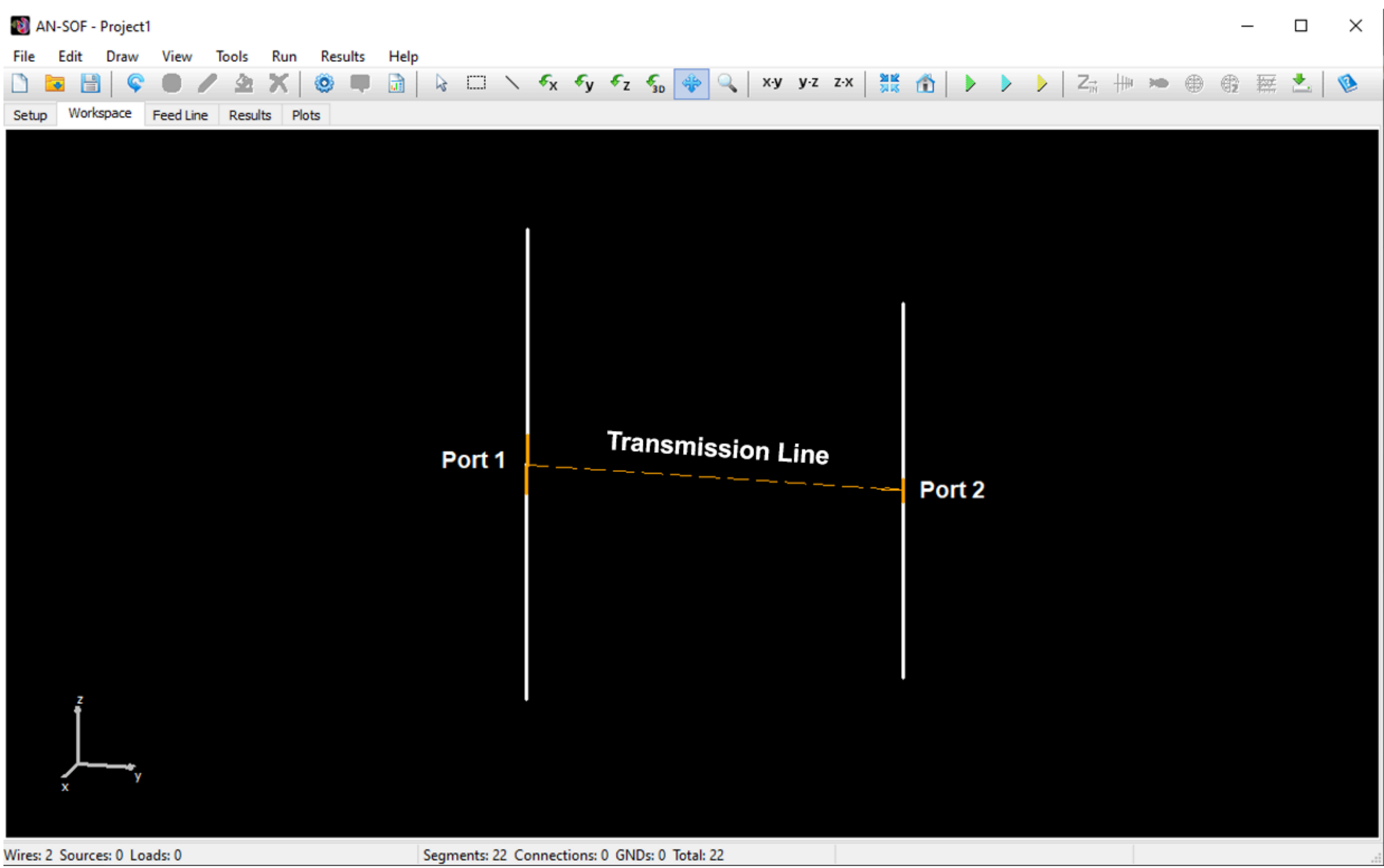


Fig. 4: Transmission line in the workspace, connecting two segments located on different wires.

While performing this procedure, you have the option to add more transmission lines directly in the "Transmission Lines" dialog window. This saves you from having to follow the steps outlined in **Adding Transmission Lines**. The advantage of adding transmission lines here is that you can edit the connections of the lines in the "Port 1" and "Port 2" columns. However, with the Draw > **Transmission Lines (Ctrl + L)** command, you can quickly edit the lines (Z0, VF, length, etc.) if you don't need to change the port connections.

A port that is already connected to a segment will show the status as **"Connected,"** while if it is not connected to any segment, it will display the status as **"FREE"**. When we are on a selected segment, a connected port will show the status as **"Here,"** which refers to the port being connected specifically to that selected segment.

To disconnect a port from a segment, enter the word **"FREE"** (without quotes, in uppercase) in the corresponding cell instead of an "X". **This allows you to use the "X" and "FREE" commands to easily connect and disconnect ports on a selected segment.**

The transmission lines that have both ports connected to segments are displayed as **straight dashed lines in orange color** in the workspace, Fig. 4. An **arrow** will indicate the direction of the line, which goes **from port 1 to 2**. Since the length of a line is another parameter that is entered, such as its characteristic impedance and velocity factor, the length of the line in the workspace may not represent the configured or "real" length of the line.

If you select a row by clicking on the row number in the Transmission Lines table, the corresponding line will be **highlighted in red in the workspace** (if it has both ports connected to segments), Fig. 5. This way, you can visually identify which line you are editing.

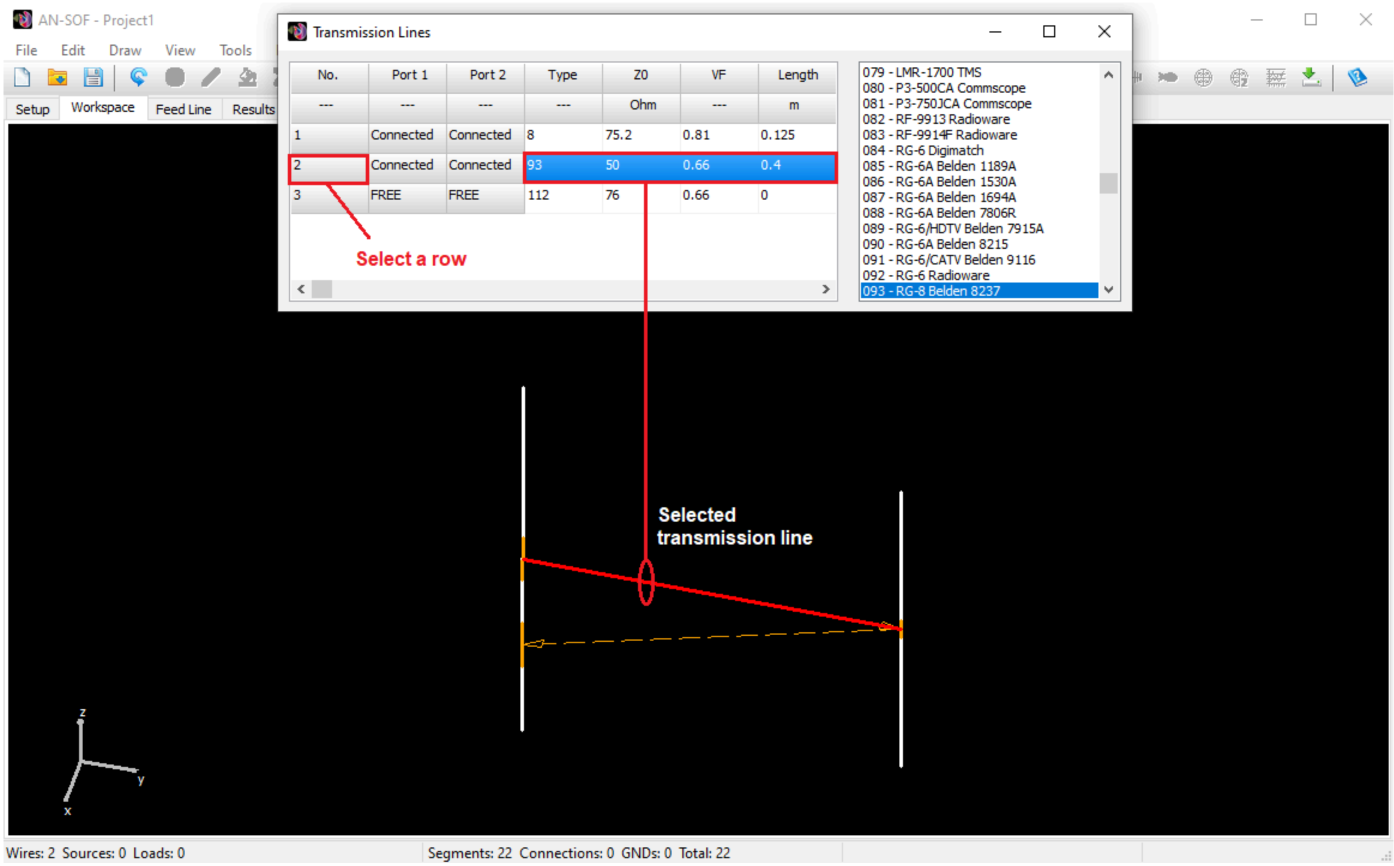


Fig. 5: By selecting a row in the Transmission Lines table, you can easily identify the corresponding transmission line in the workspace as it will be highlighted in red.

IMPORTANT Information

A transmission line with only one port connected to a wire segment will not be considered in the calculations. Instead, it exists as a row within the table, which can be used as a library of lines to select from and connect to the wire structure. Therefore, **when a port is FREE, it does not mean that the corresponding end of the transmission line is open circuited,** but rather that this line will simply be omitted in the simulation. It is sufficient for only one port to be FREE for the line to be omitted. If you need to connect a line with an open or short-circuited end, please refer to **Open and Short-Circuited Lines** for detailed instructions.

A voltage or current source can be connected to any segment where one or more transmission line ports are connected. In this case, **the sources will always be "ideal"**, i.e., with zero/infinite internal impedance (zero for voltage sources and infinite for current sources), unlike in an ordinary segment without a port connected, where sources may have non-zero/finite internal impedance (in AN-SOF, current sources should always have a finite internal impedance because this impedance is connected in parallel with the current source).

In each segment, only transmission line ports or a load impedance are allowed, but not both. If a port is connected to a segment where a load impedance already exists, this impedance will be eliminated, and vice versa. If you need to connect a load impedance in series with the port of a transmission line, connect the impedance in an adjacent segment to the port.

When there are transmission lines in the model, the **NGF (Numerical Green's Function)** option will be automatically enabled in the **Settings panel** of the Setup tabsheet. This way, calculations will be performed faster in the next simulation if only the parameters of the transmission lines are modified while the wire structure remains unchanged.

It is recommended to connect transmission lines **after drawing and segmenting the wire structure.** If the number of segments changes, the lines may become disconnected and need to be manually reconnected using the procedure described in this section.

To ensure a smooth calculation process, **AN-SOF will verify the correct connections** between the transmission lines and the wire segments. If AN-SOF detects any errors, it will promptly remove the faulty connection by setting the corresponding port to FREE state.

Open and Short-Circuited Lines

Due to the model used in the calculation engine, the transmission lines that are considered to exist in the simulation are those that have both ports connected to wire segments. Therefore, if you want to have an **open-circuited line** connected to a certain segment, the opposite port must also be connected to another wire segment. Create a **short wire with only one segment** that is no longer than 1% of the wavelength (its radius can be one-tenth of its length) and connect it to the open circuit transmission line port. This short wire should be disconnected from the rest of the structure, and the shunt admittance of the port it's connected to should be zero, Fig. 1.

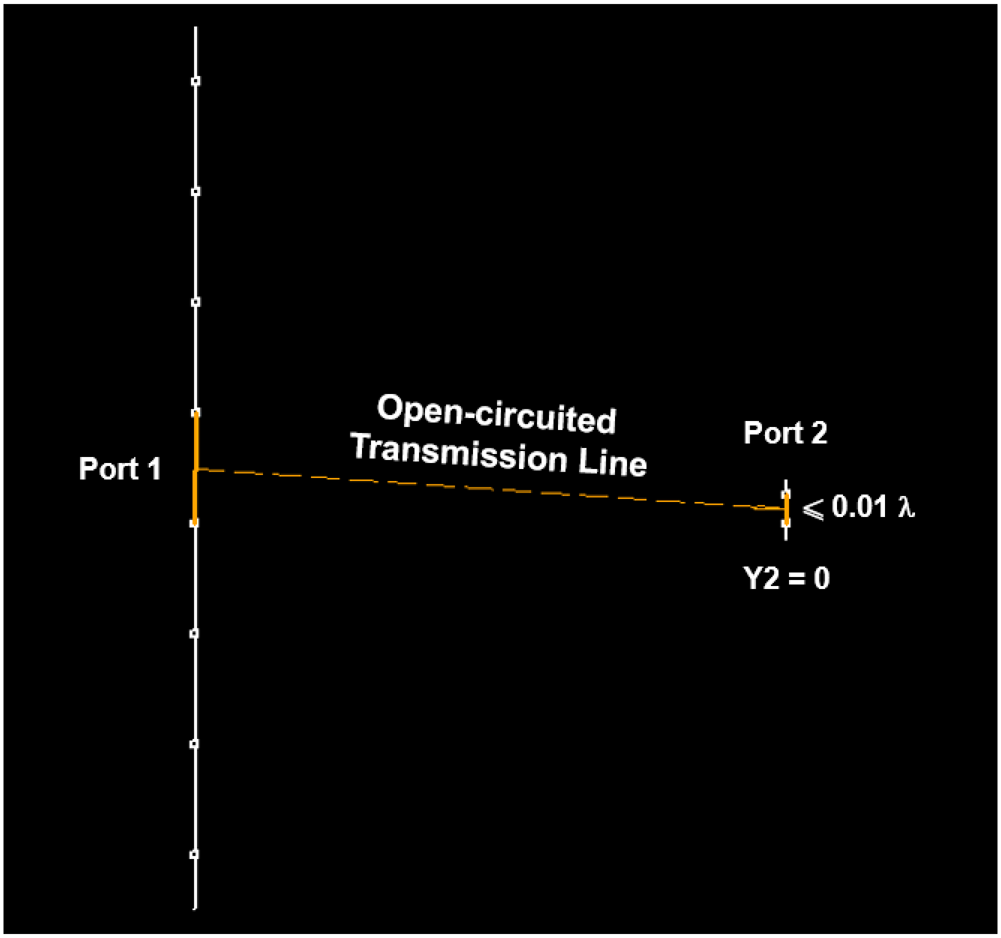


Fig. 1: Open circuit transmission line. The port that is open circuited is connected to a short wire that has only one segment and has a null shunt admittance.

On the other hand, if you need a transmission line with a **short-circuited port**, connect that port to any other segment and set a **shunt admittance** at that end that is very large, for example, **1E6 [S]**. At this end, you could connect a short wire segment created for this purpose, as is done for an open-circuited line, Fig. 2.

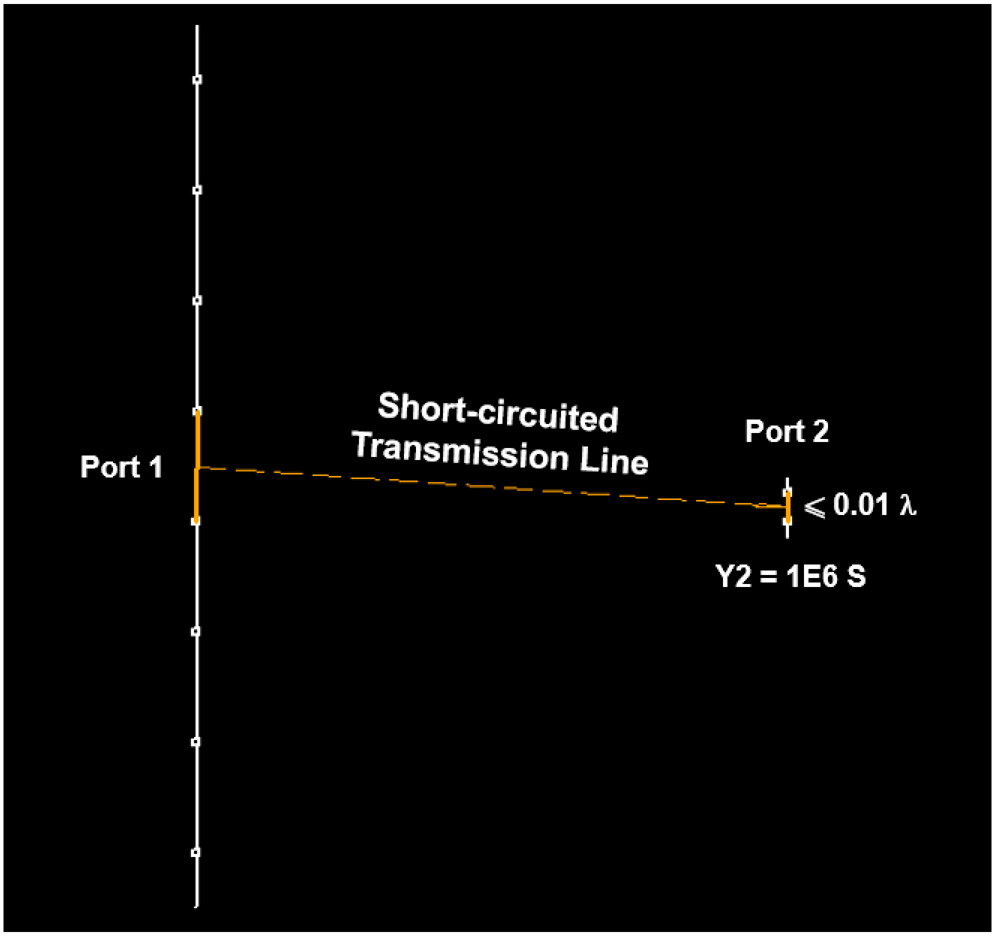


Fig. 2: Short circuited transmission line. The port that is short circuited is connected to a short wire that has only one segment and has a large shunt admittance.

When creating short wires to connect open or short circuit transmission line ports, it is advisable to move these wires away from the rest of the structure to minimize interaction with it. Enter the length of the transmission line as indicated in **Adding Transmission Lines**. Remember that the length of the line is not necessarily related to the actual distance between the segments where it is connected.

Editing Transmission Lines

The **Transmission Lines** table has a pop-up menu with keyboard shortcuts, Fig. 1. To access this menu for editing cells and rows, right-click on the table. The available commands are:

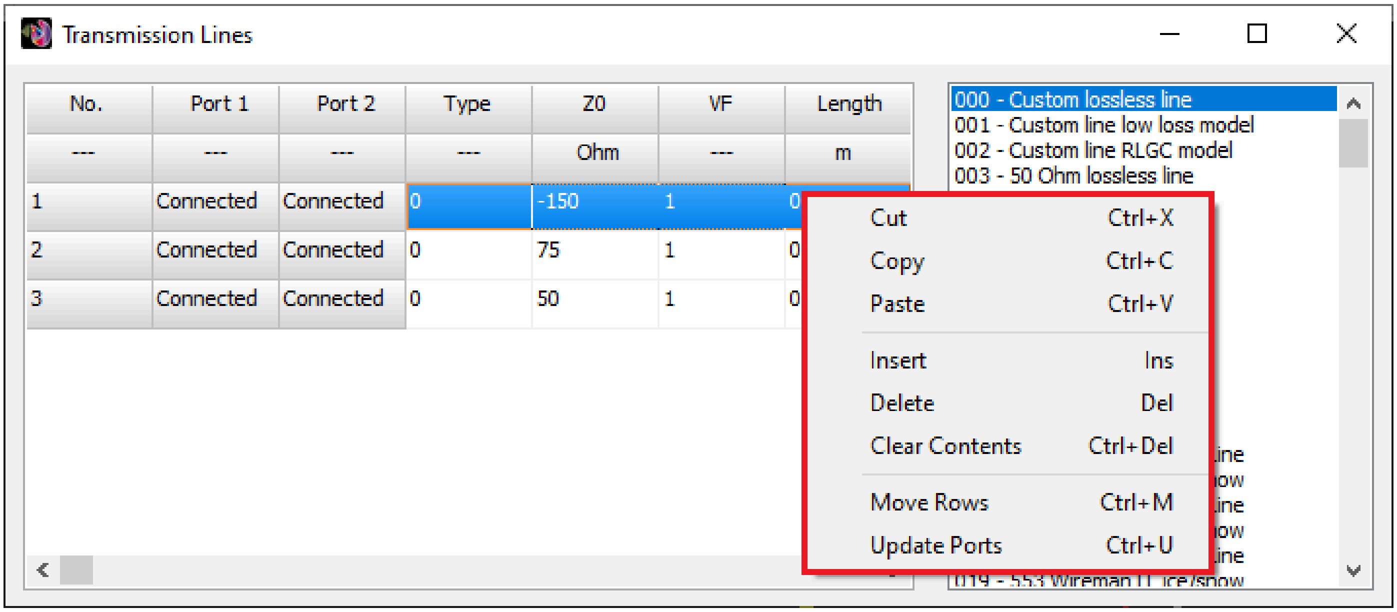


Fig. 1: Pop-up menu in the Transmission Lines table.

1. Standard **Cut (Ctrl + X)**, **Copy (Ctrl + C)**, and **Paste (Ctrl + V)** options are available for cells. A single cell can be selected by left clicking on it or by using the TAB and arrow keys on the keyboard.

2. To select a row, click on the **row number** in the left column (the **"No."** column). Use the mouse or the up and down arrow keys on the keyboard to select a single row. **Double-click on a single cell to exit row selection mode.**

3. **Cut (Ctrl + X)**, **Copy (Ctrl + C)**, and **Paste (Ctrl + V)** options also apply to a selected row. In addition, **Insert (Ins key)** and **Delete (Del key)** options can be used to add or remove rows.

4. The **Clear Contents (Ctrl + Del)** command clears the content of a selected cell or row.

5. The **Move Rows (Ctrl + M)** command allows you to enter a mode where rows can be moved up or down to order them as desired. To exit this mode, click Move Rows (Ctrl + M) again.

6. The **Update Ports (Ctrl + U)** command checks and updates the status of the transmission line ports. Use this command to verify that the lines have their ports connected to wire segments when you have made any modifications to the segmentation or geometry of the wires where there are transmission line ports.

Modeling Coaxial Cables

Coaxial transmission lines can be modeled **implicitly**, as explained in previous articles. To define a coaxial cable, one needs to know its characteristic impedance (Z0), velocity factor (VF), length, parameters that model losses (K0, K1, K2, etc.), and the shunt admittances at each end (Y1 and Y2). Additionally, each end or port of the line must be connected to **the center of a wire segment**. In this implicit model, the electromagnetic interaction between the coaxial cable shield and the wire structure is neglected, and the line ends impose boundary conditions on the voltage and current in the connected segments. However, in certain scenarios, a current can be induced that flows through **the outside of the coaxial cable shield**, known as **common-mode current**, and this current cannot be neglected. To address this, a **hybrid model** is used, which is explained in detail below.

In the hybrid model, the internal behavior of a coaxial cable is implicitly modeled using its parameters such as Z0, VF, length, etc. On the other hand, the **outer shield is modeled by adding a wire** that must be divided into segments like the rest of the structure, Fig. 1. This additional wire considers the current induced outside the coaxial cable shield.

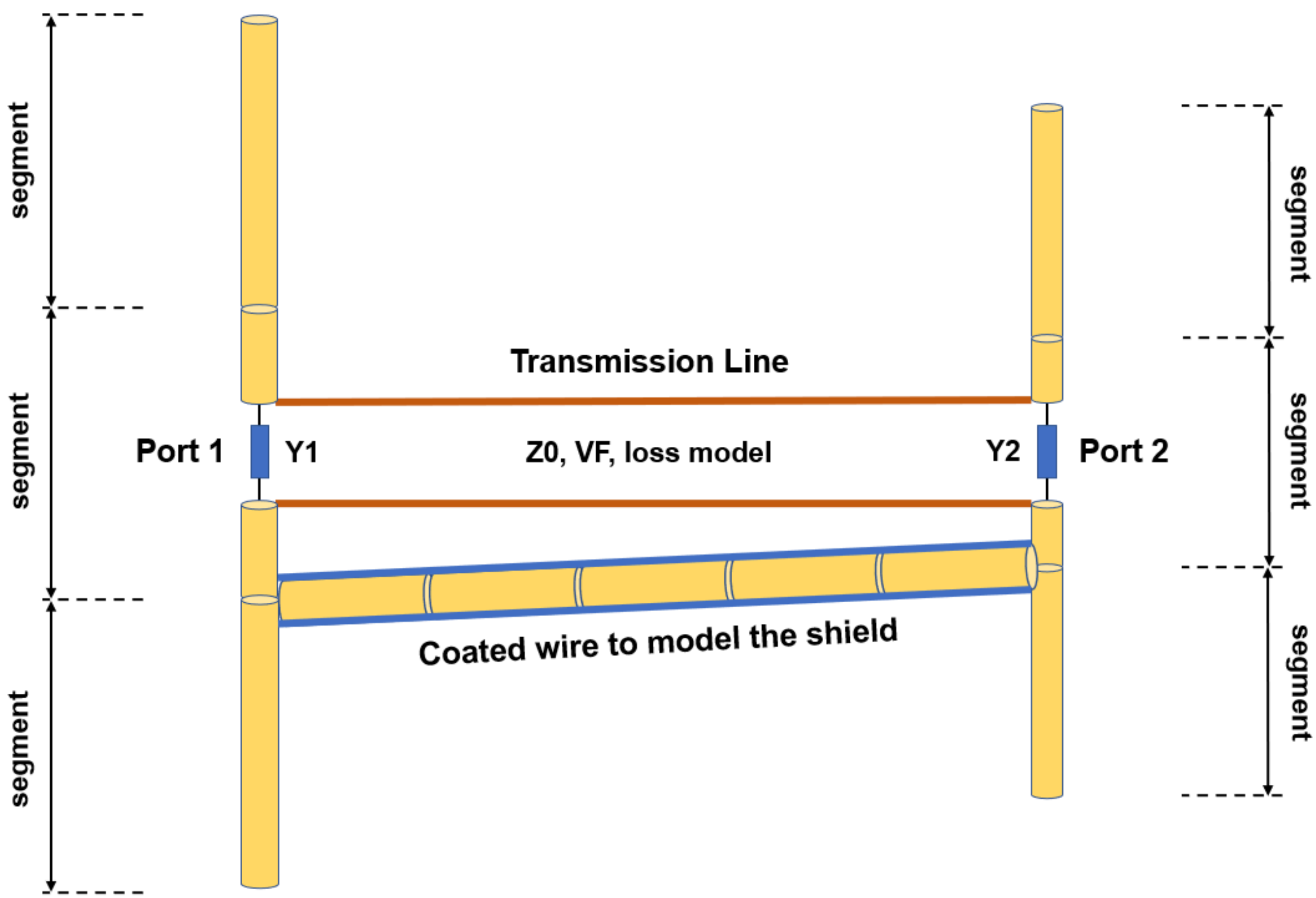


Fig. 1: An additional wire is connected between two ends of the ports of a coaxial transmission line to model the cable shield. This wire needs to be divided into segments just like the rest of the structure.

The wire representing the shield should be connected between two ends of the segments where the cable is connected, Fig. 2. Unlike transmission lines that connect in the center of the segments, wires are connected at their ends. Hence, **the additional wire representing the shield will be a segment offset from the actual position of the cable**. This is not a significant concern since the segments should be small compared to the wavelength.

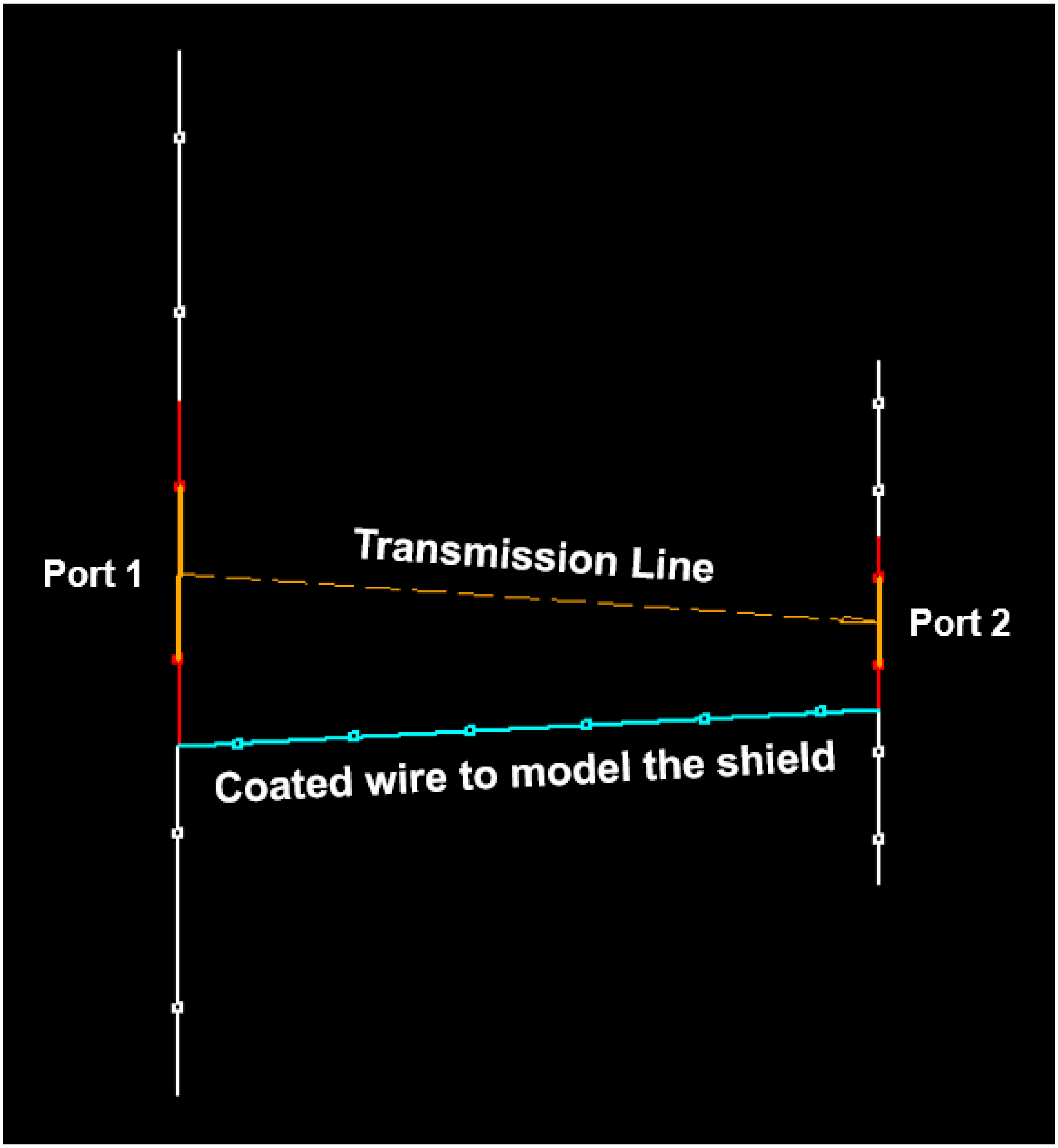


Fig. 2: Visualization of the sketch in Fig. 1 in the AN-SOF workspace. The short wires, each consisting of one segment, that are required to connect the wire representing the coaxial cable shield are highlighted in red.

Please be reminded that to connect one wire to another and **connect the ends of the coaxial cable shield**, you will need to manually divide the wires involved, as explained in **Connecting Wires**.

To simulate the **dielectric coating** of actual coaxial cables, an outer insulation can be added to the wire representing the shield, and its thickness can be input as well.
Adding a Custom Lossy Line

AN-SOF provides parameters for modeling the losses of more than 160 types of transmission lines. These parameters have been obtained by adjusting the loss model to the attenuation curves published by manufacturers. In case a particular type of cable is not listed or if the manufacturer has updated the parameters, a custom transmission line can be created using the following procedure:

1. Open the **Transmission Lines** window by going to the main menu > Draw > Transmission Lines (Ctrl + L) or follow the procedure in **Connecting Transmission Lines** to open this window by selecting a wire.

2. Select a row from the table by clicking on the row number (under the first column labeled **No.**).

3. In the panel on the right, double-click the **Custom line low loss model** option.

4. All manufacturers publish the **nominal characteristic impedance**, Z0, and the **velocity factor**, VF. Enter these values as well as the length of the line. If you enter "0" in the length cell, the linear distance between the ends of the cable will be calculated.

5. Manufacturers also publish an **attenuation table** as a function of frequency. Here is an example for the Belden 8237 cable, type RG-8/U:

Nom. Characteristic Impedance:

Impedance (Ohm)
52

Nominal Velocity of Propagation:

VP (%)
66

Nom. Attenuation:

Freq. (MHz)	Attenuation (dB/100 ft.)
1	.2
10	.6
50	1.3
100	1.9
200	2.8
400	4.2
700	5.9
900	6.9
1000	7.4
4000	23.2

6. In the cells corresponding to **Att. 1**, **Freq. 1**, **Att. 2**, and **Freq. 2**, enter the values from the attenuation table so that **the simulation frequency range is included between Freq. 1 and Freq. 2**. For example, if you are running a calculation between 150 and 170 MHz, enter Att. 1 = 1.9 dB/100 ft, Freq. 1 = 100 MHz, Att. 2 = 2.8 dB/100 ft, Freq. 2 = 200 MHz, as indicated in the table for the Belden 8237 cable, Fig. 1.

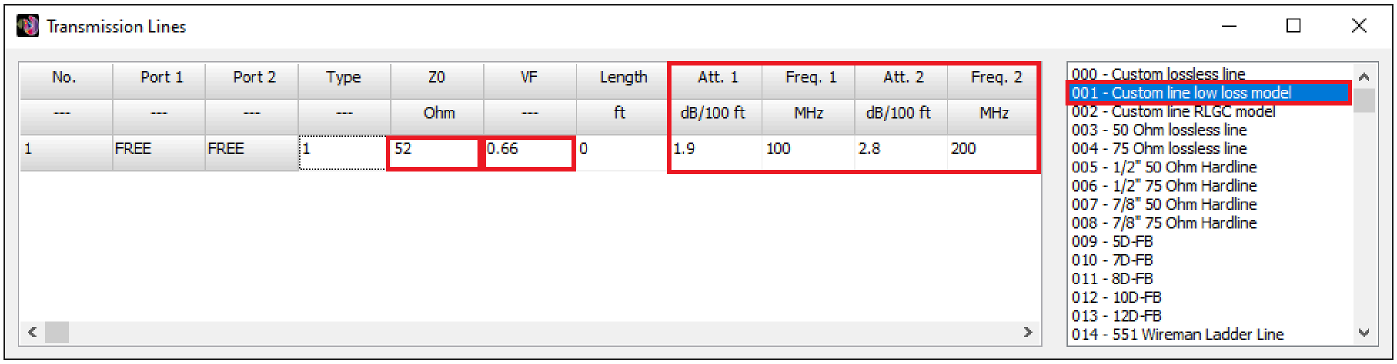


Fig. 1: Entering the values of nominal characteristic impedance, velocity factor, and attenuation for a Belden 8237 cable, type RG-8/U, when the frequency sweep of the simulation is within the range of 100 to 200 MHz.

Be careful with the units of attenuation and frequency, as they will be displayed in the units chosen in the **Preferences** window. Go to main menu > Tools > **Preferences > Units tab** to change the units for frequency and length.

Step By Step

Download Example Models

The **Models** tab in the AN-SOF interface provides a categorized library of pre-built antenna projects. These serve as excellent starting points for learning how to use the software effectively (**Fig. 1**).

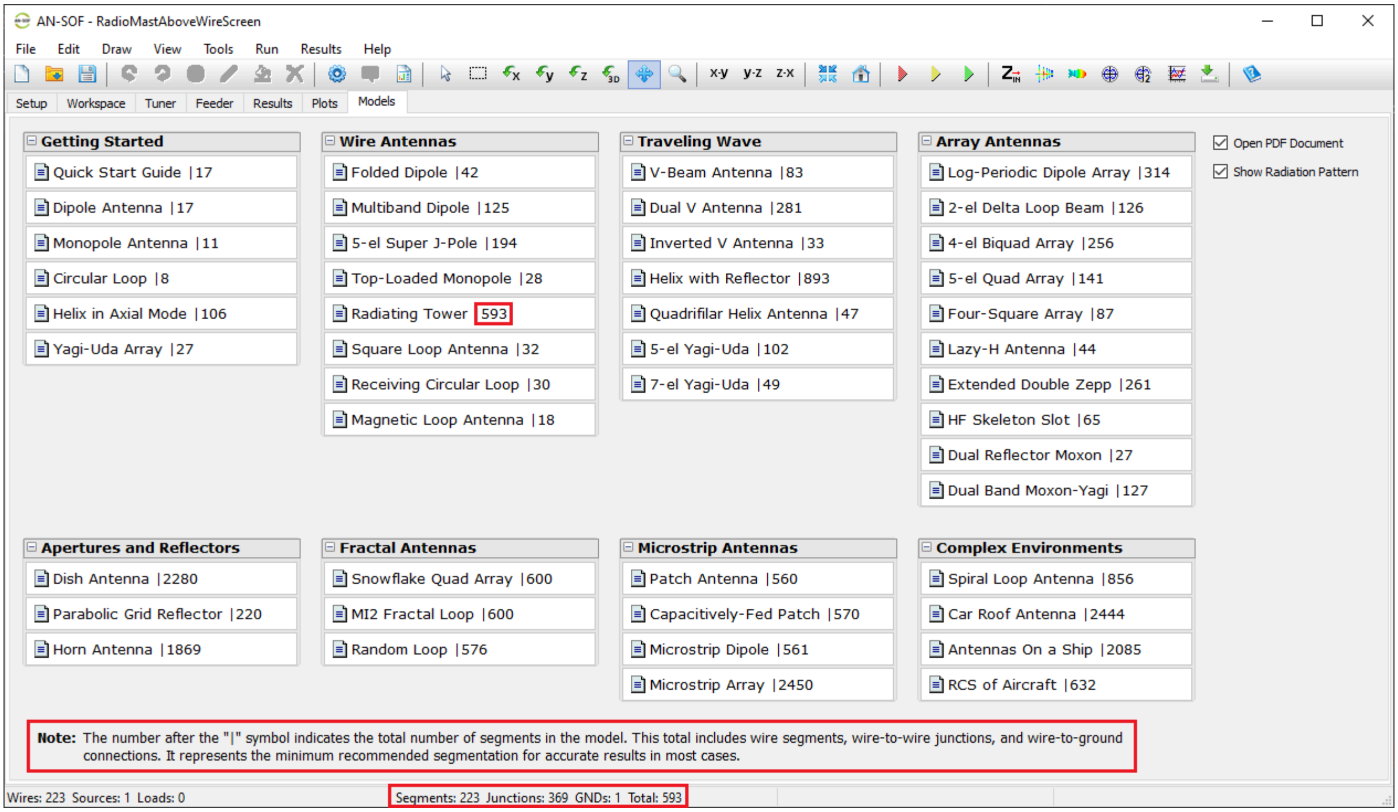


Fig. 1: *The Models tab in the AN-SOF interface.*

Interactive Learning

When you click on a model button:

- **Instant Visualization:** The project opens immediately, displaying the 3D radiation pattern and geometry.
- **Documentation:** A detailed PDF document opens automatically, explaining the underlying physics and the specific configuration of that model.

Trial Version Compatibility

The trial version of AN-SOF has a **50-segment limit** (totaling wire segments, junctions, and ground connections). Here is how you can use the examples:

- **View Mode:** All pre-computed example models, even those exceeding 50 segments, can be opened in the Trial version. You can fully inspect the 3D plots, tables, and graphs of these existing results.
- **Run Mode:** To run a new calculation or modify a design, the model must stay within the 50-segment limit.

- **Downloadable Models:** You can download these five specific models designed for the trial version:
  - **2-Element Quad**
  - **2-Element Delta Loop**
  - **HF Skeleton Slot**
  - **Inverted V**
  - **5-Element Yagi-Uda**

Download 5 Models

Expanding Your Knowledge

- **Knowledge Base:** Visit the website's **Knowledge Base** for comprehensive articles with dedicated category filters.
- **Resources & Blog:** Check these pages frequently for the latest downloadable examples and engineering insights: **Resources** and **Antennas Blog**.
- **Newsletter:** Subscribe to **"Antennas and Beyond!"** to receive updates, project ideas, and new models directly in your inbox.

Explore 5 Antenna Models with Less Than 50 Segments in AN-SOF Trial Version

Discover 5 antenna models with less than 50 segments in AN-SOF Trial Version. These examples showcase the capabilities of our software for antenna modeling and design, allowing you to evaluate its features for your projects.

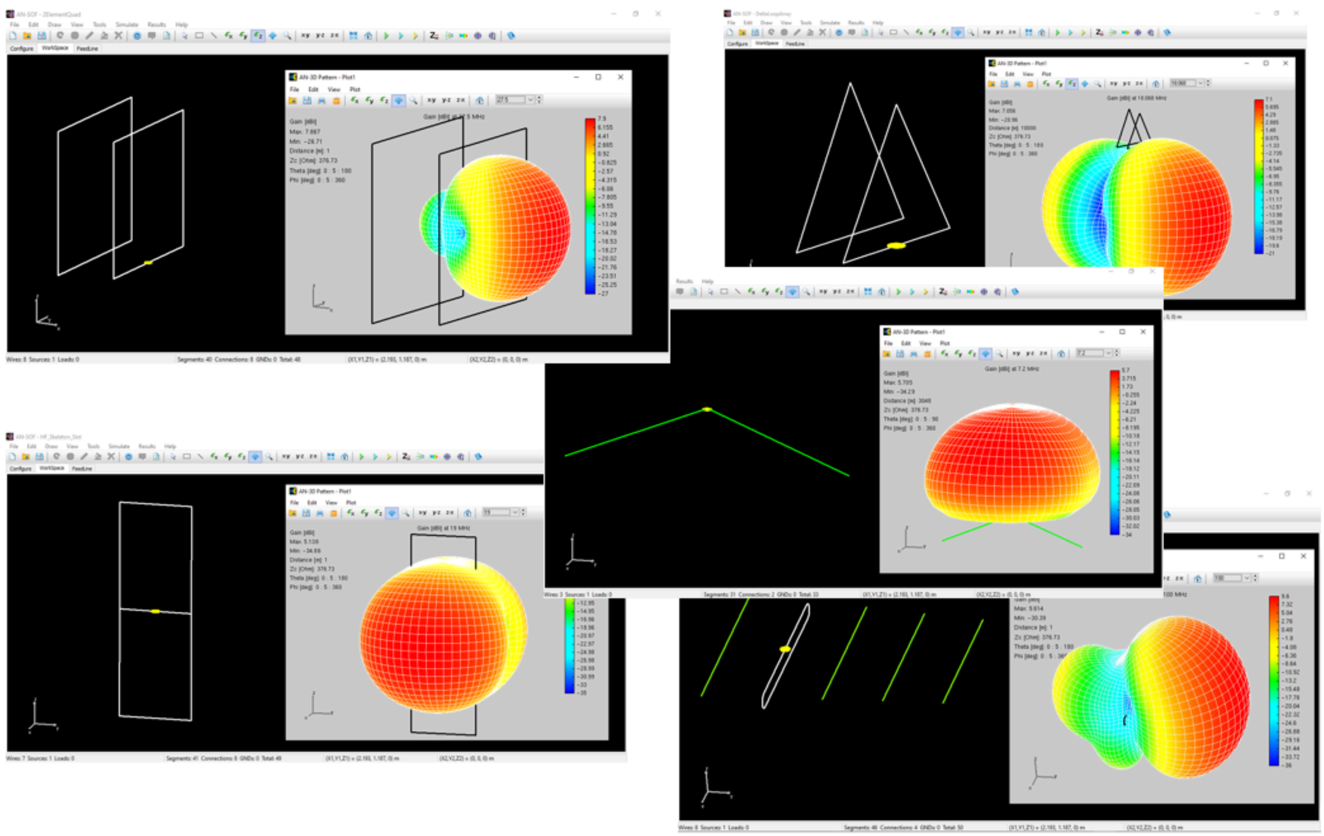


Illustration of 5 antenna models with less than 50 segments in AN-SOF Trial Version.

The trial version of **AN-SOF** is **fully featured** and **has no expiration date**. It allows users to open any **pre-calculated example file** to inspect data tables and visualize 3D radiation patterns and graphs. The sole limitation is the calculation capacity, which is capped at **50 "unknowns."** In this context, an unknown represents an electric current value to be determined by the **AN-SOF engine** at each segment, junction, or ground connection. Consequently, the sum of segments, wire-to-wire **connections**, and **ground connections** must not exceed 50 to run a new simulation in the Trial version.

The purpose of the trial version is to allow users to evaluate the full features and capabilities of AN-SOF for antenna modeling and design. You can access a variety of pre-calculated models directly through the **Models** tab in the software. Furthermore, our **Knowledge Base** features an extensive collection of **model examples** accompanied by descriptive articles. These resources are categorized by antenna type, ranging from fundamental wire structures to antennas situated in complex environments.

For more complex antenna designs, the 50-unknown limit may be reached quickly. While users can view pre-calculated examples that exceed this threshold, any modifications requiring a re-run of the simulation will be restricted if the total number of segments and connections exceeds 50. Nevertheless, the trial version remains a highly effective tool for simulating fundamental antenna projects or structures with **small electrical dimensions relative to the wavelength**.

Download the following 5 example models with less than 50 segments to make modifications to the antenna structures:

- 2 Element Quad
- 2 Element Delta Loop
- HF Skeleton Slot
- Inverted V
- 5 Element Yagi-Uda

Download Models
To ensure reliable results, **a minimum of 10 segments per wavelength** should be used. For antennas that are highly sensitive to element lengths, such as **Yagi-Uda arrays**, a higher density of approximately **50 segments per wavelength** is recommended to achieve results that closely align with physical VSWR measurements.

Explore further case studies and technical articles in the **Validation** section of our Knowledge Base. Additionally, the AN-SOF Trial Version features integrated **tuner** and **feeder** calculators, enabling users to synthesize impedance matching networks, incorporate transformers, and determine optimal feed line parameters for any given load impedance.

Ultimately, AN-SOF Trial offers a robust platform for antenna simulation, providing all the necessary tools to evaluate its performance and capabilities. With access to pre-calculated examples and advanced design utilities, users can navigate the complexities of antenna design with ease.

See Also:

Quick Start Guide: Enhancing Antenna Design Through Simulation Software

Modeling a Center-Fed Cylindrical Antenna with AN-SOF

Complete Workflow: Modeling, Feeding, and Tuning a 20m Band Dipole Antenna

About the Author

Tony Golden

RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I’m a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

[Ask me](#) | **[Email me](#)** | **[Follow me](#)**
[Antennas and Beyond!](#)
[Get Exclusive Updates](#)
Modeling a Center-Fed Cylindrical Antenna with AN-SOF

Learn how to simulate a center-fed cylindrical antenna using AN-SOF software. This step-by-step guide covers setup, geometry creation, simulation, and result analysis. Understand dipole characteristics through practical examples.

Introduction: Center-Fed Cylindrical Antenna Simulation

The center-fed cylindrical antenna serves as a fundamental example for simulation. Essentially a straight wire with a central excitation, it transitions into a half-wave dipole when its length aligns with half the wavelength of the operating frequency. The following steps outline the simulation process using AN-SOF.

Step 1: Configuring the Simulation Environment

To initiate, navigate to **Tools > Preferences** within the main menu to establish appropriate units for frequency (MHz) and length (m). Subsequently, access the **Setup tab**. Within the **Frequency panel**, select **Sweep** and configure the **Frequency Sweep** parameters as depicted in Fig. 1. The calculations will be performed at the frequencies: 50, 55, …, 295, 300 MHz. Ensure that **None** (free space) is chosen in the **Environment panel**'s **Ground Plane** box and **Discrete Sources** is selected under the **Excitation panel**.

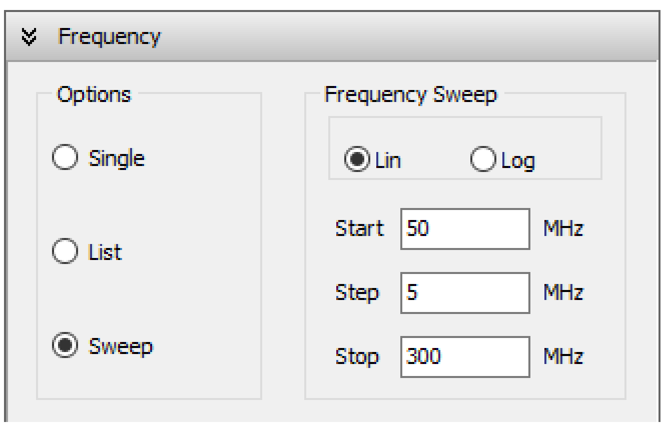


Fig. 1: Frequency sweep parameters setup.

Step 2: Creating the Antenna Geometry

To initiate the antenna geometry creation, right-click within the workspace and select **Line** from the ensuing **pop-up menu**. The 'Line' dialog box will appear. Populate the **Line** and **Attributes** pages as outlined in Figs. 2 and 3 to generate a straight wire comprising **17 segments** and a **1**

mm radius within the workspace. The wire will be drawn starting from point (0,0,-0.75) [m] and ending at point (0,0,0.75) [m], aligning with the z-axis and spanning **a length of 1.5 m**, equivalent to a **half-wavelength at 100 MHz**. Press F7 to visualize the primary axes.

Subsequently, right-click on the wire and choose **Source/Load/TL** from the context menu. Following **the procedure detailed in "Adding Sources,"** introduce a voltage source at the wire's center (segment 9). Set the source voltage to 1 (0°) V. The resulting center-fed cylindrical antenna in the AN-SOF's workspace is represented in Fig. 4.

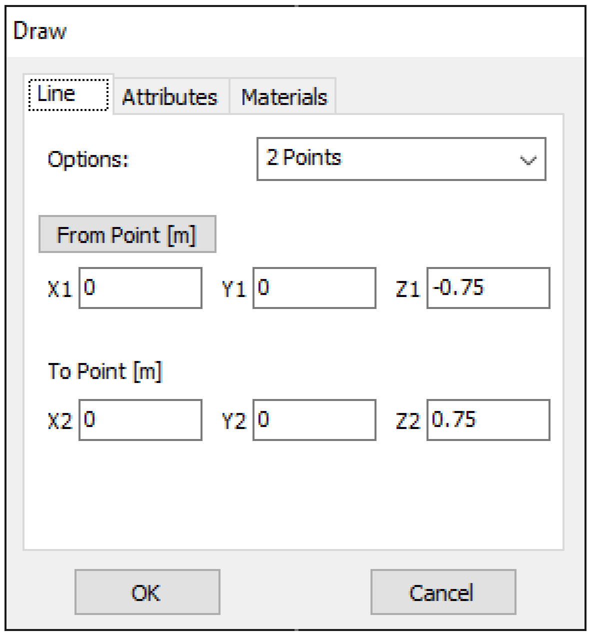


Fig. 2: Line dialog box for defining the antenna geometry.

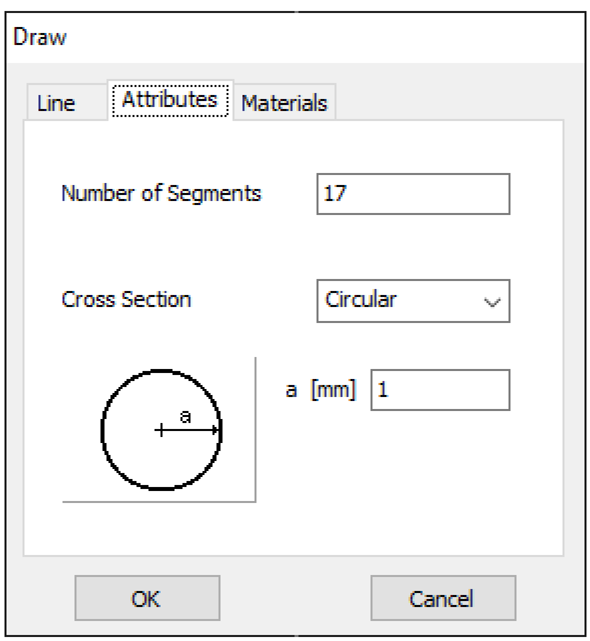


Fig. 3: Line attributes configuration.

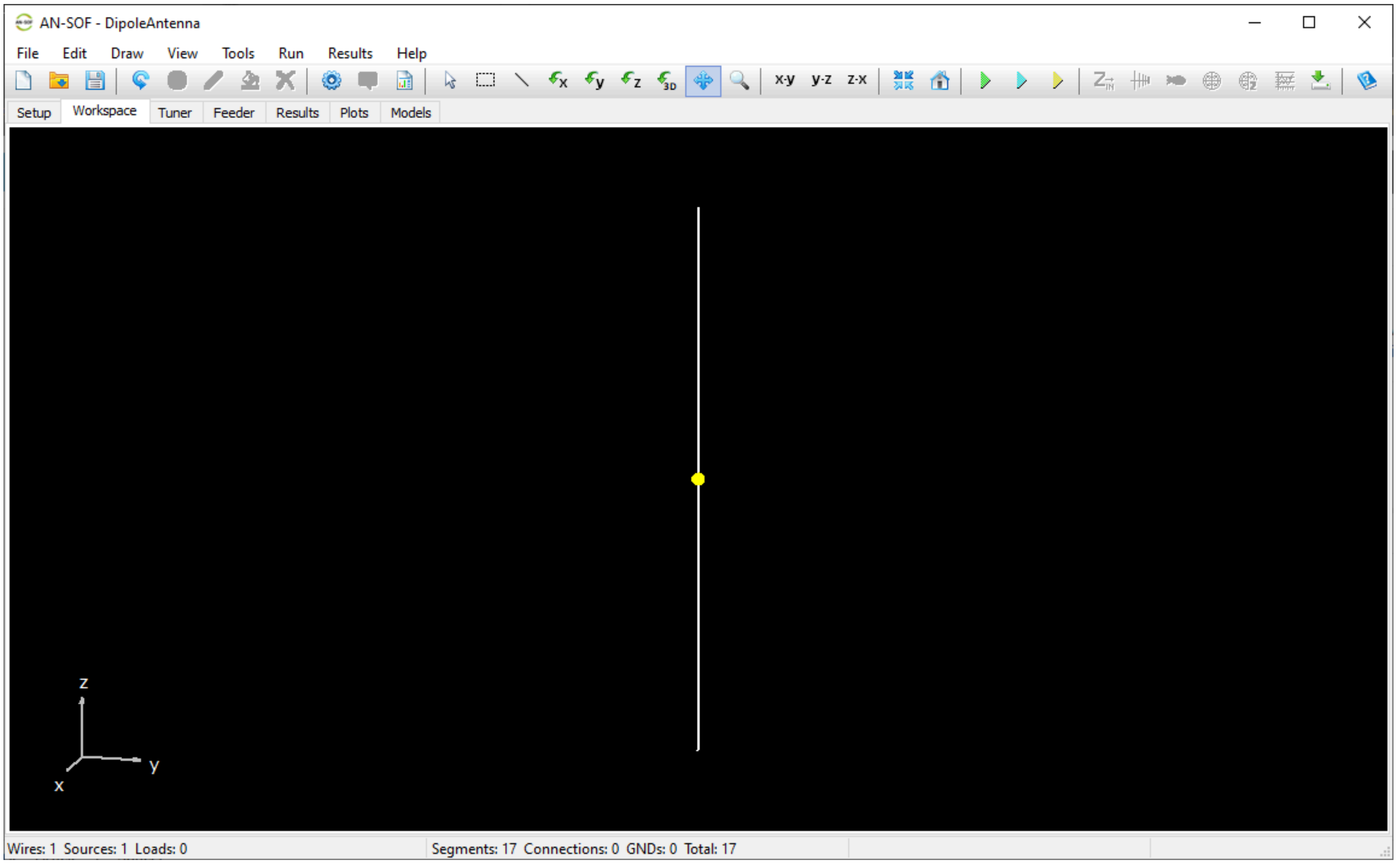


Fig. 4: Center-fed cylindrical antenna geometry.

Download Model

Step 3: Simulation Execution and Result Analysis

To initiate the simulation process, click the **Run Currents and Far-Field (F11)** button on the toolbar. Upon completion, right-click on the wire and select **Plot Currents** from the context menu, specifying the desired frequency. The resulting current distribution along the wire is graphically represented in Fig. 5. To access additional parameters of interest, refer to **the procedures outlined in "Displaying Results."**

As an illustrative example, Figures 5, 6, and 7 depict the **current distribution** at 100 MHz (amplitude in Fig. 5(a) and phase in Fig. 5(b)), **input impedance versus frequency** (real part in Fig. 6(a) and imaginary part in Fig. 6(b)), and **gain pattern in dBi** (Fig. 7) at 100 MHz.

Given that the antenna length (1.5 m) equals **half a wavelength at 100 MHz**, the current distribution in amplitude approximates **a half-cycle sine function**, aligning with the expected behavior of a half-wave dipole. A slight decrease in the amplitude and a sharp increase in the phase can be seen at the antenna center, due to the presence of the voltage source just there. The presence of the voltage source at the center disrupts the continuity of the current's slope (derivative) at that point, while the current itself remains continuous.

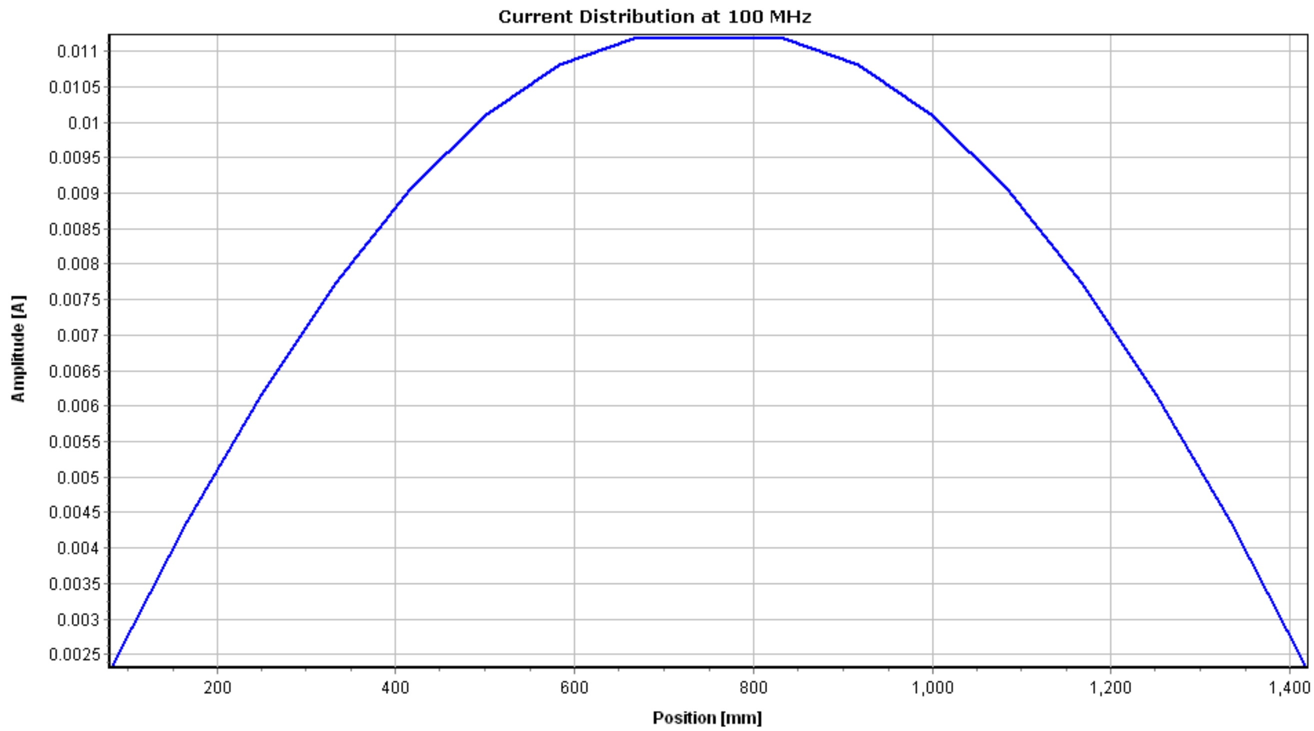


Fig. 5(a): Amplitude of the current distribution along the cylindrical antenna at 100 MHz.

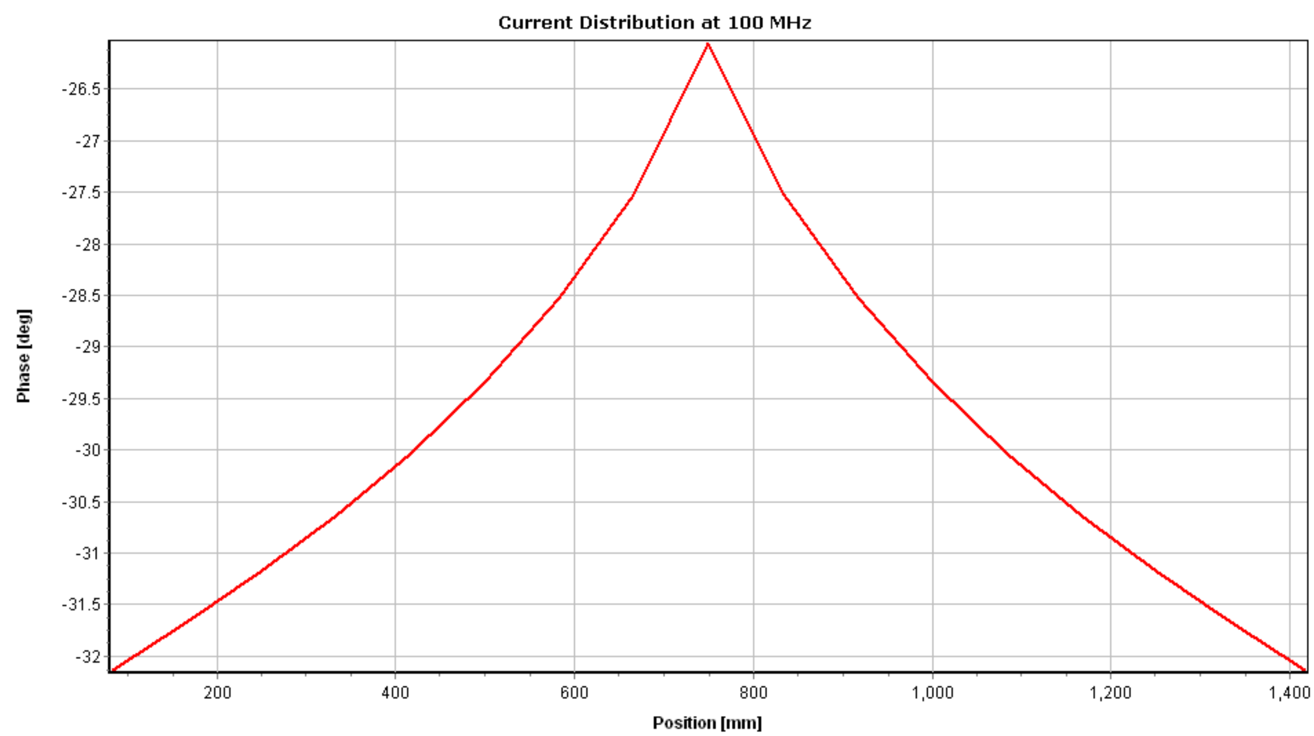


Fig. 5(b): Phase of the current distribution along the cylindrical antenna at 100 MHz.

If we look closely at Figure 6(b), which shows the **input reactance** (imaginary part of the input impedance), we can see that the curve **crosses zero just before 100 MHz**, with a **positive slope** (series resonance), then **crosses zero again just above 180 MHz**, with a **negative slope** (parallel resonance), and then **crosses zero again just below 300 MHz** with a **positive slope** (series resonance). These three points where the reactance vanishes correspond to when the physical length of the dipole approaches: λ/2, λ, and 3λ/2. The resonances do not occur exactly at integer values of half wavelength because **the thickness of the dipole is not infinitesimal**. In Figure 6(a) we can see that the input resistance is maximum at the frequency that corresponds to the parallel resonance. All these are the expected and classical behaviors of a dipole of finite thickness.

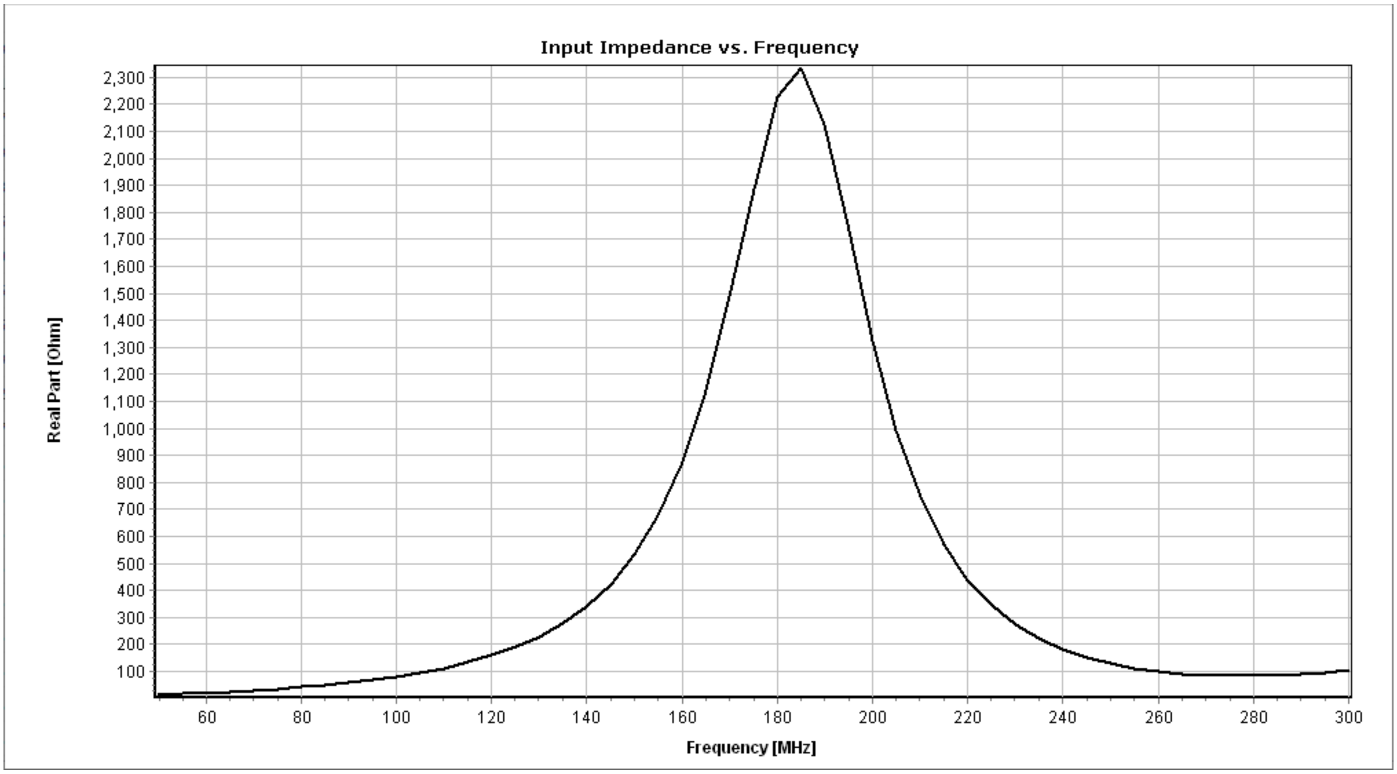


Fig. 6(a): Real part of the input impedance vs. frequency.

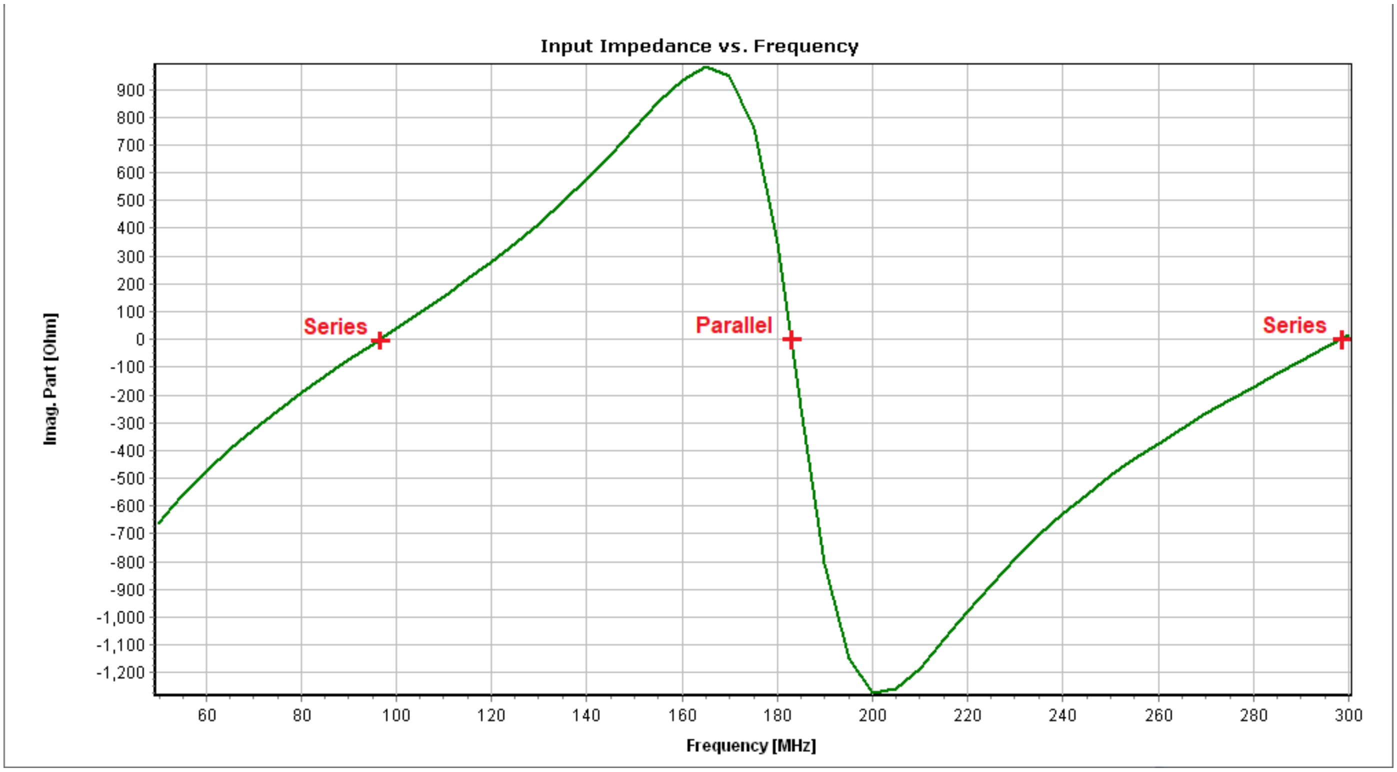


Fig. 6(b): Imaginary part of the input impedance vs. frequency.

Regarding the gain pattern in Fig. 7, it is **donut-shaped as expected for a half-wave dipole**, with **a maximum of 2.17 dBi**. We should remember that the theoretical peak gain of an infinitesimally thin half-wave dipole in free space with a perfect sinusoidal current distribution is 2.15 dBi (corresponding to a numerical gain of 1.64). The obtained gain in AN-SOF is 0.02 dBi higher than the theoretical value due to the finite radius of the cross-section of the dipole.

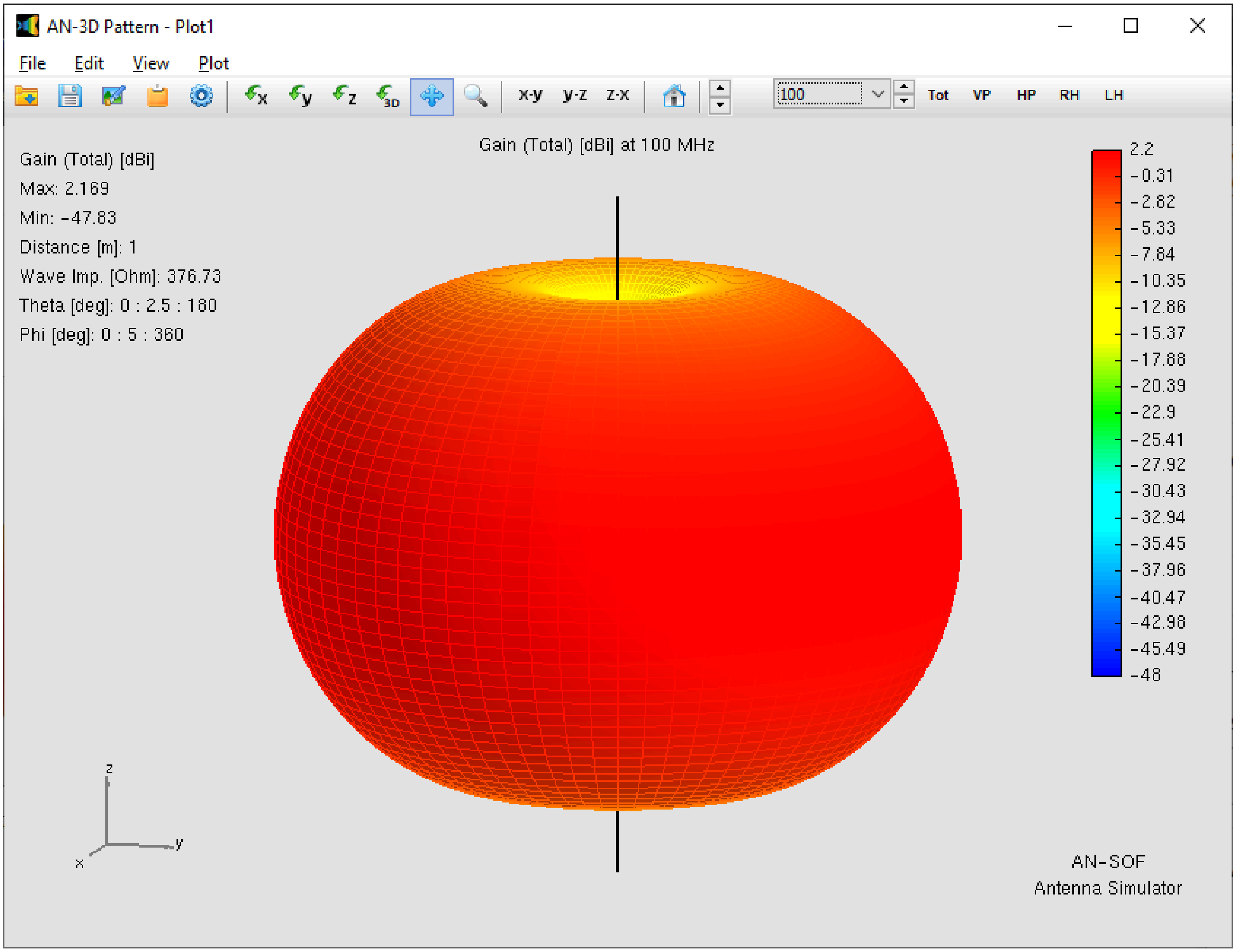


Fig. 7: Gain pattern (dBi) at 100 MHz.

Conclusion

This tutorial provided a step-by-step guide to simulating a center-fed cylindrical antenna using **AN-SOF Antenna Simulator**. By following the outlined procedures, users can efficiently model this fundamental antenna type and analyze its key characteristics.

The simulated results align with the expected behavior of a half-wave dipole, demonstrating the software's accuracy in predicting current distribution, input impedance, gain, and radiation patterns. The influence of the antenna's finite thickness on the resonance frequencies and gain was also highlighted.

This example serves as a foundation for more complex antenna designs. By understanding the simulation process for this simple geometry, users can apply similar principles to model and analyze a wide range of antenna structures.

See Also:

[Linear Antenna Theory: Historical Approximations and Numerical Validation]

About the Author

Tony Golden

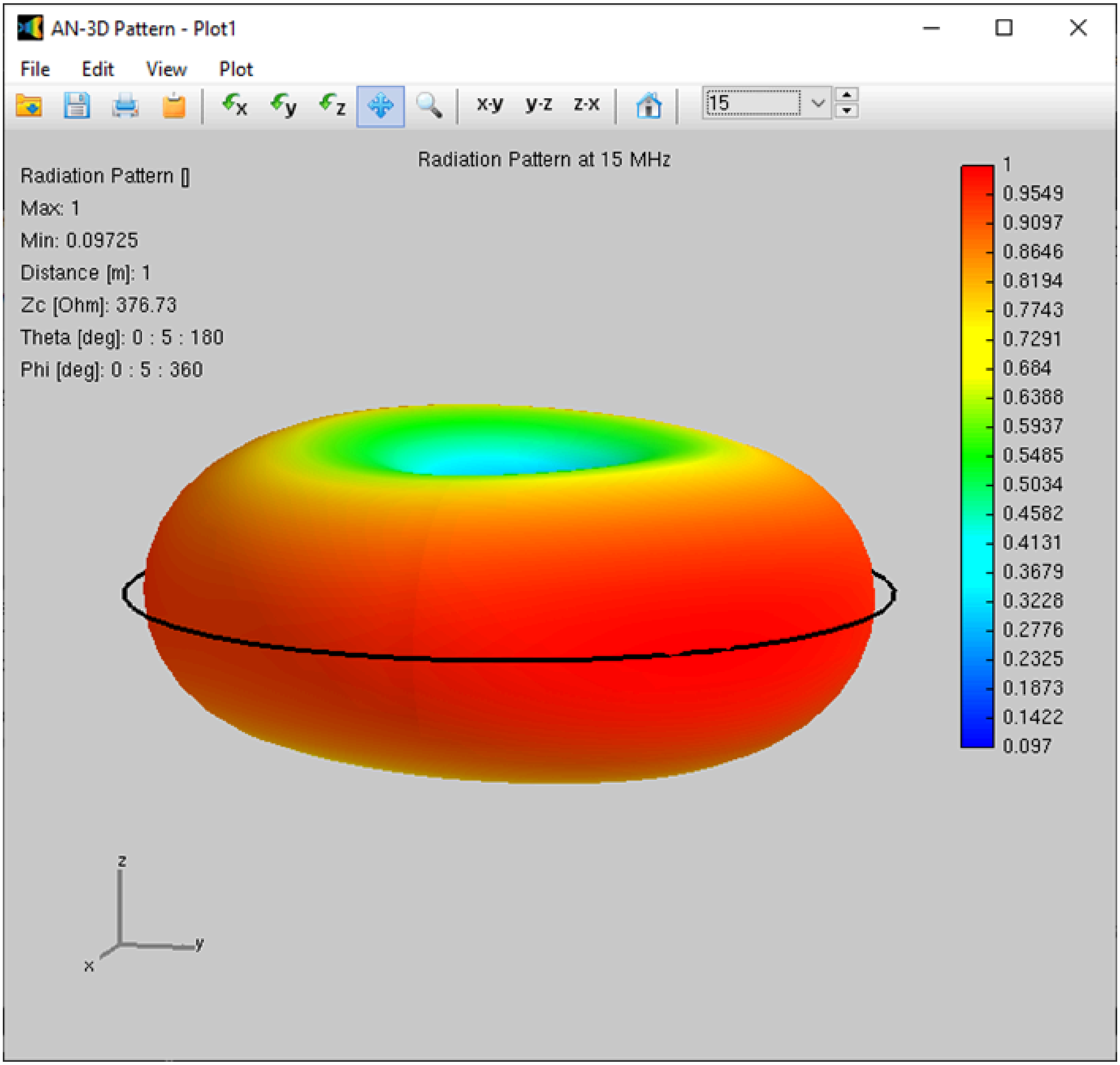
RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I'm a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

Ask me | **Email me** | **Follow me**

Modeling a Circular Loop Antenna in AN-SOF: A Step-by-Step Guide

This step-by-step guide empowers you to simulate circular loop antennas in AN-SOF. We'll configure the software, define loop geometry, and explore how its size relative to wavelength affects radiation patterns and input resistance. Gain valuable insights into this fundamental antenna type!



*T*his article provides a step-by-step guide to modeling a **circular loop antenna** using **AN-SOF software**. Circular loops are a common antenna type, and their analysis requires **curved segments** within the simulation environment. The guide will detail the configuration process, including defining the loop geometry, setting up the frequency sweep, incorporating a voltage source, and analyzing the key parameters like radiation pattern and input resistance. This guide is valuable for RF engineers, ham radio enthusiasts, students, and antenna design professionals seeking to utilize AN-SOF for circular loop antenna simulations.

1. Specifying the Simulation Setup

This section outlines the initial setup steps required to model a circular loop antenna in AN-SOF. We'll configure a frequency sweep to analyze the antenna's behavior across a specified range.

Frequency Sweep:

1. Navigate to the **Setup** tab and select **Sweep** within the **Frequency** panel.
2. Choose **Lin** for a linear frequency sweep. This allows for evenly spaced data points across the desired range.
3. Define the sweep parameters:
   - **Start frequency:** 3 MHz
   - **Step:** 1 MHz (adjust as needed based on desired resolution)
   - **Stop frequency:** 30 MHz

These settings establish a linear sweep from 3 MHz to 30 MHz with 1 MHz increments between each data point (as shown in Fig. 1).

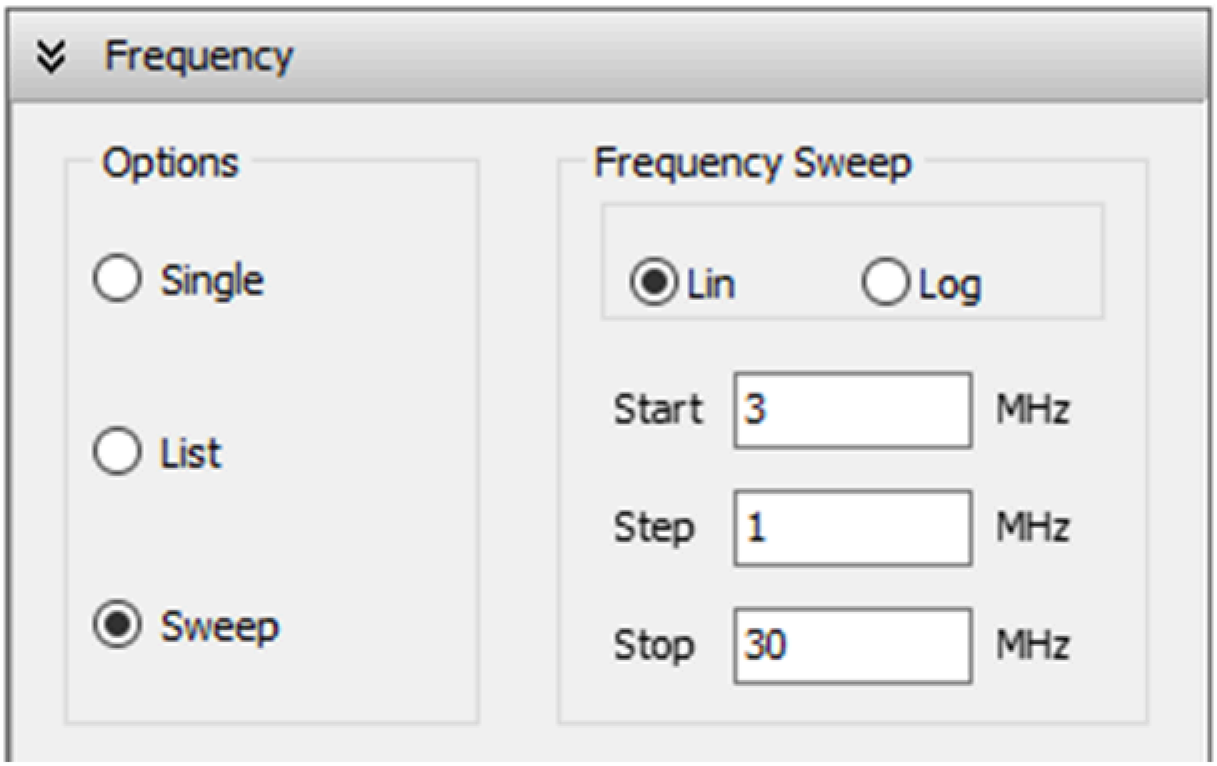


Fig. 1: Configuring the Frequency Sweep in AN-SOF.

Additional Settings:

- **Environment:** Ensure that **None** is selected in the **Ground Plane** box within the **Environment** panel. This removes any ground plane influence from the simulation, which might not be relevant for a free-space loop antenna.
- **Excitation:** In the **Excitation** panel, verify that **Discrete Sources** is selected. This indicates that we'll define a lumped source (voltage or current) to excite the antenna later in the modeling process.

By following these steps, we've established the foundation for our loop antenna simulation by configuring the frequency sweep and essential simulation settings in AN-SOF. The next section will delve into defining the geometry of the circular loop itself.

2. Defining the Circular Loop Geometry

This section focuses on creating the circular loop geometry within the AN-SOF workspace:

1. **Access the Draw Menu:** Navigate to the **Workspace** tab. Right-click on an empty area within the workspace and select **Circle** from the **pop-up menu**.
2. **Specify Loop Parameters:** The Draw dialog box for the circle will appear (Fig. 2). Define the following parameters for your loop antenna **using the provided tabs**:
   - **Center:** (Cx, Cy, Cz) = (0, 0, 0) (**Circle tab**)
   - **Radius:** 0.5 meters (**Circle tab**)
   - **Segments:** 8 (**Attributes tab**)
   - **Cross-section type:** Circular (**Attributes tab**)
   - **Cross-section radius:** 5 millimeters (**Attributes tab**)

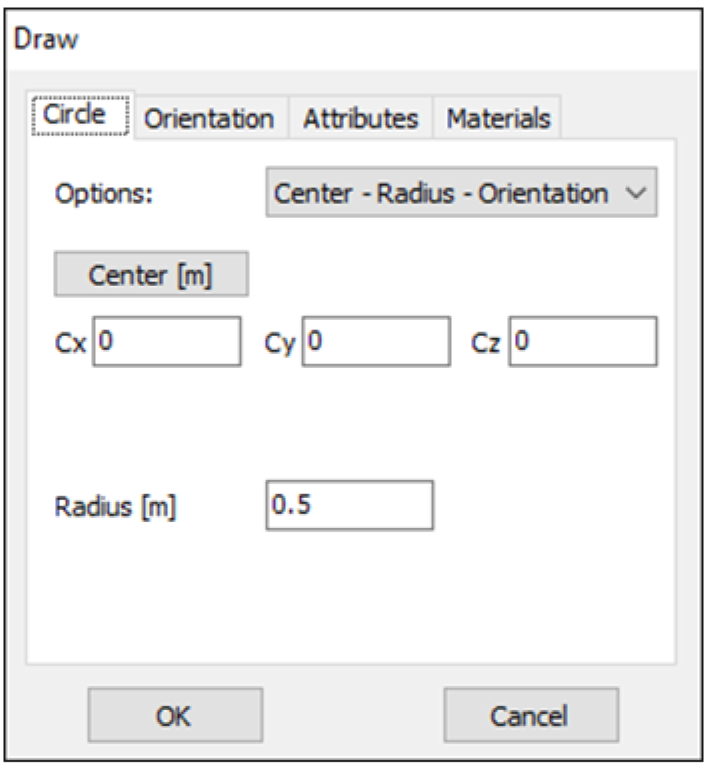


Fig. 2(a): Setting loop dimensions in AN-SOF Draw dialog (Circle tab).

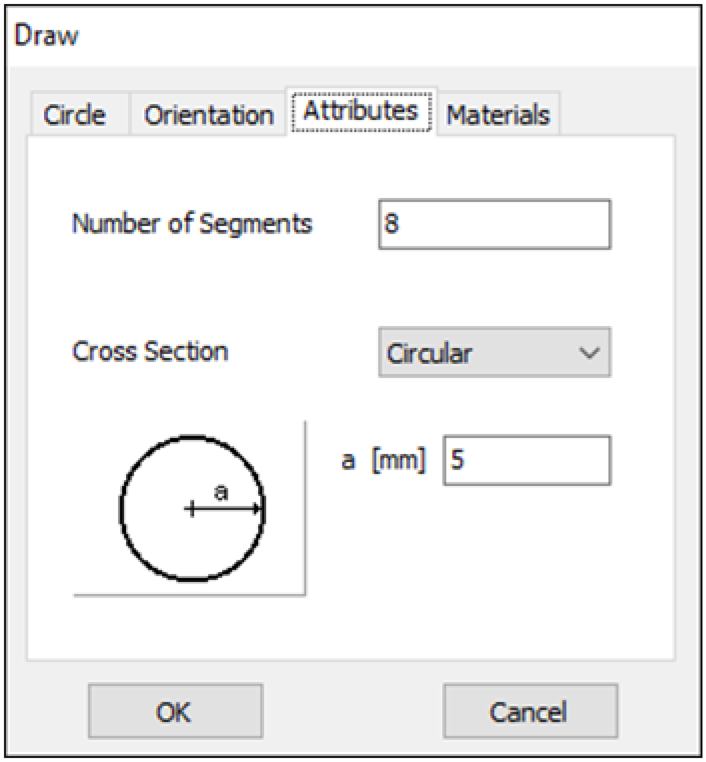


Fig. 2(b): Defining loop segmentation and wire cross-section in AN-SOF Draw dialog (Attributes tab).

[Download Model](#)

Segment Selection: The number of segments used to discretize the loop circumference is crucial for accurate simulation results. While 8 segments are a reasonable starting point, a convergence study might be necessary to ensure sufficient accuracy, especially for electrically large loops. As a rule of thumb, aim for 10-20 segments per wavelength at the highest frequency of interest.

Electrical Size Considerations: It's important to consider the loop's electrical size relative to the wavelength. At 30 MHz (the highest frequency in your sweep), the wavelength (λ) is indeed 10 meters, and the loop's circumference (0.314λ) is close to one-third of a wavelength. This suggests the loop might not be electrically small at the high end of the frequency range. This characteristic will affect the antenna input impedance and radiation pattern.

Assigning the Excitation Source:

1. **Right-click** on the circular loop within the AN-SOF workspace.
2. From the pop-up menu, select **Source/Load**.
3. Choose to add a **voltage source** and position it at the **first segment** of the loop.

For detailed instructions on source placement and parameter definition, refer to the AN-SOF documentation's '[Adding Sources](#)' section.

3. Running the Simulation and Analyzing Results

This section guides you through initiating the simulation process and analyzing the obtained results in AN-SOF:

1. **Run Simulation:** Locate the **Run Currents and Far-Field (F11)** button on the toolbar and click it. This initiates the simulation, calculating the current distribution on the loop and its far-field radiation pattern across the defined frequency sweep.
2. **Visualizing the Radiation Pattern:** Once the simulation is complete, click the **Far-Field 3D Plot** button on the toolbar. This will display the radiation pattern of the loop antenna in a 3D format (AN-3D Pattern application similar to Fig. 3).
3. **Frequency-Dependent Analysis:** The AN-3D Pattern toolbar offers functionalities to explore the radiation pattern's behavior at different frequencies within the sweep range.
    - **Frequency selection dropdown menu:** This menu allows you to directly choose a specific frequency point to view its corresponding radiation pattern.
    - **Frequency navigation buttons:** Utilize the up and down arrow buttons on the toolbar to navigate through the calculated frequencies and observe the dynamic changes in the radiation pattern. As expected for a circular loop antenna, the pattern should exhibit a doughnut-like shape at lower frequencies.
4. **Input Resistance Analysis:** Navigate to the **Results tab** within AN-SOF. Here, you should observe a very low input resistance value, likely around 0.000195 Ohm at 3 MHz.
5. **Comparison with Theoretical Radiation Resistance:** The well-known formula for the radiation resistance, R_r, of an electrically small loop antenna is $R_r = 31{,}200 (A/\lambda^2)^2$. Applying this formula with the loop's area (A) and the wavelength (λ) at 3 MHz, you obtain a theoretical value of $R_r \approx 0.000192$ Ohm. The close agreement between the simulated and theoretical values at 3 MHz demonstrates that the loop behaves according to the small loop antenna model at lower frequencies within the sweep range.

Important Note:

It's important to remember that the formula used for radiation resistance applies to electrically small loops. As mentioned earlier, the chosen loop dimensions might not be electrically small across the entire frequency sweep (especially at 30 MHz). This will lead to deviations between the theoretical and simulated results at higher frequencies.

Figure 3 illustrates the frequency dependence of the loop antenna's 3D radiation pattern. Subfigures (a), (b), and (c) depict the patterns at 3 MHz, 15 MHz, and 30 MHz, respectively.

By following these steps, you've successfully run the simulation, analyzed the radiation pattern, and compared the input resistance with theoretical expectations.

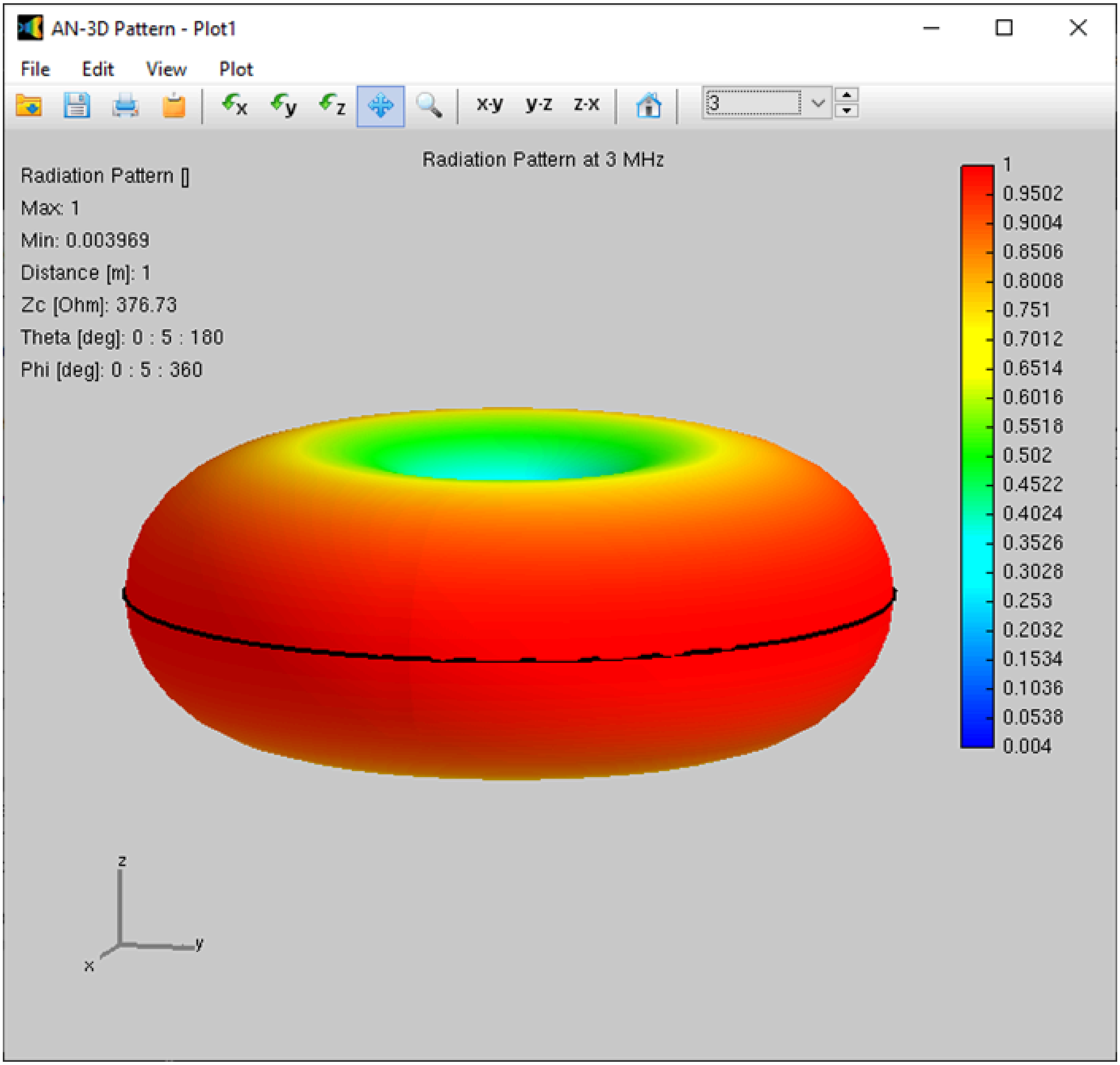


Fig. 3(a): Doughnut-shaped radiation pattern of the loop antenna at 3 MHz.

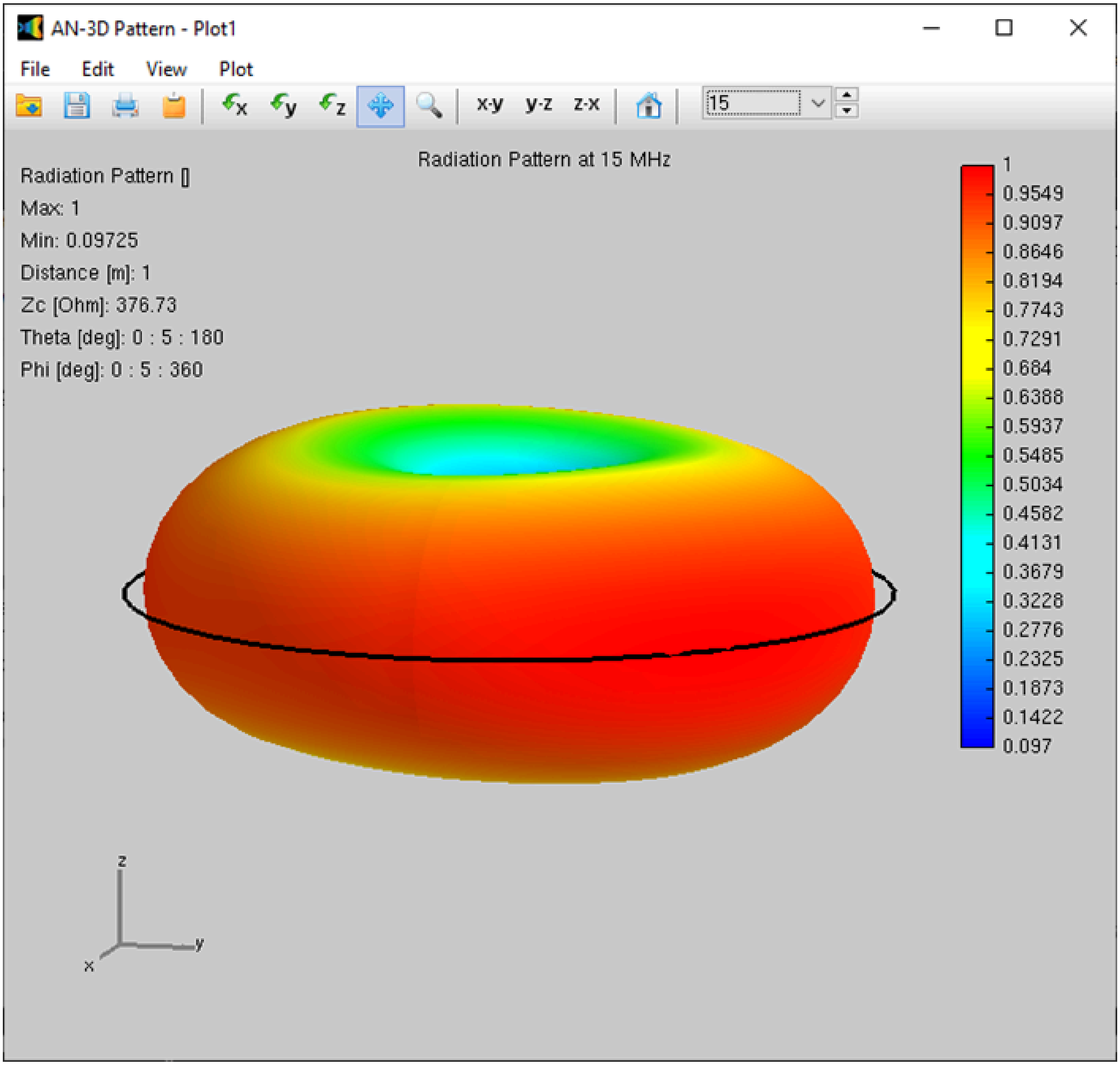


Fig. 3(b): Radiation pattern of the loop antenna at 15 MHz, showing a transition from the low-frequency pattern.

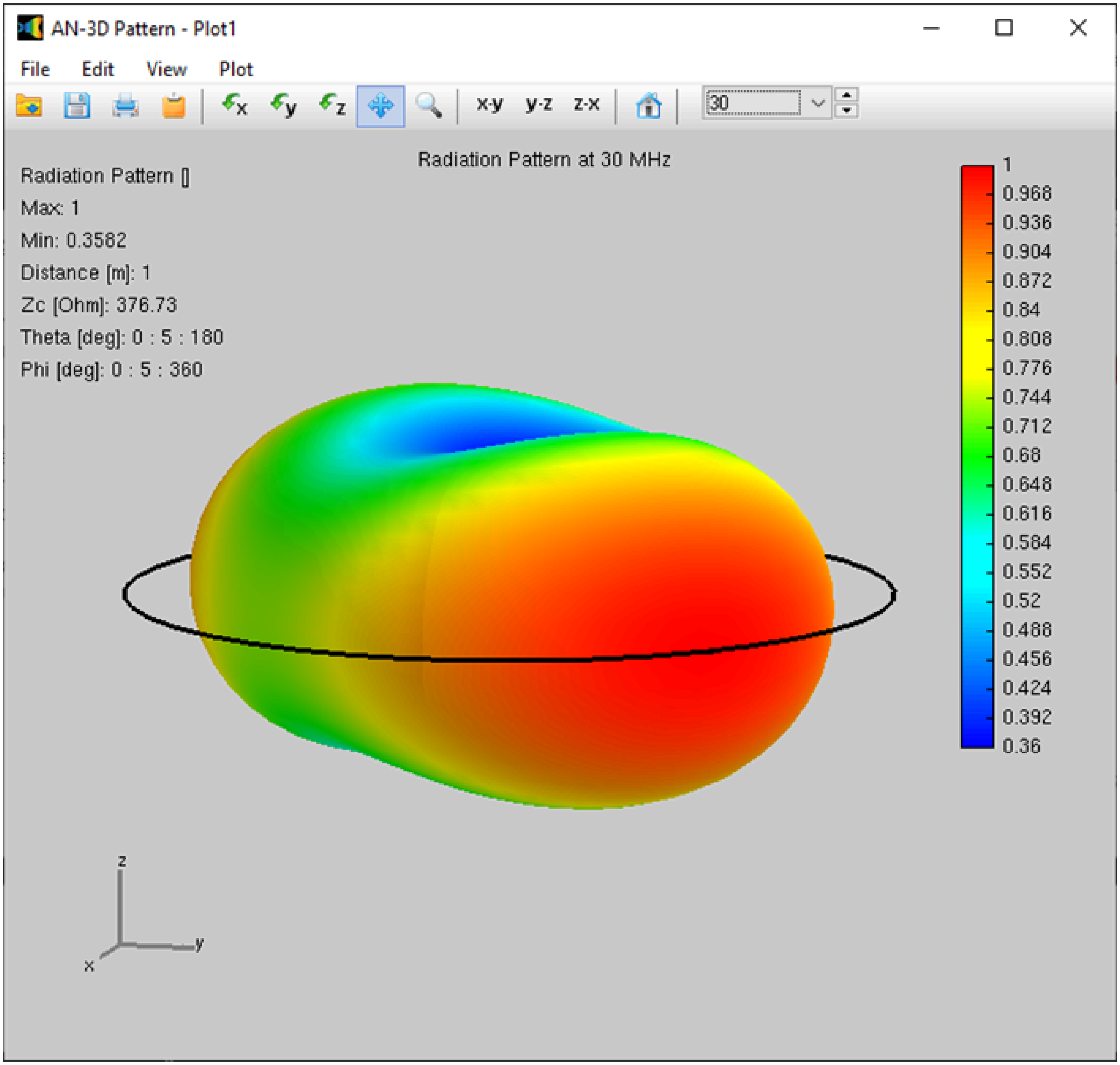


Fig. 3(c): Radiation pattern of the loop antenna at 30 MHz.

See Also:

- [Precision Modeling of Small Loop Antennas: Validating the Conformal Method of Moments (CMoM)]
- [Input Impedance and Directivity of Large Circular Loops: Theory vs. Numerical Simulation]
- [Experimenting with Half-Wave Square Loops: Simulation and Practical Insights]

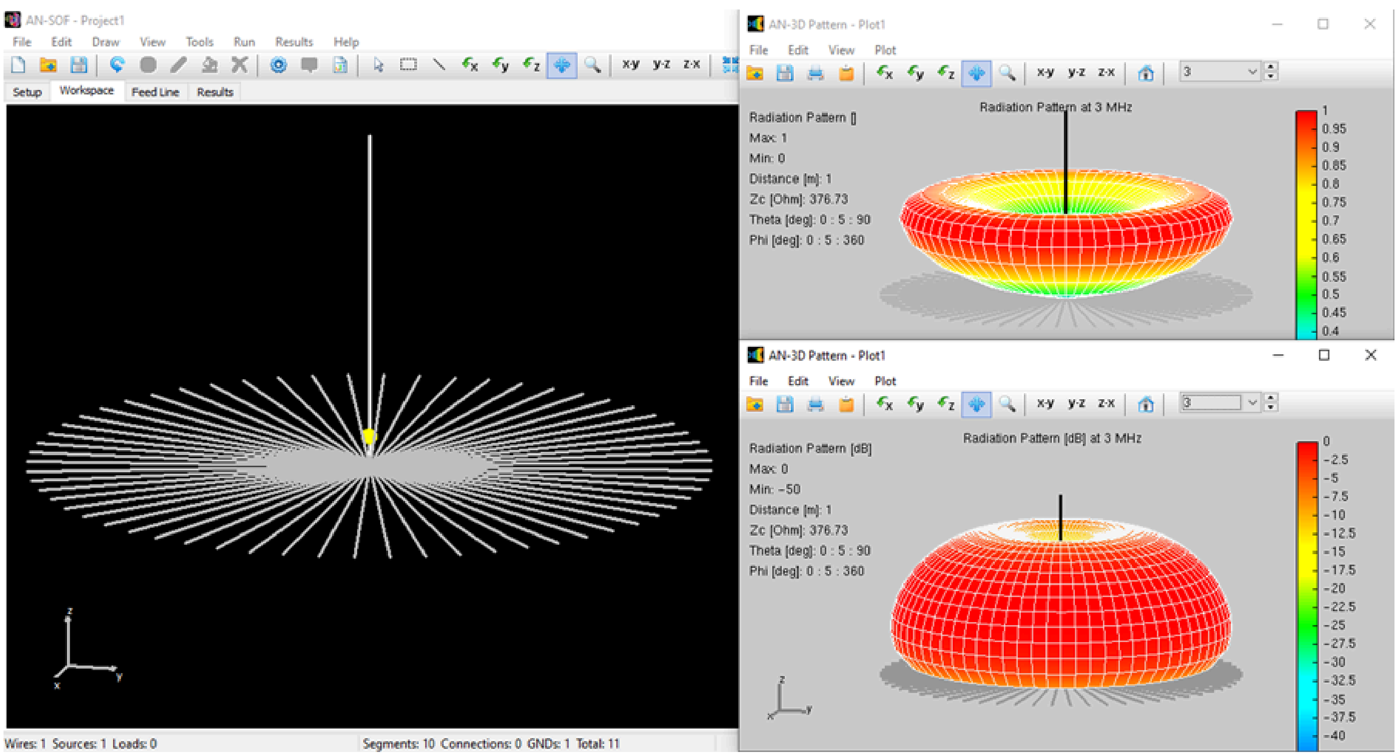
About the Author

Tony Golden
RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I'm a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

[Ask me] | **[Email me]** | **[Follow me]**
[Antennas and Beyond!]
[Get Exclusive Updates]
Monopole Antennas Over Imperfect Ground: Modeling and Analysis with AN-SOF

Explore the design and simulation of monopole antennas over imperfect ground using AN-SOF. Learn how ground conditions impact performance and optimize efficiency for LF/MF broadcasting applications.



Introduction

A **monopole antenna** is a class of radio antenna consisting of **a single radiating element**, typically mounted vertically over **a conductive ground plane**. It is one of the simplest and most widely used antenna designs, particularly in applications where space and simplicity are critical. The monopole can be thought of as **half of a dipole antenna:** by introducing a ground plane, one of the dipole's two radiating elements is effectively replaced by the ground plane's mirror image, creating a virtual “second leg.” This results in a structure that is electrically equivalent to a dipole but requires only half the physical height, making monopoles particularly advantageous for low-frequency and compact installations.

Monopole antennas are commonly used in a variety of applications, including AM/FM broadcasting, LF (Low Frequency) and MF (Medium Frequency) communication systems, mobile and base station antennas, and amateur radio setups. Their omnidirectional radiation pattern in the horizontal plane makes them ideal for broadcasting and communication over wide areas. However, their performance is highly dependent on the quality of the ground plane, as losses in the ground can significantly reduce efficiency and gain.In this article, we will simulate a monopole antenna in the form of a **radio mast operating over imperfect ground**, typical of LF and MF broadcasting scenarios. We will explore the impact of ground conditions on antenna performance and demonstrate how to optimize the design for improved efficiency.

Step 1 | Setup

Navigate to the **Setup** tab and set the operating frequency to **3 MHz** in the **Frequency panel**. Next, go to the **Environment** panel > **Ground Plane** box and select the **Real** option, as shown in **Fig. 1**. Choose the **Radial wire ground screen** and **Poor ground** options. Note that the soil conductivity will automatically be set to **0.001 S/m** and the relative permittivity (dielectric constant) to **5**.

Finally, configure the number of radials, their length, and their radius as illustrated in **Fig. 1**. For radio masts, it is common practice to use a constant input power as a reference, such as **1 kW**. Proceed to the **Excitation panel**, select **Discrete Sources**, then choose **Set Input Power** and enter **1,000 W**, as shown in **Fig. 2**.

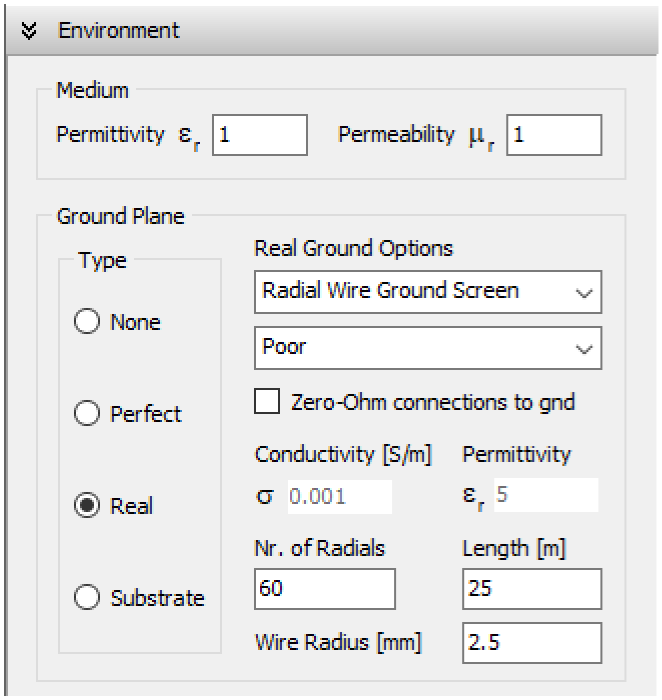


Fig. 1: Configuring a radial wire ground screen in the Environment panel of AN-SOF.

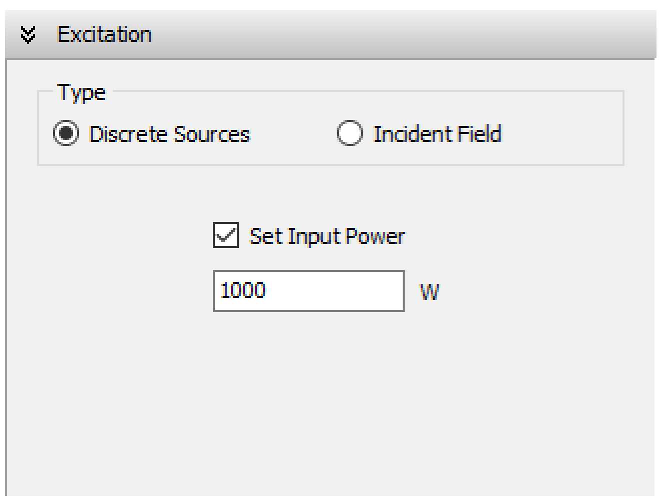


Fig. 2: Configuring discrete sources in the Excitation panel with an input power of 1,000 W.

Step 2 | Draw

Right-click on the workspace and select [Line](#) from the [pop-up menu](#). Specify a vertical wire with a height of **25 meters** (equivalent to 1/4 wavelength at 3 MHz) and a triangular cross-section, as illustrated in **Fig. 3**. Although the recommended minimum number of segments is **3**, we will divide the wire into **10 segments** to achieve higher resolution in the current distribution. Note that the wire will automatically be connected to the ground at the origin **(0, 0, 0)**.

Next, right-click on the wire and select the **Source/Load** command from the pop-up menu. Place a voltage source on the first segment to ensure the source is connected to the base of the mast. For further details, refer to the **Adding Sources** section.

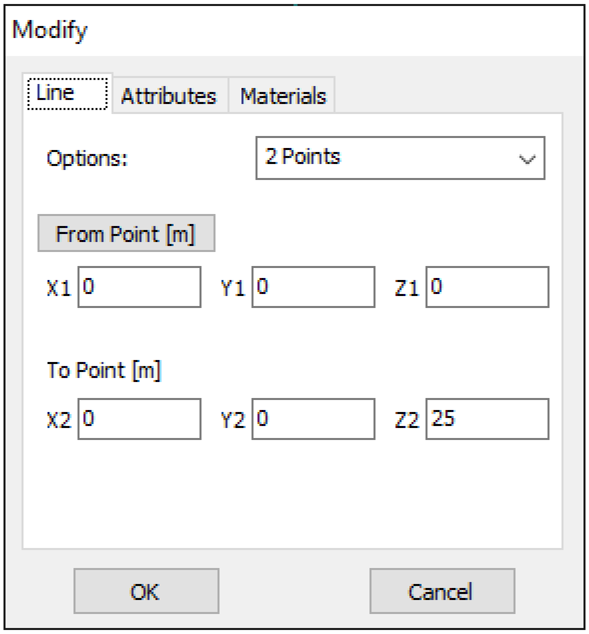


Fig. 3(a): Defining a vertical wire in the Line configuration page.

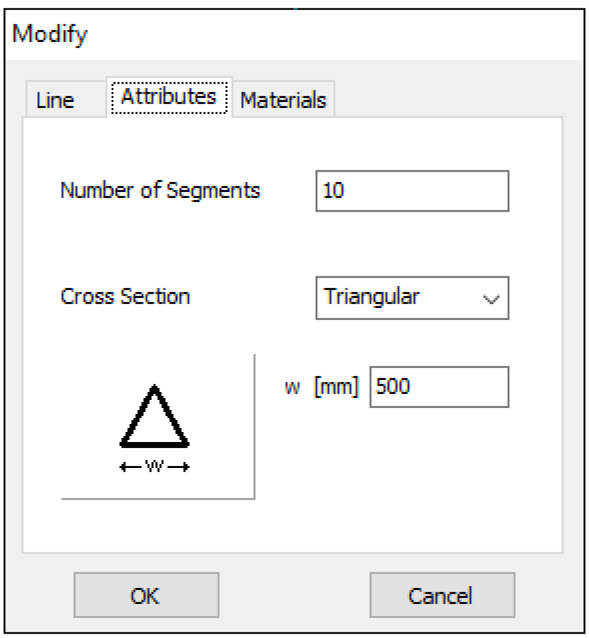


Fig. 3(b): Configuring a triangular cross-section in the Attributes page.

Step 3 | Run

Click on the **Run Currents and Far-Field (F11)** button located on the toolbar. Once the calculations are complete, click on the **Far-Field 3D Plot** button on the toolbar to display the radiation pattern. In the **AN-3D Pattern** window, select **Radiation Pattern** from the **Plot** menu to generate the normalized radiation pattern (dimensionless). Then, choose the **Radiation Pattern [dB]** option to view the pattern in decibel scale. Note that the far field exhibits a null on the **xy-plane** due to losses in the ground plane, as shown in **Fig. 4**.

The antenna efficiency is defined as the ratio of radiated power to input power. Navigate to the **Results tab** to view key parameters such as input impedance, VSWR, directivity, gain, and efficiency, as illustrated in **Fig. 5**. You will observe that the efficiency is relatively low, which consequently results in low gain, as a significant portion of the input power is dissipated in the ground. In this example, we have intentionally chosen a **Poor soil** condition. To improve antenna efficiency, experiment with different soil types, increase the number of radial wires, and adjust their lengths.

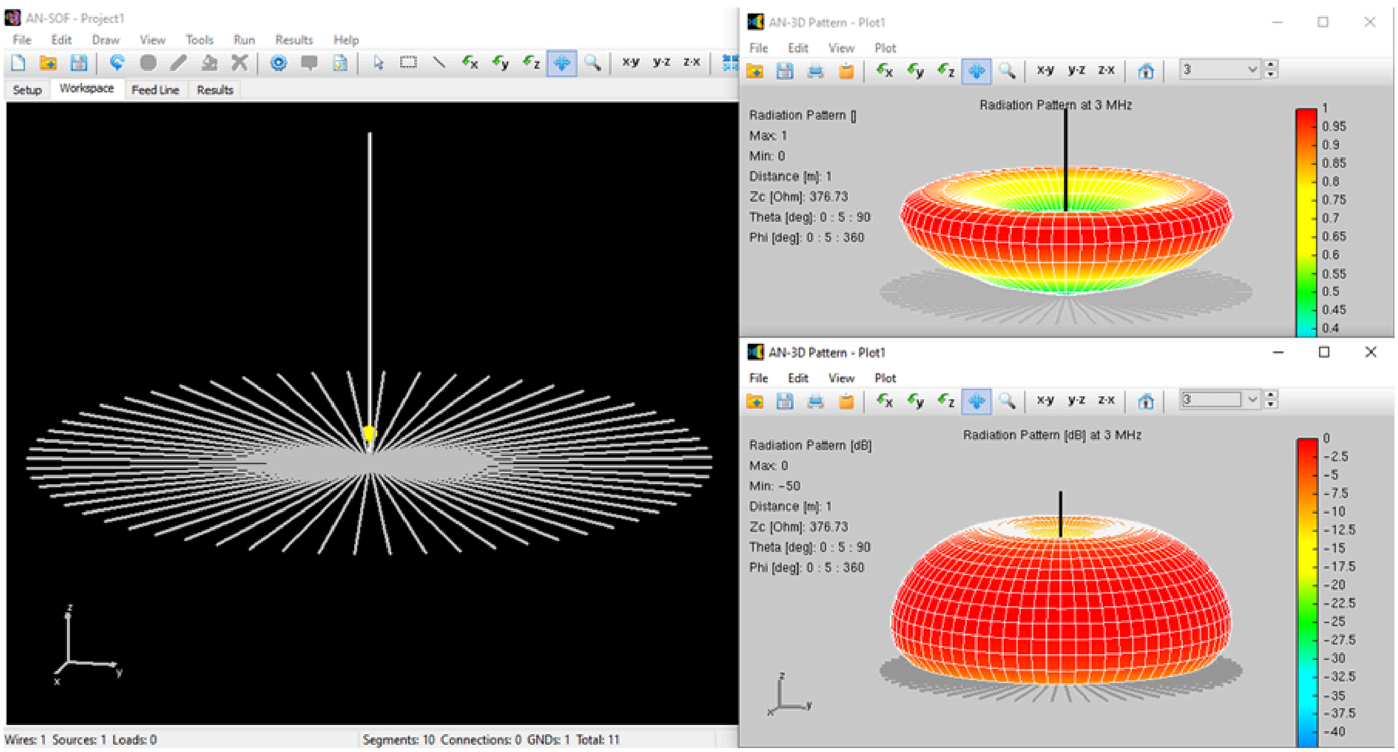


Fig. 4: Quarter-wave monopole antenna model (left) with its 3D radiation pattern: linear scale (top right) and decibel scale (bottom right), simulated using AN-SOF.

Download Model

Setup | Workspace | Tuner | Feeder | Results | Plots | Models

No.	Freq.	Rin	Xin	VSWR	S11	Dir.	Gain	Eff.
---	MHz	Ohm	Ohm	---	dB	dBi	dBi	%
1	3	52.2053	30.0022	1.78716	-10.982	5.19411	-2.71064	16.2004

Fig. 5: Results tab displaying input impedance and radiation parameters for the quarter-wave monopole antenna over a radial wire ground screen.

Conclusion

Simulating monopole antennas using the **AN-SOF Antenna Simulator** offers significant advantages for engineers and designers. By leveraging its advanced modeling capabilities, users can accurately predict antenna performance, optimize designs, and evaluate the impact of various ground conditions without the need for costly and time-consuming physical prototypes.

The ability to analyze parameters such as radiation patterns, input impedance, gain, and efficiency provides invaluable insights, enabling the development of high-performance monopole antennas tailored to specific applications.

Whether you're designing for LF/MF broadcasting, mobile communications, or amateur radio, AN-SOF empowers you to explore design trade-offs and achieve optimal results with precision and efficiency.

See Also:

Modeling a Center-Fed Cylindrical Antenna with AN-SOF

An Efficient Approach to Simulating Radiating Towers for Broadcasting Applications

About the Author

Tony Golden

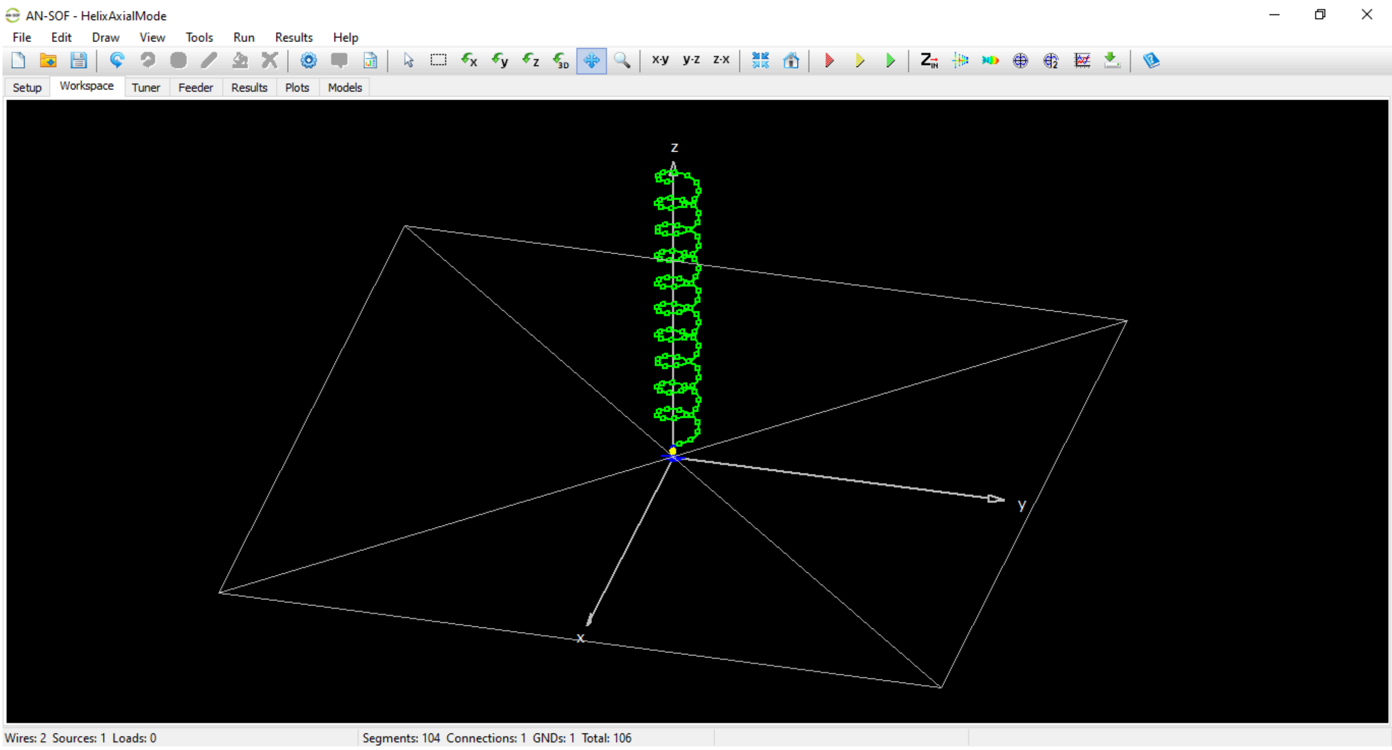
RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I'm a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

Ask me | **Email me** | **Follow me**

Antennas and Beyond!
Get Exclusive Updates
Modeling Helix Antennas in Axial Radiation Mode Using AN-SOF

Perfect for Beginners: Quick Guide to Helix Antenna Simulation.
Master axial-mode helix design in AN-SOF with this easy step-by-step tutorial. Learn ground plane setup, helix creation, and radiation pattern analysis. Start modeling professional antennas today!



Helix Antenna in Axial Mode

The helix antenna demonstrates the importance of **curved segments for accurate geometry representation**. When the helix length approaches or exceeds the operating wavelength, it operates in *axial mode* – characterized by endfire radiation along its axis. This requires a **ground plane reflector** for proper operation.

Step 1: Setup

1. **Frequency Setup**:
   - Navigate to **Setup tab > Frequency panel**
   - Set operating frequency: **100 MHz**
2. **Ground Plane Configuration**:
   - Go to **Environment panel > Ground Plane box**
   - Select: **Perfect**
   - Set position: **Z = 0** (xy-plane) (**Fig. 1**)
3. **Excitation**:

   Ensure **Discrete Sources** is selected in **Excitation panel**

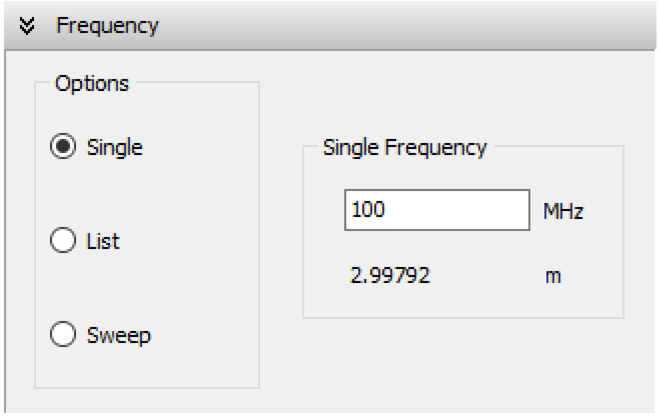


Fig. 1(a): Setting the operating frequency for the helix antenna.

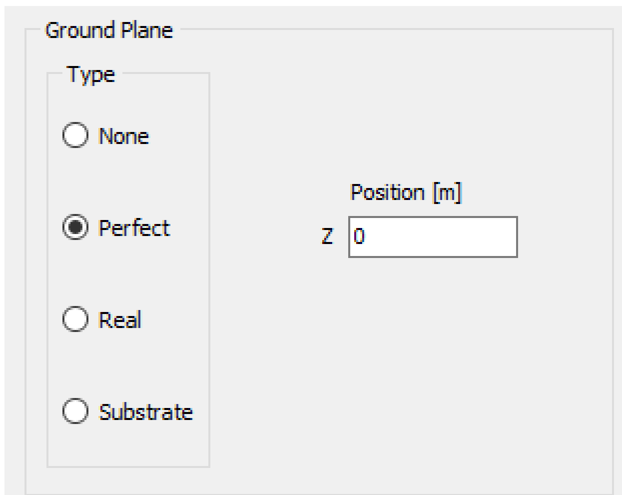


Fig. 1(b): Setting the ground plane for the helix antenna.

Step 2: Drawing the Structure

1. **Helix Creation**:
   - Right-click on the workspace > Select **Helix** from **pop-up menu**
   - *Draw Helix* dialog appears (**Fig. 2**)
   - Start point: **(0, 0, 0.3) m** (above ground plane)
   - Orientation: **Z-axis**
2. **Axial Mode Dimensions**:
   - Set parameters per antenna textbooks:
     - Radius
     - Pitch (turn spacing)
     - Number of turns *(shown in Fig. 2)*
   - **Attributes tab**:
     - Segments: **103** (recommended)
     - Cross-section: **Circular**, radius = **3 mm**

3. **Ground Connection**:
   - Right-click helix > **Start Point to GND**
   - *Draw Line* dialog auto-populates connection points (**Fig. 3**)
   - Set:
     - Segments: **2**
     - Radius: **3 mm**
4. **Source Placement**:
   - Right-click vertical wire > **Source/Load/TL**
   - Connect voltage source to segment nearest ground plane (Refer to: **Adding Sources** guide)

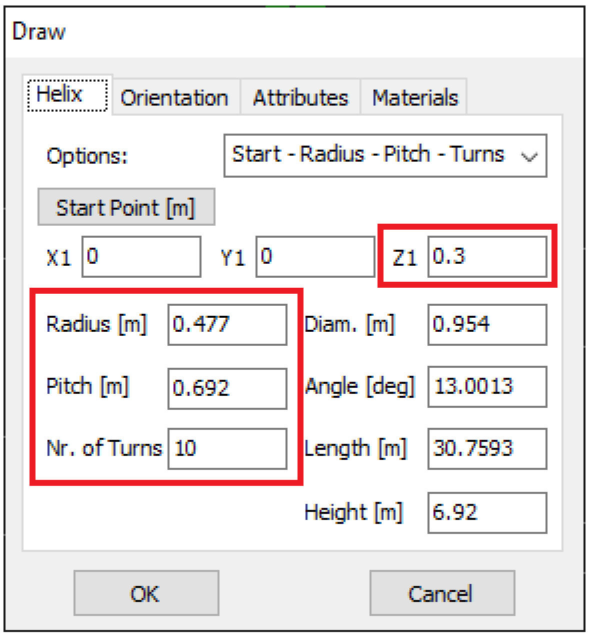


Fig. 2(a): Specifying the helix dimensions.

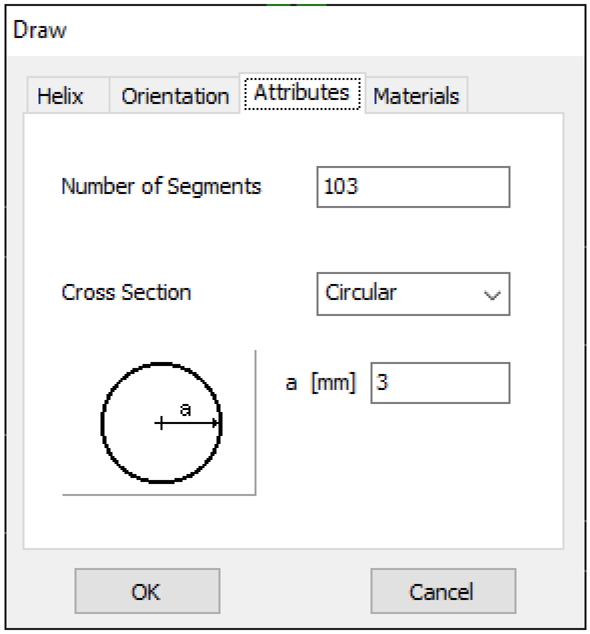


Fig. 2(b): Specifying the helix segmentation and cross-section.

[Download Model]

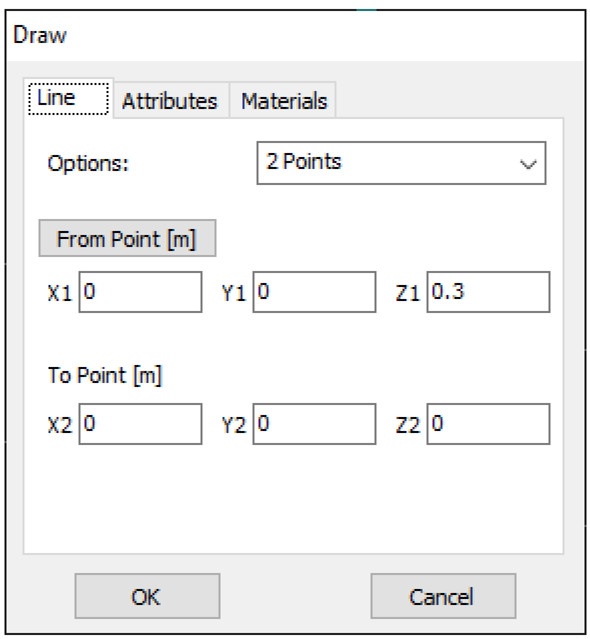


Fig. 3(a): Specifying the vertical wire that connects the helix to the ground plane.

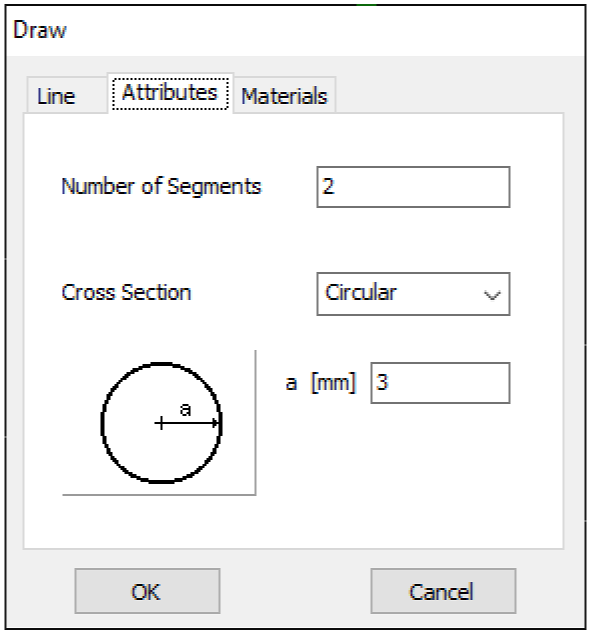


Fig. 3(b): Specifying the segments and cross-section for the vertical wire.

Step 3: Simulation & Analysis

1. **Run Simulation**:
   - Click **Run Currents and Far-Field (F10)**
   - View 3D radiation pattern via **Far-Field 3D Plot** button (**Fig. 4a**)
   - Observe main lobe along helix axis (axial mode characteristic)
2. **Polarization Analysis**:
   - In **AN-3D Pattern Plot**:

     Compare *E-right* vs. *E-left* components (**Figs. 4b, 4c**)
   - For accurate comparison:

     Set matching scale maxima (*Edit* > *Preferences*)
3. **Left-Handed Variant**:
   - Create by specifying **negative turn count**
   - Re-run simulation and compare polarization components

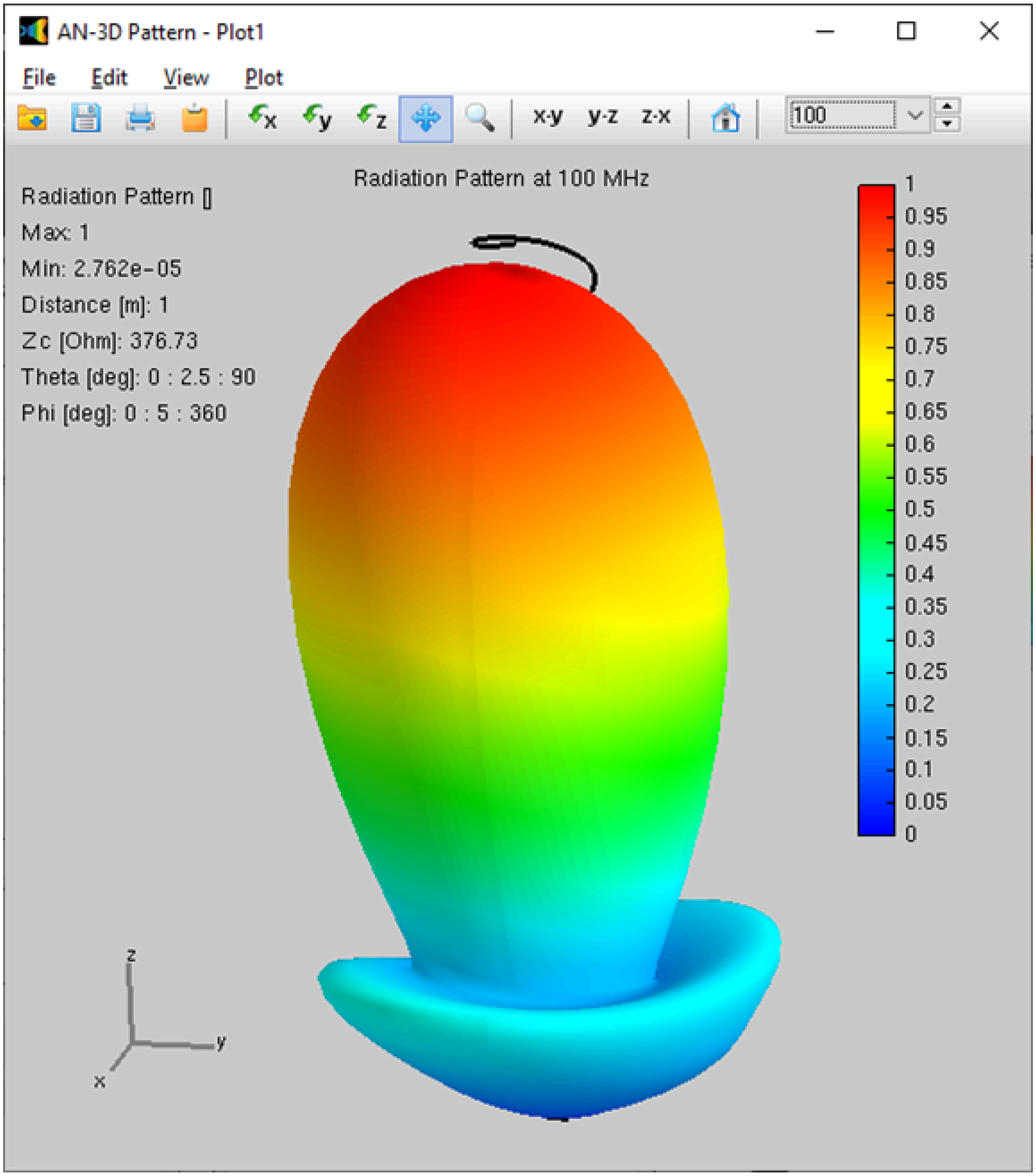


Fig. 4(a): Normalized radiation pattern of the helix.

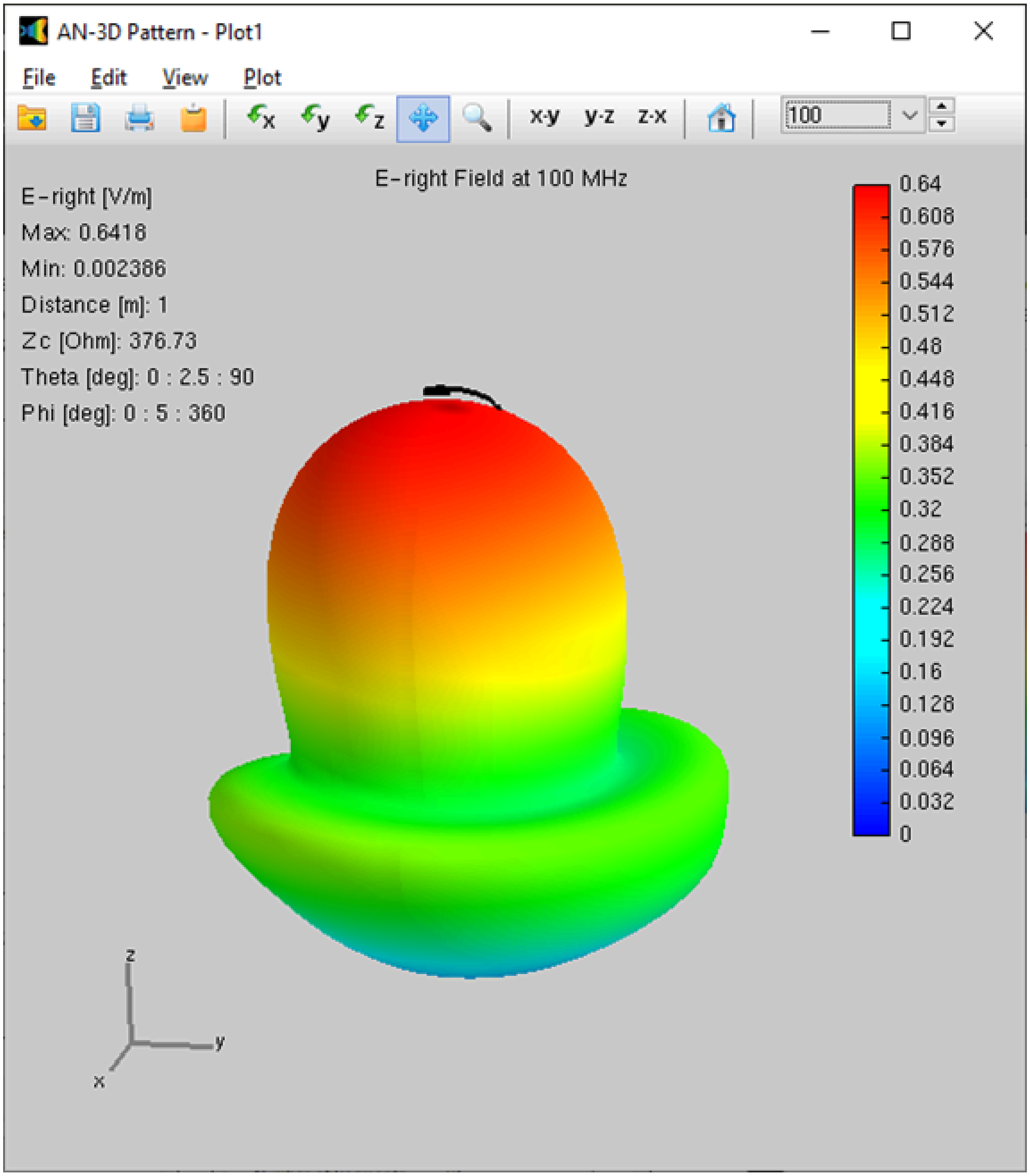


Fig. 4(b): Right-handed circularly polarized component of the far-field.

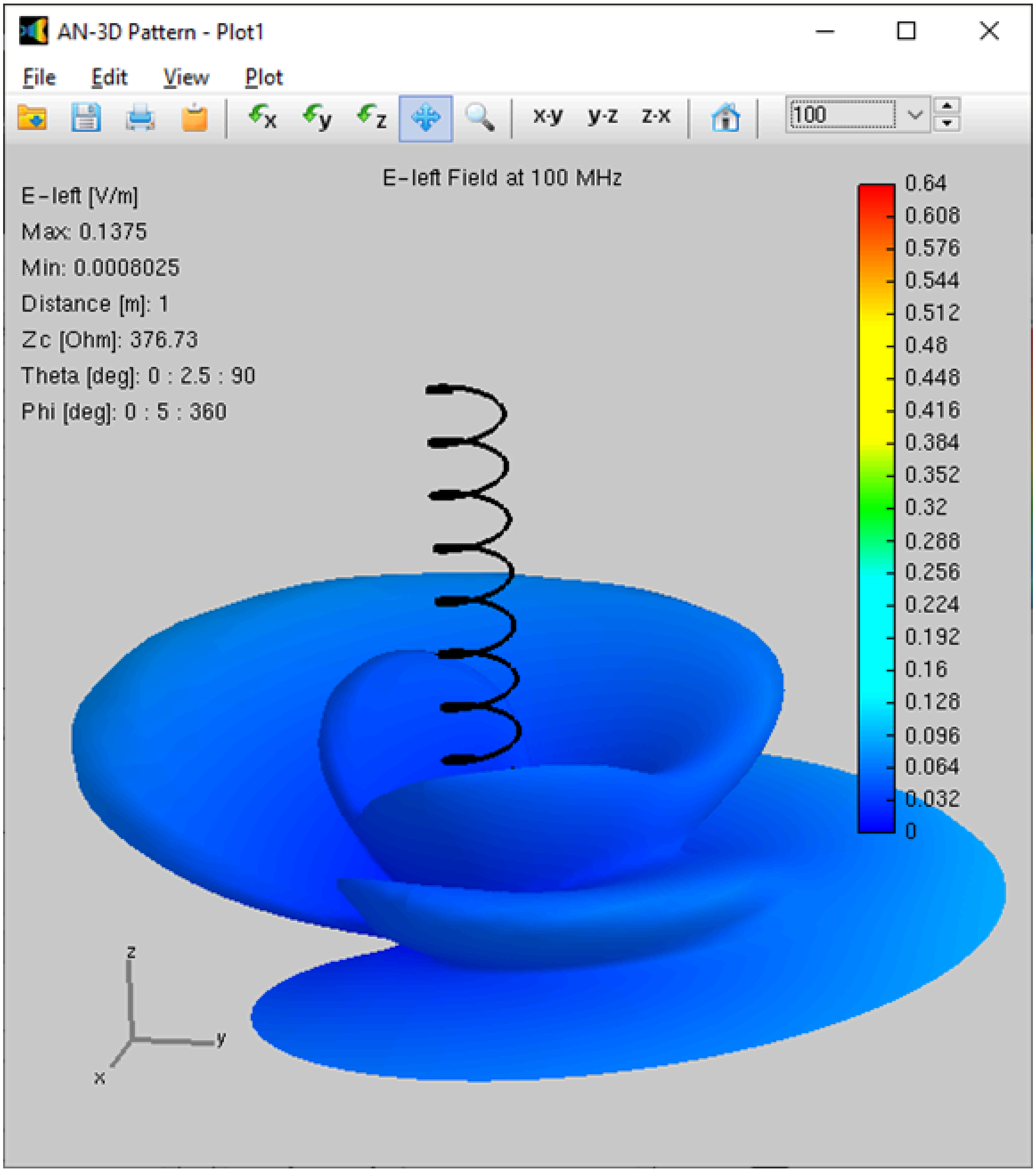


Fig. 4(c): Left-handed circularly polarized component of the far-field.

Key Features Demonstrated

- Curved segment modeling for helical structures
- **Axial mode** radiation characteristics
- Circular polarization analysis
- **Ground plane** integration
- **Parametric comparison** (right vs. left-handed)

The complete simulation showcases AN-SOF's capability to model complex antenna behavior with precise geometrical control.

See Also:

- **Efficient NOAA Satellite Signal Reception with the Quadrifilar Helix Antenna**
- **DIY Helix High Gain Directional Antenna: From Simulation to 3D Printing**

About the Author

Tony Golden

RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I'm a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

Ask me | **Email me** | **Follow me**

Antennas and Beyond!
Get Exclusive Updates
Step-by-Step: Modeling Basic Yagi-Uda Arrays for Beginners

Master Yagi-Uda simulation in AN-SOF! This quick guide walks you through modeling a 3-element array (reflector, driven element, director). Analyze radiation patterns with professional results.

Introduction to Yagi-Uda Antennas

The Yagi-Uda antenna, commonly called simply "Yagi," is a directional antenna array developed in 1926 by Japanese researchers Hidetsugu Yagi and Shintaro Uda. This elegant design consists of three key elements:

1. A **driven element** (typically dipole)
2. **Reflectors** (usually 1-2 elements)
3. **Directors** (multiple elements)

Through careful spacing of these elements, Yagis achieve high directivity and gain in one direction while being relatively simple to construct. They revolutionized radio communication and remain popular today for applications ranging from TV reception to amateur radio.

Yagi-Uda Simulation Basics

Now that you've mastered **cylindrical antenna** basics, let's progress to antenna arrays. This guide walks you through simulating a classic **3-element Yagi-Uda design** (**Fig. 1**) featuring:

Director (front element)

Reflector (rear element)

Driven element (center dipole)

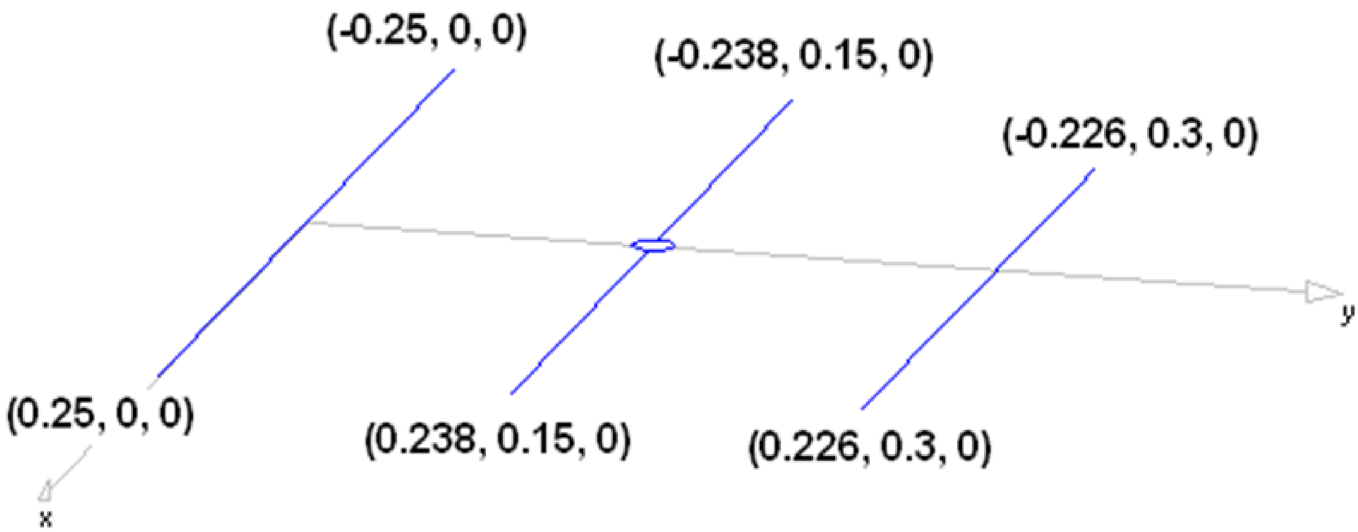


Fig. 1: Yagi-Uda array geometry definition (coordinates in meters).

Download Model

Step 1: Simulation Setup

1. **Frequency Configuration**:
   - **Setup tabsheet > Frequency panel**
   - Set: **300 MHz** (UHF range ideal for Yagis)
2. **Environment Settings**:
   - **Environment panel > Ground Plane**: Select **None** (standalone array)
   - **Excitation panel**: Confirm **Discrete Sources** is active

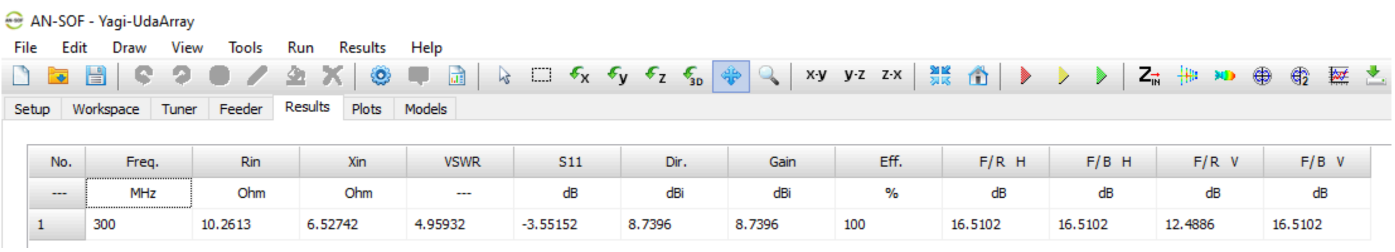
Step 2: Building the Array

1. **Element Construction**:
   - Draw each wire individually (as in **cylindrical antenna tutorial**)
   - Use coordinates from **Fig. 1** for precise spacing
2. **Segment Configuration**:
   Uniformly set for all elements:
     - Segments: **15** (balanced accuracy/speed)
     - Radius: **5 mm** (typical for UHF)

3. **Source Attachment**:
    - Right-click **driven element** > **[Source/Load/TL]**
    - **[Connect voltage source]** to **center segment**
      (Pro Tip: Middle placement ensures symmetrical excitation)

Step 3: Running & Analyzing

1. **Execute Simulation**:
    - Click **Run Currents and Far-Field (F10)**
    - Observe **8.7 dBi peak gain** in **[Results tab]** (**Fig. 2**)
2. **Radiation Pattern**:
    Click **[Far-Field 3D Plot]** to visualize:
    - Forward-directive beam (**Fig. 3**)
    - Characteristic side/null patterns



No.	Freq.	Rin	Xin	VSWR	S11	Dir.	Gain	Eff.	F/R H	F/B H	F/R V	F/B V
---	MHz	Ohm	Ohm	---	dB	dBi	dBi	%	dB	dB	dB	dB
1	300	10.2613	6.52742	4.95932	-3.55152	8.7396	8.7396	100	16.5102	16.5102	12.4886	16.5102

Fig. 2: Results tab showing 8.7 dBi peak gain for the Yagi-Uda array.

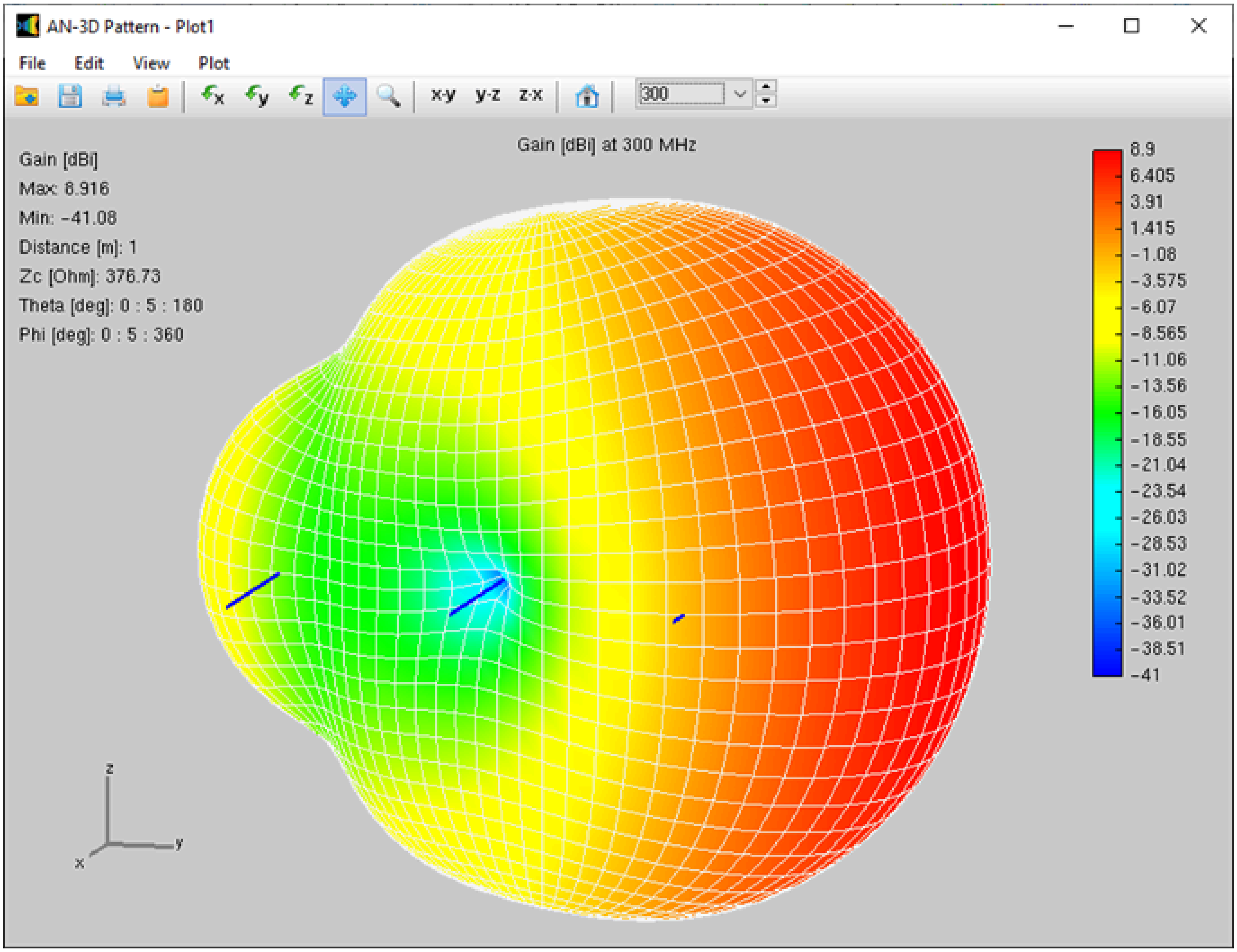


Fig. 3: Yagi-Uda array gain pattern (dBi) at 300 MHz.

Why This Matters

- Teaches fundamental array principles
- Demonstrates directional gain enhancement
- Provides baseline for more complex designs
  (Try modifying element spacing/numbers to see performance changes!)

See Also:

[Front-to-Rear and Front-to-Back Ratios: Applying Key Antenna Directivity Metrics]

About the Author

Tony Golden

RF ENGINEER & PHYSICS PH.D. With 25+ years in Computational Electromagnetics, I'm a passionate researcher focused on antenna modeling and design. As Founder of Golden Engineering LLC, I develop accessible, high-performance simulation tools that help RF engineers optimize their designs, educators teach complex concepts, and hobbyists bring antenna projects to life.

Have a question?

Ask me | **Email me** | **Follow me**
Antennas and Beyond!
Get Exclusive Updates

Background Theory

The AN-SOF Calculation Engine

The AN-SOF calculation engine is a high-performance electromagnetic solver written in C++ using double-precision arithmetic. It has been specifically developed to overcome **the limitations of traditional wire-antenna modeling** and to provide superior accuracy for complex metallic structures.

Conformal Method of Moments (CMoM)

At its core, the engine solves the **Electric Field Integral Equation (EFIE)** in the frequency domain. While standard solvers utilize the traditional Method of Moments (MoM) with straight-wire approximations, AN-SOF employs the **Conformal Method of Moments (CMoM)**.

In this approach, **curved wires** are modeled using **conformal segments** that exactly follow the physical contour of the structure. Traditional linear approximations are often inefficient, requiring a high number of unknowns and excessive computer memory to represent a curve. By using curved segments, AN-SOF significantly reduces the number of unknown currents, simulation time, and memory overhead, allowing for the analysis of much larger and more complex problems.

Overcoming Legacy MoM Limitations

Many legacy MoM codes, especially those based on the **thin-wire kernel** and linear geometry approximations, suffer from several critical drawbacks that AN-SOF is designed to eliminate:

- **Impedance Divergence:** Traditional codes often produce **divergent input impedances** as segment density increases.
- **Poor Convergence for Curved Structures:** Helices, loops, and spirals are often poorly represented by straight-wire segments, leading to inaccurate results.
- **Proximity Singularities:** Numerical errors frequently occur when two parallel wires are in close proximity or when a wire is near a **lossy ground plane**.

Exact Kernel Formulation

By combining CMoM with an **exact kernel formulation**, AN-SOF removes the reliance on the "thin-wire approximation." This provides several distinct advantages for the user:

1. **Enhanced Accuracy:** The precision of **current distribution** and **far-field calculations** is significantly increased.
2. **Resource Efficiency:** Lower memory usage and **faster computation times** facilitate the modeling of larger designs on standard hardware.
3. **Broad Frequency Range:** The engine maintains stability across an extreme spectrum, from **very low frequencies** (60 Hz power circuits) to **high-frequency microwave applications**.

Electric Field Integral Equation

The foundation of modern antenna simulation lies in solving the **Electric Field Integral Equation (EFIE)**. For metallic structures with ideal conductivity, the EFIE relates the incident electromagnetic wave to the currents induced on the surface of the object.

The Fundamental Equation

In the frequency domain, the EFIE is expressed as follows:

$$\hat{\mathbf{n}}\times\left[j\omega\mu_0\int_S \mathbf{J}_s(\mathbf{r}')G(\mathbf{r},\mathbf{r}')d^2\mathbf{r}'+\frac{\nabla}{j\omega\varepsilon_0}\int_S \nabla'\cdot\mathbf{J}_s(\mathbf{r}')G(\mathbf{r},\mathbf{r}')d^2\mathbf{r}'\right]=\hat{\mathbf{n}}\times\mathbf{E}^{inc}(\mathbf{r}) \quad (1)$$

Where:

- $\mathbf{E}^{inc}$: The incident electric field on the PEC (Perfect Electrical Conductor) surface S.
- $\hat{\mathbf{n}}$: The unit vector normal to the surface at the observation point $\mathbf{r}$.
- $\mathbf{J}_s(\mathbf{r}')$: The unknown surface electric current density at the source point $\mathbf{r}'$.
- G: The free-space Green's function.

The free-space Green's function is given by:

$$G(\mathbf{r},\mathbf{r}')=\frac{e^{-jkR}}{4\pi R} \quad (2)$$

where $R=|\mathbf{r}-\mathbf{r}'|$ is the distance between the source and observation points.

As usual, $k=\omega/c$ is the wavenumber, $\omega=2\pi f$ is the angular frequency, f is the frequency in Hertz (Hz), and $c=1/\sqrt{\mu_0\varepsilon_0}$ is the speed of light in free space. The permeability and permittivity of free space are denoted by μ_0 and ε_0, respectively. The free-space wavelength is defined as $\lambda=c/f$.

The EFIE expresses the boundary condition on a surface S, which is assumed to be a Perfect Electrical Conductor (PEC); specifically, it enforces a **zero tangential electric field** on that surface.

From Surfaces to Wires

When modeling wire antennas, we assume the wire radius (a) is much smaller than the wavelength ($a \ll \lambda$). This allows us to simplify the surface integral into a **line integral** along the axis of the wire. The resulting equation for a curvilinear wire C is:

$$\hat{\mathbf{s}} \cdot \frac{j}{\omega \varepsilon_0} \int_C \left[k^2 I(s') K(s,s') \hat{\mathbf{s}}' + \frac{dI(s')}{ds'} \nabla K(s,s') \right] ds' = \hat{\mathbf{s}} \cdot \mathbf{E}^{inc} \quad (3)$$

where $\hat{\mathbf{s}}$ is the tangential unit vector along the curve C. In this specialized form, $I(s')$ represents the total current flowing along the wire's path C, and $K(s,s')$ is the **Kernel** of the integral equation, which is given by:

$$K(s,s') = \frac{1}{4\pi^2} \int_0^{2\pi} \int_0^{2\pi} G(\mathbf{r},\mathbf{r}') \, d\phi' \, d\phi \quad (4)$$

where ϕ and ϕ' represent the angular positions of the observation and source points, respectively, on the wire's circular cross-section.

By averaging the EFIE over the wire circumference, we obtain the curvilinear EFIE presented in Eq. (3), utilizing the **exact kernel** defined in Eq. (4).

The current distribution $I(s)$ represents the **average value** of the surface current density $\mathbf{J}_s$ in the axial direction, while current in the transverse direction is neglected. This remains a valid assumption provided the wire radius is small relative to the wavelength λ.

Parametric Modeling and the Exact Kernel

Accurate simulation of curved wires—such as helices, spirals, or loops—requires a **parametric description** of the wire's geometry. By defining the **position vector** $\mathbf{r}(s)$ and its **first derivative** (the tangent vector), the solver maintains **exact geometric information** that is lost in traditional straight-wire approximations.

The wire axis C is described by parametric equations, which can be expressed in a Cartesian coordinate system as:

$$\mathbf{r}(s) = \hat{\mathbf{x}} x(s) + \hat{\mathbf{y}} y(s) + \hat{\mathbf{z}} z(s) \quad (5)$$

In this equation, $\mathbf{r}(s)$ points from the origin to any point on the wire, as shown in **Fig. 1**. The parameter s varies over a real interval $s_{min} \leq s \leq s_{max}$.

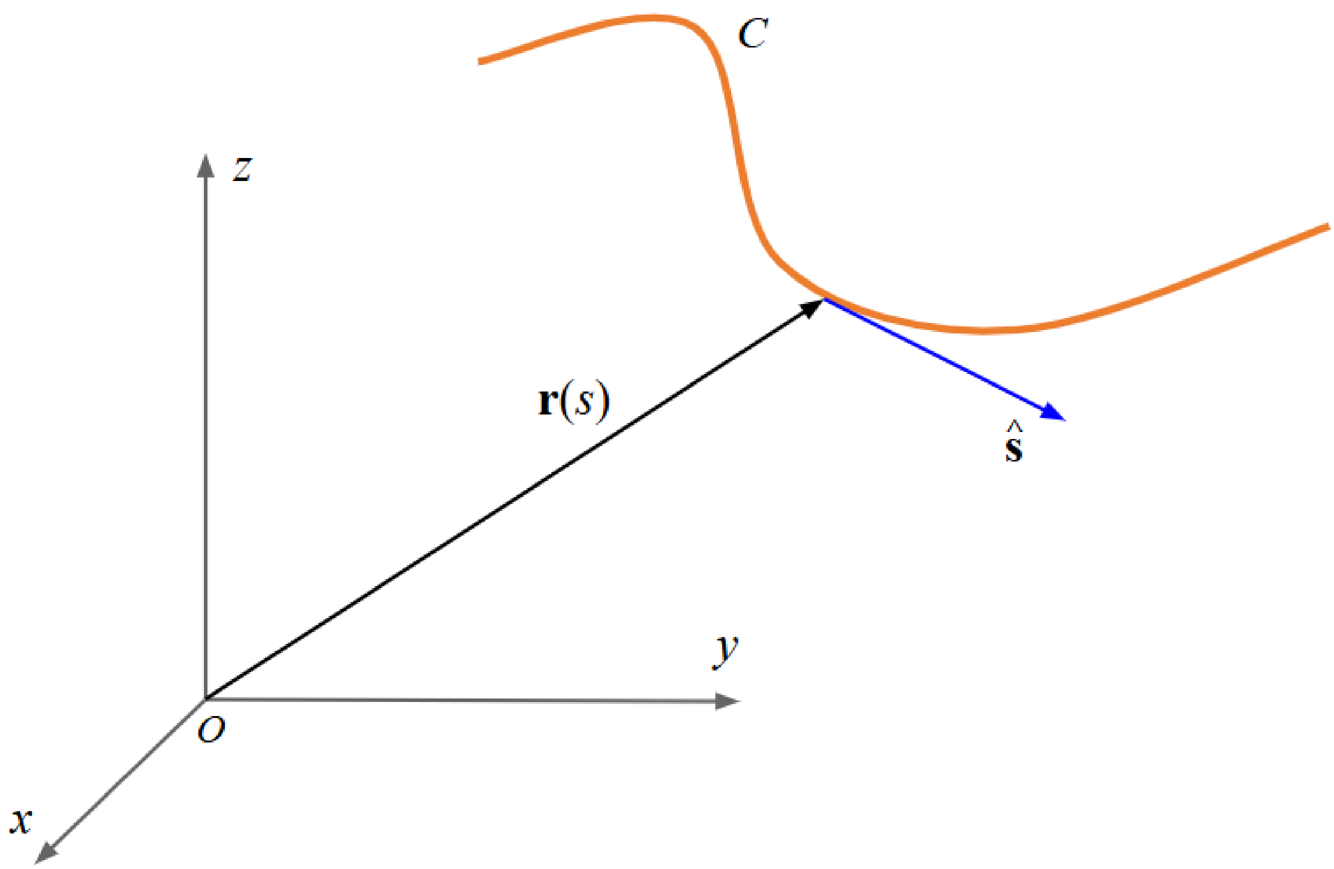


Fig. 1: *Parametric description of a curved wire. The tangent unit vector is obtained from the first derivative of the position vector* $\mathbf{r}(s)$*.*

The tangent unit vector $\hat{\mathbf{s}}$ is obtained from the first derivative of Eq. (5):

$$\frac{d\mathbf{r}}{ds} = \hat{\mathbf{x}}\frac{dx}{ds} + \hat{\mathbf{y}}\frac{dy}{ds} + \hat{\mathbf{z}}\frac{dz}{ds} \Longrightarrow \hat{\mathbf{s}} = \frac{d\mathbf{r}}{ds}\left|\frac{d\mathbf{r}}{ds}\right|^{-1} \quad (6)$$

This parametric description is essential for the accurate modeling of curvilinear wires and general wire structures. Approximating the geometry with straight-wire segments results in **a loss of geometric information** that can never be fully restored. However, this information is preserved when a parametric representation is used to describe the wire's locus.

The Thin-Wire Approximation

In many legacy codes, the exact kernel in Eq. (4) is simplified using the **thin-wire approximation** (**Fig. 2**):

$$K(s,s') \approx \frac{e^{-jkR}}{4\pi R}, \quad R = \sqrt{|\mathbf{r}(s) - \mathbf{r}(s')|^2 + a^2} \quad (7)$$

where a is the wire radius.

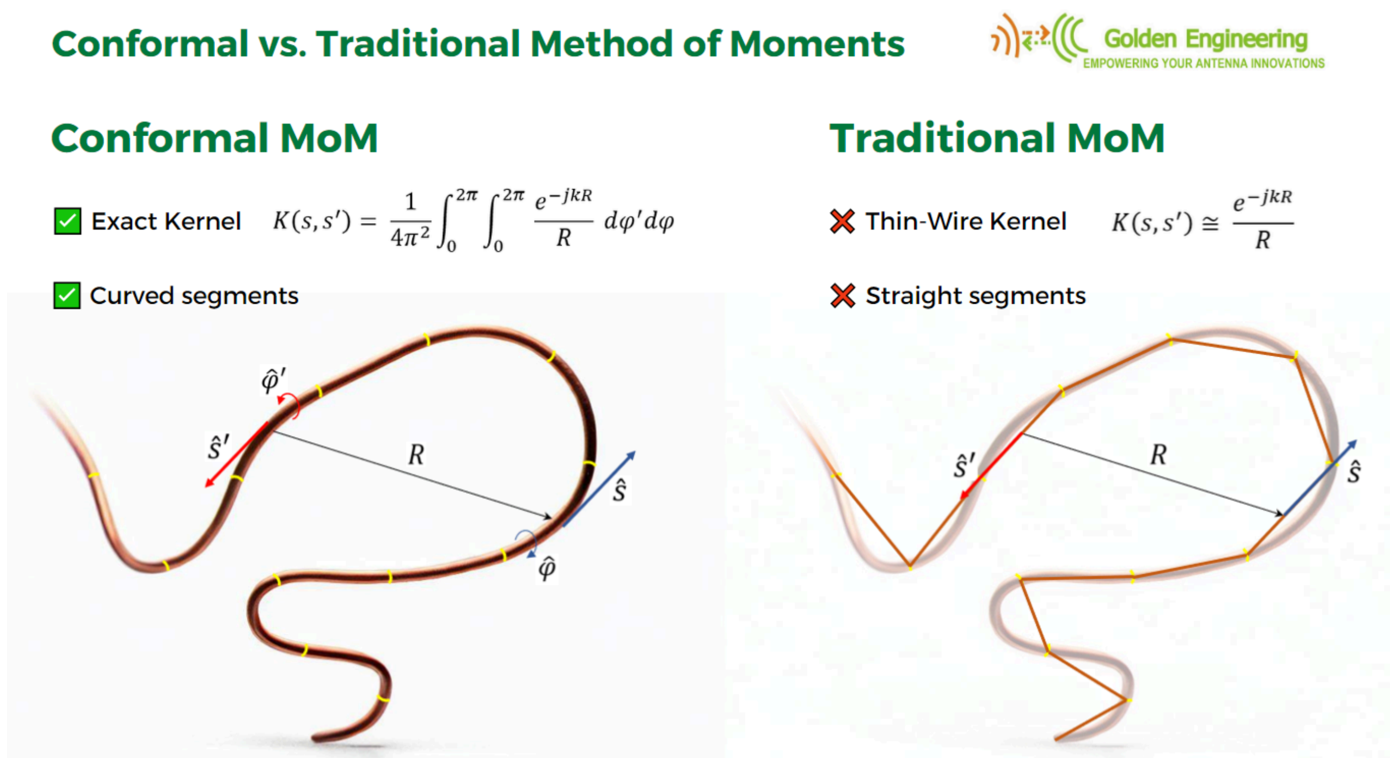


Fig. 2: *(Left) Exact kernel and exact curvilinear geometry description in the CMoM implemented in AN-SOF. (Right) Thin-wire kernel and straight segment approximation implemented in legacy MoM codes.*

If the wire is straight and s represents the length parameter along the wire, and when the observation point $\mathbf{r}(s)$ and the source point $\mathbf{r}(s')$ are both on the same straight wire, the distance R reduces to the standard thin-wire approximation for straight wires:

$R=\sqrt{(s-s')^2+a^2}$ (8)

While this approximation is computationally efficient, it introduces numerical artifacts when segments are very short or when wires are in close proximity. AN-SOF's use of the **Exact Kernel** avoids these simplifications, ensuring that the distance R between source and observation points is calculated precisely across the entire tubular surface of the wire. This approach leads to superior convergence and numerical stability.
The Exact Kernel

The kernel is the mathematical core of the **Electric Field Integral Equation (EFIE)** solved by AN-SOF. It determines how the current at one point on a wire influences the field at another. While several approximations exist to simplify this calculation, AN-SOF utilizes an **Exact Kernel** to maintain high numerical precision where other solvers fail.

Defining the Kernel

The kernel is obtained by averaging the free-space Green's function $G(\mathbf{r},\mathbf{r}')$ over the circumference of the wire's cross-section. Mathematically, it is expressed as:

$$K(s,s') = \frac{1}{4\pi^2} \int_0^{2\pi} \int_0^{2\pi} G(\mathbf{r},\mathbf{r}')\, d\phi'\, d\phi \quad (1)$$

In this equation:

- s and s' are coordinates along the wire axis (observation and source points).
- ϕ and ϕ' are the angular positions on the wire's surface.
- The result represents the interaction between a source point $\mathbf{r}=(s,\phi)$ and an observation point $\mathbf{r}'=(s',\phi')$ on the actual tubular surface of the wire.

The Thin-Wire Approximation

Because the exact integral is difficult to solve analytically, many legacy **Method of Moments (MoM)** codes use the **thin-wire approximation**:

$$K(s,s') \approx \frac{e^{-jkR}}{4\pi R}, \quad R = \sqrt{|\mathbf{r}(s) - \mathbf{r}(s')|^2 + a^2} \quad (2)$$

where a is the wire radius.

This approximation assumes that **the current is concentrated on the wire axis** and the observation point is on the surface, or vice versa. While efficient for very thin wires, this simplification breaks down as the wire becomes thicker relative to the segment length.

Consequences of Approximation

When a wire is divided into segments where **the diameter is larger than the segment length** (a ratio >1), the thin-wire approximation introduces significant numerical errors:

1. **Current Oscillations:** The current distribution begins to exhibit non-physical "ripples" or oscillations, particularly near discrete sources (feed points) and at the wire ends.
2. **Impedance Divergence:** As the mesh is refined (more segments added), the input impedance does not settle on a single value. Instead, **it diverges**, making it impossible to obtain a reliable result.

Figure 1 shows the current distribution in amplitude along a center-fed half-wave dipole obtained using both the thin-wire approximation and the exact kernel. The antenna has been divided into segments with a diameter three times greater than their lengths, resulting in **very thick wire segments**.

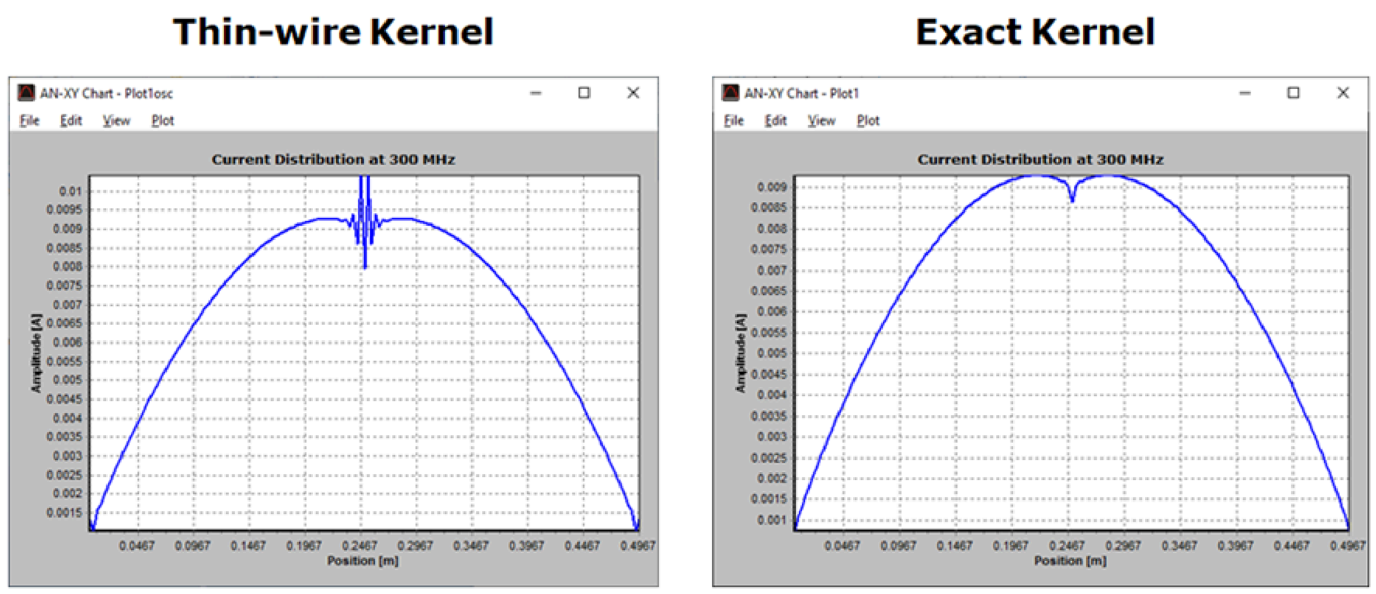


Fig. 1: *Current distribution along a center-fed half-wave dipole divided into segments with a diameter-to-segment length ratio of 3.*

The thin-wire kernel exhibits the well-known oscillatory effect on the current distribution near the position of discrete sources and at wire ends when the segment diameter-to-length ratio is greater than 1. As shown, these oscillations disappear when the exact kernel is used instead of the thin-wire approximation.

The AN-SOF Advantage: Stability and Convergence

By using the **Exact Kernel**, AN-SOF eliminates these numerical artifacts. The current distribution remains smooth and physically accurate, even for segments with a high diameter-to-length ratio.

As shown in comparisons between AN-SOF and NEC-2 (which relies on the thin-wire kernel), AN-SOF demonstrates **monotonic convergence** (see **Fig. 2**). Whether using **a finite-gap or delta-gap source**, the resistance and reactance values stabilize as the number of segments (N) increases, providing a trustworthy solution for complex, high-precision antenna designs.

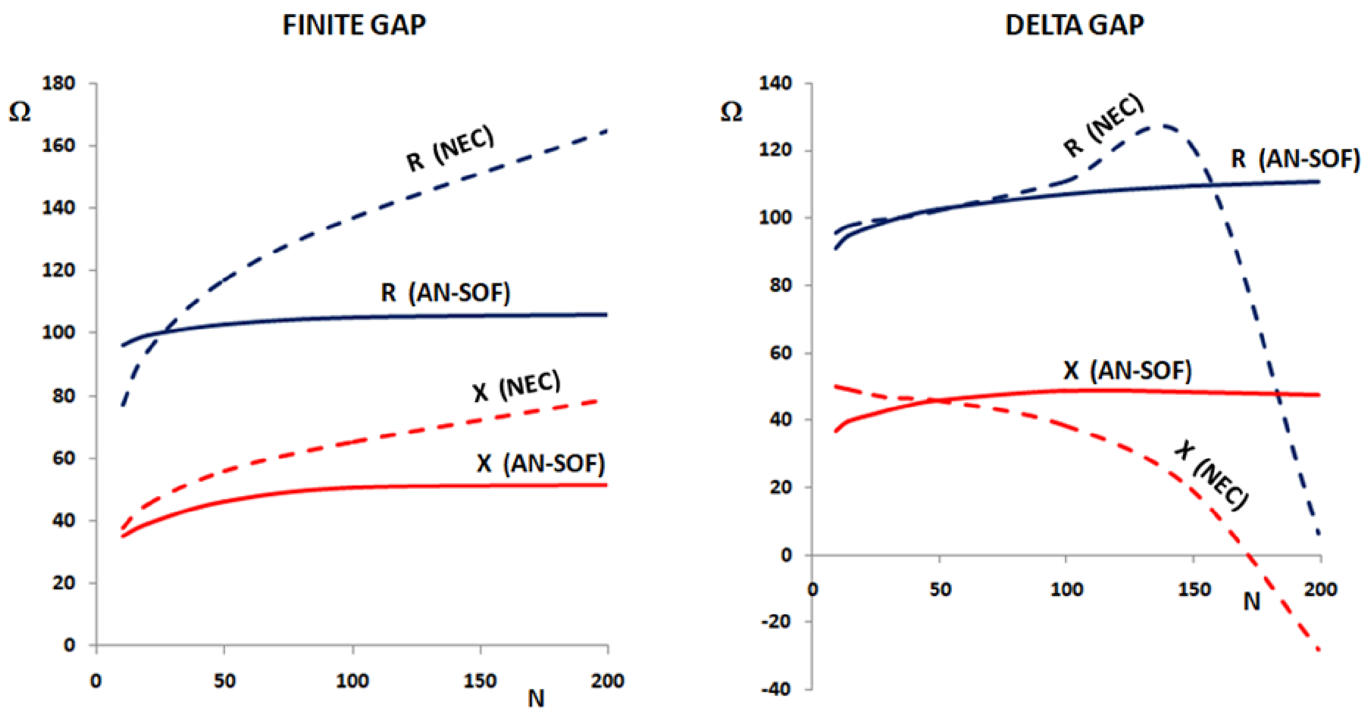


Fig. 2: *Comparison between AN-SOF and NEC-2 results for the input impedance of a center-fed half-wave dipole as a function of the number of segments per arm. Parameters: radius a=0.005λ, source gap =0.025λ (left) and source gap →0 (right).*

The Method of Moments

The Method of Moments (MoM) is the numerical technique used by AN-SOF to transform the [Electric Field Integral Equation (EFIE)](#) into a **system of linear equations**. Once digitized, these equations can be solved using standard algebraic methods to determine the current distribution on a structure.

From Calculus to Algebra

To solve the EFIE numerically, we represent the integral linear operator as L. The fundamental equation is expressed as:

$\mathrm{L}(I) = E_T \quad (1)$

Where E_T is the tangential component of the incident electric field on the surface of the wire. Because the exact current distribution (I) is unknown, it is approximated as a finite sum of N **basis functions** (F_n) with unknown amplitudes (I_n):

$I \approx \sum_n I_n F_n \quad (2)$

By substituting this expansion into Eq. (1) and utilizing the linearity of the operator L, we get:

$\sum_n I_n \mathrm{L}(F_n) = E_T \quad (3)$

Weighting and Matrix Formation

To solve for the N unknown amplitudes, Eq. (3) is weighted with a set of N linearly independent **testing functions** (T_m), also known as **weighting functions**, as follows:

$$\sum_n I_n \int T_m \mathrm{L}(F_n)\,du = \int T_m E^T du \quad (4)$$

where the integrals are evaluated over the wire surface, whose generally **curvilinear axis** is described parametrically by the real variable u. The testing process ensures that the number of independent equations equals the number of unknowns, resulting in the matrix equation:

$$[\mathbf{Z}][\mathbf{I}]=[\mathbf{V}] \quad (5)$$

Where:

- $[\mathbf{Z}]$ **(Impedance Matrix):** An $N \times N$ matrix where each element $Z_{mn}=\int T_m \mathrm{L}(F_n)\,du$.
- $[\mathbf{I}]$ **(Current Matrix):** An $N \times 1$ column vector containing the unknown current amplitudes I_n.
- $[\mathbf{V}]$ **(Voltage Matrix):** An $N \times 1$ column vector where each element $V_m=\int T_m E^T du$.

AN-SOF employs advanced linear algebra algorithms to solve this fully occupied system of equations.

Basis and Testing Functions in AN-SOF

The MoM process begins by dividing the wire structure into N segments. AN-SOF specifically utilizes **triangular basis functions** and **pulse testing functions** (**Fig. 1**).

- **Triangular Basis Functions:** Used to approximate the current distribution smoothly across segments.
- **Pulse Testing Functions:** Used to satisfy the boundary conditions along the segments.

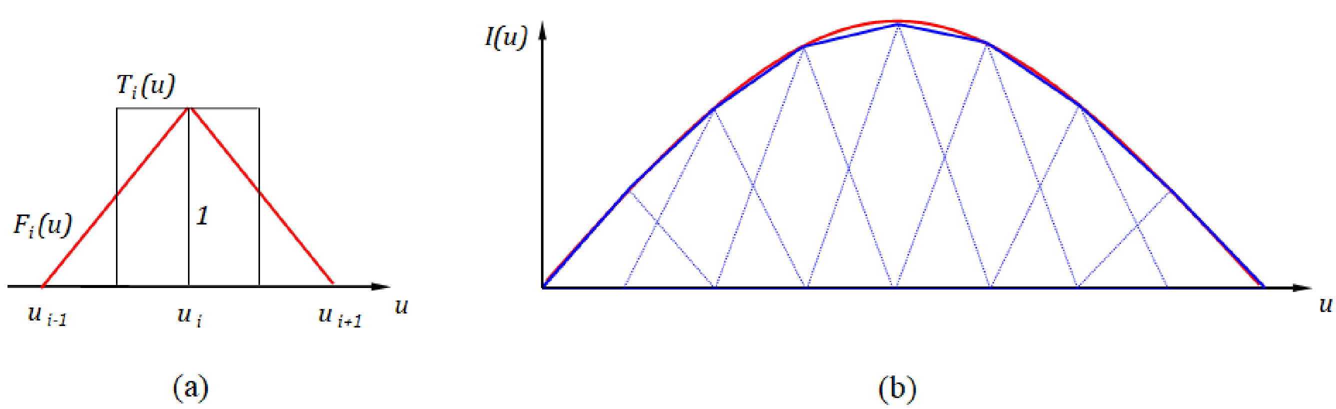


***Fig. 1:** (a) Triangular basis functions, $F_i(u)$, and pulse testing functions, $T_i(u)$. (b) Current distribution approximated by triangular functions.*

The Conformal Method of Moments (CMoM)

When a wire is curved and described parametrically, the support for the basis and testing functions follows that curve. This results in the Conformal Method of Moments (CMoM). Unlike legacy codes that force basis functions onto straight-wire approximations, CMoM ensures the mathematical model adheres strictly to the physical geometry.

Once the system is solved and the currents (I_n) are known, the engine can accurately compute secondary parameters such as input impedance, far-field radiation patterns, and near-field distributions.
Excitation of the Structure

In the Method of Moments (MoM), the excitation represents the right-hand side of the matrix equation $[\mathbf{Z}][\mathbf{I}]=[\mathbf{V}]$. This “Voltage Vector” $[\mathbf{V}]$ defines the external electromagnetic energy applied to the structure, which can take the form of discrete voltage sources or incident external fields.

Discrete Voltage Sources

When a voltage generator is placed on a specific wire segment (the i-th segment), the corresponding element in the voltage column vector is set to the voltage V_i of that generator. This results in a vector where all elements are zero except for the driven segment:

$$[\mathbf{V}]=\begin{bmatrix} 0 \\ \vdots \\ V_i \\ \vdots \\ 0 \end{bmatrix}$$

The Delta-Gap Source Model

In a **delta-gap** model, the source is assumed to exist at a single point (an infinitesimal gap). In a numerical simulation, this source occupies the entire length of the segment where it is placed. As you increase the number of segments (N) to refine the mesh, the length of that segment diminishes, approaching zero. While mathematically convenient, this can lead to convergence issues.

The Finite-Gap Source Model

To model a **finite-gap** source, which more accurately represents physical connectors, a specific, short wire segment is defined to house the excitation. Unlike the delta-gap model, when the overall antenna mesh is refined, the length of this specific source segment **remains fixed**. This approach ensures the convergence of the input impedance, which is calculated by dividing the source voltage by the computed current at that specific segment.

Incident Plane Wave Excitation

When the structure is excited by an external source, such as a distant transmitting antenna, it is modeled as an **incident plane wave**. In this scenario, every segment of the wire is excited simultaneously by the incoming field.

The incident field $\mathbf{E}^i$ at any point $\mathbf{r}$ is defined as:

$$\mathbf{E}^i(\mathbf{r}) = \mathbf{E}_0 e^{-j\mathbf{k}\cdot\mathbf{r}}$$

Where:

- $\mathbf{k}$: The wavevector, indicating the direction of propagation.
- $k = |\mathbf{k}|$: The wavenumber.
- $\mathbf{r}$: The evaluation point on the structure.

To fill the voltage vector $[\mathbf{V}]$, the solver calculates the contribution for each segment m by integrating the incident field along the segment's length:

$$V_m = \int_{s_m} \mathbf{E}^i(\mathbf{r}(s)) \cdot \hat{\mathbf{s}} \, ds$$

In this integral, $\mathbf{r}(s)$ represents the **parametric description** of the curved wire axis, and $\hat{\mathbf{s}}$ is the unit vector tangent to the wire at that point. This ensures that only the tangential component of the electric field, the part responsible for inducing current, is accounted for in the calculation (**Fig. 1**).

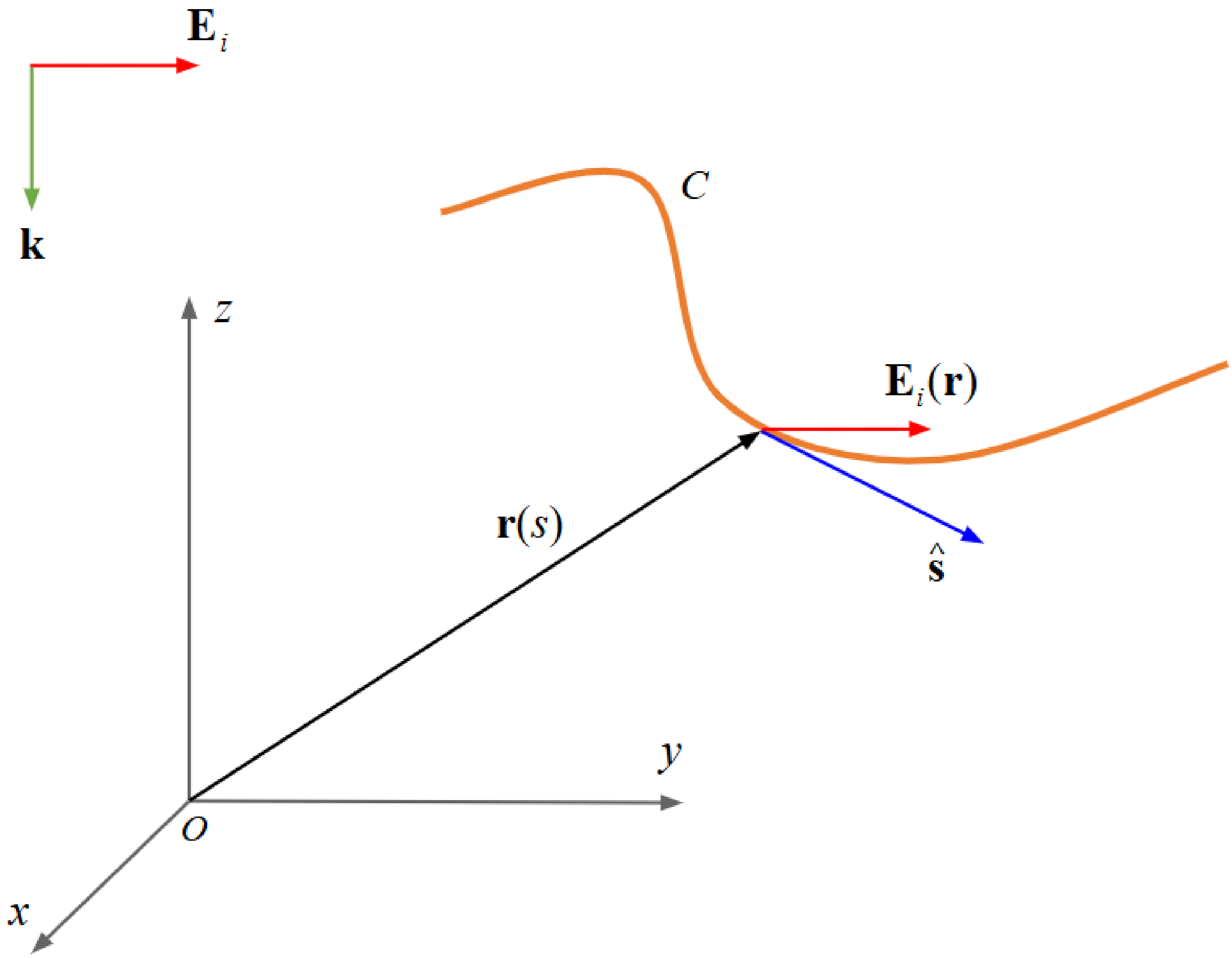


Fig. 1: *Incident plane wave exciting a curvilinear wire.*

Curved vs. Straight Segments

The choice between using curved segments or straight-wire approximations is fundamental to the accuracy and efficiency of electromagnetic simulations. The following study illustrates the advantages of **curved segments** regarding the stability and convergence properties of numerical solutions.

Helix Antenna Model

To compare these two approaches, we examine a center-fed helical antenna (see **Fig. 1**) operating in free space (normal mode). The specific dimensions of the model are the following:

- **Helix Radius:** 0.0273λ
- **Pitch:** 0.0363λ
- **Number of Turns:** 10
- **Wire Radius:** 0.001λ

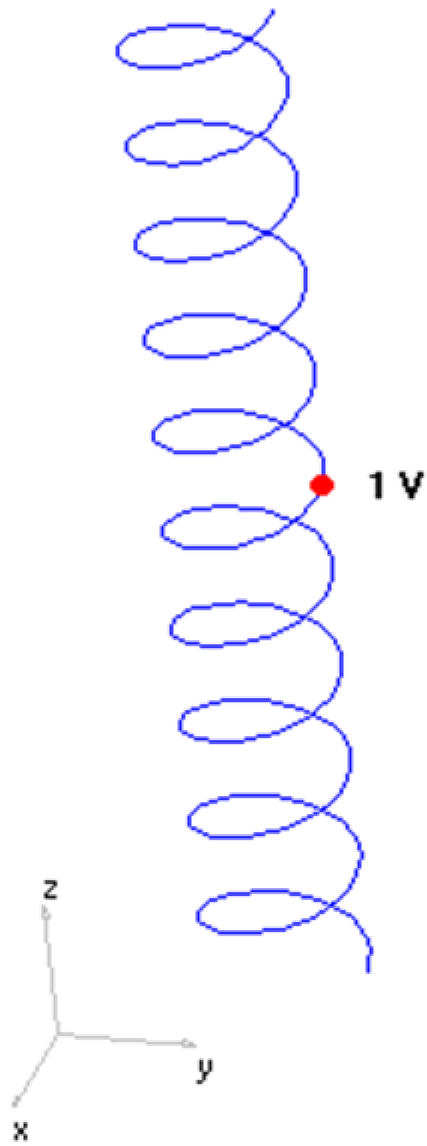


Fig. 1: *Center-fed helix antenna model.*

Input Impedance Convergence

The performance of the **Conformal Method of Moments (CMoM)** in AN-SOF is compared against a standard straight-wire approximation. We investigate the convergence of the **input impedance** as the number of segments (N) increases.

Figure 2(a) shows the **Input Resistance** convergence. The straight-wire approximation converges much more slowly than the AN-SOF curve, which settles rapidly into a stable value.

Figure 2(b) displays the **Input Reactance** convergence. The difference is even more pronounced here: the straight-wire model converges very slowly toward a value below the asymptote of the AN-SOF curve. In contrast, the curved segment model in AN-SOF demonstrates smooth, monotonic, and rapid convergence.

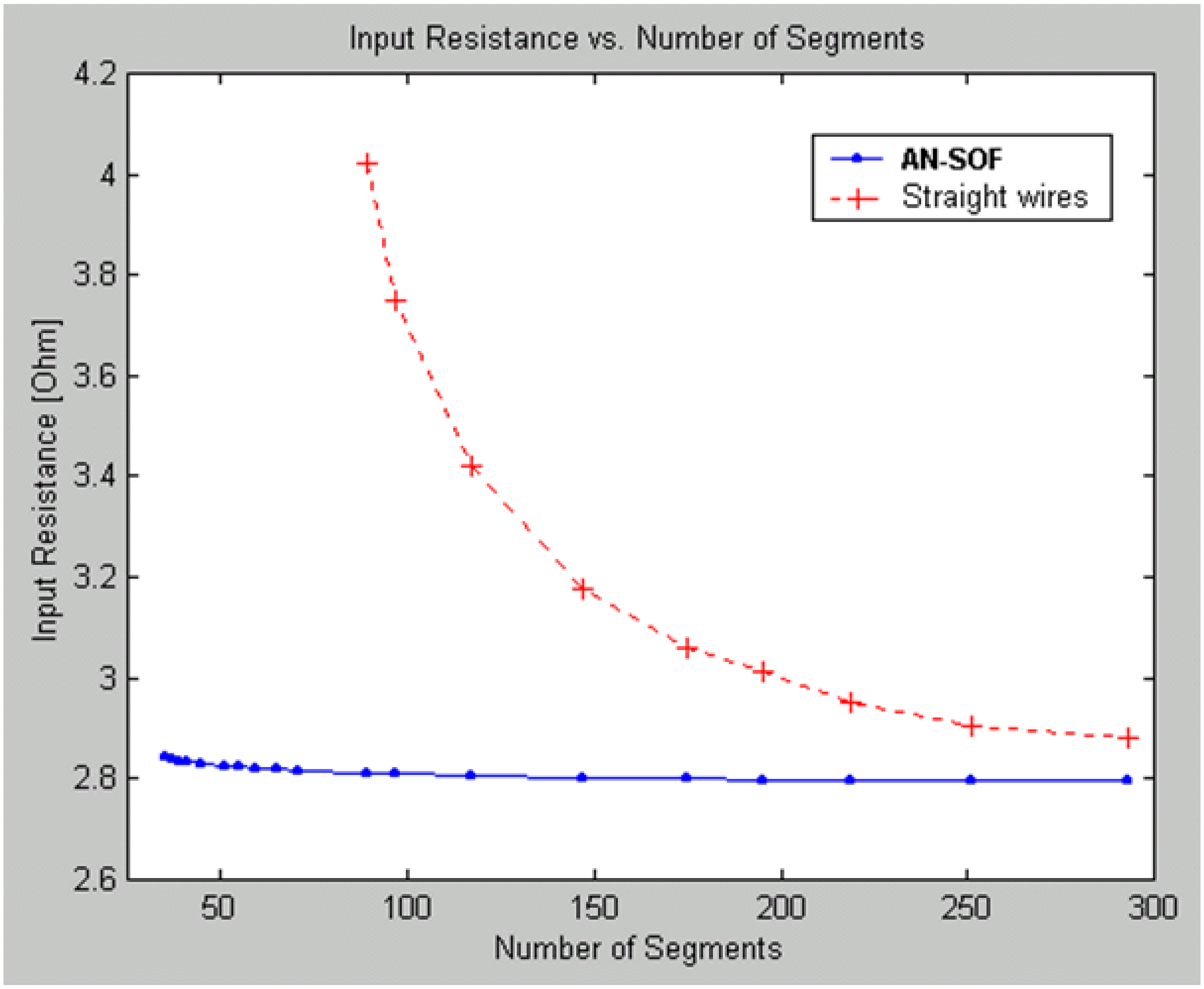


Fig. 2(a): *Resistance convergence plot for the normal-mode helix.*

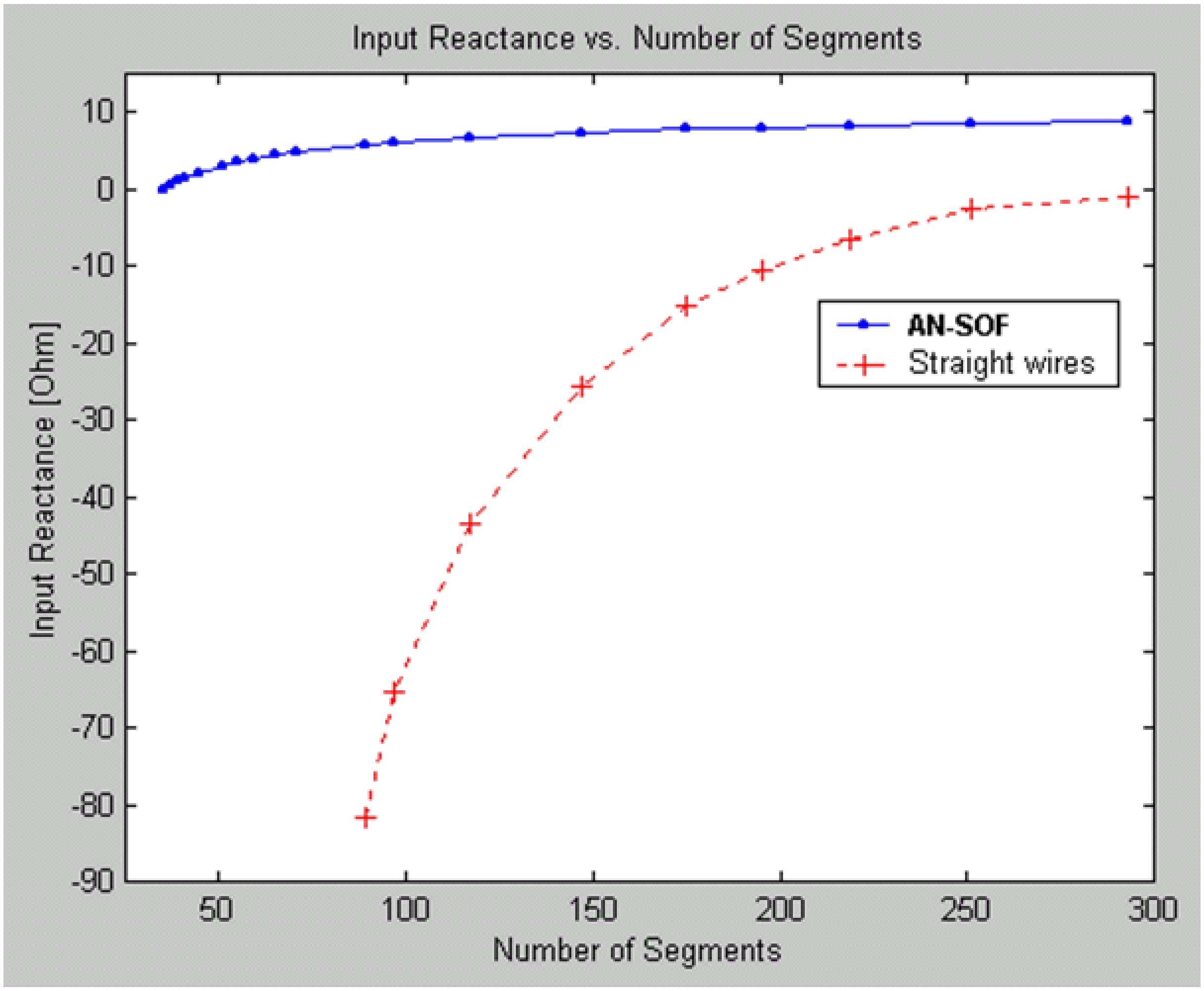


Fig. 2(b): Reactance convergence plot for the normal-mode helix.

Input Admittance Convergence

The same comparison is applied to the **input admittance** (conductance and susceptance) to further validate the results.

In **Fig. 3(a)**, the **Input Conductance** results are plotted. The straight-wire approximation starts to increase smoothly as the number of segments increases and then diverges rapidly. In contrast, the AN-SOF curve decays fast to a convergent value.

In **Fig. 3(b)**, the **Input Susceptance** is shown. Again, the improved convergence rate of the curved segments is evident, while the straight-wire approximation diverges.

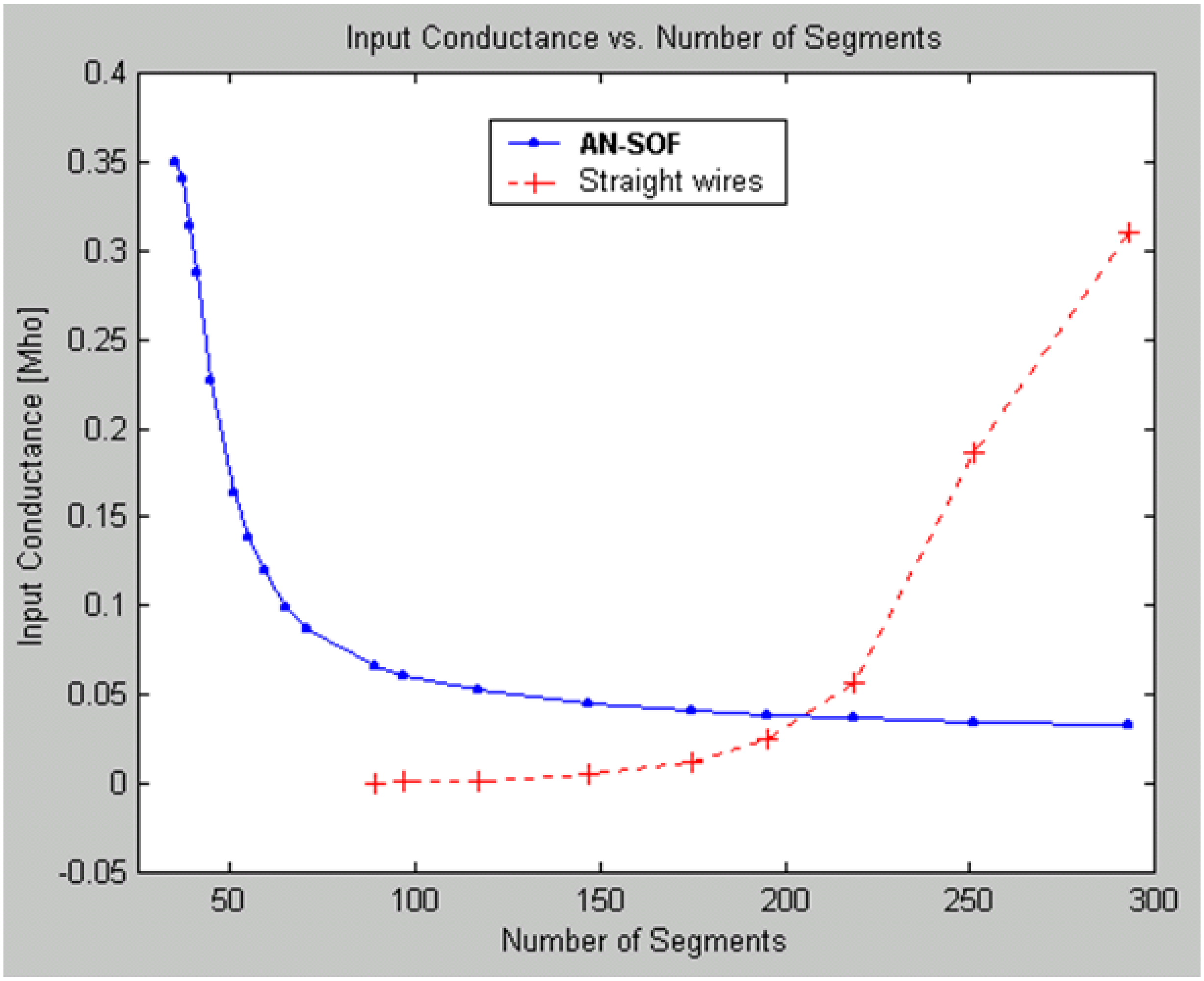


Fig. 3(a): *Conductance convergence plot for the normal-mode helix.*

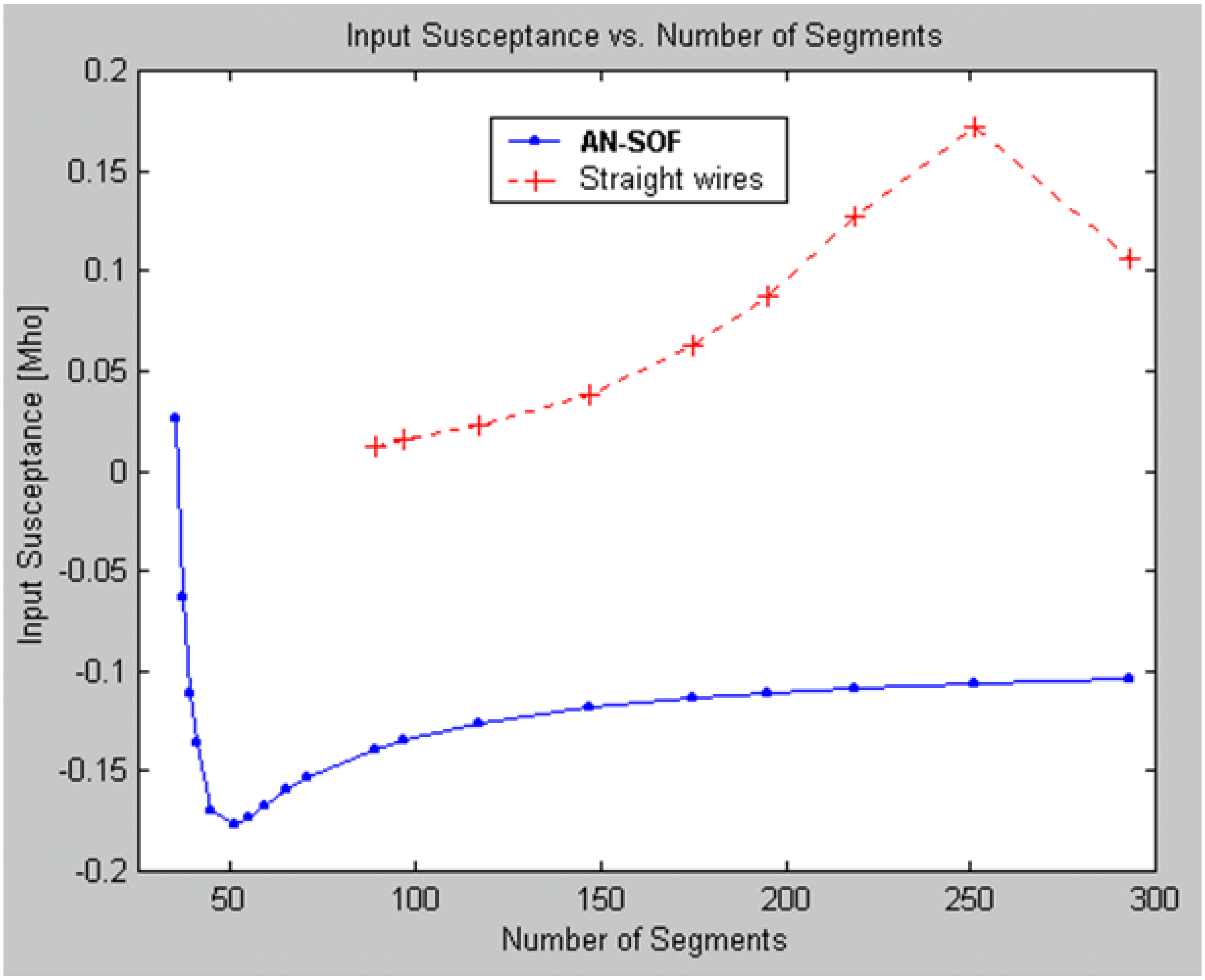


Fig. 3(b): *Susceptance convergence plot for the normal-mode helix.*

Practical Advantages

As demonstrated by these benchmarks, the use of curved segments provides three critical advantages:

- **Convergent Input Impedance and Admittance:** As the number of segments is increased, the input impedance and admittance for the CMoM implemented in AN-SOF converge. In contrast, the legacy straight-wire segment approximation does not guarantee convergence, and in the case of the normal-mode helix, it diverges.
- **Reduced Simulation Time:** Accurate results are obtained with fewer segments, leading to faster computation.
- **Memory Efficiency:** Because fewer unknowns are required to describe complex geometries like helices, memory overhead is greatly reduced.

By modeling the true physical contour of the structure through the CMoM, AN-SOF ensures that the numerical solution remains stable and reliable, avoiding the artifacts and slow convergence associated with traditional linear approximations.

Frequently Asked Questions

Licensing FAQ

1. Do I need a key for the AN-SOF Trial version?

No, you don't. Simply run AN-SOF and start using it. If prompted for a license, go to the AN-SOF main menu > Help > **Activation Key**. Click the **Trial Key** button in the displayed window, followed by the **Activate** button. AN-SOF will restart and be ready for use in trial mode.

2. Is there a time limit for the Trial version?

The trial period lasts **30 days**, during which software evaluation support is available. However, you can continue using AN-SOF Trial after the evaluation period ends.

The Trial version is limited to **50 segments** (wire segments + wire-to-wire junctions + wire-to-ground connections) for calculations. However, all example files included in the installation directory are pre-computed and can be opened without restrictions to display tables and graphs.

You can also download additional example files with fewer than 50 segments from **this link**.

3. Will AN-SOF stop working when my plan ends?

No, AN-SOF PRO is a lifetime license and will continue to function on the same PC where it was installed and activated, even after your plan expires. However, if you choose not to renew your plan, you will no longer have access to the following:

- **Software Updates:** You will no longer be able to download the latest versions, including bug fixes and security updates.
- **Technical Support:** You will no longer receive technical assistance from our team.
- **Installer Access:** Installers for discontinued versions will only be available for **3 years after discontinuation**. After this period, they will no longer be provided. As technology evolves, we cannot guarantee long-term compatibility.
- **Activation Key Replacement:** If you lose your activation key for a discontinued version or need to transfer AN-SOF to another PC, **we cannot issue a new key more than three years after the version has been discontinued**.

While AN-SOF will continue to work on your installed PC (assuming the OS remains unchanged), we recommend renewing your plan to ensure access to the latest features, technology improvements, and ongoing support.

4. What happens if I don't renew my plan?

If you don't renew your plan, the following will apply:

- **Continued Functionality:** AN-SOF will keep working on the same PC where it was installed and activated, but only with the last version available before your plan expired.
- **No Updates:** You won't receive new versions, bug fixes, security patches, or feature updates.
- **Limited Support:** Technical support will no longer be available.
- **Version Discontinuation:** Your installed version will eventually be discontinued. **Three years after discontinuation, its installer and activation key will no longer be available**.

To maintain full functionality, updates, and support, we recommend renewing your plan before it expires. **Customers who renew early receive special benefits and discounts.**

5. Do you offer a refund for AN-SOF purchases?

Due to the availability of a **fully functional trial version with no time limit**, all sales of AN-SOF software licenses are final and **non-refundable**. This policy helps us ensure that users have a fair opportunity to try out the software before making a purchase. We encourage all users to thoroughly evaluate the trial version before purchasing a license. You can find detailed information about the conditions and process for exceptions to the non-refundable policy on our dedicated **Refund Policy** page.

6. Are consulting services included in the Gold and Platinum plans?

No, consulting services are not included, as we do not offer consulting of any kind. However, our **technical support**—available with both plans—goes beyond resolving software issues. We assist customers in improving their models and expanding their understanding of antenna modeling techniques to maximize the capabilities of AN-SOF. While we provide tips and guidance, this should not be considered consulting.

Technical FAQ

1. What are the minimum PC requirements?

Windows Vista/7/8/10/11. 2GHz CPU, 2GB RAM, 1GB free disk space.

2. Can AN-SOF be run on a Mac computer?

The supported operating system is Microsoft Windows. We have no plans to release a Mac version. Macintosh users can run a program called **Parallels >**. Parallels Desktop for Mac is desktop virtualization software that allows Microsoft Windows applications to run on an Apple Mac computer.

3. Is there a version of AN-SOF for Linux?

No. The supported operating system is Microsoft Windows. We have no plans to release a Linux version. You can use a Windows emulator like Wine, CrossOver Linux, Vmware Workstation or whatever you find on the market.

4. Does AN-SOF support parallel processing?

No, it doesn't. AN-SOF has been developed to run on home computers running Windows(R) OS, so numerical calculation strategies have been implemented to take care of the available RAM memory and at the same time obtain reliable results.

5. What is the upper frequency limit?

AN-SOF does not have a fixed upper frequency limit. What matters is the **electrical size** of the structure, meaning its dimensions measured in **wavelengths**, rather than the specific frequency or band. As the structure becomes larger in terms of wavelengths, the size of the matrix equation to be solved increases, which requires more memory and computation time on a given PC. In practice, the limit is determined by your computer resources and the electrical size of the model, not by frequency itself.

6. How does AN-SOF divide the wires into segments? Is a higher density of segments needed in tapered wires?

By default, AN-SOF calculates the minimum recommended number of segments for each wire depending on its length in wavelengths. Various **convergence analyzes >** show that **10 wire segments per wavelength** is sufficient for most cases. Regarding tapered wires, in old algorithms like NEC it was necessary to increase the density of segments near the connections between wires when there is a radius jump. This is not necessary in AN-SOF as it is not NEC based. See the advantages of AN-SOF **here >**.

7. Can arrays be built quickly by duplicating and copying parts of a structure?

Yes. Go to **main menu > Edit > Copy Wires >** to duplicate or make the desired number of copies of the selected wires. There is also the **Stack Wires** command which allows us to repeat a design along a given direction. By using this command in combination with the **Scale Wires >** command we can quickly build Yagi-like arrays.

8. Can parametric design be done with AN-SOF, that is, run simulations with variable geometric parameters such as the separation between dipoles in an array?

Yes. Parametric design is possible by running a **Bulk Simulation >**. We prefer that the user chooses the programming language to generate a sequence of files in NEC format with one or more variable parameters. Calculations on these files can then be run automatically in bulk. **Scilab >** is a free numerical calculation software tool with which we can program **scripts** that generate multiple descriptions of an antenna with variable parameters. Download an example of a Yagi-Uda antenna with variable element spacing from **this link >**.

9. Is the wire grid model well suited for surfaces?

In addition to specializing in wire structures composed of wire grids, AN-SOF also allows the modeling of **solid metallic surfaces**. **Solid surfaces** are viewed in the AN-SOF workspace as if they were wire grids; however, they are actually made up of **flat strips**. These strips have widths automatically calculated to completely cover a metal surface without leaving holes. Currently, there is a limit in the size of the grid/surface that can be modeled (see FAQ #10).

10. Can AN-SOF model electrically large antennas like horns and parabolic dishes?

AN-SOF is equipped to model solid metallic surfaces, including parabolics and horns. Currently, there is a limitation of approximately 10 square wavelengths of surface. As long as your antenna's surface area stays within this threshold, you can run simulations. This limitation pertains to the surface area of the antenna measured in square wavelengths, rather than a restriction on the frequency range. This limitation also applies to grids (patches, plates, cylinders, spheres, etc.).

11. Does AN-SOF support load impedances?

Yes, it does. Resistance, inductance, and capacitance elements can be added to the structure to model the connection of lumped load impedances.

12. Are near E- and H-fields available in tables and for exportation as Excel or Google Sheets files?

Yes, they are. The computed near E and H fields can be visualized in 2D and 3D plots as well as in tables and exported as CSV (Comma Separated Values) files. Cartesian, cylindrical and spherical near field components can be obtained.

13. Is AN-SOF based on a NEC engine?

No. AN-SOF is an independent implementation of the Method of Moments (MoM) for wire structures. NEC is an old Fortran calculation engine that has a lot of limitations. Many of these limitations have been removed in AN-SOF by implementing the so-called **Conformal Method of Moments with Exact Kernel >** in a completely new object-oriented C++ code. See further details **here >**.

14. Can NEC files be imported into AN-SOF?

Yes. Most of the NEC commands are supported. Download NEC example files to import into AN-SOF from **here >**.

15. Can dielectric materials be modeled with AN-SOF?

Dielectric material can be added as insulation or coating to metallic wires, and microstrip antennas can be patterned on a dielectric substrate. However, modeling volumes composed entirely of dielectric materials is not currently supported.

16. What types of PCBs and microstrip antennas can AN-SOF simulate?

While AN-SOF was originally designed for simulating wire structures using the **Method of Moments (MoM)**, its calculation engine has been extended to support **simple PCB and microstrip antenna designs** that meet the following requirements. For structures that do not comply with these conditions, a **Finite Element Method (FEM) solver** (not included in AN-SOF) should be used instead.

Key Limitations & Requirements:

1. **Single-Layer, Lossless Substrate Only**
    - Only **one dielectric layer** is supported (multilayer substrates are not).
    - The substrate must be **lossless** (loss tangent is assumed zero).
    - **No holes or cutouts** are permitted in the substrate.
2. **Finite-Size Substrate Constraints**
    - The substrate must be **rectangular**.
    - Traces must be **at least 5× the trace width away from the substrate edges**.
3. **Ground Plane & Vias**
    - A **perfect electric conductor (PEC) ground plane** is mandatory and cannot be removed.
    - **Vertical vias** can be modeled as **short vertical wires** connecting traces to the ground plane (e.g., for antenna feeding).
4. **No Slot-Based Designs**
    Slot antennas or patches with slots **cannot be modeled** due to software limitations.

We currently **have no plans to integrate an FEM solver into AN-SOF**. For advanced PCB or antenna designs—such as multilayer structures, lossy substrates, or slotted geometries—we recommend using alternative simulation tools.
Troubleshooting

1. I get the error “Current License file is not valid for this version. AN-SOF will run in trial mode.”

You have entered an invalid activation key. Find the AN-Key app and launch it. Press the “Trial Key” button and then “Activate”. Restart AN-SOF. Follow the instructions in the AN-Key window **to request a valid key** corresponding to your serial number if you have purchased a license. Please note that the key you used to activate a previous version of AN-SOF may not be valid for the latest version. **Request a new key >**.

2. I get the error “The License file does not exist. AN-SOF will run in trial mode.”

Find the AN-Key app and launch it. Press the “Trial Key” button and then “Activate”. Restart AN-SOF. Follow the instructions in the AN-Key window **to request a valid key** corresponding to your serial number if you have purchased a license. Please note that the key you used to activate a previous version of AN-SOF may not be valid for the latest version. **Request a new key >**.

3. I have entered the correct activation key, but AN-SOF continues to run in trial mode.

Uninstall AN-SOF. Then go to C:\ and delete all the folders whose names start with “AN-SOF Professional”. Reinstall AN-SOF. Find the AN-Key app and launch it. Follow the instructions in the AN-Key window **to request a valid key** corresponding to your serial number if you have purchased a license. If the problem persists, open a support case **here >**. Please note that the key you used to activate a previous version of AN-SOF may not be valid for the latest version.
Request a new key >.

4. AN-SOF or one of its applications does not work.

Uninstall AN-SOF. Then go to C:\ and delete all the folders whose names start with “AN-SOF Professional”. Reinstall AN-SOF.

5. When running AN-SOF or any of its applications nothing is displayed on the screen.

Press Ctrl + Alt + Del and run the Task Manager. Right click on the application that is not working and choose “End Task”.

6. When I try to run AN-SOF, I get the error “The feature you are trying to use is on a network resource that is unavailable”.

Navigate to the folder that you specified when using the installer. The default folder is “C:\AN-SOF Professional X”, where X is the AN-SOF version. Then, launch **ANSOF.exe** directly from that location. You can create a shortcut to this file on the Windows desktop if you wish.
Articles Index Directory

Indexed Articles

A

- Wire Materials
- Wire Properties

Disclaimer of Warranty

This Knowledge Base and associated software are provided “as is” without warranty of any kind, expressed or implied. Golden Engineering LLC disclaims all warranties, including, but not limited to, the implied warranties of merchantability, fitness for a particular purpose, and non-infringement.

Golden Engineering LLC does not warrant that:

- The technical descriptions, procedures, and software are free from errors or defects.
- The equations, programs, and procedures are consistent with any standard of merchantability.
- The equations, programs, and procedures will meet your requirements for any application.

Golden Engineering LLC is not liable for any damages arising from the use of the Knowledge Base or associated software, including, but not limited to:

- Direct, indirect, incidental, consequential, or punitive damages.
- Damages for loss of profits, data, or use.
- Damages for personal injury or property damage.

Any use of the Knowledge Base or associated software is at your own risk.