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

antennasimulator.com/index.php/an-sof-user-guide

AN-SOF Antenna Simulation Software

Fast and Easy-to-Use Software 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.

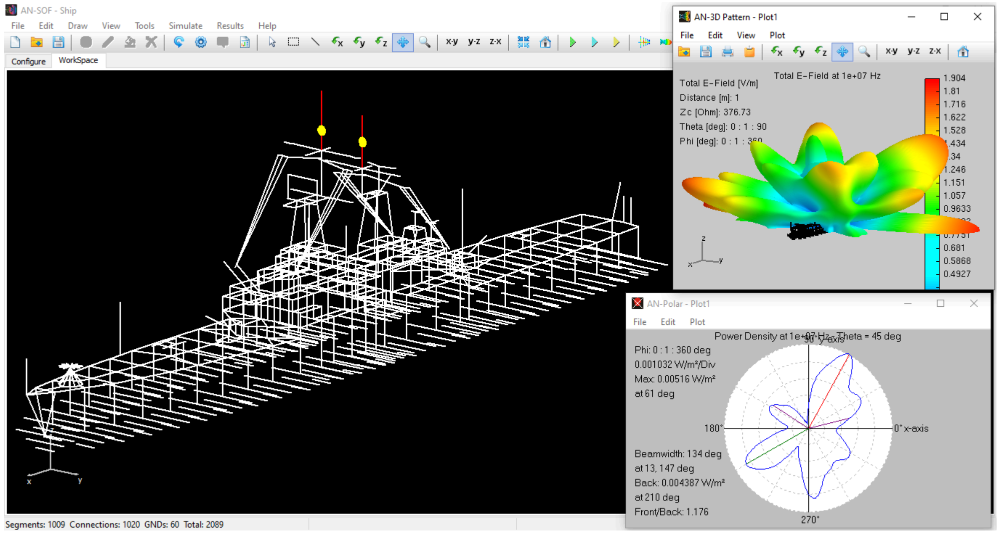
Table of Contents

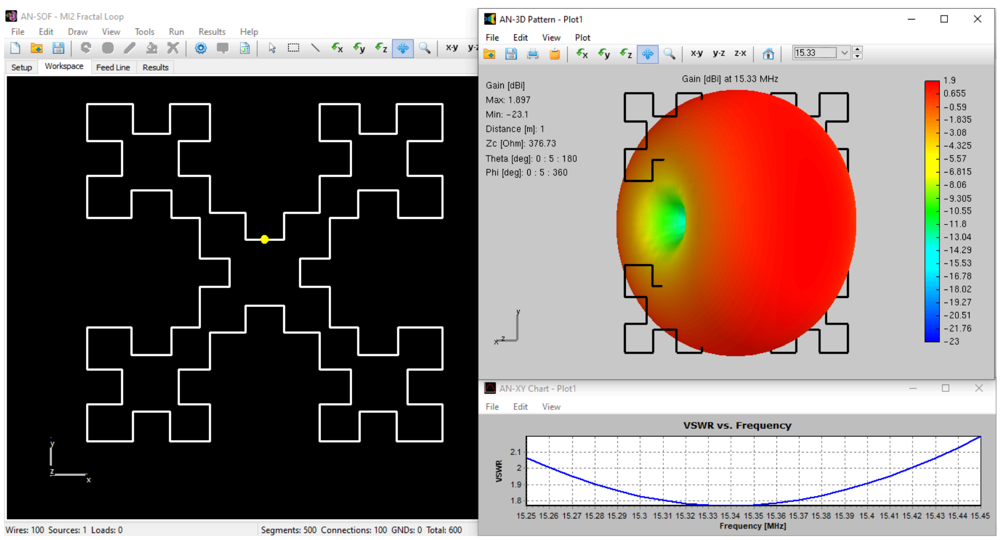
Getting Started

Quick Start Guide

Enhancing Antenna Design Through Simulation Software

An **antenna model** is a representation of a real-world antenna in a computer program. This type of model should not be confused with a scale model, which is sometimes built to measure the radiation characteristics of a larger physically-sized, identical antenna. Due to the mathematical complexity involved in modeling, **computer software** is often programmed to predict and analyze **antenna performance**.





Computer simulation in the industry is used to overcome challenges and drive innovation in the **product creation** and **development** processes. A computer model offers the advantage of being easily modified, redesigned, broken, destroyed, and rebuilt multiple times without wasting materials. Therefore, the design process can achieve a significant **reduction in the cost** of building successive physical models with the aid of simulation software.

AN-SOF is a comprehensive simulation software suite for **antenna modeling and design**. It facilitates the design of various wire antennas, such as dipoles, monopoles, yagis, log-periodic arrays, helices, spirals, loops, horns, fractals, phased arrays, and many other antenna types. Additionally, AN-SOF supports the modeling of feeding systems using **transmission lines**, allowing for a detailed analysis of antenna configurations. The software is capable of simulating antennas positioned above **lossy ground planes** or broadcast antennas above **radial wire ground screens**.

Moreover, AN-SOF's calculation method has been expanded to include single-layer **microstrip patch antennas** and the computation of radiated emissions from **Printed Circuit Boards** (PCBs). Consequently, AN-SOF can be effectively utilized for **Electromagnetic Compatibility (EMC) Applications**. The software accommodates **passive circuits** with lumped impedances and **non-radiated networks**, enabling a 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.

Embrace the excitement of this fascinating field with AN-SOF at your disposal!

With AN-SOF, the possibilities for antenna analysis and optimization are extensive, providing a comprehensive toolkit for antenna design and performance evaluation.

> **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.

The structure’s geometry can be easily drawn in AN-SOF using the mouse, menus, and user-friendly dialog windows. Wires are drawn in a 3D space, where tools are available to zoom, move, and rotate the structure.

To plot the results from a simulation, a suite of integrated applications allows us to display graphs: **AN-XY Chart**, **AN-Smith**, **AN-Polar**, and **AN-3D Pattern**. These tools can also be executed independently for subsequent graphic processing.

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 >**.

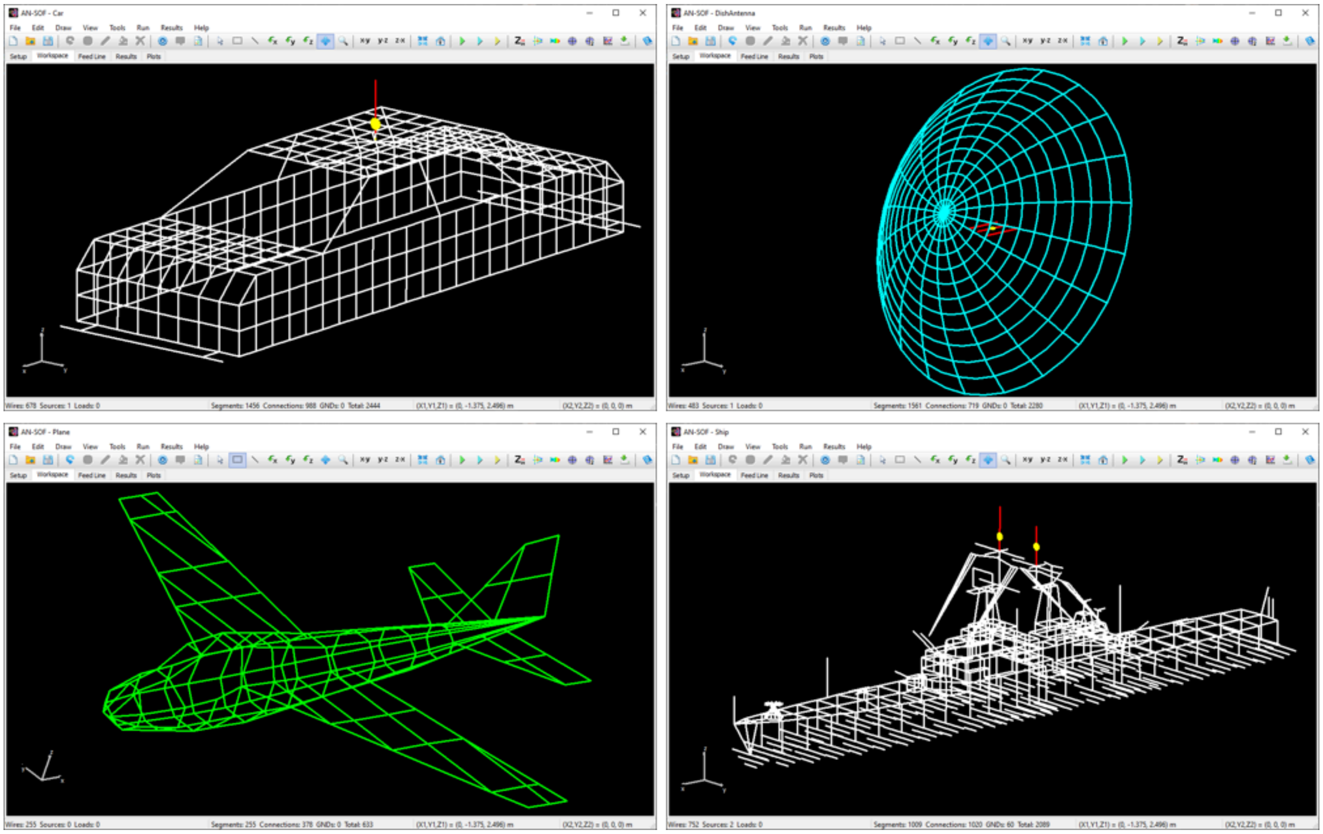


Fig. 1: Computer models of a car, a parabolic reflector, an airplane, and a ship using wire grids.

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.

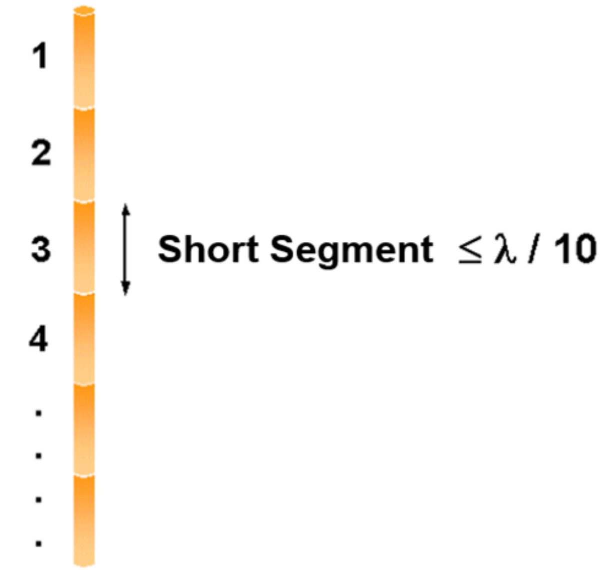


Fig. 2: A straight wire divided into short segments relative to the wavelength.

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.

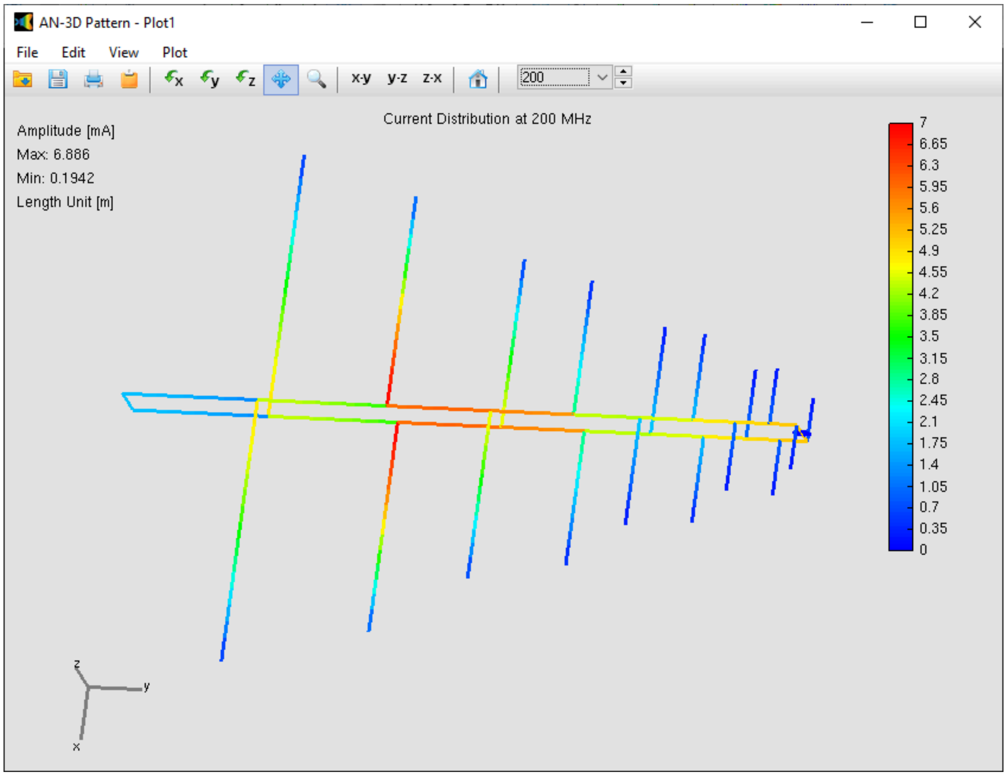


Fig. 3: Current distribution on a log-periodic antenna. The color map on the structure indicates the amplitude of the electric currents.

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.

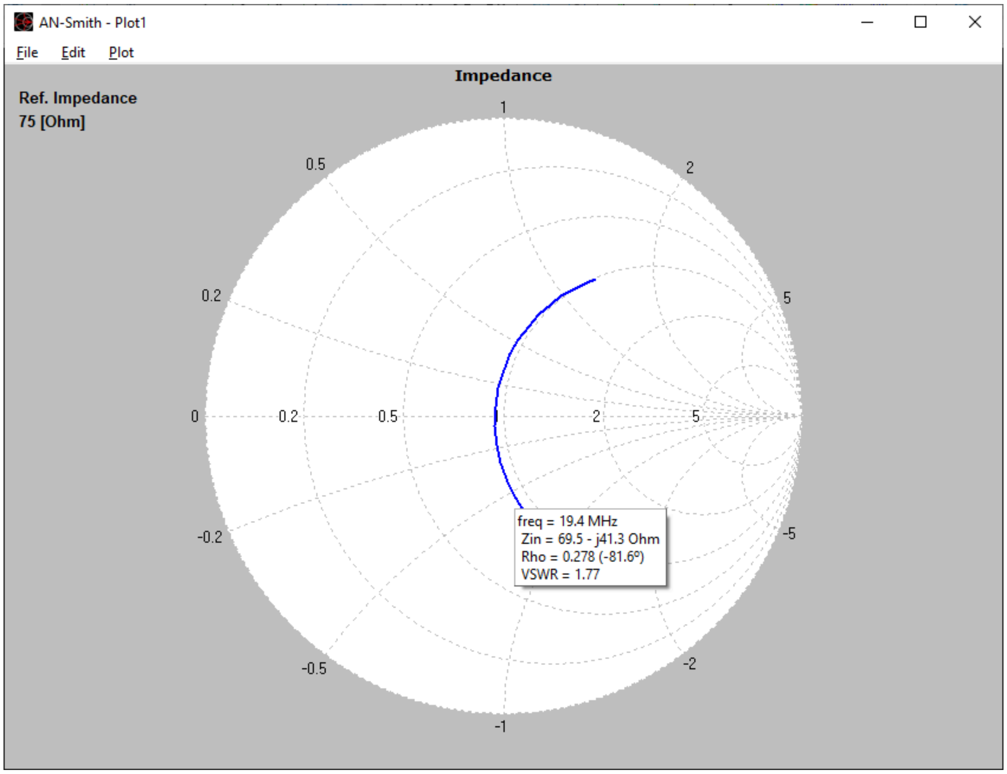


Fig. 4: Impedance plotted as a function of frequency on a Smith Chart, where the VSWR can be obtained by clicking on the curve.

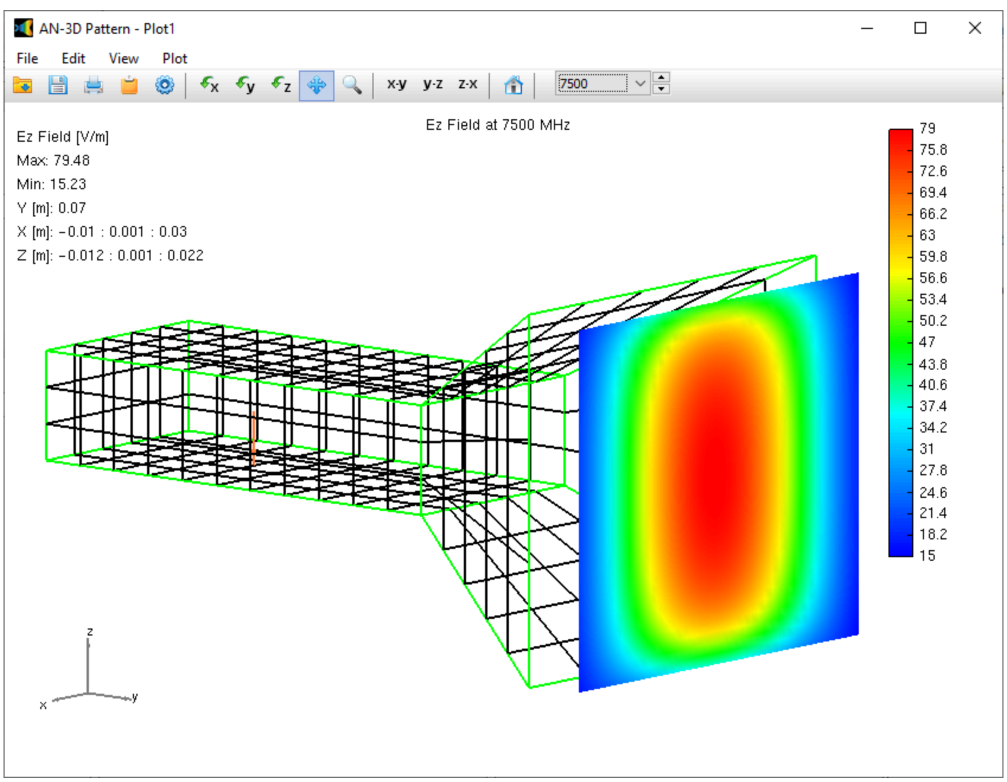


Fig. 5: Near electric field in the vicinity of a Horn antenna.

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.

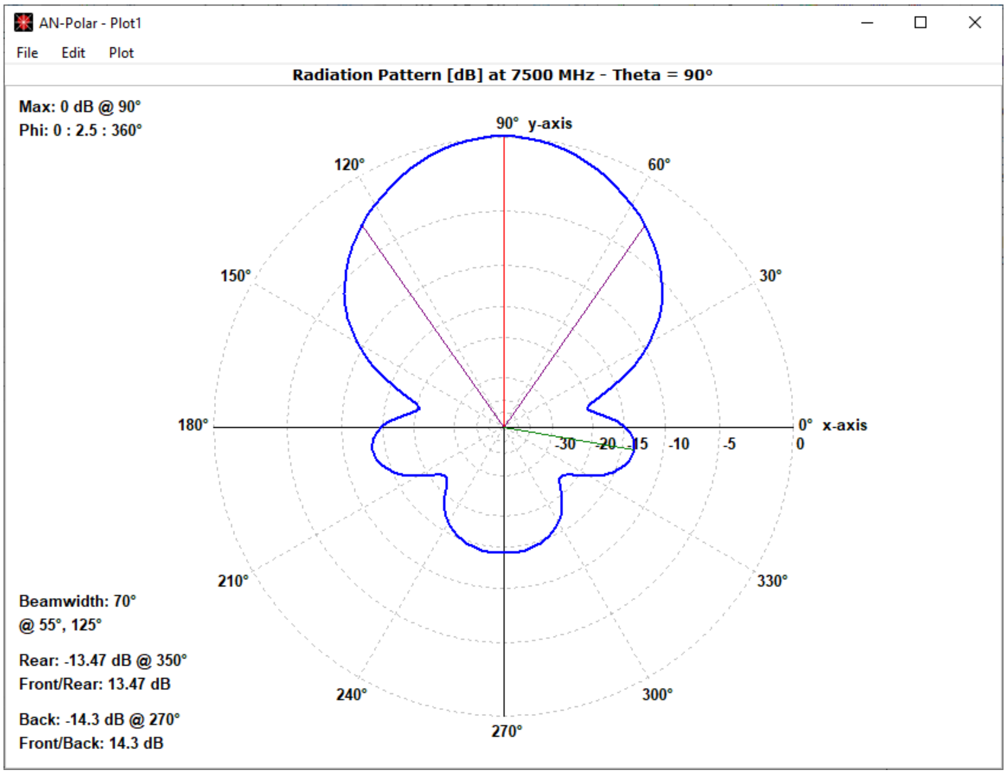


Fig. 6: Far-field pattern represented in a polar diagram, indicating beamwidth, front-to-rear ratio, and front-to-back ratio.

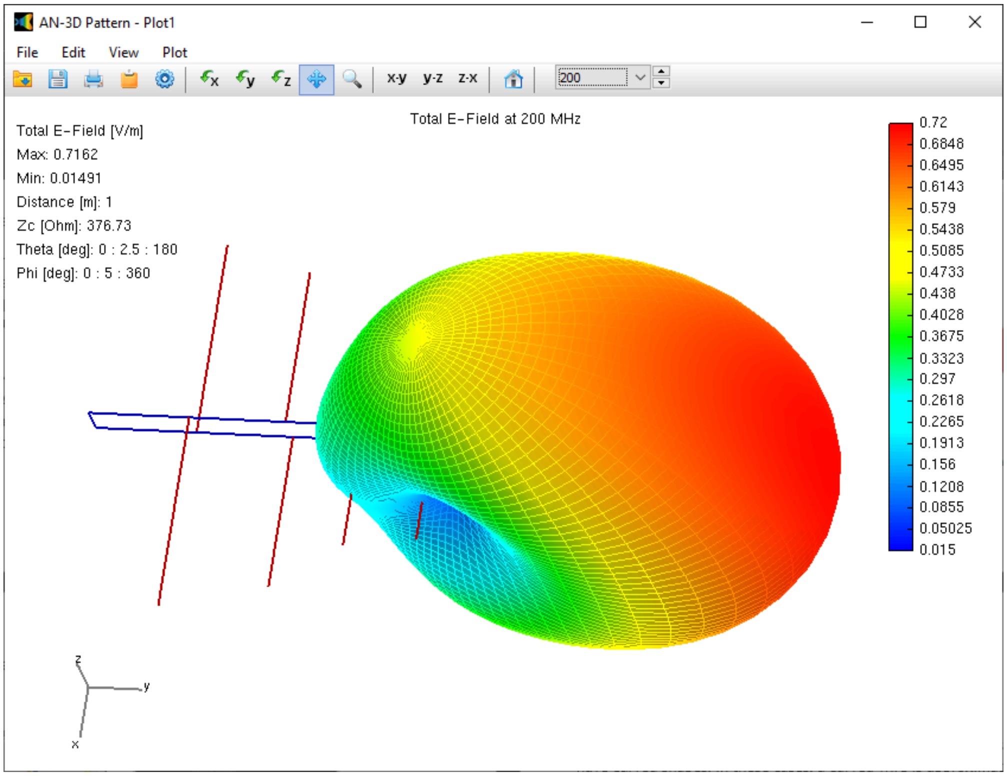


Fig. 7: Far-field pattern represented in a 3D plot, superimposed onto the antenna geometry.

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**.

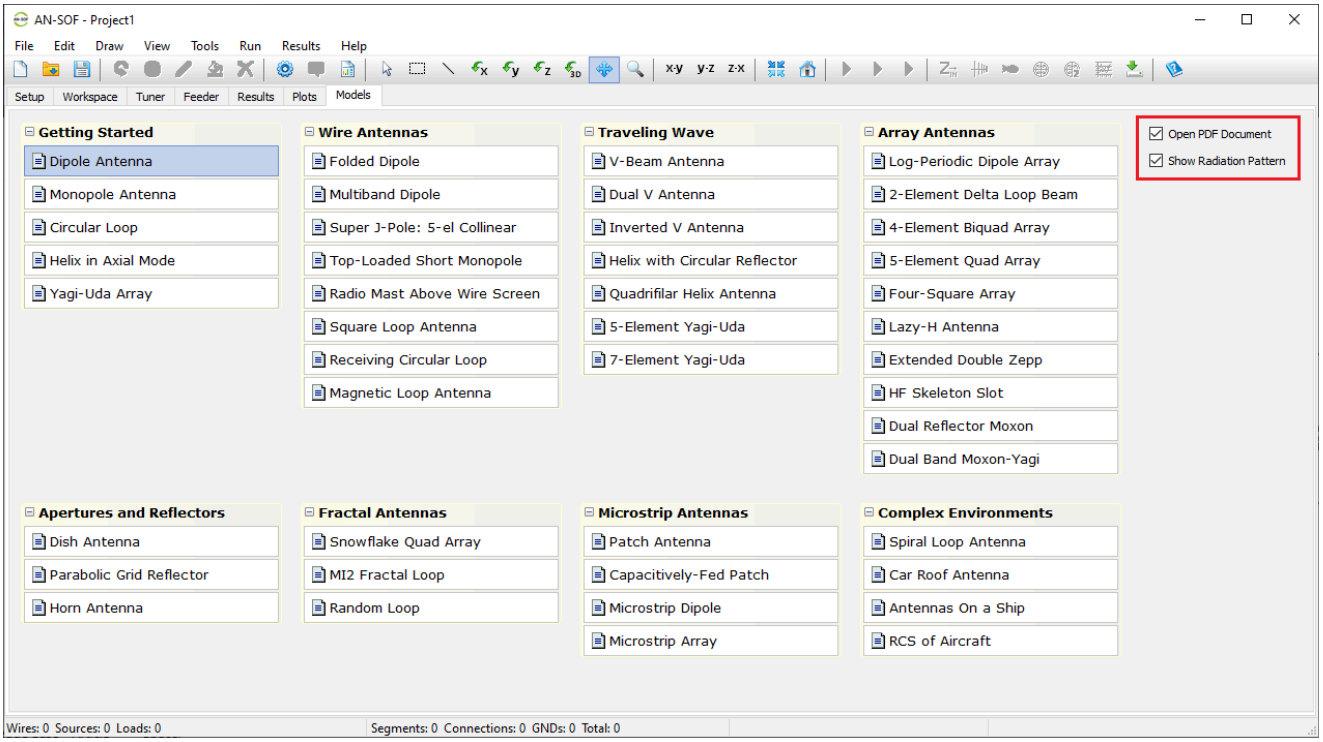


Fig. 8: The Models tab in the AN-SOF interface.

Please note that the AN-SOF Trial version supports models with up to 50 segments plus connections. If you modify an example model and it exceeds this limit, you will need an active license for the **AN-SOF PRO Edition** to re-run 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).

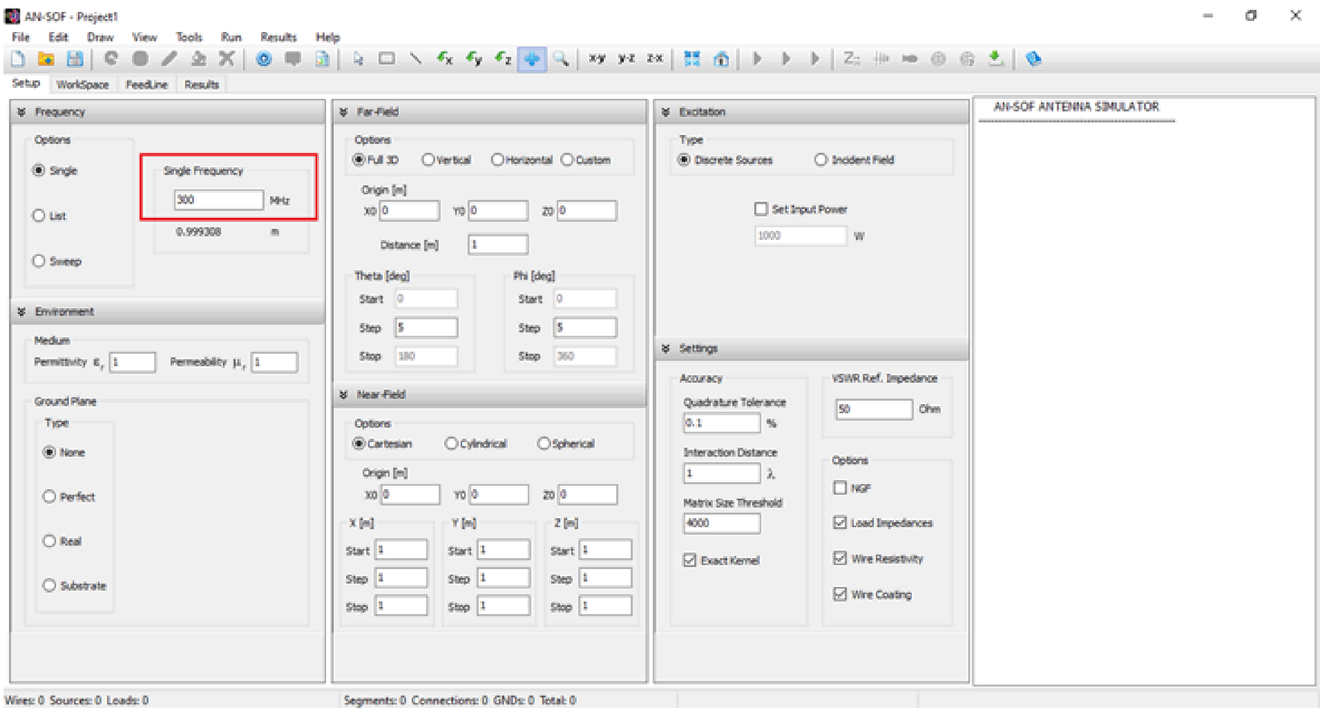


Fig. 9: The Single Frequency option in the Setup tab, where a frequency of 300 MHz is set.

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.

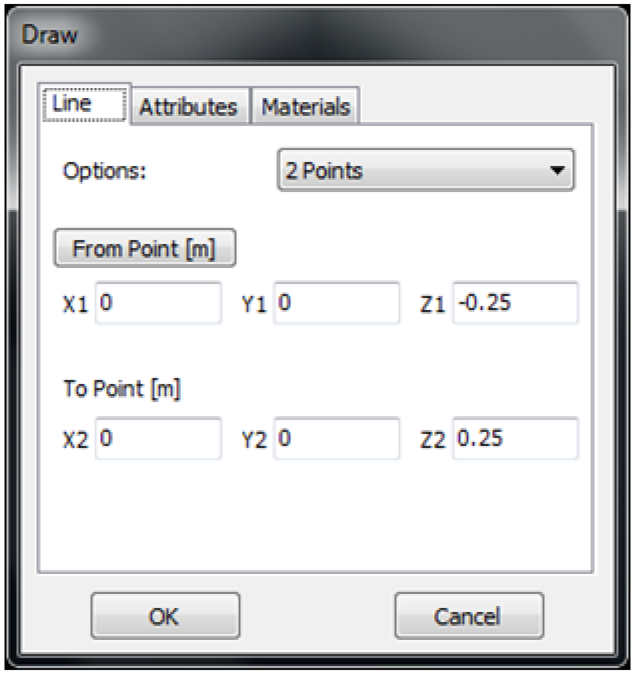


Fig. 10: The Line tab in the Draw dialog box for drawing a straight line.

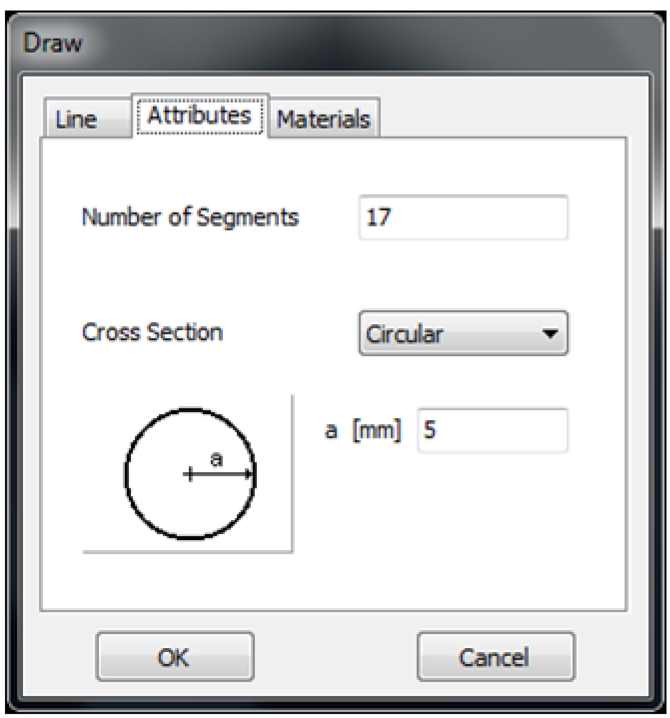


Fig. 11: The Attributes tab in the Draw dialog box, where you can set the number of segments and wire radius.

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.

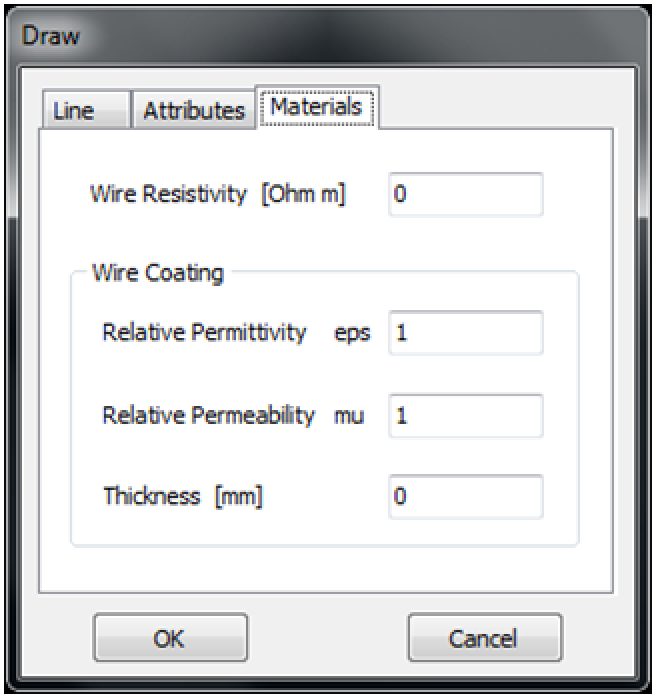


Fig. 12: The Materials tab in the Draw dialog box, used for setting the wire resistivity.

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).

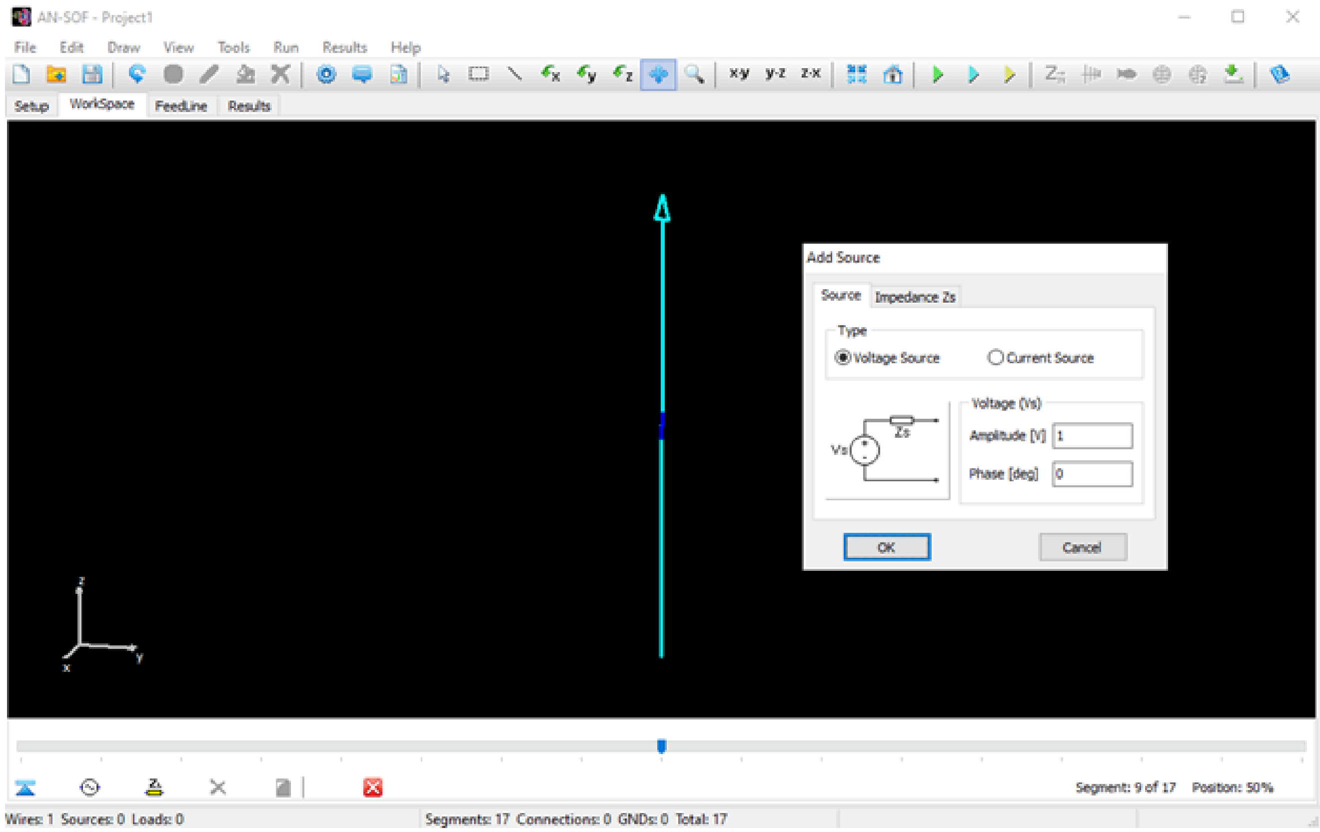


Fig. 13: The Add Source dialog box appears after clicking the Add Source button in the Source/Load toolbar at the bottom of the screen.

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).

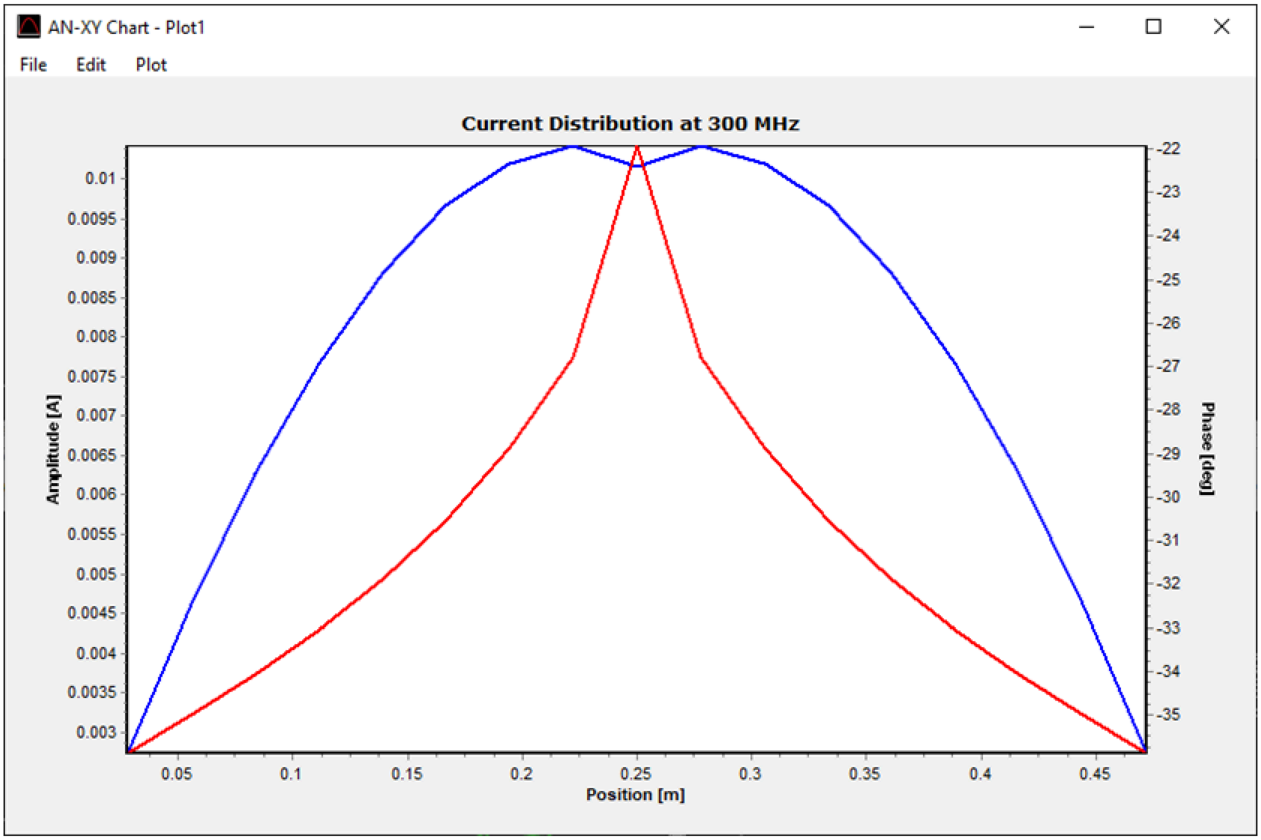


Fig. 14: Current distribution in amplitude and phase along a half-wave dipole.

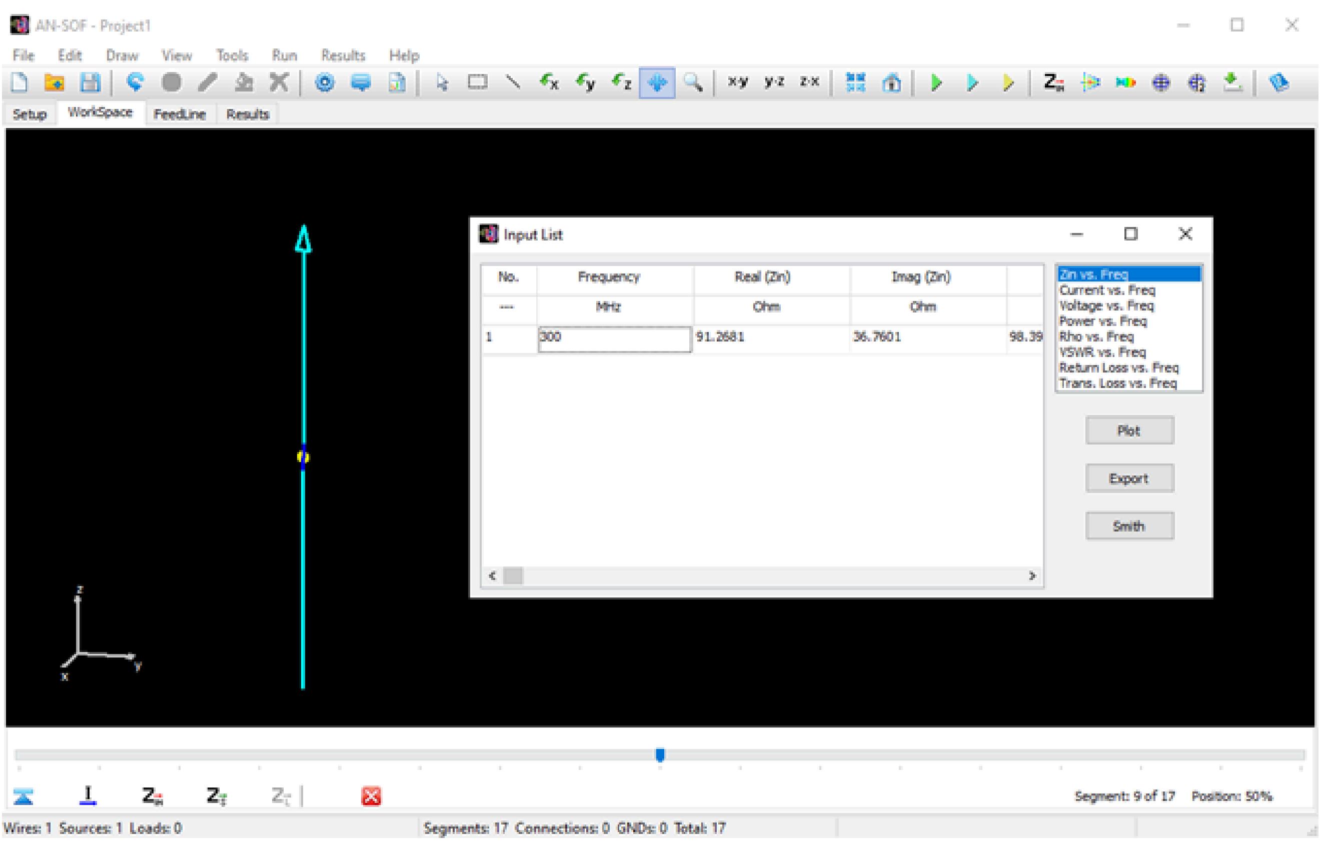


Fig. 15: The Input List dialog box displaying the input impedance.

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).

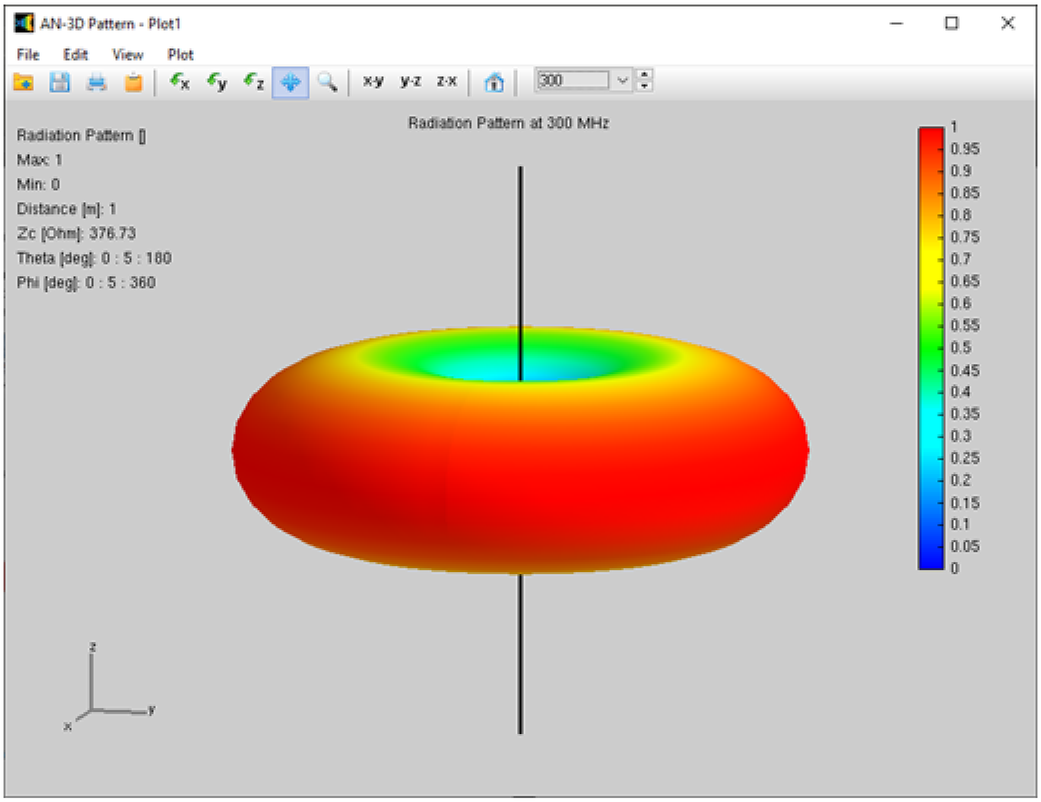


Fig. 16: The radiation pattern of a half-wave dipole exhibits a donut shape.

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.

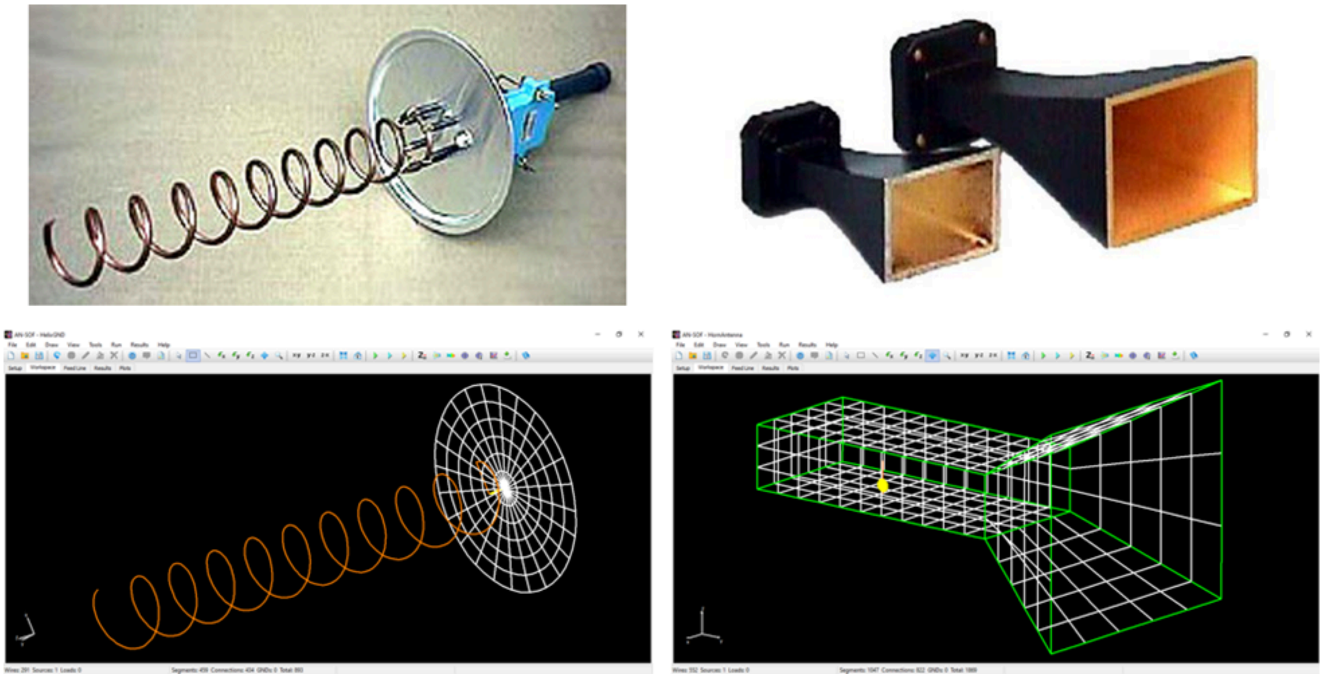


Fig. 1: Antennas modeled by means of wires and wire grids.

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.

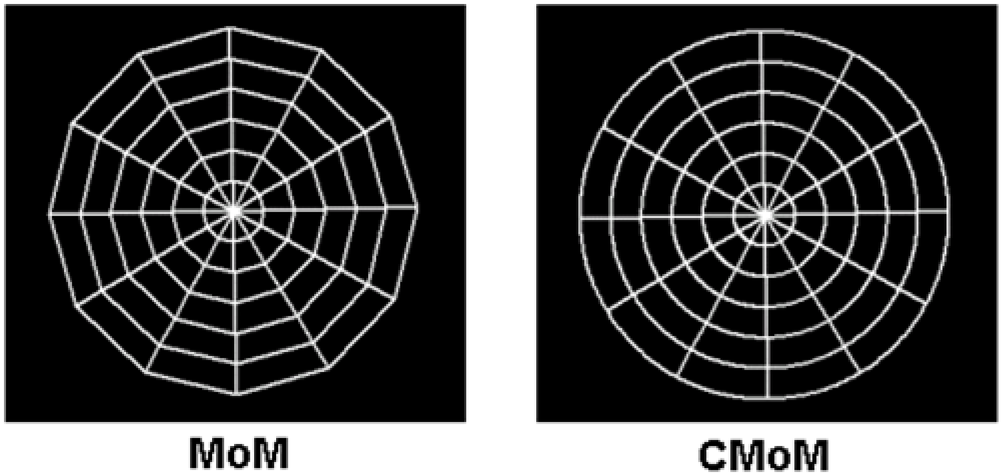


Fig. 2: Modeling of a disc by means of the MoM and CMoM methods.

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

Wire Grids and Solid Surfaces

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**.

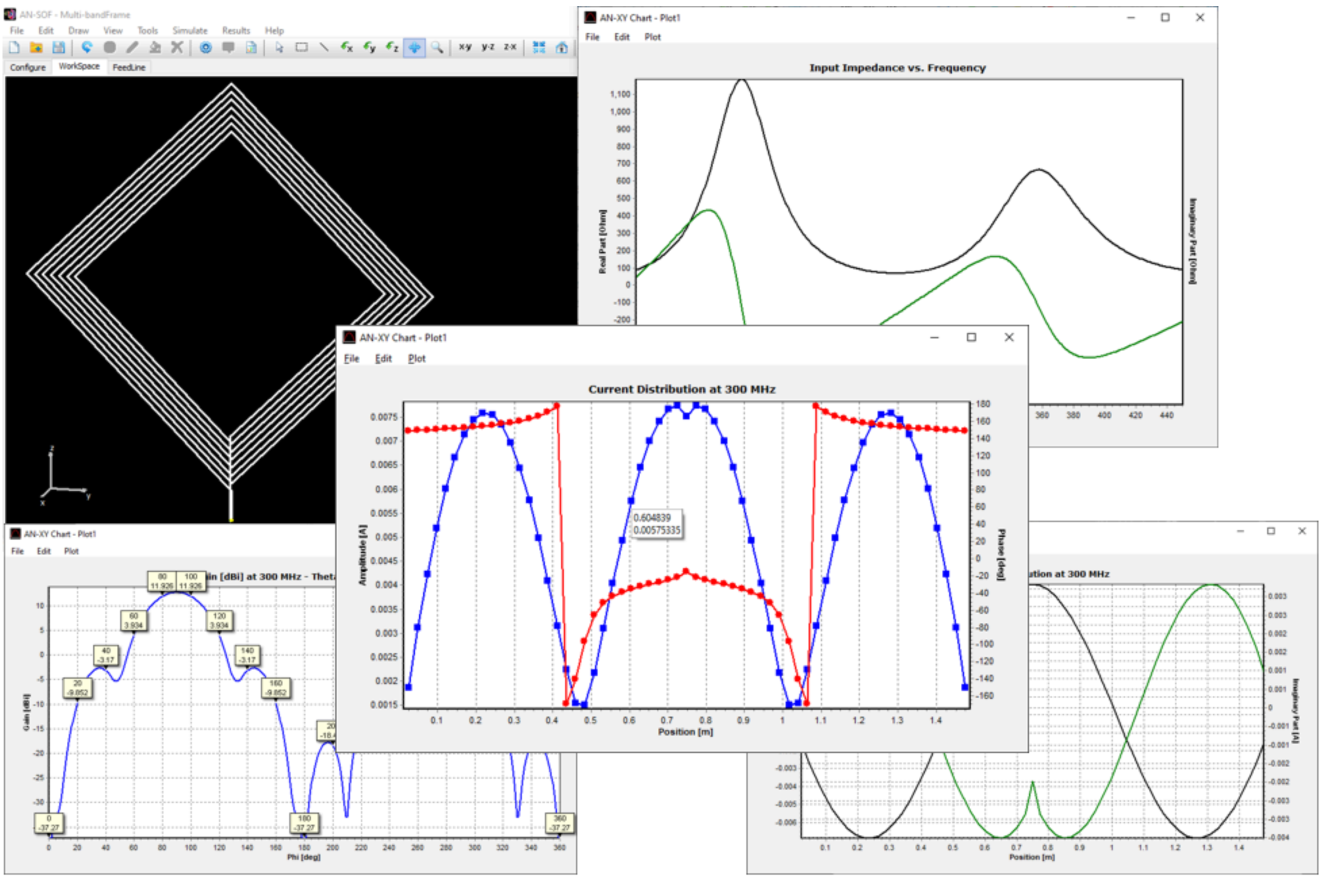
7. **3D radiation patterns** can be viewed from arbitrary angles with zoom functions, colored mesh and surface representations, and a color bar scale. 3D patterns can be plotted with specially designed lighting and illumination for enhanced visualization of simulation results.
8. **Far-field patterns** 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:

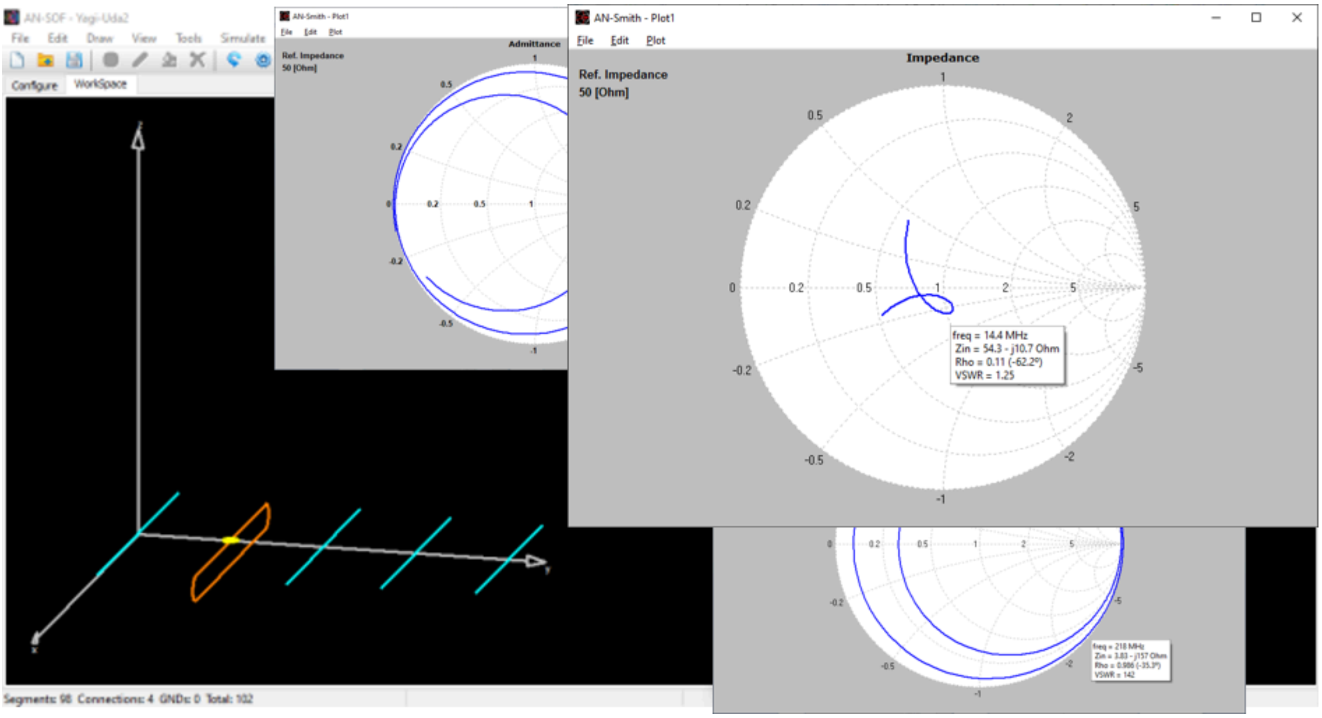
AN-XY Chart app

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.



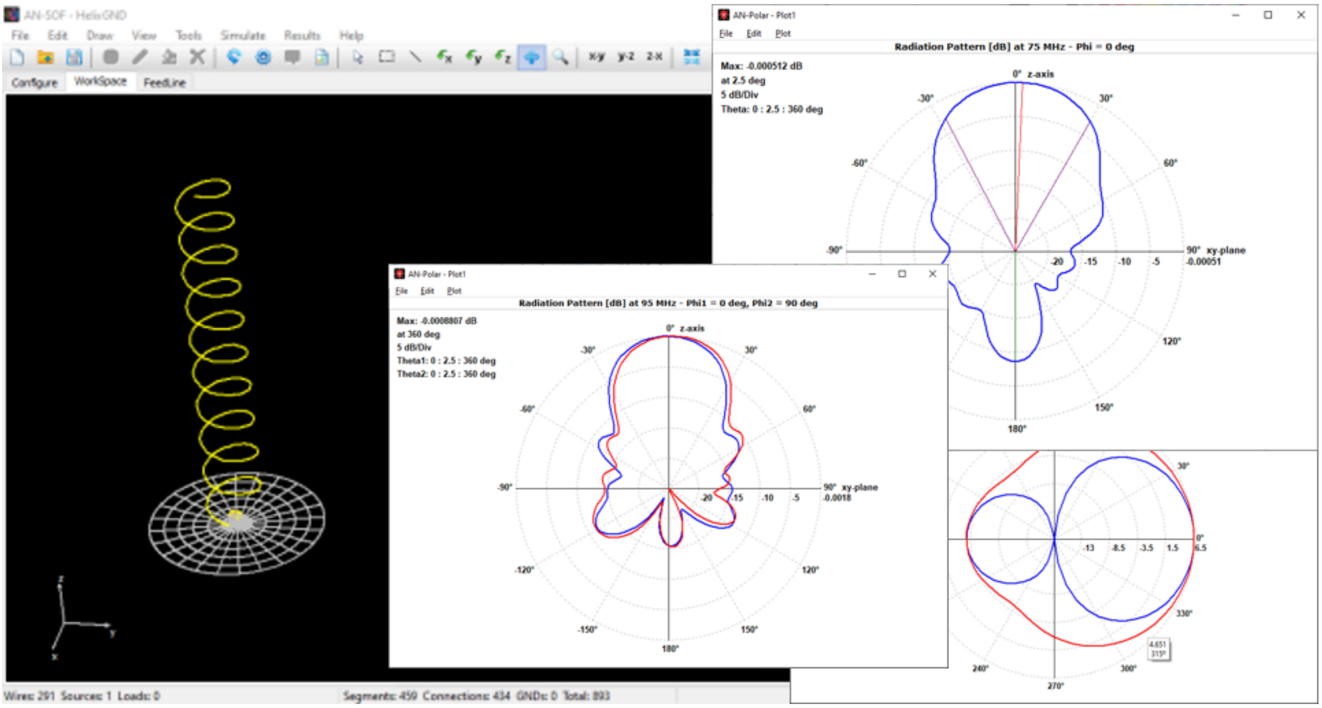
AN-Smith app

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

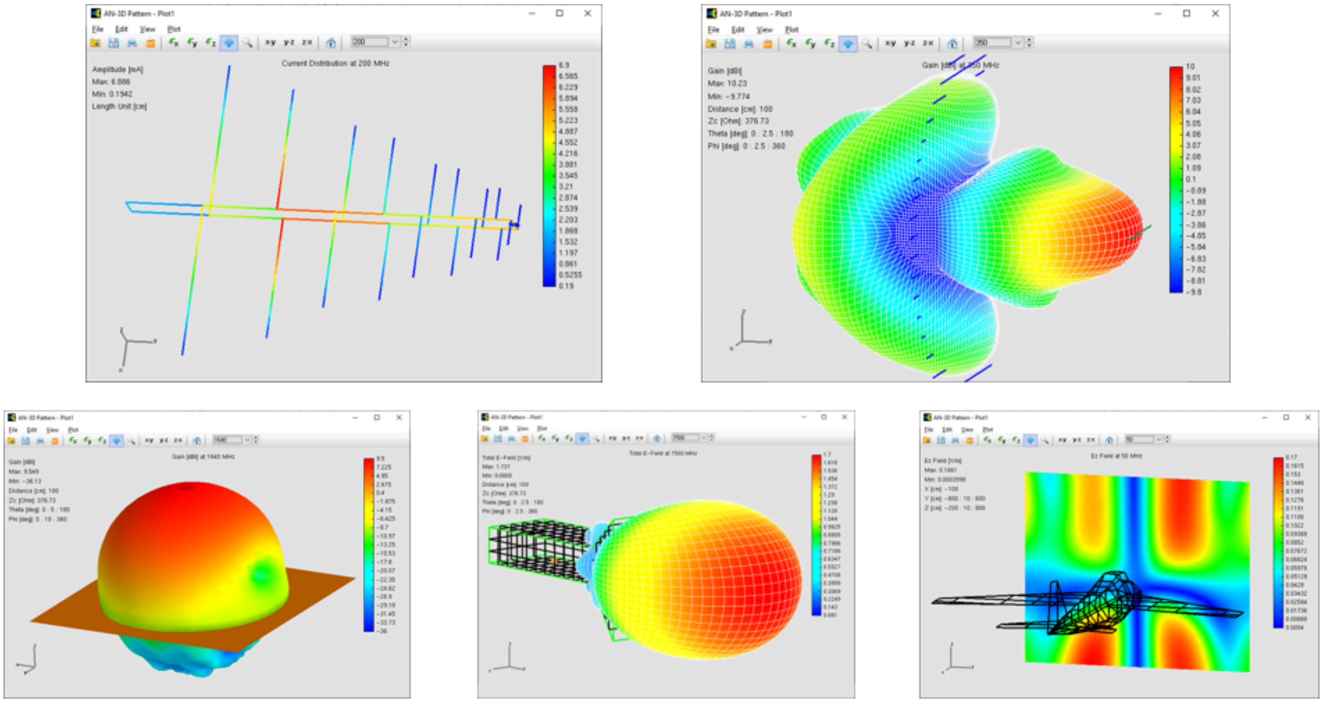
Plot on a **polar diagram** the radiation pattern versus the azimuth (horizontal) or zenith (vertical) angles. The maximum, -3dB and minimum radiation levels are shown within the chart as well as the **beamwidth** and **front-to-rear/back ratio**. Click on the graph to quickly obtain the values of the 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 is started, the initial screen contains the following components:

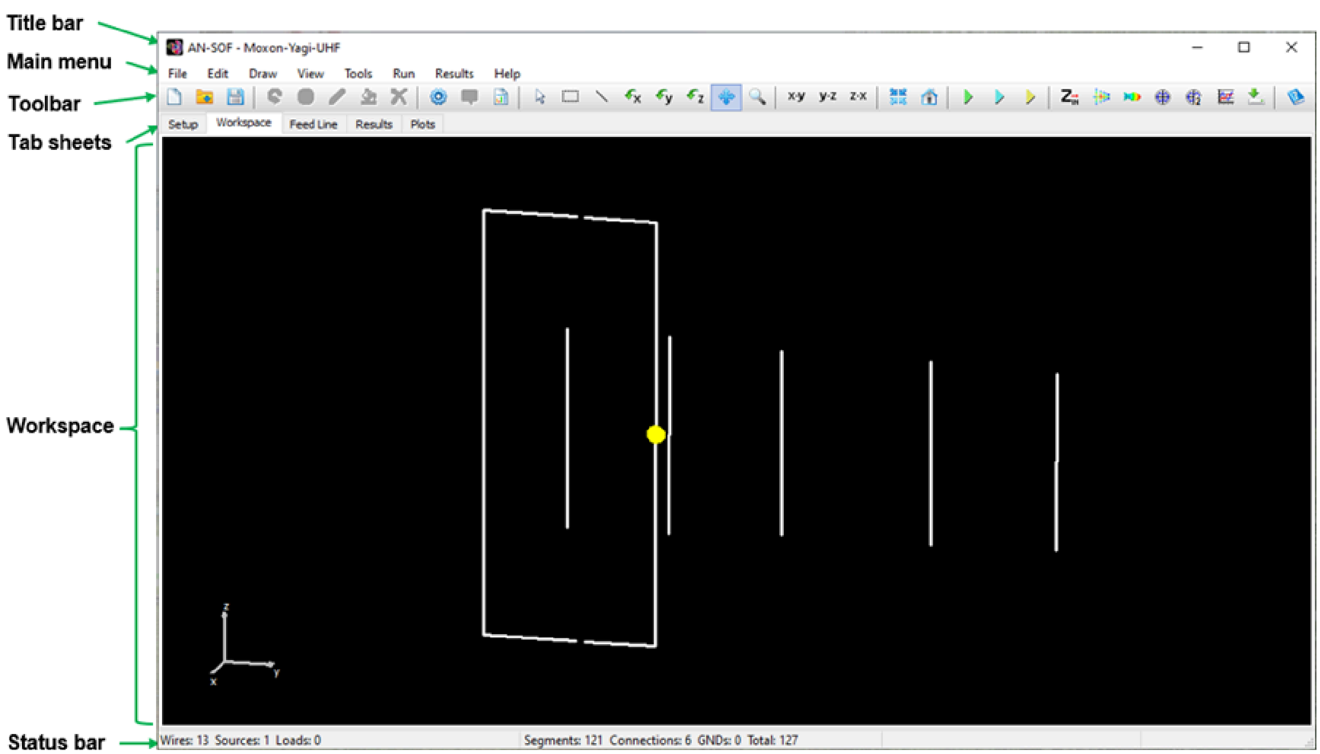


Fig. 1: The AN-SOF interface.

The **title bar** contains the name of the currently active project (.emm file).

The **main menu** bar contains the File, Edit, Draw, View, Tools, Run, Results, and Help menus.

The **main toolbar** contains icons that represent commands.

The **tab sheets** allow us to quickly switch between pages, from Setup to Plots.

The **workspace** is the page where the wire structure can be drawn in a 3D space.

The **status bar** contains information about the number of segments, connections, and ground points.

File Menu

Use the File menu to open, save, close, and print new or existing projects. This menu has the following commands:

New… (Ctrl + N)

Creates a new project.

Open… (Ctrl + O)

Displays the Open dialog box for opening an existing project (.emm file).

Save (Ctrl + S)

Saves the currently active project using its current name.

Save As…

Saves the currently active project using a new name. Also saves a new project using a name specified by the user.

Import Wires

Displays the Import dialog box for importing a list of wires in either AN-SOF (.wre files), NEC, DXF (CAD files) or MM format.

Export Wires

Displays the Export dialog box for exporting wires to a NEC or DXF file.

Copy Workspace

Sends the project workspace to the clipboard as a bitmap image.

Print… (Ctrl + P)

Sends the project workspace to the printer.

Exit (Ctrl + Q) Closes the project that is open and then exits AN-SOF.

Edit Menu

Use the Edit menu commands to edit and handle wires and wire grids. This menu has the following commands:

Undo (Ctrl + Z)

Returns the project to the status before a command was executed.

Source/Load (Ctrl + Ins)

Displays the Source/Load toolbar for exciting or loading the selected wire. This command is enabled when a wire is selected.

Modify (Ctrl + M)

Displays the Modify dialog box for modifying the selected wire or wire grid. This command is enabled when a wire or wire grid is selected.

Wire Color

Displays a Windows(R) dialog box for changing the color of the selected wires. This command is enabled when a wire or group of wires is selected.

Delete (Ctrl + Del)

Deletes the selected wire, wire grid or group of wires with all sources and loads placed on it. This command is enabled when a wire, wire grid or group of wires is selected.

Copy Start Point

Copies the starting point of the selected wire. This point can then be used as the starting point of a second wire, which will be connected to the first one. This command is enabled when a wire is selected.

Copy End Point

Copies the ending point of the selected wire. This point can then be used as the starting point of a second wire, which will be connected to the first one. This command is enabled when a wire is selected.

Start Point to GND

Draws a vertical wire between the start point of the selected wire and the ground plane. This command is shown when a ground plane is included in the model, and it is enabled when a wire is selected.

End Point to GND

Draws a vertical wire between the end point of the selected wire and the ground plane. This command is shown when a ground plane is included in the model, and it is enabled when a wire is selected.

Copy Wires

Displays the Copy Wires dialog box for copying the selected wire or group of wires. The copied wires can then be pasted in a different position. This command is enabled when a wire or group of wires is selected.

Move Wires

Displays the Move Wires dialog box for moving the selected wire or group of wires to a different position. This command is enabled when a wire or group of wires is selected.

Rotate Wires

Displays the Rotate Wires dialog box for rotating the selected wire or group of wires around the chosen axis. This command is enabled when a wire or group of wires is selected.

Scale Wires

Displays the Scale Wires dialog box for scaling the selected wire or group of wires according to the specified scale factor. This command is enabled when a wire or group of wires is selected.

Stack Wires

Displays the Stack Wires dialog box for stacking the selected wire or group of wires along the specified direction and according to the given number of wires in the stack. This command is enabled when a wire or group of wires is selected.

Draw Menu

Use the Draw menu commands to create and draw wires and wire grids. This menu has the following commands:

Line

Opens the Line dialog box for drawing a line or straight wire.

Arc

Opens the Arc dialog box for drawing an arc.

Circle

Opens the Circle dialog box for drawing a circle or circular loop.

Helix

Opens the Helix dialog box for drawing a helix or helical wire.

Quadratic

Opens the Quadratic dialog box for drawing a quadratic wire.

Archimedean Spiral

Opens the Archimedean Spiral dialog box for drawing an Archimedean spiral.

Logarithmic Spiral

Opens the Logarithmic Spiral dialog box for drawing a logarithmic spiral.

Wire Grid

Creates a new wire grid in the workspace. This option has a sub-menu with the following commands:

- **Patch**: Opens the Draw dialog box for drawing a rectangular grid on the xy-plane.
- **Plate**: Opens the Draw dialog box for drawing a plate or bilinear surface.
- **Disc**: Opens the Draw dialog box for drawing a disc.
- **Flat Ring**: Opens the Draw dialog box for drawing a flat ring or a disc with a hole at its center.
- **Cone**: Opens the Draw dialog box for drawing a cone.
- **Truncated Cone**: Opens the Draw dialog box for drawing a truncated cone.
- **Cylinder**: Opens the Draw dialog box for drawing a cylinder.
- **Sphere**: Opens the Draw dialog box for drawing a sphere.
- **Paraboloid**: Opens the Draw dialog box for drawing a parabolic surface.

Tapered Wire

Creates a new tapered wire in workspace. This option has a sub-menu with the same commands as the wire options described above, but each wire can have a stepped radius along its length.

Tabular Input (Ctrl + T)

Opens a table to enter linear wires, sources and loads in spreadsheet format.

View Menu

Use the View menu commands to display or hide different elements of the AN-SOF interface, zoom the wire structure, and view additional information about the project and wires. This menu has the following commands:

Wire Properties… (Ctrl + W)

Displays the Wire Properties dialog box for viewing information about the selected wire. This command is enabled when a wire is selected.

Project Details…

Displays the Project Details dialog box for viewing information about the project that is open.

Zoom In (Ctrl + I)

Increases the size of the view in the workspace (also roll the mouse wheel to zoom).

Zoom Out (Ctrl + K)

Decreases the size of the view in the workspace (also roll the mouse wheel to zoom).

Reset Zoom Scale

Resets the zoom and resizes the view of the structure in the workspace.

Axes (Ctrl + A)

Displays the Axes dialog box for changing the appearance of the axes in the workspace. Press F7 to switch between small and main axes.

X-Y Plane / Y-Z Plane / Z-X Plane

Shows a view of the xy-plane/ yz-plane/ zx-plane parallel to the screen.

Center

Centers the view of the structure in the workspace (double click on the workspace to center the view).

Initial View (Home)

Returns the workspace to the initial view.

Drawing Panel

Displays a panel to the left of the workspace that contains buttons for quicker access to commands for drawing wires and wire grids.

Tools Menu

Use the Tools menu commands to display 3D, polar, rectangular, and Smith charts and to check the wires. This menu has the following commands:

3D Chart

Executes the AN-3D Pattern application for opening 3D plot files (.p3d).

Polar Chart

Executes the AN-Polar application for opening polar plot files (.plr).

Rectangular Chart

Executes the AN-XY Chart application for opening rectangular plot files (.plt).

Smith Chart

Executes the AN-Smith application for opening Smith chart files (.sth).

Check Individual Wires

Checks the segment length, cross-section size and thin-wire ratio of each wire. Wires in warning/error will be highlighted in yellow/red.

Check Wire Spacing

Checks the spacing between wires. Wires in warning/error will be highlighted in yellow/red.

Delete Duplicate Wires

Deletes duplicate or overlapping wires.

Calculator

Executes the Microsoft Windows(R) Calculator application.

Preferences

Displays the Preferences dialog box for setting up the preferred options for unit systems, workspace color, pen width, confirmation questions, etc.

Run Menu

Use the Run menu commands to run the calculations. This menu has the following commands:

Run ALL (F10)

Runs the calculation of the current distribution, far- and near-fields.

Run Currents and Far-Field (F11)

Runs the calculation of the current distribution and far-fields.

Run Currents and Near-Field (F12)

Runs the calculation of the current distribution and near electric and magnetic fields.

Run Currents

Runs the calculation of the current distribution on the wire structure. This command is disabled when the currents are already computed.

Run Far-Field

Runs the calculation of the far-field generated by the currents flowing on the wire structure. This command is enabled when the currents are already computed.

Run Near E-Field

Runs the calculation of the near electric field generated by the currents flowing on the wire structure. This command is enabled when the currents are already computed.

Run Near H-Field

Runs the calculation of the near magnetic field generated by the currents flowing on the wire structure. This command is enabled when the currents are already computed.

Run Bulk Simulation Opens a dialog box for choosing multiple files in NEC format at the same time. The file extension must be “.nec”. AN-SOF will import these input files and compute the corresponding output results. The results will be saved as CSV files in the same directory as the NEC input files.

Results Menu

Use the Results menu commands to visualize the results from a simulation. This menu has the following commands:

Plot Current Distribution

Executes the AN-3D Pattern application for plotting the current distribution as a colored pattern on the wire structure.

Plot Currents

Executes the AN-XY Chart application for plotting the currents vs. position along the selected wire. This command is enabled when a wire has been selected.

List Currents

Displays the List Currents toolbar for listing the currents vs. frequency at the chosen segment on the selected wire. If the segment has a source on it, the list of input impedances, voltages, and powers as a function of frequency can be shown. This command is enabled when a wire has been selected.

List Input Impedances

Displays a table with the list of input impedances vs. frequency, including reflection coefficient, VSWR, return loss and transmission loss at the antenna terminals.

Plot Far-Field Pattern

This option has a sub-menu with the following commands:

- **3D Plot**: Executes the AN-3D Pattern application for plotting a 3D view of the radiation patterns.
- **Polar Plot 1 Slice**: Displays the Radiation Pattern Cut dialog box for selecting a 2D slice of the 3D far-field pattern. Then, the selected 2D pattern cut will be plotted in polar coordinates by the AN-Polar application.
- **Polar Plot 2 Slices**: Displays a dialog box for selecting two slices of the 3D far-field pattern. Then, the selected 2D patterns will be plotted in polar coordinates by the AN-Polar application.

- **2D Rectangular Plot**: Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut will be plotted in rectangular coordinates by the AN-XY Chart application.

Plot Far-Field Spectrum

Displays the Select Far-Field Point dialog box for selecting a point in space where the far-field components will be shown versus frequency. Then, the far-field spectrum will be plotted in rectangular coordinates by the AN-XY Chart application.

List Far-Field Pattern

Displays a table showing the total E-field and its components (E-theta, E-phi, E-right, E-left) at the grid of angles theta and phi specified in the Far-Field panel of the Setup tabsheet. This table can be exported as a CSV file.

List Far-Field Spectrum

Displays the Select Far-Field Point dialog box for selecting a point in space where the far-field components will be shown versus frequency. Then, this far-field spectrum will be listed in a table with different columns for the total E-field and the field components: E-theta and E-phi (spherical components) and the right and left polarized components.

Power Budget/RCS

Displays the Power Budget dialog box for listing the total input power, consumed and radiated powers, power densities, efficiency, directivity and gain vs. frequency. In the case of plane wave excitation, the Radar Cross Section (RCS) vs. frequency will be displayed.

Plot Near E-Field Pattern

This option has a sub-menu with the following commands:

- **3D Plot**: Executes the AN-3D Pattern application for plotting a 3D view of the near electric field components.
- **2D Plot**: Displays the Near-Field Cut dialog box for selecting a 2D cut of the near electric field pattern. Then, the selected 2D pattern cut will be plotted by the AN-XY Chart application.

Plot Near E-Field Spectrum

Displays the Select Near-Field Point dialog box for selecting a point where the near electric field components will be shown versus frequency. Then, this near-field spectrum will be plotted in rectangular coordinates by the AN-XY Chart application.

List Near E-Field Pattern

Displays a table showing the total near E-field and its components at the grid of points specified in the Near-Field panel of the Setup tabsheet. This table can be exported as a CSV file.

List Near E-Field Spectrum

Displays the Select Near-Field Point dialog box for selecting a point where the near electric field components will be shown versus frequency. Then, this near-field spectrum will be listed in a table with different columns for the field components.

Plot Near H-Field Pattern

This option has a sub-menu with the following commands:

- **3D Plot**: Executes the AN-3D Pattern application for plotting a 3D view of the near magnetic field components.
- **2D Plot**: Displays the Near-Field Cut dialog box for selecting a 2D cut of the near magnetic field pattern. Then, the selected 2D pattern cut will be plotted by the AN-XY Chart application.

Plot Near H-Field Spectrum

Displays the Select Near-Field Point dialog box for selecting a point where the near magnetic field components will be shown versus frequency. Then, the near-field spectrum will be plotted in rectangular coordinates by the AN-XY Chart application.

List Near H-Field Pattern

Displays a table showing the total near H-field and its components at the grid of points specified in the Near-Field panel of the Setup tabsheet. This table can be exported as a CSV file.

List Near H-Field Spectrum

Displays the Select Near-Field Point dialog box for selecting a point where the near magnetic field components will be shown versus frequency. Then, the near-field spectrum will be listed in a table with different columns for the field components.

Help Menu

Use the Help menu to access the user guide, request technical support, activate a license, or view the version of AN-SOF. This menu has the following commands:

User Guide

Displays the AN-SOF user guide in PDF format.

AN-SOF Home Page

Goes to the AN-SOF web page at **www.antennasimulator.com** in the default web browser.

Knowledge Base

Goes to the **knowledge base >** where you can search for categorized information.

Email to Tech Support

Executes the default e-mail client to send a technical support request to **info@antennasimulator.com**.

Chat to Tech Support

Goes to the live chat page in the default web browser.

Activation Key

Executes the AN-Key application to activate a license.

Check for Updates

Goes to the website where the **latest AN-SOF releases >** are posted.

About AN-SOF

Shows copyright and version information.

Main Toolbar

The main toolbar has the following icons and associated commands:

Fig. 2: Main Toolbar.

New (Ctrl + N)

Creates a new project.

Open (Ctrl + O)

Displays the Open dialog box to open an existing project (.emm file).

Save (Ctrl + S)

Saves the currently active project using its current name.

Undo (Ctrl + Z)

Returns the project to the status before a command was executed.

Source/Load (Ctrl + Ins)

Displays the Source/Load toolbar for adding a source or load to the selected wire. This command is enabled when a wire has been selected.

Modify (Ctrl + M)

Displays the Modify dialog box for modifying the selected wire or group of wires. This command is enabled when a wire or group of wires has been selected.

Wire color

Displays a Windows(R) dialog box for changing the color of the selected wire or group of wires. This command is enabled when a wire or group of wires has been selected.

Delete (Ctrl + Del)

Deletes the selected wire, wire grid or group of wires with all sources and loads placed on it. This command is enabled when a wire, wire grid or group of wires has been selected.

Preferences

Displays the Preferences dialog box for setting up the preferred options for unit systems, workspace color, pen width, confirmation questions, etc.

Wire Properties (Ctrl + W)

Displays the Wire Properties dialog box for viewing information about the selected wire. This command is enabled when a wire has been selected.

Project Details

Displays the Project Details dialog box for viewing information about the currently active project.

Select Wire

Enables the selection mode where a wire can be selected individually by left clicking on it.

Selection Box

Enables the selection mode where a group of wires can be selected expanding a box with the mouse (left mouse button pressed).

Draw Line

Enables the drawing mode where a line can be dragged with the mouse (left mouse button pressed). This mode is enabled when the X-Y, Y-Z or Z-X view has been chosen. The coordinates of the starting and ending points of the line will be shown in the status bar.

Rotate around X/Y/Z/3D

Enables the 3D rotation of the view or around the x/y/z-axis by moving the mouse.

Move

Enables the movement of the view by moving the mouse (left mouse button pressed).

Zoom

This allows you to expand a rectangle and select the area of the screen you wish to zoom in on. Additionally, you can use the mouse wheel to adjust the zoom level of the view.

X-Y / Y-Z / Z-X Plane

Shows a view of the xy/yz/zx-plane parallel to the screen.

Center

Centers the view of the structure on the workspace.

Initial View (Home)

Returns the workspace to the initial view.

Run ALL (F10)

Runs the calculation of the current distribution, far- and near-fields.

Run Currents and Far-Field (F11)

Runs the calculation of the current distribution and far-fields.

Run Currents and Near-Field (F12)

Runs the calculation of the current distribution and near-fields.

List Input Impedances

Shows a table with the input impedances vs. frequency. Reflection coefficient, VSWR, return and transmission losses at the antenna terminals are also tabulated.

Plot Current Distribution

Executes the AN-3D Pattern application for plotting the current distribution as a colored pattern on the wire structure.

Far-Field 3D Plot

Executes the AN-3D Pattern application for plotting a 3D view of the radiation pattern.

Far-Field Polar 1 Slice

Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut will be plotted in a polar chart by the AN-Polar application.

Far-Field Polar 2 Slices

Displays a dialog box for selecting two slices of the 3D far-field pattern. Then, the selected 2D patterns will be plotted in a polar chart by the AN-Polar application.

Far-Field 2D Plot

Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut will be plotted in rectangular coordinates by the AN-XY Chart application.

Export Results

Opens a dialog box to save the results displayed in the “Results” tab as a CSV file.

User Guide

Opens the user guide file in PDF format.
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.

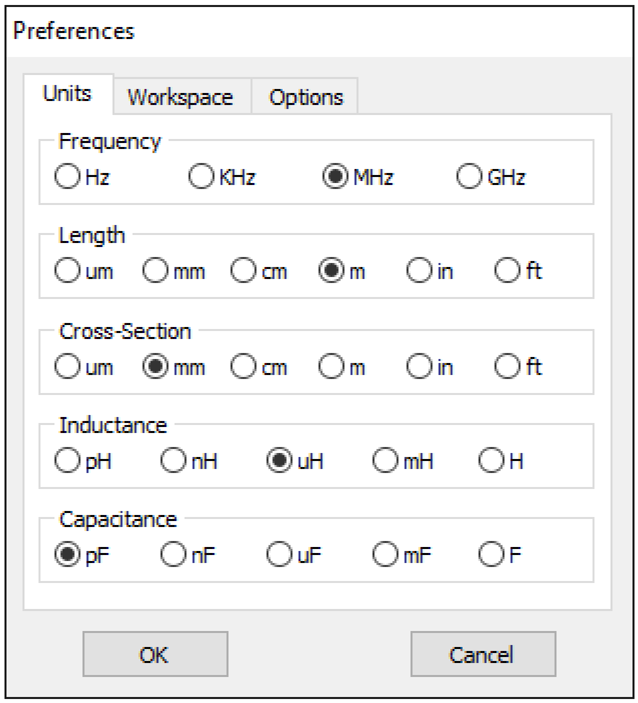


Fig. 1: Units tab in the Preferences dialog box where frequencies, lengths, wire cross-sections, inductances, and capacitances can be set.

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.

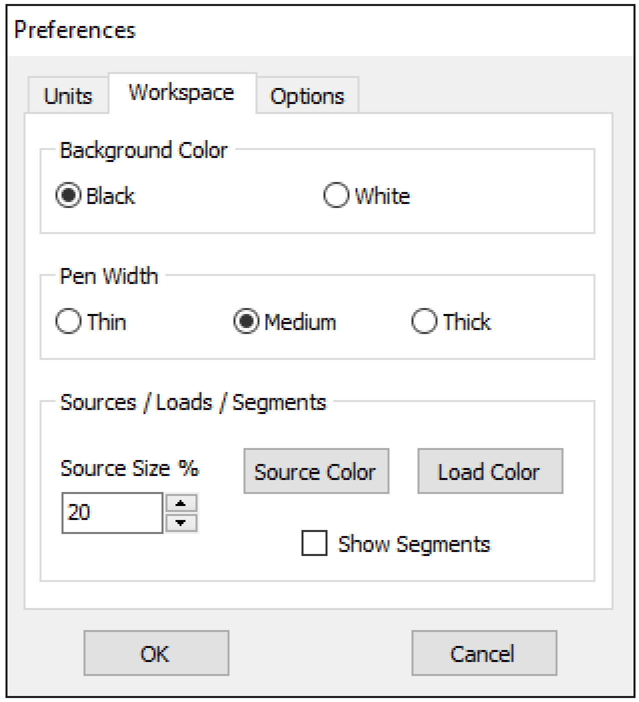


Fig. 2: Preferences dialog box showing the Workspace tab, where the workspace background color, pen width, and appearance of sources/loads can be set.

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.

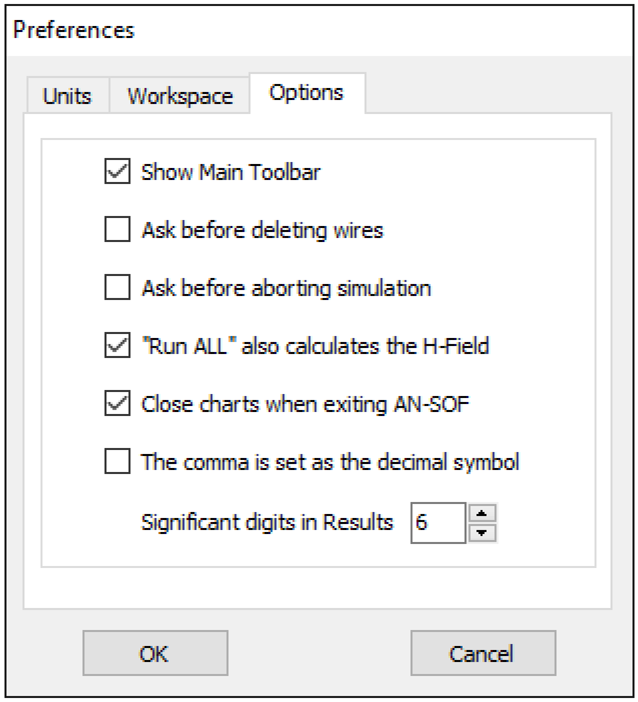


Fig. 3: Options tab in the Preferences dialog box, where various additional settings can be configured.

> **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.

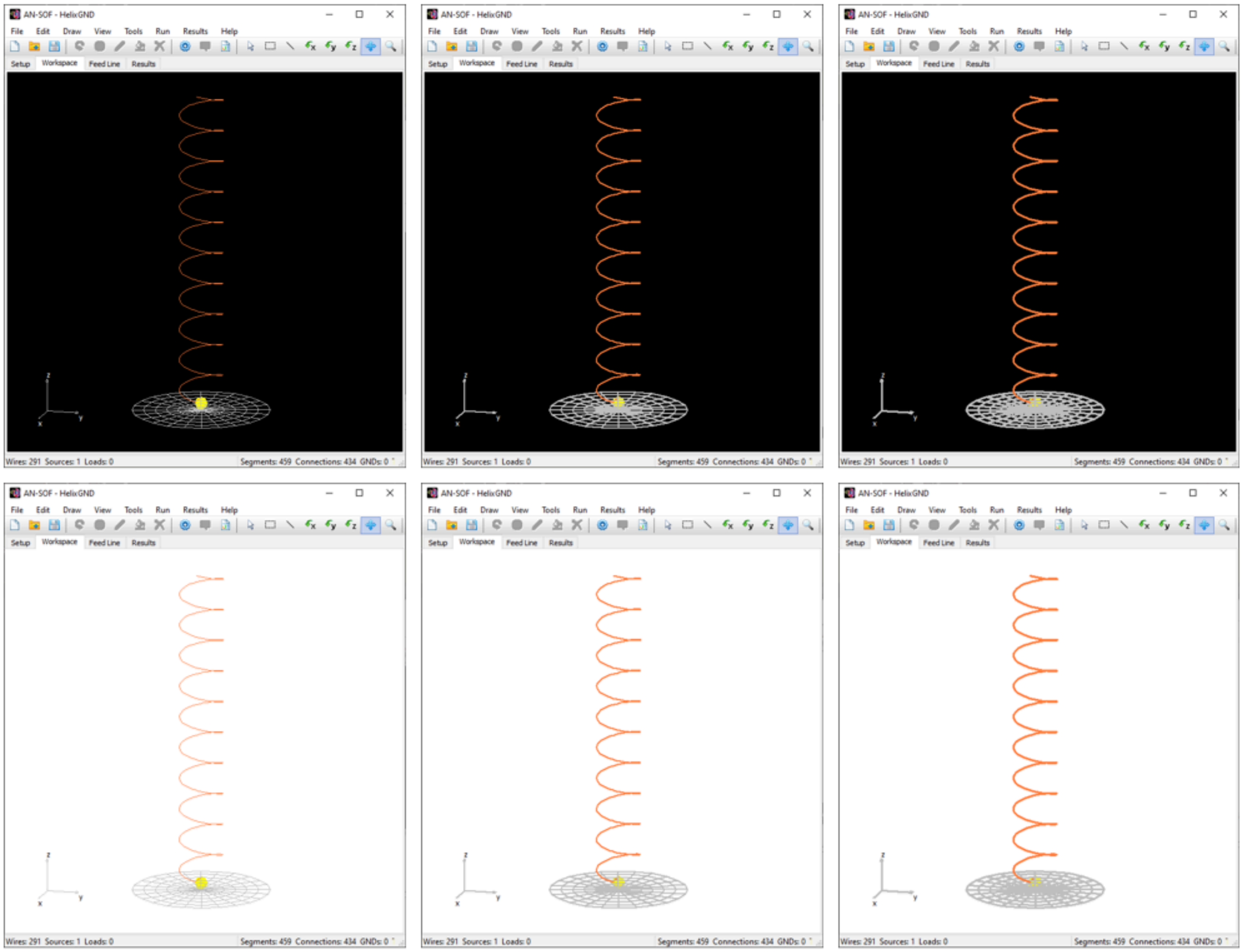


Fig. 1: Display options in the workspace.

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**.

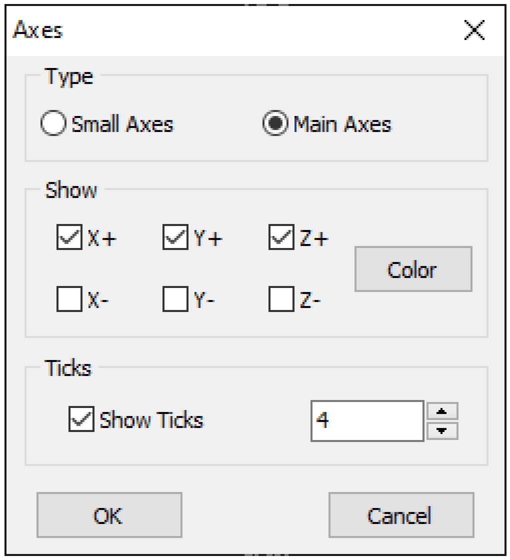


Fig. 2: Axes dialog box, showing options to display positive and negative axes.

> **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:

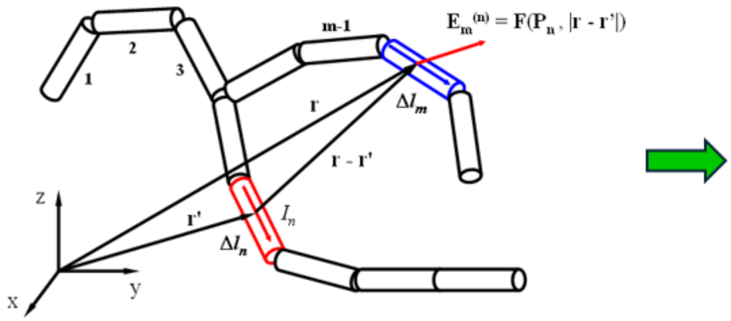
1. Click the **Move** button on the toolbar.
2. Move the mouse while holding the left button.

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.



$$\begin{bmatrix} V_1 \\ V_2 \\ \cdots \\ V_N \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} & \cdots & Z_{1N} \\ Z_{21} & Z_{22} & \cdots & Z_{2N} \\ \cdots & \cdots & \cdots & \cdots \\ Z_{N1} & Z_{N2} & \cdots & Z_{NN} \end{bmatrix} \cdot \begin{bmatrix} I_1 \\ I_2 \\ \cdots \\ I_N \end{bmatrix}$$

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**.

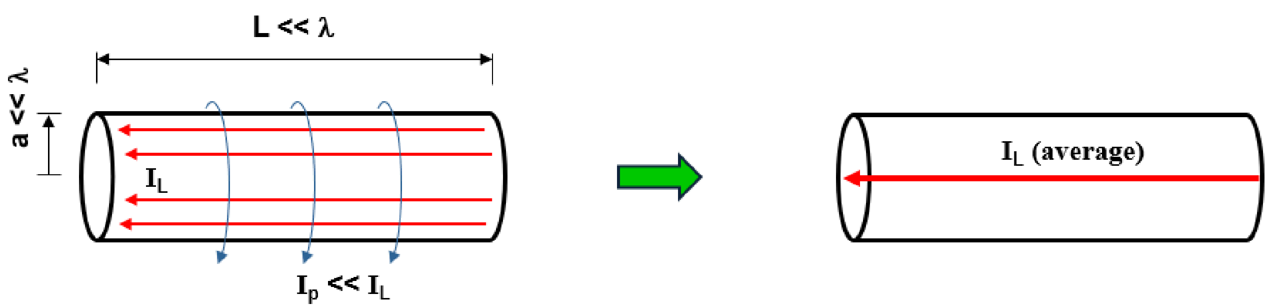


Fig. 2: Illustration of the thin-wire approximation for a wire segment in the Method of Moments.

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.

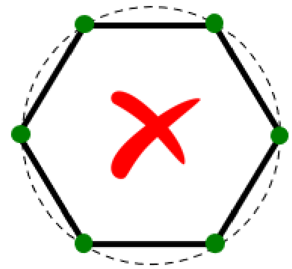
Eliminating assumption 1 has a significant impact on the **accuracy** of calculations, particularly in the current distribution near the antenna's feed point or terminals, where obtaining precise values for input impedance and standing wave ratio (SWR) is crucial. In addition to discarding assumption 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:

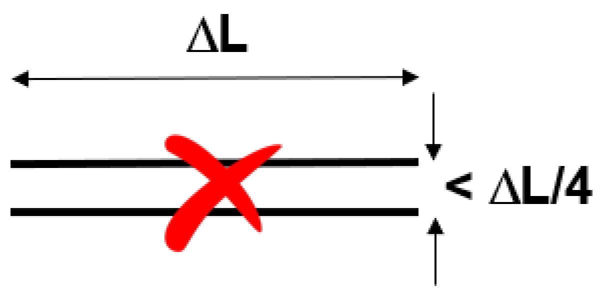
1. No curved wires:

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.



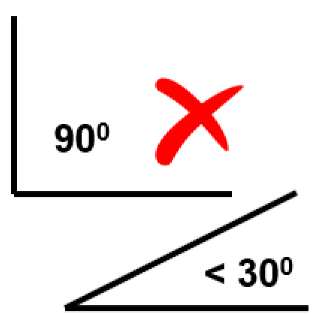
2. Wire spacing limitation:

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.



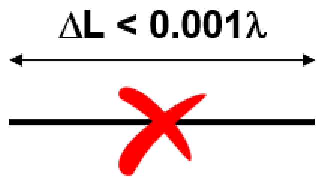
3. Issues with bent wires:

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.



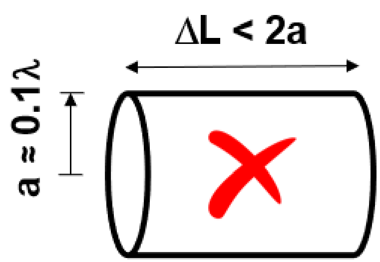
4. Short segment constraint:

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.



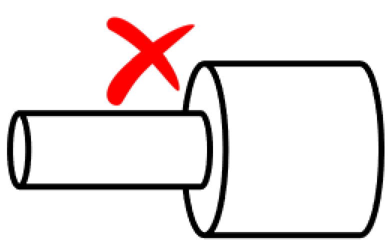
5. Thin wire requirement:

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.



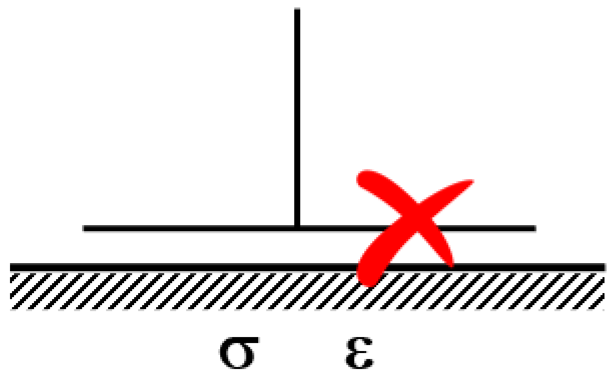
6. Tapered wire issues:

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

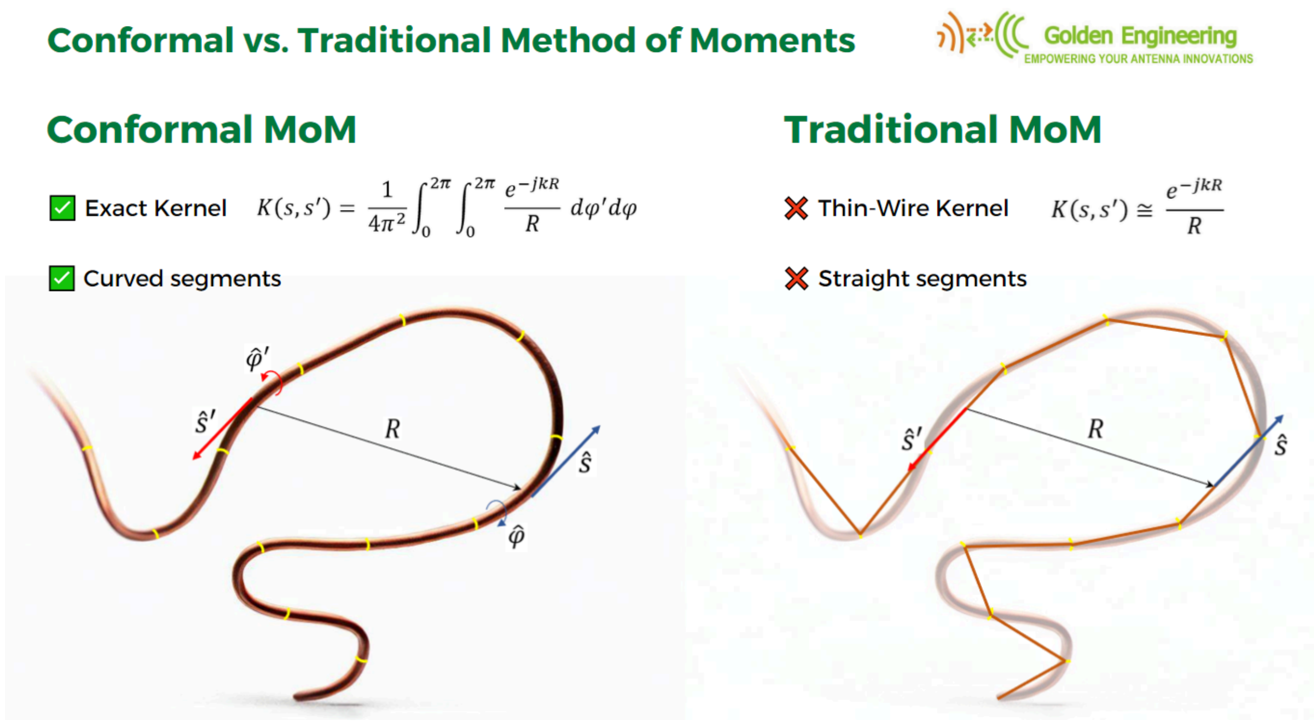


7. Proximity to lossy ground plane affects horizontal wires:

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

The simulation parameters can be set in the Setup tabsheet. This page has the following panels: **Frequency**, **Environment**, **Far-Field**, **Near-Field**, **Excitation**, and **Settings**, Fig. 1.

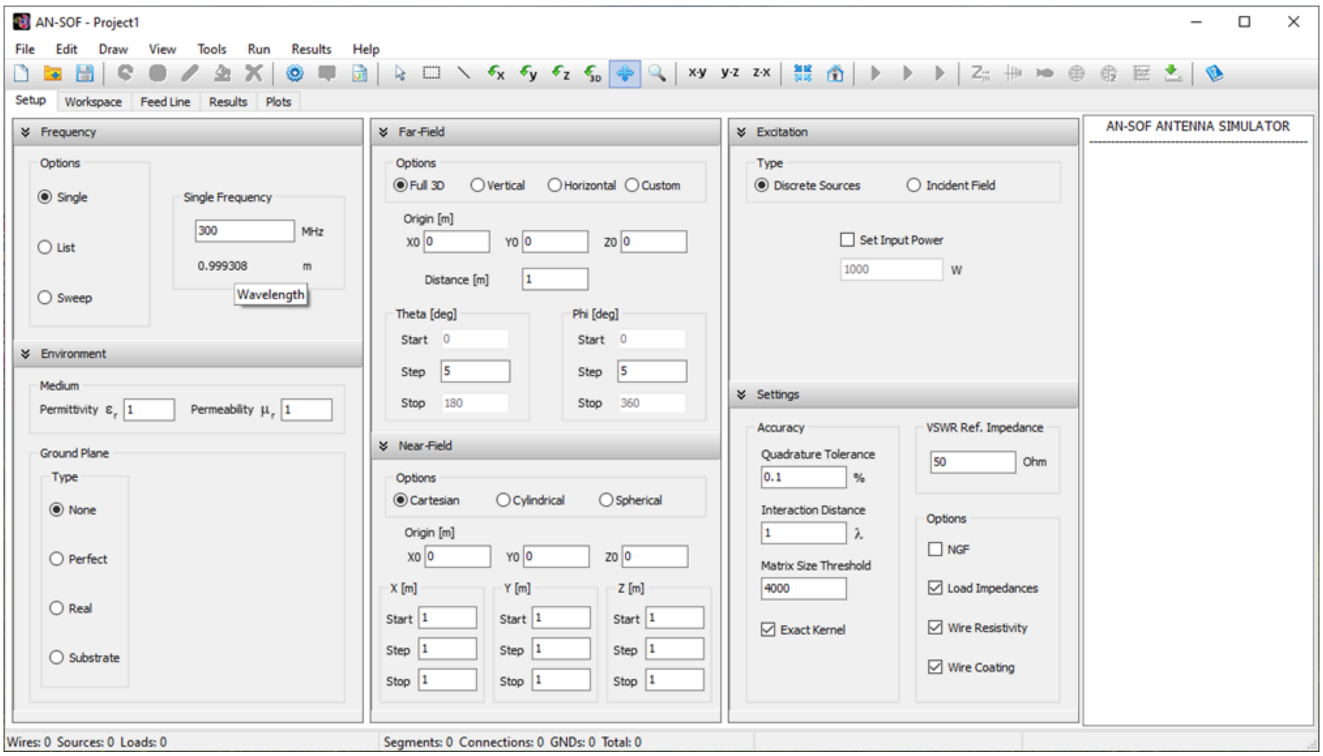


Fig. 1: Setup tab where the simulation parameters can be set.

In the **Frequency panel >**, the project operating frequencies can be specified.

In the **Environment panel >**, the relative permittivity and permeability of the surrounding medium and the type of ground plane can be set.

In the **Far-Field panel >**, the angular ranges for the calculation of the far-field can be set.

In the **Near-Field panel >**, the observation points for the calculation of the near-field can be set.

In the **Excitation panel >**, the type of excitation for the structure can be set. When discrete sources are chosen as excitation, the total input power can be specified. When an incident field is chosen as excitation, the incoming direction and polarization for the incident wave can be specified.

In the **Settings panel >**, additional parameters can be set, such as the reference impedance for VSWR and the accuracy of the calculations.

On the right side of the Setup page there is a **Note panel** to write notes associated to the project. These notes will be saved in a text file in the same path as the project file and with the same name as the project.

Specifying the Frequencies

Go to the Setup tab in the main window and select the **Frequency** panel.

The Frequency panel has three options: **Single**, **List** and **Sweep**. By choosing one of these options the simulation can either be performed for a single frequency, for frequencies taken from a list or for a frequency sweep.

If **Single** is chosen, enter the frequency in the *Single Frequency* box, as shown in Fig. 1. The **wavelength** will be shown below the frequency.

If **List** is chosen, write the list of frequencies in the *Frequency List* box, Fig. 2. A list from a text file can be read by pressing the Open button. The frequency list can also be saved to a text file by pressing the Save button.

If **Sweep** is selected, it can either be linear or logarithmic. For a linear sweep the start, step and stop frequencies must be set, Fig. 3. For a logarithmic frequency sweep the start, stop and a multiplication factor must be set, Fig. 4.

The frequency unit can be changed going to **Tools > Preferences** in the main menu and choosing a suitable unit in the **Units** page of the **Preferences** dialog box. Refer to **Preferences >**.

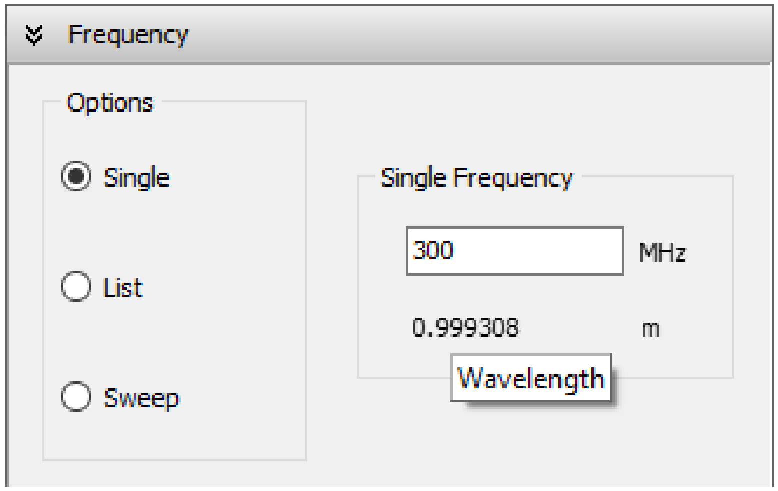


Fig. 1: Frequency panel in the Setup tabsheet. A single frequency is set.

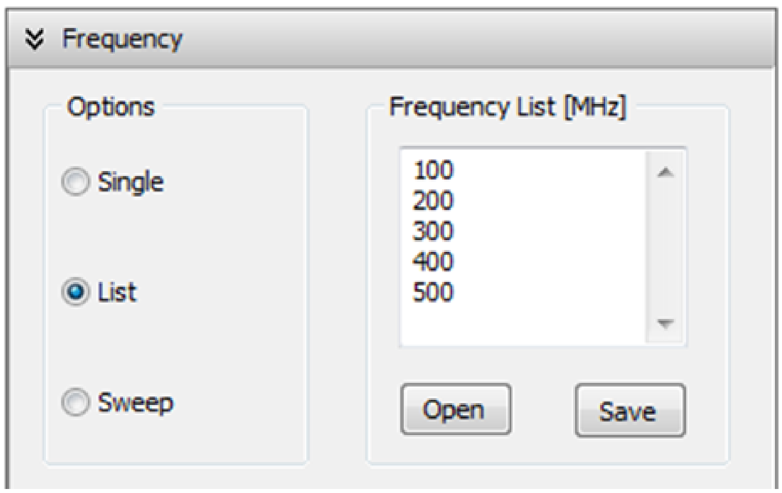


Fig. 2: Frequency panel in the Setup tabsheet. A list of frequencies is set.

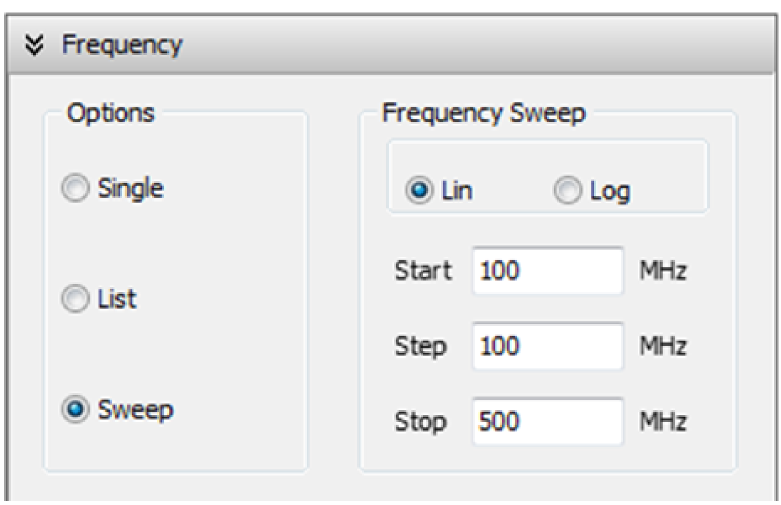


Fig. 3: Frequency panel in the Setup tabsheet. A linear frequency sweep is set.

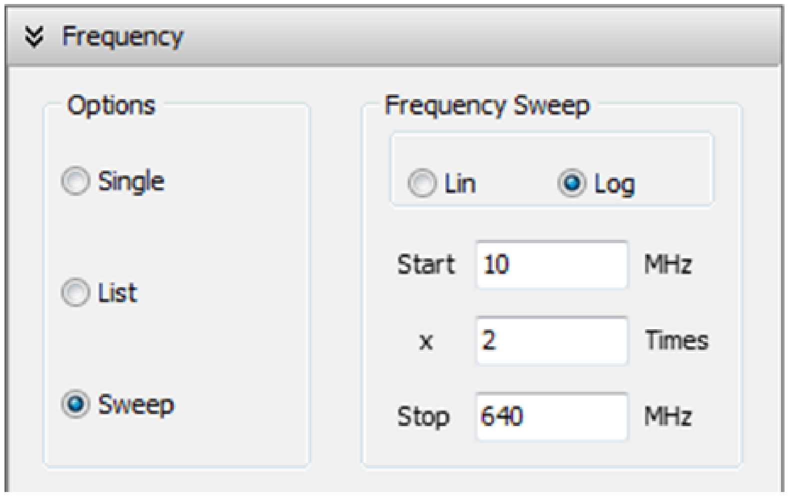


Fig. 4: Frequency panel in the Setup tabsheet. A logarithmic frequency sweep is set.

Defining the Environment

Ground Plane Options

Navigate to the **Setup** tab in the main window and access the **Environment** panel. You can adjust the **relative permittivity** and **permeability** of the surrounding medium within the **Medium** box, as shown in Fig. 1.

There are four ground plane options available:

None

When the **None** ground plane is selected, the simulation will be conducted in **free space**, with the relative **permittivity** and **permeability** values set in the **Medium** box (see Fig. 1).

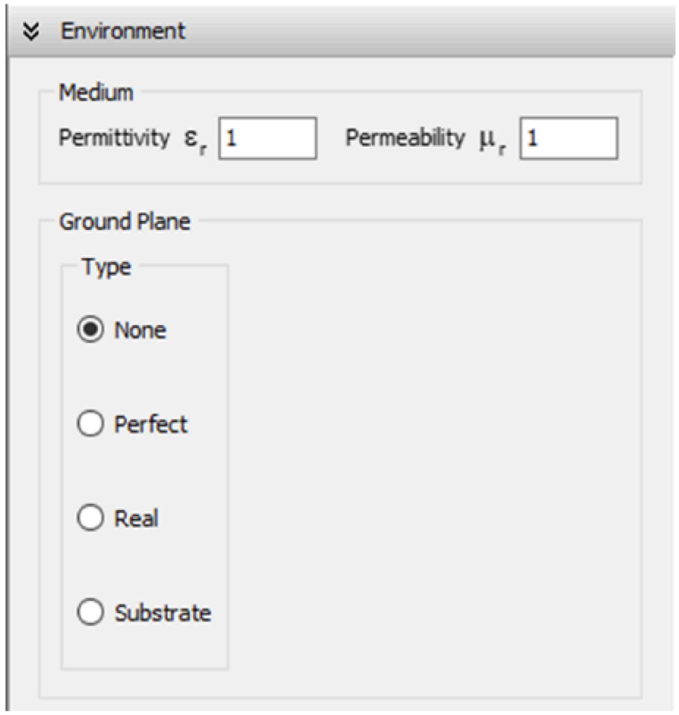


Fig. 1: Medium and Ground Plane boxes in the Environment Panel. None ground plane is chosen (free space).

Perfect

An **infinitely large perfectly electrically conducting (PEC) ground plane** will be positioned at the specified height from the xy-plane ("Z Position" in Fig. 2). Consequently, the ground plane will be parallel to the xy-plane. The "Z" position determines the height of the ground plane above the xy-plane, with a negative Z indicating placement below the xy-plane.

When the **Perfect** option is selected, **all wires must be positioned above the perfect ground plane**. In simpler terms, all wires must have a Z-coordinate greater than or equal to the specified position. AN-SOF does not verify wires for potential crossings with the PEC ground plane or for placement at the bottom of the plane. Additionally, it does not support horizontal wires lying directly on the ground plane. However, **it does allow for connections to be established from wire ends to the ground plane**.

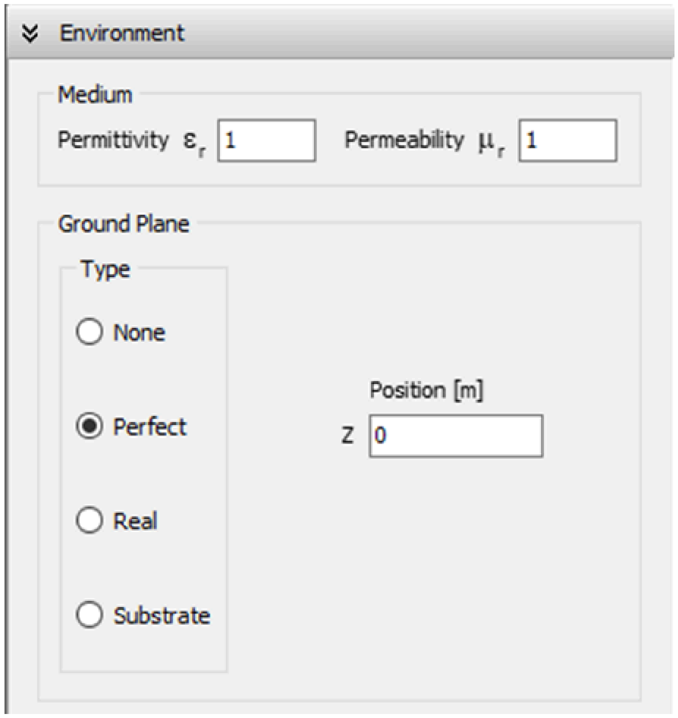


Fig. 2: A perfect ground plane is placed at Z = 0 (xy-plane).

Real

A **real ground plane**, with user-defined **conductivity** and **relative permittivity** (relative permeability set to 1), will be situated on the xy-plane at **z = 0**, as shown in Fig. 3. There are three available options for real ground calculations: **Sommerfeld-Wait/Asymptotic**, **Reflection Coefficients/Asymptotic**, and **Radial wire ground screen**.

All wires must be positioned above the ground plane (z = 0). Horizontal wires placed directly on the ground plane are not supported. However, wire end connections to the ground plane can be established when either the “Sommerfeld-Wait/Asymptotic” or “Radial wire ground screen” options are selected.

The “Reflection Coefficients/Asymptotic” option exclusively permits connections to the ground plane for vertical wires, resulting in perfect zero-Ohm connections. In cases involving horizontal wires, they must be separated by at least one free space wavelength from the ground plane. In such situations, it is essential to verify the validity of the results. AN-SOF does not automatically verify whether these conditions are satisfied within a model.

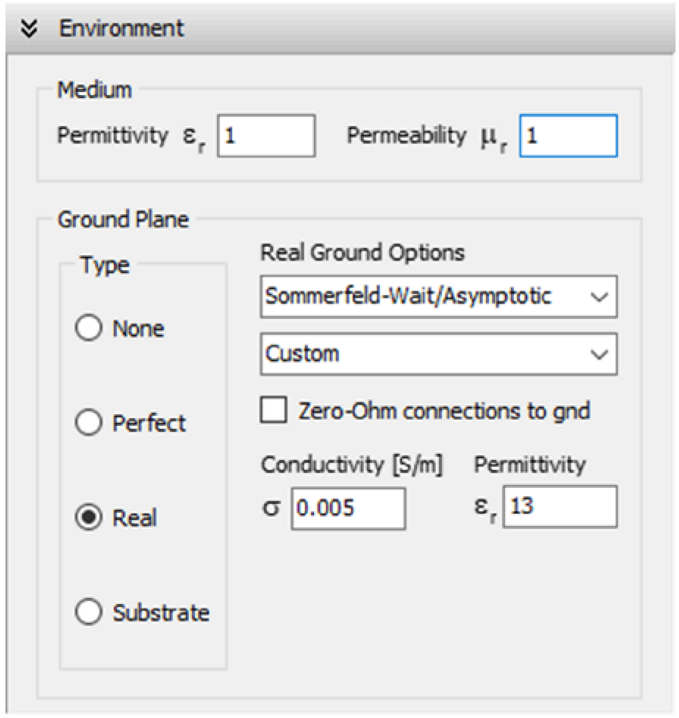


Fig. 3: The parameters of a real ground plane are set.

Real Ground Options

Sommerfeld-Wait/Asymptotic

This option involves calculating the currents flowing through the antenna/wire structure using a model that includes a perfect ground plane and incorporates equivalent loss impedances to address power dissipation in the ground plane, particularly when wires are in close proximity to or connected to the ground. Developed by Prof. James R. Wait, this model is particularly effective for obtaining the input impedance of low-frequency (LF) and medium-frequency (MF) antennas, especially in scenarios where the ground conductivity is high within those frequency bands. Additionally, the finite conductivity and permittivity of the ground are employed to calculate the near-field and far-field radiation from the structure, utilizing the Sommerfeld-Norton asymptotic expressions and Fresnel's reflection coefficients, respectively.

Connections to the ground are permitted, either at the start or end point of a wire with z = 0, and they are considered imperfect by default. This means that currents flowing between the ground and the grounded wires result in power losses in the ground. However, if you select the **"Zero-Ohm connections to ground"** option, wire connections to the ground will be treated as perfect, with no power dissipation occurring at the connection point.

Reflection Coefficients/Asymptotic

In this option, the ground parameters have an impact on the current distribution on the antenna or wire structure above the ground. This influence is determined through a generalization of Fresnel's reflection coefficients, which means that the input impedance of a transmitting antenna is also influenced by the real ground conditions. Moreover, the near and far fields are affected by the finite ground conductivity and its dielectric constant. The near fields are computed using the Sommerfeld-Norton asymptotic expressions, allowing us to calculate the electric and magnetic field as a function of distance from the transmitting antenna. This enables us to observe the attenuation resulting from ground losses. The far-field, on the other hand, is computed using standard Fresnel's reflection coefficients.

Vertical wire connections to the ground are permitted, but they are treated as lossless connections.

Radial wire ground screen

In this option, a ground screen consisting of buried radial wires will be positioned beneath the ground plane. The screen is centered at the origin of coordinates and features user-specified parameters, including the number of radial wires, wire length (or radius of the circular screen), and wire radius.

The ground screen model influences the current distribution on the antenna/wire structure by calculating the power dissipated in the ground plane-wire screen system. Consequently, the presence of the screen and the finite ground conductivity will impact the input impedance of a transmitting antenna located above the ground screen. Additionally, the finite ground conductivity and permittivity are employed to compute the near- and far-fields radiated from the structure, utilizing the Sommerfeld-Norton expressions and the Fresnel's reflection coefficients, respectively.

Connections to the ground are permitted, either at the start or end point of a wire with z = 0, and they are considered imperfect by default. This means that currents flowing between the ground and the grounded wires result in power losses in the ground. However, if you select the **"Zero-Ohm connections to ground"** option, wire connections to the ground will be treated as perfect, with no power dissipation occurring at the connection point.

Substrate

A **dielectric substrate**, with a user-defined **permittivity**, will be positioned **beneath the xy-plane (z = 0)**, as shown in Fig. 4. The substrate can either extend infinitely or have finite dimensions in the xy-plane. It is essential to specify the **slab thickness**, denoted as 'h,' along the z-axis. **A perfectly electrically conducting (PEC) ground plane will be situated at z = -h, just below the dielectric slab**, as illustrated in Fig. 5. To facilitate setting the substrate's permittivity, choose from a drop-down list with common materials (e.g., FR4, RT/Duroid, Rogers RO slabs).

When the **Substrate** option is selected, **all wires must be positioned on the xy-plane (z = 0)**. These wires can represent flat traces of planar or patch antennas printed on the dielectric substrate, microstrip lines, or PCB (Printed Circuit Board) traces. The only exception to this rule is for vertical wires, which can be employed to connect wire strips at z = 0 to the PEC ground plane at z = -h. Typically, a voltage or current source is connected to these vertical wires to power the system, whether it's an antenna or a PCB.

It's important to note that **the PEC ground plane beneath the dielectric substrate cannot be omitted from the model**, meaning that ungrounded substrates are not supported with this option. Wires positioned above the xy-plane (with z-coordinates > 0) or below the PEC ground plane of the substrate (with z-coordinates < -h) are not supported. AN-SOF does not automatically verify compliance with these conditions.

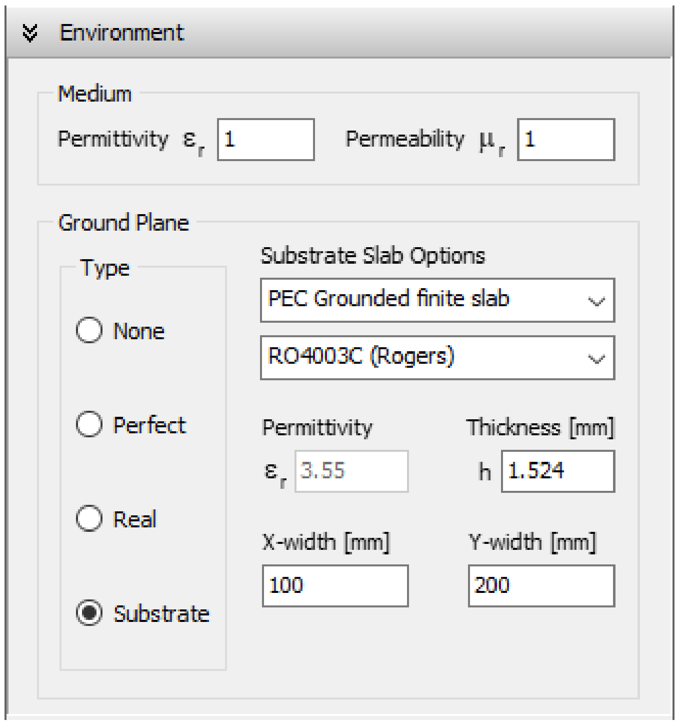


Fig. 4: The parameters of a finite dielectric substrate are set. A perfect ground plane will be placed at z = -h.

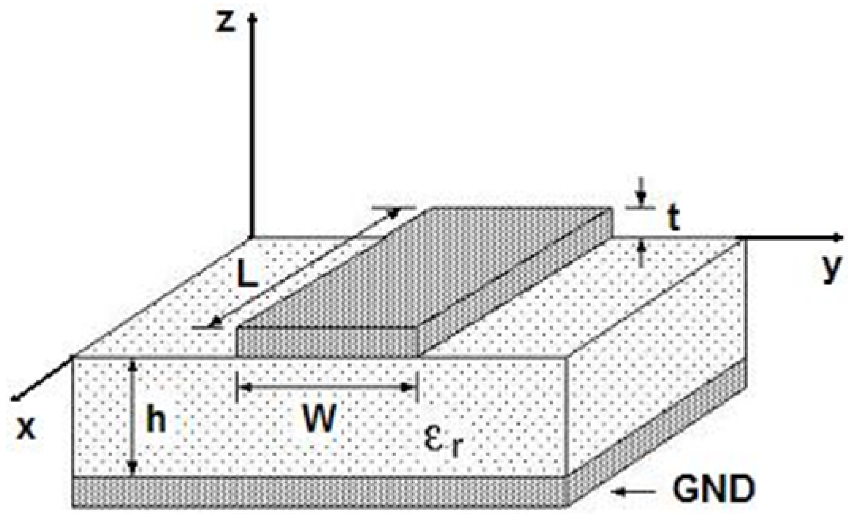


Fig. 5: Dielectric substrate below the xy-plane. A microstrip line is set over the xy-plane.

Far Field Parameters

The Far-Field Panel

Go to the Setup tab in the main window and select the **Far-Field** panel, Fig. 1.

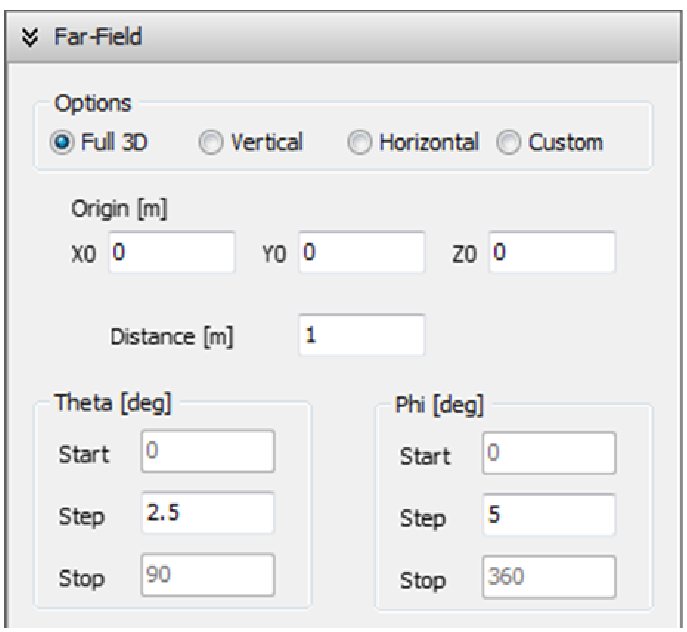


Fig. 1: Far-Field panel in the Setup tabsheet.

The far field can be computed after having calculated the current distribution previously. Thus, the parameters set in the Far-Field panel have no effect in 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, which allows us 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 can be 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

It is 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.

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

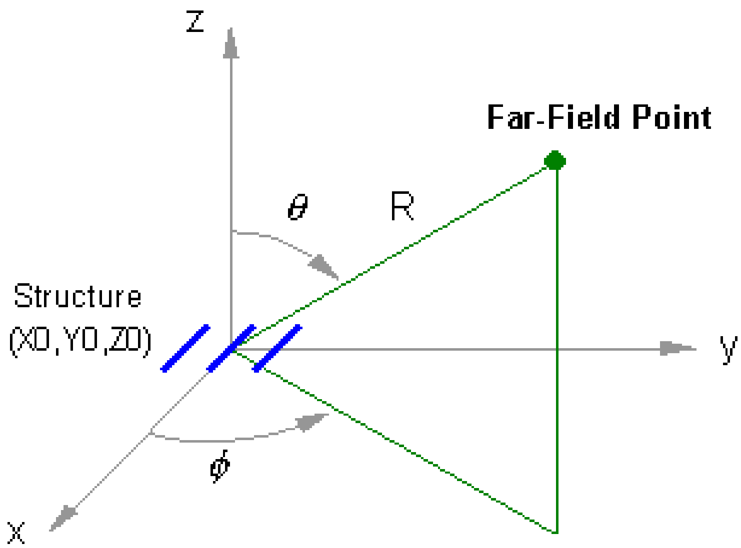


Fig. 2: Spherical coordinates (R,Theta,Phi) of a far-field point.

Near Field Parameters

Near-Field Panel

Go to the Setup tab in the main window. Then, select the Near-Field panel.

The near field can be computed after having calculated the current distribution previously. Thus, the parameters set in the Near-Field panel have no effect in the determination of the currents and can be set at any time. However, the near field must be recalculated every time these parameters are modified. The Near-Field panel has three options: *Cartesian*, *Cylindrical*, and *Spherical*. By choosing one of these options near-fields can either be calculated in Cartesian, Cylindrical or Spherical coordinates.

Cartesian Coordinates

If the **Cartesian** option is chosen, the following parameters can be set for near-field calculations, Fig. 1:

Origin (X0,Y0,Z0)

It is the origin of the Cartesian coordinates used to define the observation points where near fields will be calculated.

X

This box is used to set x-coordinates of the observation points where near-fields will be calculated. The start, step and stop x-coordinates must be set. Start and stop x-coordinates are measured from X0.

Y

This box is used to set y-coordinates of the observation points where near-fields will be calculated. The start, step and stop y-coordinates must be set. Start and stop y-coordinates are measured from Y0.

Z

This box is used to set z-coordinates of the observation points where near-fields will be calculated. The start, step and stop z-coordinates must be set. Start and stop z-coordinates are measured from Z0.

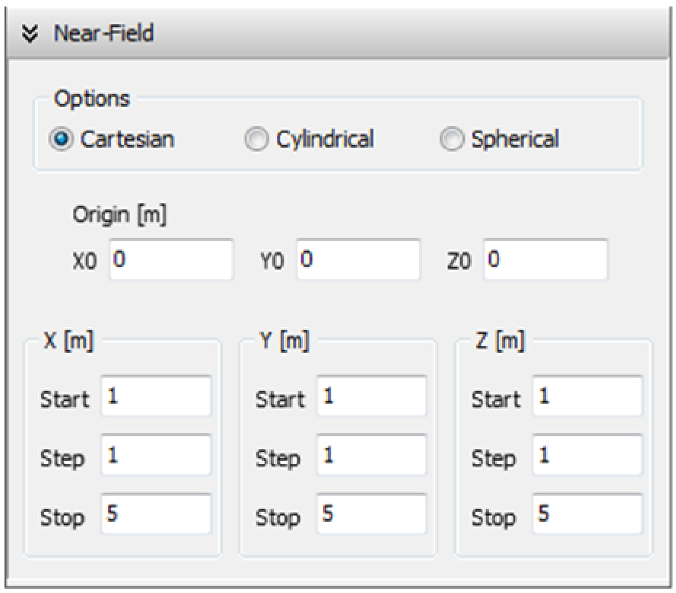


Fig. 1: Near-Field panel in the Setup tabsheet. The Cartesian option is selected.

Cylindrical Coordinates

If the **Cylindrical** option is chosen, the following parameters can be set for near-field calculations, Fig. 2:

Origin (X0,Y0,Z0)

It is the origin of the Cylindrical coordinates used to define the observation points where near fields will be calculated.

R

This box is used to set the distances or R-coordinates of the observation points where near-fields will be calculated. The start, step and stop R-coordinates must be set. Start and stop distances or R-coordinates are measured from the origin (X0,Y0,Z0).

Phi

This box is used to set the azimuth angles or phi-coordinates of the observation points where near-fields will be calculated. The start, step and stop phi-coordinates must be set in degrees.

Z

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

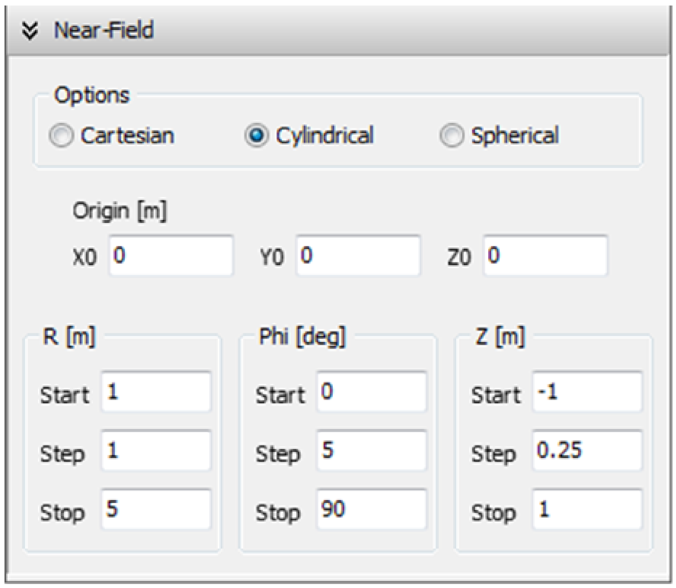


Fig. 2: Near-Field panel in the Setup tabsheet. The Cylindrical option is selected.

Spherical Coordinates

If the **Spherical** option is chosen, the following parameters can be set for near-field calculations, Fig. 3:

Origin (X0,Y0,Z0)

It is the origin of the Spherical coordinates used to define the observation points where near fields will be calculated.

R

This box is used to set the distances or R-coordinates of the observation points where near-fields will be calculated. The start, step and stop R-coordinates must be set. Start and stop distances or R-coordinates are measured from the origin (X0,Y0,Z0).

Theta

This box is used to set zenith angles or theta-coordinates of the observation points where near-fields will be calculated. The start, step and stop theta-coordinates must be set in degrees.

Phi

This box is used to set azimuth angles or phi-coordinates of the observation points where near-fields will be calculated. The start, step and stop phi-coordinates must be set in degrees.

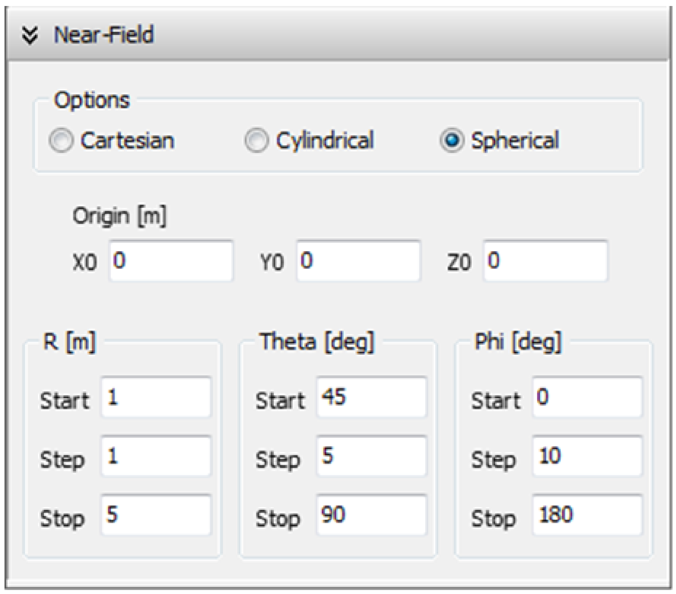


Fig. 3: Near-Field panel in the Setup tabsheet. The Spherical option is selected.

Defining the Excitation

Excitation Panel

Go to the Setup tab in the main window and select the **Excitation** panel. There are two types of excitations: *Discrete Sources* and *Incident Field*, Fig. 1.

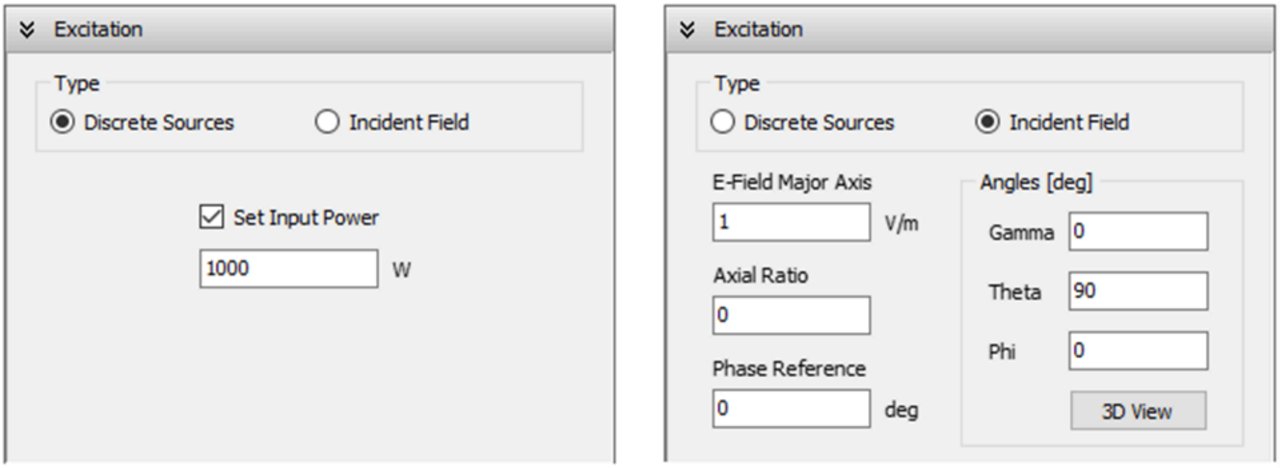


Fig. 1: Excitation panel in the Setup tabsheet.

Discrete Sources

The discrete generators placed at the wire structure will be used to calculate the current distribution. The total input power in Watts can be specified, so the voltage/current sources will be adjusted accordingly to achieve the specified input power. If the input power is not specified, then the voltage/current sources will be constant, and the input power will be an output result from calculations.

Incident Field

An incident plane wave will be used as the excitation of the structure. The direction of incidence and polarization of the incoming field can be set in this panel.

The following parameters must be set for the incident wave excitation:

E-Field Major Axis

In the case of linear polarization, it is the amplitude, in Volts per meter (rms value), of the incoming electric field. For an elliptically polarized plane wave, it is the major axis of the polarization ellipse.

Axial Ratio

It is the ratio of the minor axis to the major axis of the polarization ellipse. If the axial ratio is positive (negative) a right-handed (left-handed) ellipse is obtained. If the axial ratio is set to zero, a linearly polarized wave will be obtained.

Phase Reference

It is the phase, in degrees, of the incident plane wave at the origin of coordinates. Its value only shifts all phases in the structure by the same amount.

Gamma

For a linearly polarized wave, it is the polarization angle, in degrees, of the incident electric field measured from the plane of incidence to the direction of the electric field vector, as it is shown in Fig. 2. For an elliptically polarized wave, Gamma is the angle between the plane of incidence and the major ellipse axis.

Theta

It is the zenith angle, in degrees, of the incident direction.

Phi

It is the azimuth angle, in degrees, of the incident direction.

The definition of these parameters is illustrated in Fig. 2.

When the **3D View** button is pressed a user interface is enabled in the workspace, where the direction of arrival of the plane wave and its polarization can be specified easily, Fig. 3.

> **Note**
>
> **When an incident plane wave is used as excitation, all discrete sources, if any, will not be considered in the simulation.**

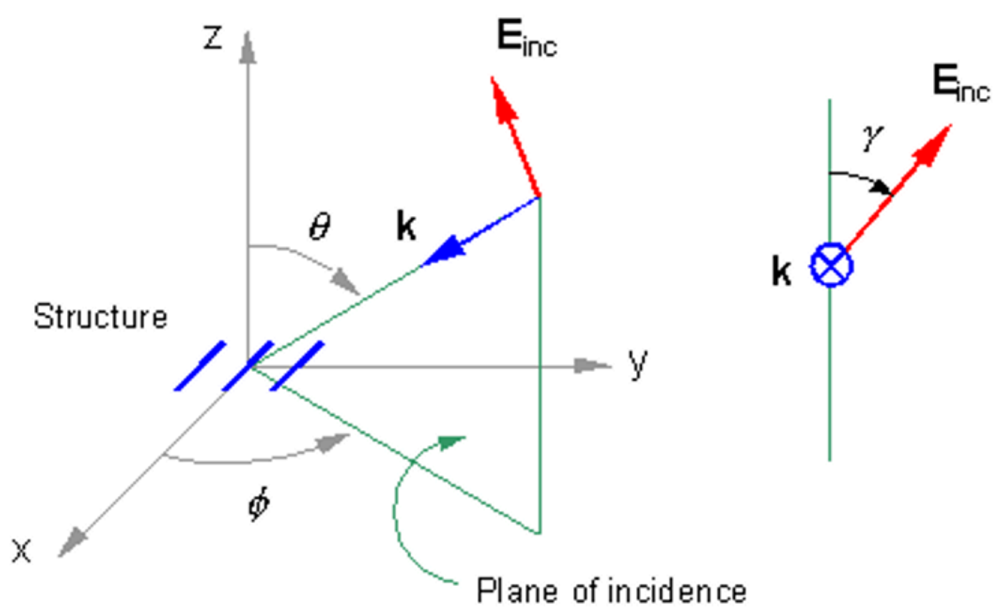


Fig. 2: Definition of the incident plane wave.

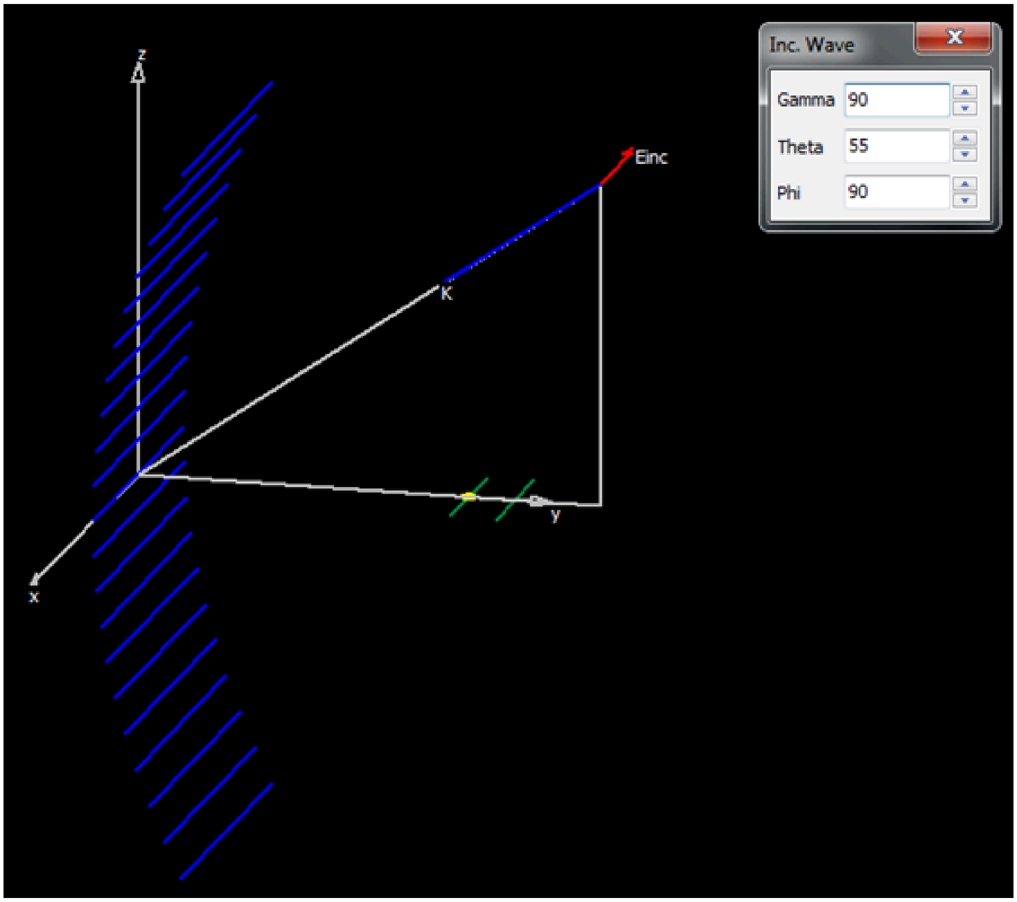


Fig. 3: 3D View user interface for the incident field definition. In the case of elliptical polarization, the electric field vector Einc indicates the major ellipse axis.

The Settings Panel

Go to the Setup tab in the main window and select the **Settings** panel, Fig. 1.

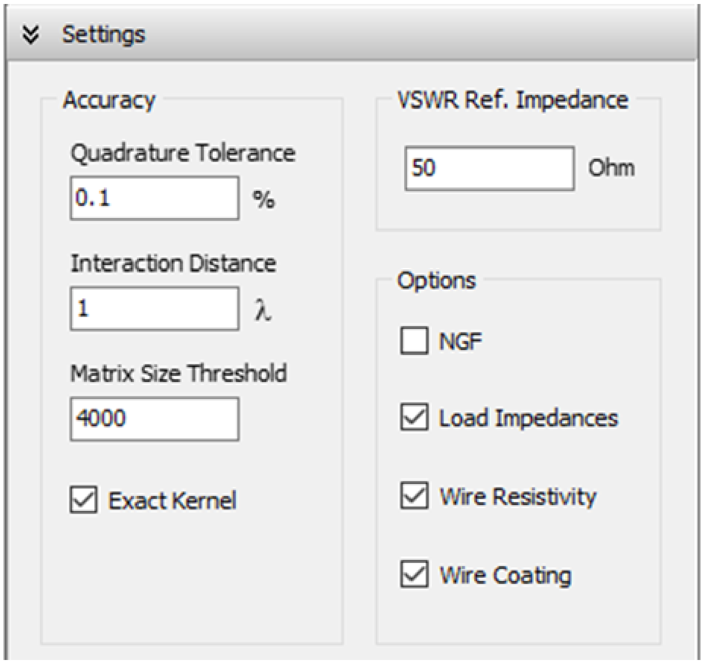


Fig. 1: Settings panel in the Setup tabsheet.

The accuracy of the integrals involved in the calculations can be set in the Settings panel. The **Quadrature Tolerance** is the error in the evaluation of interactions between wire segments which are separated by a distance less than the **Interaction Distance**.

The **Interaction Distance** is the maximum distance in wavelengths between segments for which an error less than the Quadrature Tolerance is guaranteed in the integrations. The interaction between all wire segments further apart than the Interaction Distance is computed using a third-degree polynomial approximation to the involved integrals, which is more accurate for curved segments than the Hertzian dipole approximation used in the traditional Method of Moments. Therefore, the Interaction Distance could be set to zero for a faster simulation when wire segments are not too close to each other, but results will be less accurate. A convergence test for various values of this parameter is recommended.

For most cases, a quadrature tolerance between 0.1% and 1% and an interaction distance between 0.25 and 1.0 wavelengths will be enough for obtaining accurate results.

In AN-SOF, all calculations are done with double precision. The **Matrix Size Threshold** allows us to simulate big antenna problems when the size of the structure compromises the available memory space. For instance, by setting the Matrix Size Threshold to 4,000, the set of linear equations associated to the Z-matrix of the antenna system will be computed using single precision for a matrix size bigger than 4,000 x 4,000. This will impact the accuracy of the calculations but will save memory. In practice, the error will be not significant.

The **Exact Kernel** option allows us to use the exact Kernel for the Electric Field Integral Equation associated to the structure. This option must be chosen when relatively thick wire segments are used to describe the wire structure. If the Exact Kernel option is unchecked, an extended thin-wire approximation will be used for the kernel. If all wire segments are thin enough, then the computation will be a little faster using the extended thin-wire kernel. Refer to **The Exact Kernel** for further information.

In the Settings panel, the **Reference Impedance for VSWR** calculations can also be set. A default value of 50 Ohm is set.

Besides, the following options for the type of simulation are available in the Options box:

> If **NGF** is checked, the Numerical Green's Function calculation is performed in the simulation, that is, the LU-decomposed matrix of the system is stored in a file in the first simulation. Then, by using the stored information, new simulations are performed faster than the first one. Check this option if you need to change the amplitude values of voltage/current sources frequently.
>
> If **Load Impedances** is checked, lumped impedances will be considered in the simulation. With this option all the lumped loads can be disabled or enabled at the same time.
>
> If **Wire Resistivity** is checked, the finite resistivity of the wires will be considered in the simulation. Any wire has its own resistivity in [Ohm meter] that can be set when the wire is drawn. This option allows us considering the whole structure as a perfect electric conductor when it is unchecked.
>
> If **Wire Coating** is checked, the coating materials of the wires will be considered in the simulation. Any wire has its own coating specified by a dielectric permittivity, magnetic permeability, and thickness, which can be set when the wire is drawn. When this option is unchecked, the wire coating will not be considered in the simulation.

Project Details

Go to **View > Project Details** in the main menu to display the Project Details window, where a summary of the project information is shown, Fig. 1. There is also a button on the toolbar to access this window.

The text in the Project Details window can be selected and copied to the clipboard in the usual way (Ctrl+C and Ctrl+V commands).

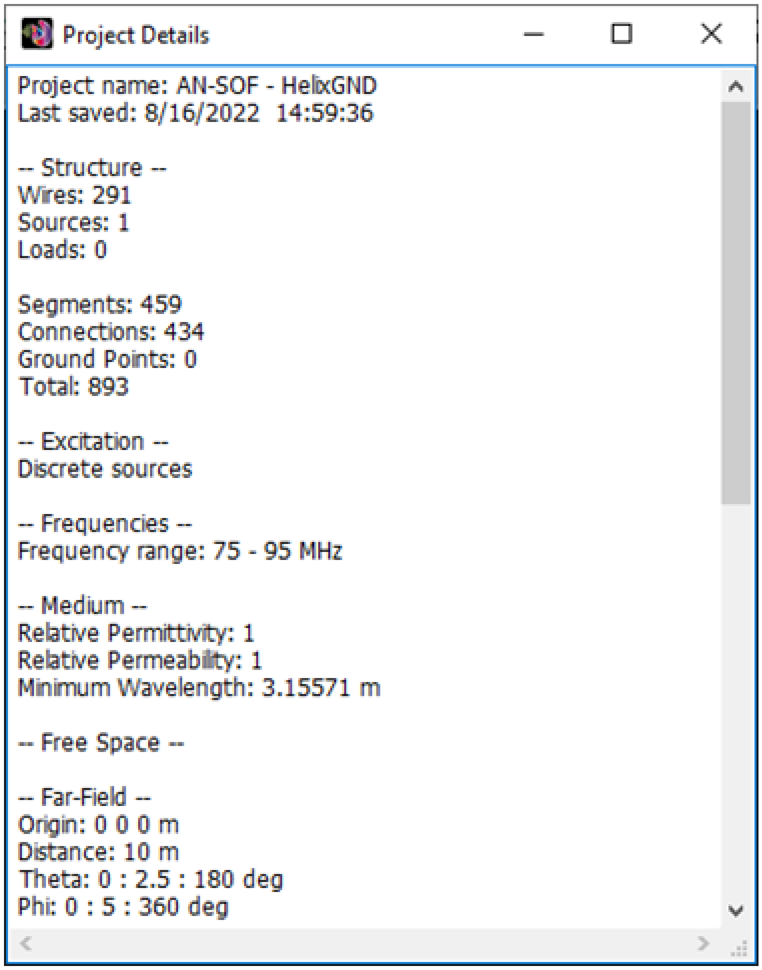


Fig. 1: Project Details window.

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	Unselect a wire
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 Main/Small axes
F8	Select a wire in order of creation
F9	Select a wire in reverse order of creation
F10	Run ALL
F11	Run currents and far-field
F12	Run currents and near-field
Ctrl + A	Display the Axes dialog box
Ctrl + I	Zoom in
Ctrl + K	Zoom out
Ctrl + M	Modify the selected wire
Ctrl + N	Create a new project
Ctrl + O	Open a project file
Ctrl + P	Print the workspace
Ctrl + Q	Exit AN-SOF

Key	Action
Ctrl + R	Run Currents
Ctrl + S	Save the project
Ctrl + T	Tabular input of linear wires
Ctrl + W	Show properties of the selected wire
Ctrl + Del	Delete the selected wire or group of wires
Ctrl + Ins	Display the Source/Load toolbar

Drawing Wires

Types of Wires

AN-SOF has different types of wires. Each wire type has its own geometrical parameters, attributes and materials that can be set in a specific Draw dialog box. This dialog box allows us drawing a new wire in the workspace.

Choosing **Draw** in the main menu shows the following commands:

Menu Options

The commands to draw wires can be accessed from three menus:

- Main menu > **Draw**.
- **Popup menu** by right clicking on the workspace.
- Main menu > View > **Drawing Panel**.

Wire Attributes

The **Attributes** page is part of the Draw dialog box for the selected wire type (see Fig. 1). On the Attributes page, you can specify the following attributes:

Number of Segments

Every wire must be divided into a certain **number of segments**. During the simulation process, AN-SOF needs to determine the unknown current on each segment. When you access the Attributes page, a default Number of Segments is displayed. This default number is calculated based on the wire's length and the shortest wavelength, but you can modify it as needed.

> **Note**
>
> If you set the **Number of Segments to zero**, AN-SOF will automatically compute the minimum recommended number of segments for the wire. This calculation assumes **10 segments per wavelength**, considering the shortest wavelength in a frequency sweep.

Cross-Section

The Cross-Section of the wire can be chosen from a combo-box. There are six cross-section types available: **Circular**, **Square**, **Flat**, **Elliptical**, **Rectangular**, and **Triangular**. AN-SOF computes an equivalent radius for the five last cases. Infinitesimally thin wires are not allowed, so the cross-section radius must be greater than zero.

The Draw dialog box for any wire type has its own Attributes page with the same features as those described here.

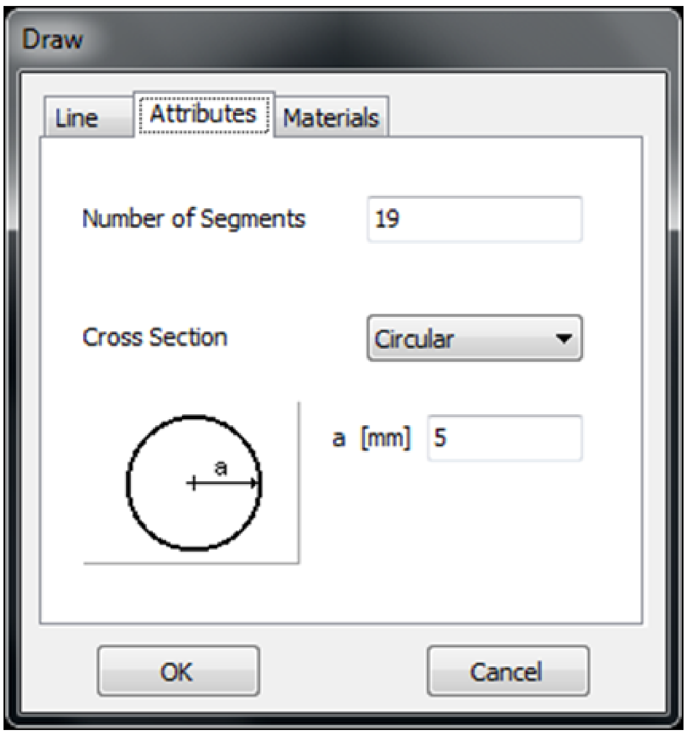


Fig. 1: Attributes page in the Draw dialog box for the Line.

Wire Materials

The **Materials** page belongs to the Draw dialog box of the chosen wire type, Fig. 1.

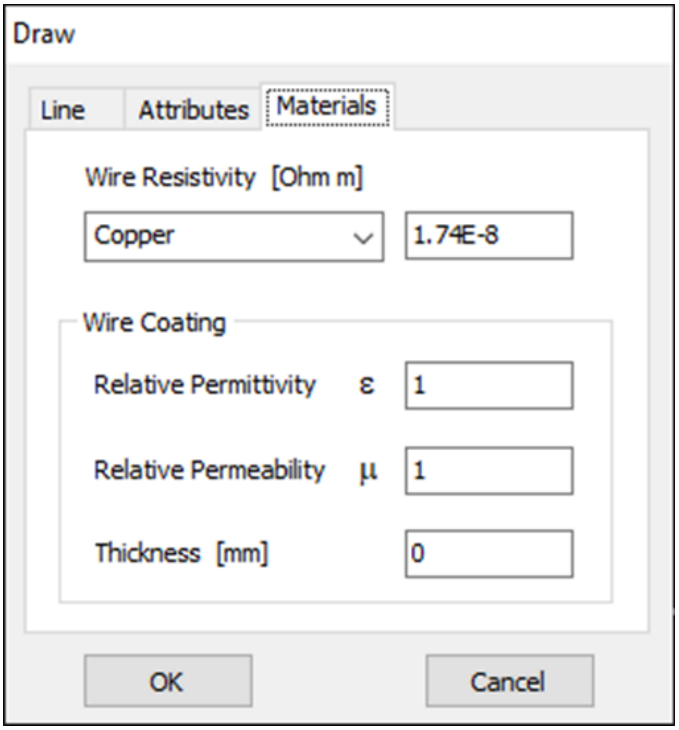


Fig. 1: Materials page in the Draw dialog box for the Line.

In the Materials page the following attributes can be specified:

Wire Resistivity

A resistivity in [Ohm meter] can be specified for the wire. The following list of most common metals is available for choosing:

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

The corresponding resistivity value will be automatically displayed for the chosen metal. Choose the **Custom** option to set a resistivity value if it is not in the list. Choose **Perfect (PEC)** to set a perfect electrically conducting metal.

The resistivity is used for computing a distributed impedance per unit length along the wire, which considers the **skin effect**. The equivalent radius for wires of non-circular cross section will be used to compute the impedance per unit length along the wires.

The resistivity of wires is considered in the simulation if the option **Wire Resistivity** is checked in the **Settings panel** of the **Setup** tabsheet.

Wire Coating

Wires can have **insulation** or **coating material**. The cross section of a coated wire is circular, so the equivalent radius will be used for wires having a non-circular cross section. In this case, the material the coating is made of can be set by the following parameters:

- **Relative Permittivity**: It is the dielectric constant of the coating material relative to the permittivity of vacuum.
- **Relative Permeability**: It is the magnetic permeability of the coating material relative to the permeability of vacuum.
- **Thickness**: It is the thickness of the coating shield. It can be set to zero when no coating is used.

The wire coating is considered in the simulation if the option **Wire Coating** is checked in the **Settings panel** of the **Setup** tabsheet.
Enabling/Disabling Resistivity

If wires with non-zero resistivity have been drawn previously and the whole structure must now be considered as a perfect electric conductor, all resistivities can be disabled without modifying the definitions of the wires.

Go to the Setup tabsheet in the main window and select the **Settings** panel, Fig. 1. If the option **Wire Resistivity** in this panel is checked, the resistivities are enabled. Uncheck the Wire Resistivity option to disable all of them.

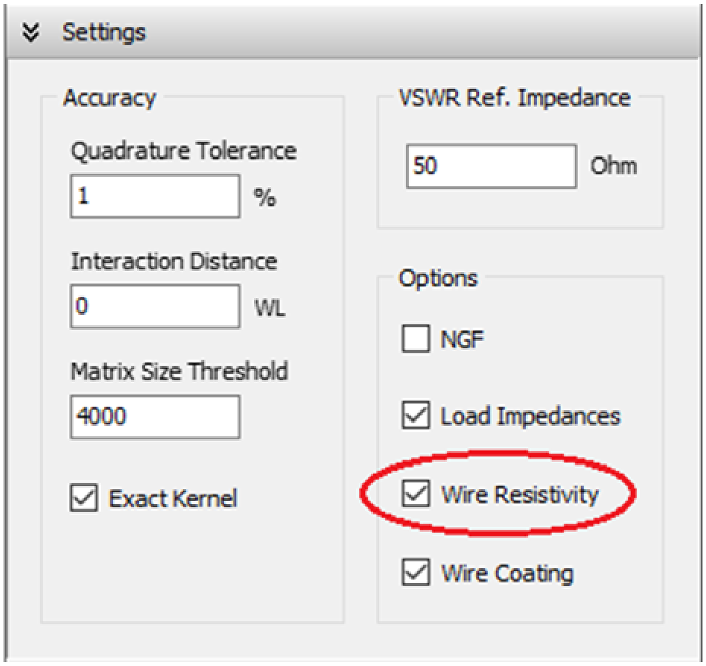


Fig. 1: Wire Resistivity option in the Settings panel of the Setup tabsheet. If this option is checked, all resistivities are enabled, otherwise they are disabled.

Enabling/Disabling Coating

If wires with a coating shield or insulation have been drawn previously and the whole structure must now be considered as composed of bare conductive wires, all coatings can be disabled without modifying the definitions of the wires.

Go to the Setup tabsheet in the main window and select the **Settings** panel, Fig. 1. If the option **Wire Coating** in this panel is checked, the coatings are enabled. Uncheck the Wire Coating option to disable all of them.

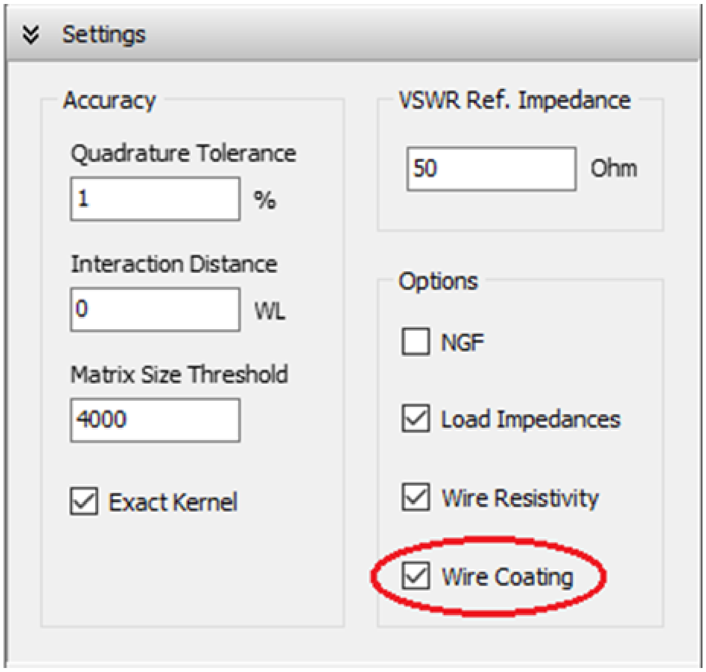


Fig. 1: Wire Coating option in the Settings panel of the Setup tabsheet. If this option is checked, all coatings are enabled, otherwise they are disabled.

Cross-Section Equivalent Radius

The wire cross-section can be chosen from a combo-box in the **Attributes** page of the Draw dialog box for the chosen wire type, Fig. 1.

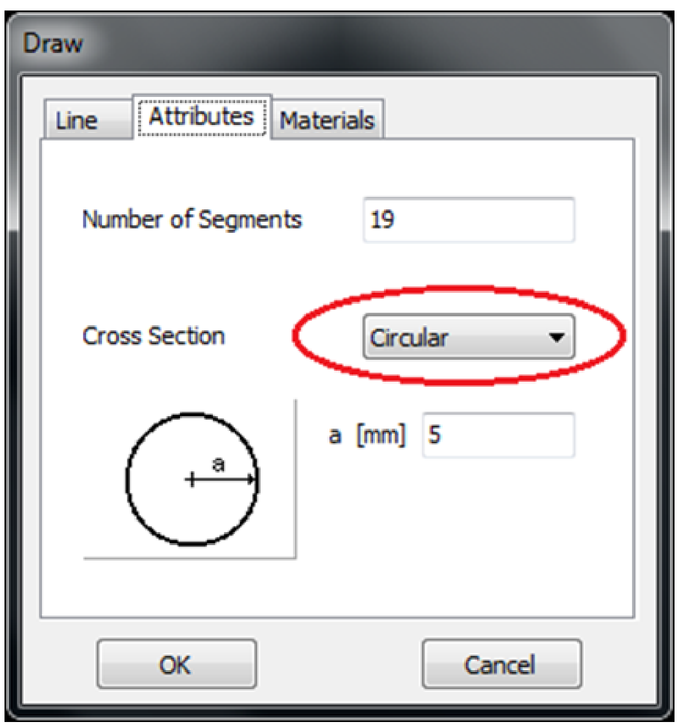
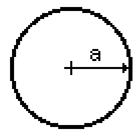


Fig. 1: Cross-section combo-box in the Attributes page of the Draw dialog box. A circular cross section of radius “a” is chosen.

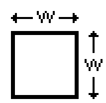
There are six cross-section types available: **Circular**, **Square**, **Flat**, **Elliptical**, **Rectangular**, and **Triangular**. AN-SOF computes an equivalent radius for the non-circular cross-sections. The equivalent radius is the radius of a circular cross-section that produces the same average electromagnetic fields around the wire and on its surface.

The cross-sections and their equivalent radii are the following:



Circular

A positive and non-zero radius “a” must be set. The equivalent radius is “a”.

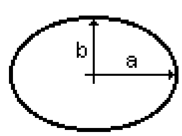


Square

A positive and non-zero width “w” must be set. The equivalent radius is 0.59017 w.

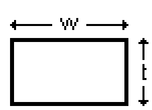
Flat

A positive and non-zero width “w” must be set. The equivalent radius is w/4.



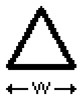
Elliptical

The semi-axes “a” and “b” must be positive and non-zero. The equivalent radius is (a + b)/2.



Rectangular

The widths “w” and “t” must be positive and non-zero. The equivalent radius is computed using a polynomial and logarithmic approximation to the solution of an integral equation.



Triangular

A positive and non-zero width “w” must be set. The equivalent radius is 0.42 w.
Exporting Wires

You can export linear wires from AN-SOF to a text file in NEC format (extension .nec) by navigating to File > **Export Wires** in the main menu. Linear wires will be saved as **GW** lines. Additionally, the exported file will include **GE** (ground connections), **GN** (ground plane), **TL** (transmission line), **LD** (load impedances and wire conductivity), **IS** (wire insulation), **FR** (frequency), **EX** (excitation), **EK** (exact kernel), and **RP** (radiation pattern) cards.

Moreover, the exported file can be saved as a **Scilab** script, with a .sce extension. The exported file will contain programming code that can be adjusted to create a new project, allowing for variations in parameters such as wire lengths and positions, frequencies, and ground conditions.

Adding Wires

Line

The “Line” refers to a linear or straight wire.

To access the “Line” dialog box for drawing a line, navigate to **Draw > Line** in the main menu. This dialog box contains three pages: **Line**, **Attributes**, and **Materials** (Fig. 1).

Line Page

The **Line** page allows you to set the geometrical parameters for the line. Two options are available: **2 Points** and **Start – Direction – Length**.

The **2 Points** option enables you to define the line by specifying two points: “From Point” and “To Point” (Figs. 1 and 2).

If **Start – Direction – Length** is selected, the line will be drawn starting from the **Start Point**, in the direction given by the **Theta** and **Phi** angles in spherical coordinates, and ending at a point defined by the **Wire Length** measured along that direction (Figs. 3 and 4).

After setting the geometrical parameters on the **Line** page, you can select the **Attributes page** to specify the **Number of Segments** and **Cross-Section**. The **Materials page** allows you to set the wire **Resistivity** and **Coating**.

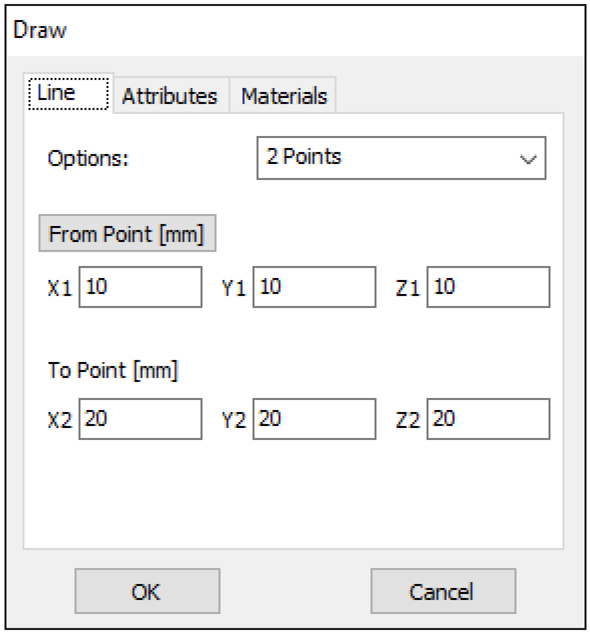


Fig. 1: "2 Points" option in the Line page of the Draw dialog box for the Line.

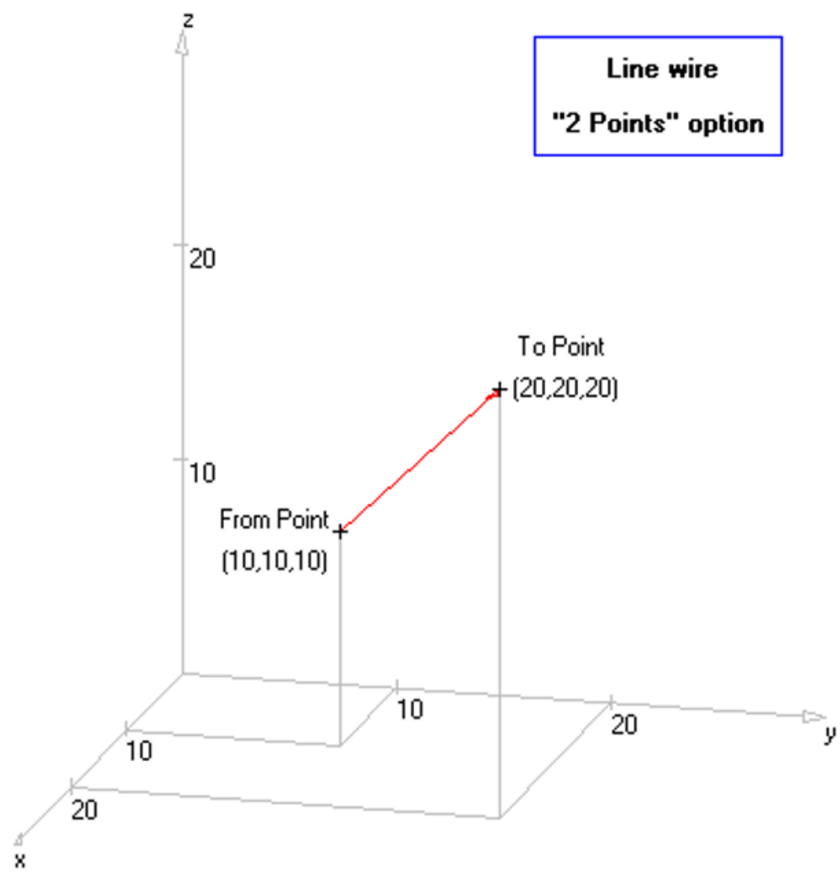


Fig. 2: A Line drawn using the "2 Points" option with the parameters shown in Fig. 1.

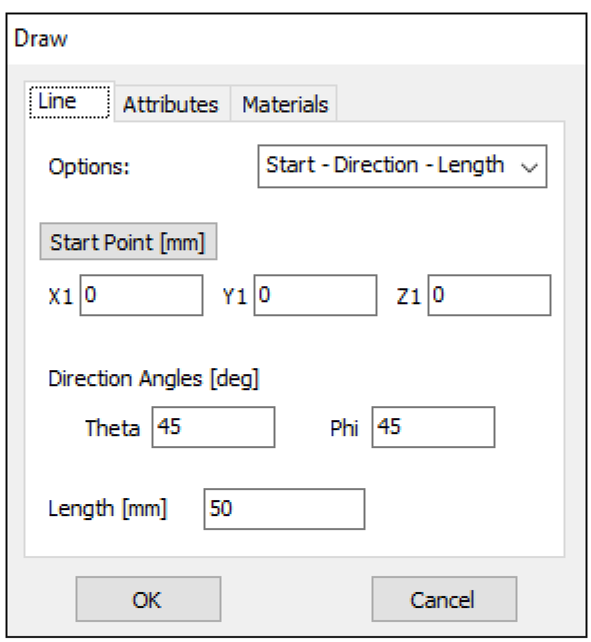


Fig. 3: "Start – Direction – Length" option in the Line page of the Draw dialog box for the Line.

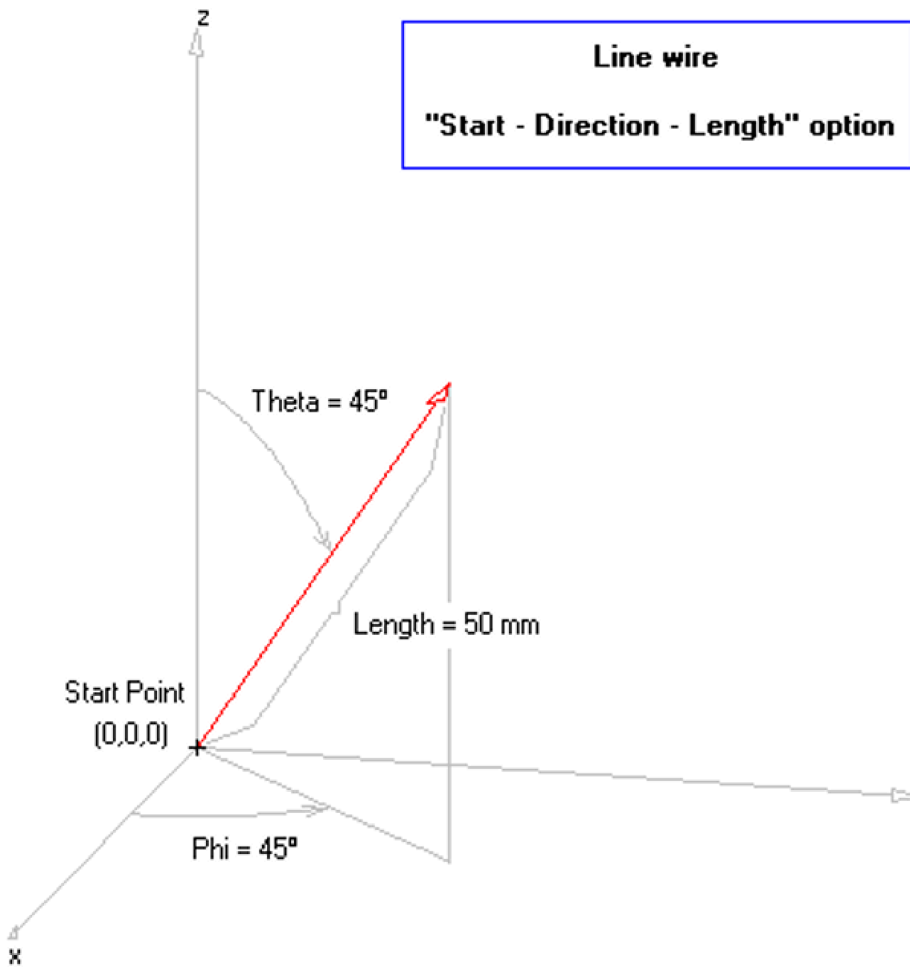


Fig. 4: A Line drawn using the "Start – Direction – Length" option with the parameters shown in Fig. 3.

Arc

The "Arc" refers to a circular arc.

To access the "Arc" dialog box for drawing an arc, navigate to **Draw > Arc** in the main menu. This dialog box contains three pages: **Arc**, **Attributes**, and **Materials** (Fig. 1).

Arc Page

The **Arc** page allows you to set the geometrical parameters for the arc. Two options are available: **3 Points** and **Start – Center – End**.

The **3 Points** option enables you to define the arc by specifying three points: a **Start Point**, a **Second Point**, and an **End Point**. An arc starting from the **Start Point**, passing through the **Second Point**, and ending at the **End Point** will be drawn on the workspace (Figs. 1 and 2).

If **Start – Center – End** is selected, the arc will be drawn starting from the **Start Point**, with the center specified by **Center** and ending at a point determined by the **End Point** (Figs. 3 and 4). The **End Point** determines the arc's aperture angle and the plane in which it lies. Note that the **End Point** may not coincide with the actual ending point of the arc.

After setting the geometrical parameters on the **Arc** page, you can select the **Attributes page** to specify the **Number of Segments** and **Cross-Section**. The **Materials page** allows you to set the wire **Resistivity** and **Coating**.

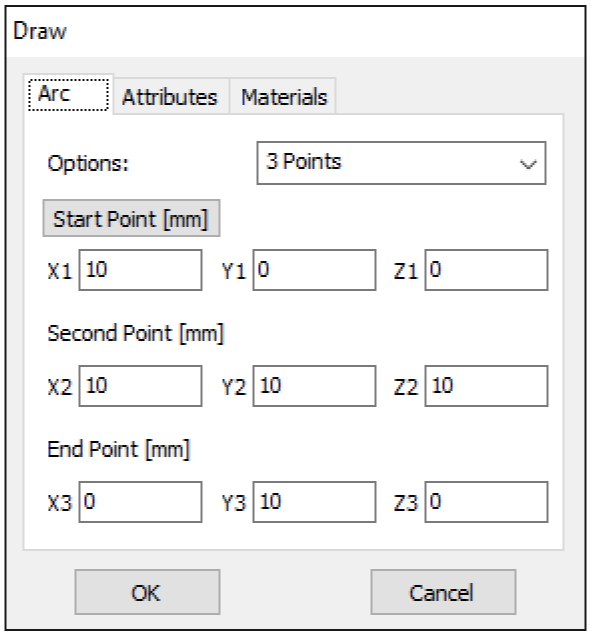


Fig. 1: "3 Points" option in the Arc page of the Draw dialog box for the Arc.

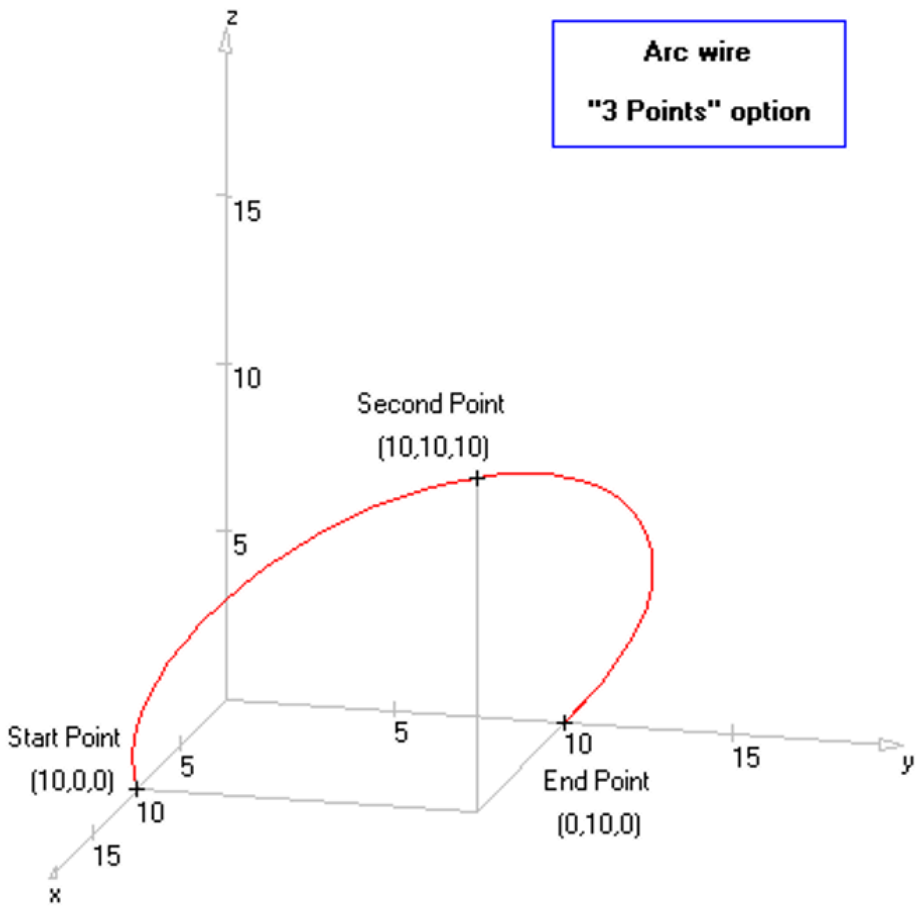


Fig. 2: An Arc drawn using the "3 Points" option with the parameters shown in Fig. 1.

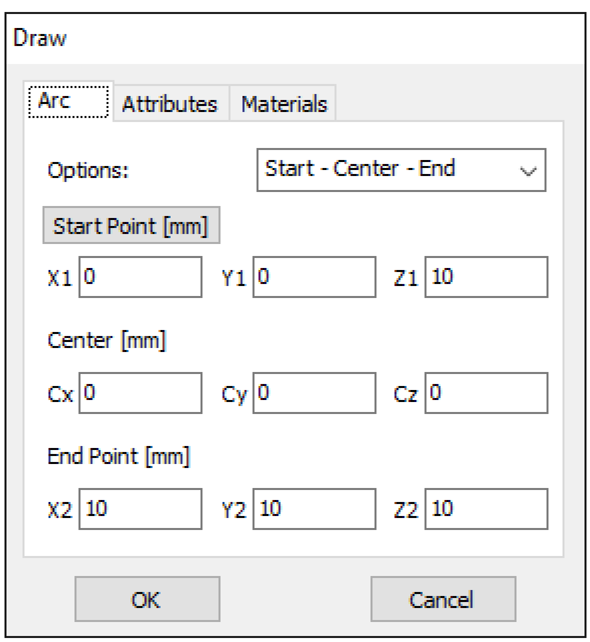


Fig. 3: "Start – Center – End" option in the Arc page of the Draw dialog box for the Arc.

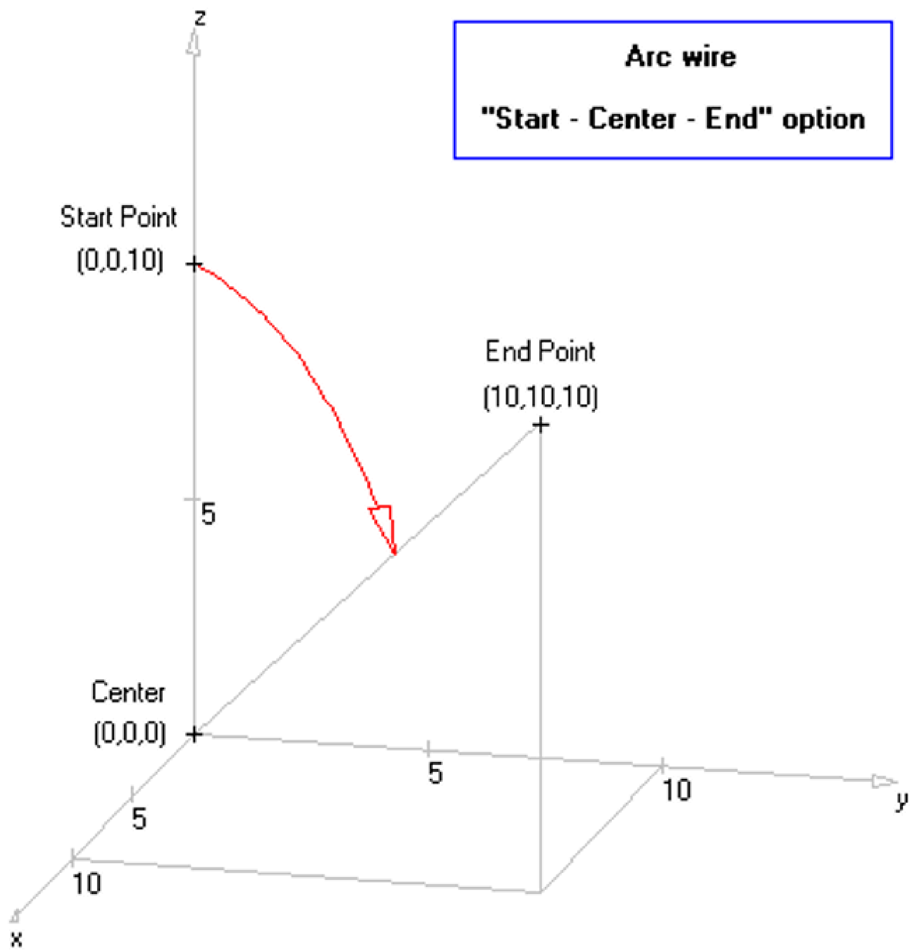


Fig. 4: An Arc drawn using the "Start – Center – End" option with the parameters shown in Fig. 3.

Circle

The Circle refers to a circular loop.

Go to **Draw > Circle** in the main menu to display the Draw dialog box for the Circle. This dialog box has four pages: **Circle**, **Orientation**, **Attributes** and **Materials**.

The Circle page

In the **Circle** page the geometrical parameters for the Circle can be set. There are two options: *Center – Radius – Orientation* and *3 Points*.

The **Center – Radius – Orientation** option allows us entering the Circle by giving its Center, Radius, and axis, Figs. 1 and 2. The circle axis can be set in the **Orientation** page, Fig. 3.

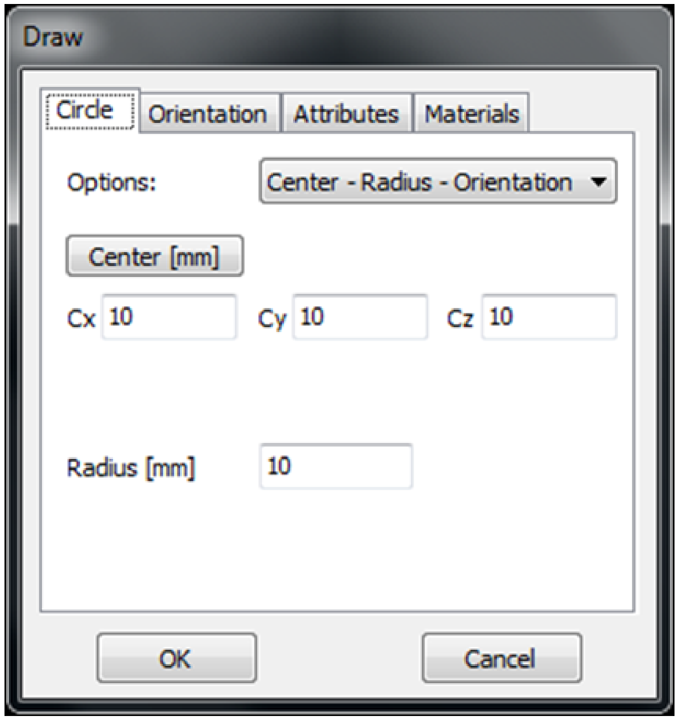


Fig. 1: "Center – Radius – Orientation" option in the Circle page of the Draw dialog box.

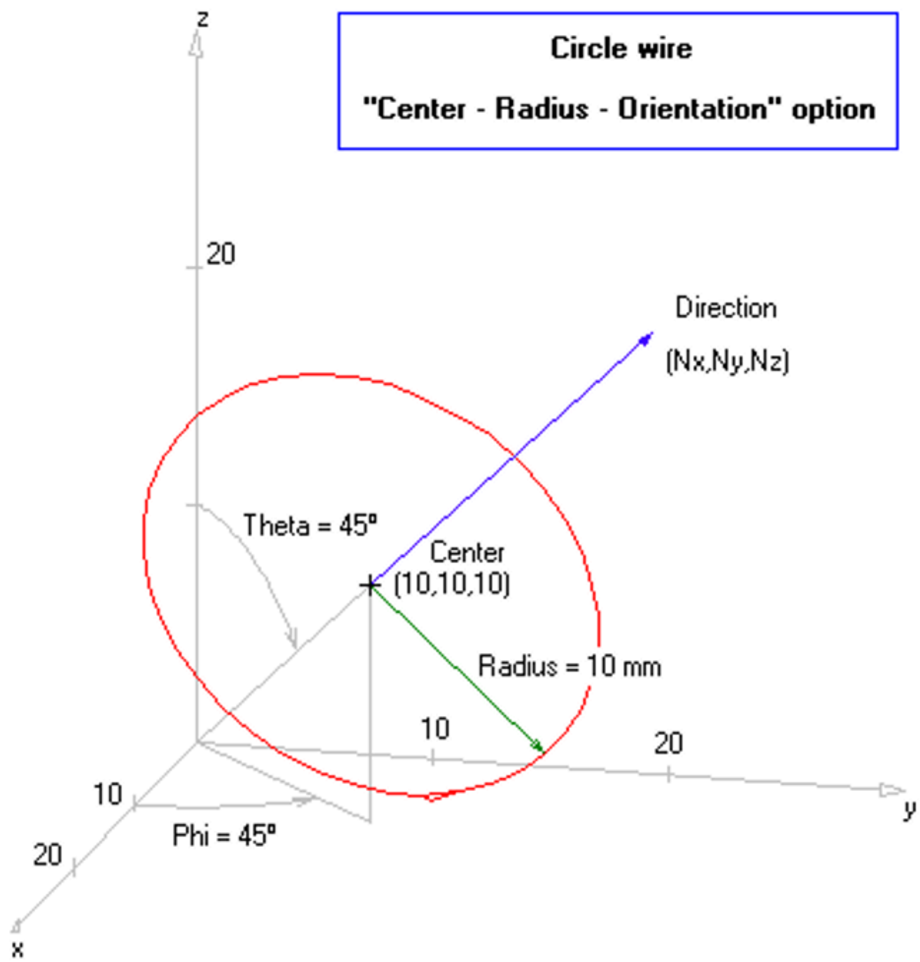


Fig. 2: A Circle drawn using the "Center – Radius – Orientation" option.

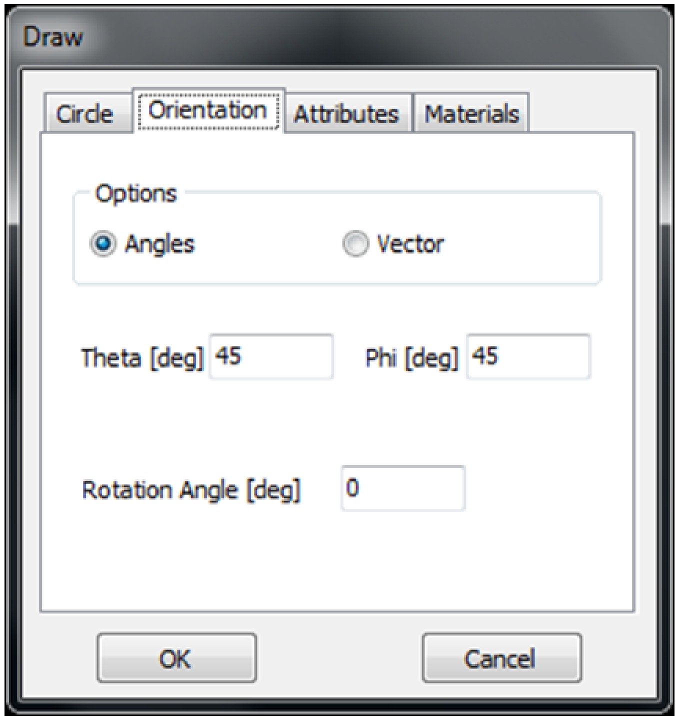


Fig. 3: Orientation page in the Draw dialog box for the Circle.

If the **3 Points** option is chosen, the Circle will be drawn starting from First Point, passing through Second Point and Third Point, and ending at First Point, Figs. 4 and 5. Thus, the circle starts and ends at the same point. The Orientation page will be invisible when the 3 Points option is chosen.

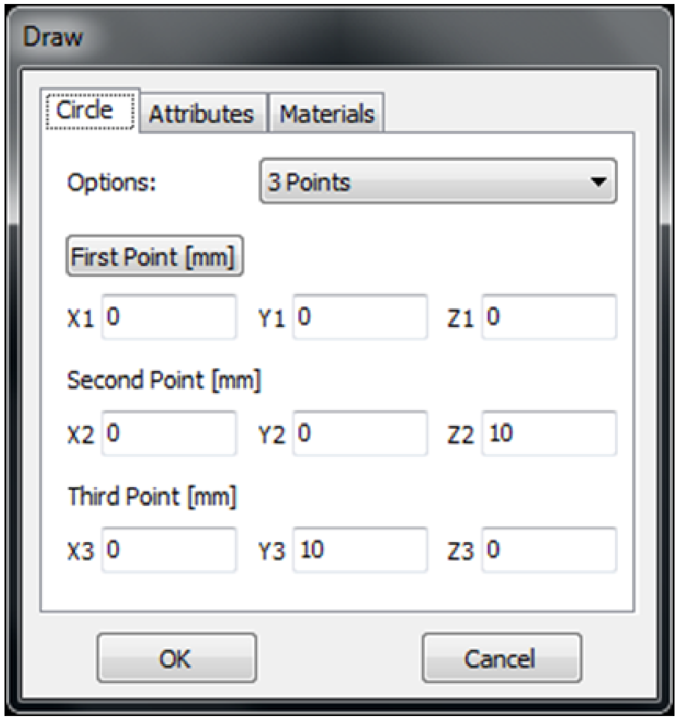


Fig. 4: "3 Points" option in the Circle page of the Draw dialog box.

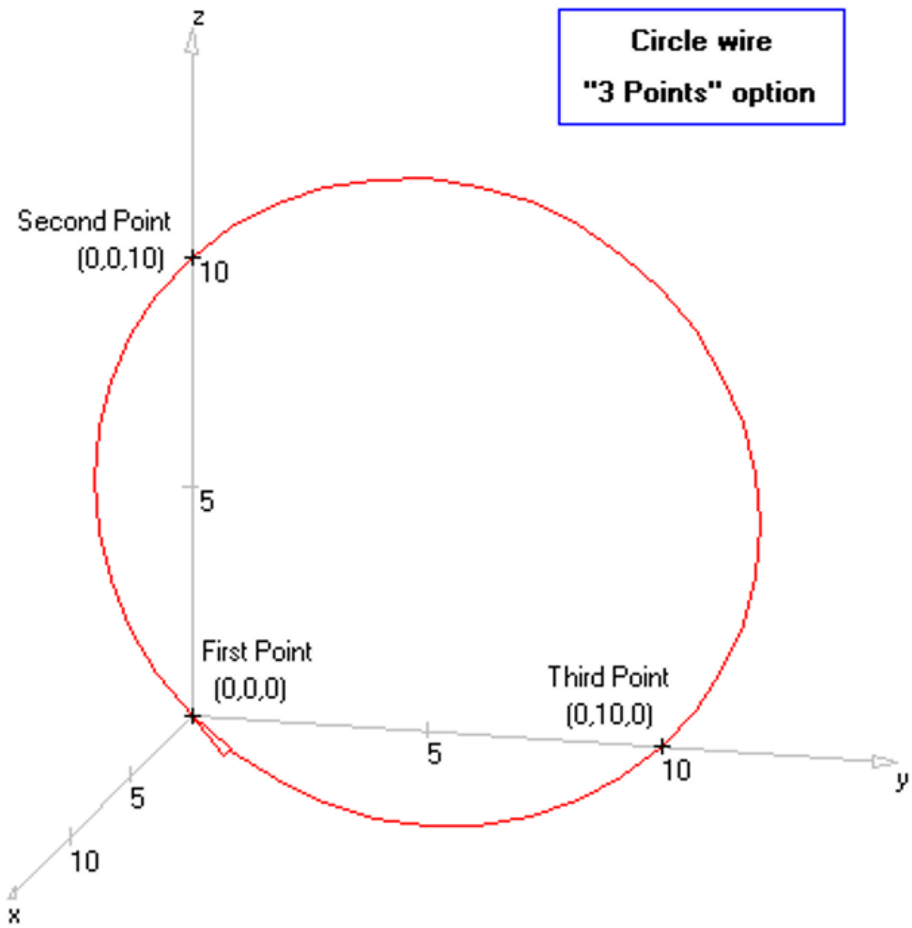


Fig. 5: A Circle drawn using the “3 Points” option.

Once the geometrical parameters in the Circle and Orientation pages have been set, the **Attributes >** page can be selected, where the number of segments and cross-section can be set. The wire resistivity and coating can be set in the **Materials >** page.

The Orientation page

In the **Orientation** page the orientation for the Circle can be set. There is a box with two options: *Angles* and *Vector*, Fig. 3.

If **Angles** is selected, the circle axis can be defined by given an orthogonal direction to the rest plane of the circle. Thus, the Theta and Phi angles determine the axis direction in spherical coordinates.

If **Vector** is selected, the circle axis can be defined by given an orthogonal vector to the rest plane of the circle. Thus, the Nx, Ny, and Nz components of that vector determine the axis direction.

The circle can be rotated around its axis by given the **Rotation Angle**.

Helix

The “Helix” refers to a wire curved into a circular helical shape.

To access the “Helix” dialog box for drawing a helix, navigate to **Draw > Helix** in the AN-SOF main menu. This dialog box contains four tabs: **Helix**, **Orientation**, **Attributes**, and **Materials**.

Helix Page

The **Helix** page allows you to set the geometrical parameters for the helix. Two options are available: **Start – Radius – Pitch – Turns** and **Start – End – Radius – Turns**.

The **Start – Radius – Pitch – Turns** option enables you to define the helix by specifying its **Start Point**, **Radius**, **Pitch**, and **Number of turns**, as shown in Figures 1 and 2. The **Pitch** represents the spacing between turns. A positive (negative) pitch results in a right-handed (left-handed) helix. The **Number of turns** does not need to be an integer, allowing you to enter fractions of turns. Alternatively, you can enter the **Diameter**, **Pitch Angle**, and **Wire Length** instead of the radius-pitch-number of turns combination. When entering the **Radius – Pitch – Turns** combination, the **Diameter – Pitch Angle – Wire Length** set will be automatically calculated, and vice versa. In any case, the helix’s **axial height** is displayed automatically (calculated from the input data and cannot be entered).

The orientation of the helix axis can be set on the **Orientation** page (Fig. 3), as described below.

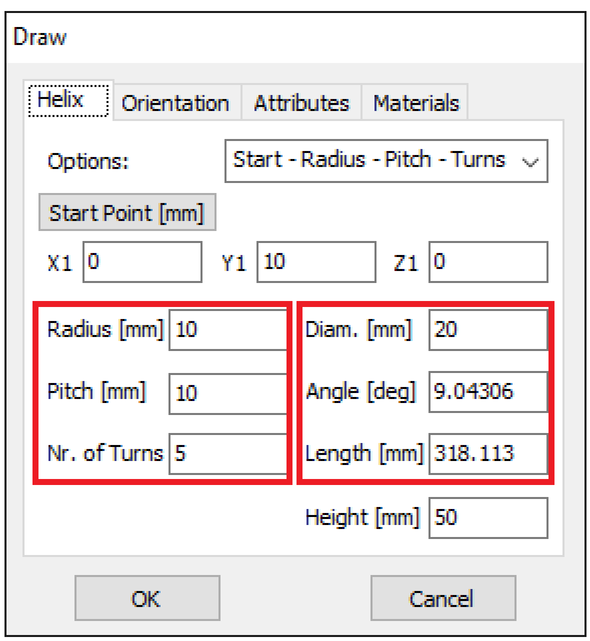


Fig. 1: “Start – Radius – Pitch – Turns” option in the Helix page of the Draw dialog box for the Helix.

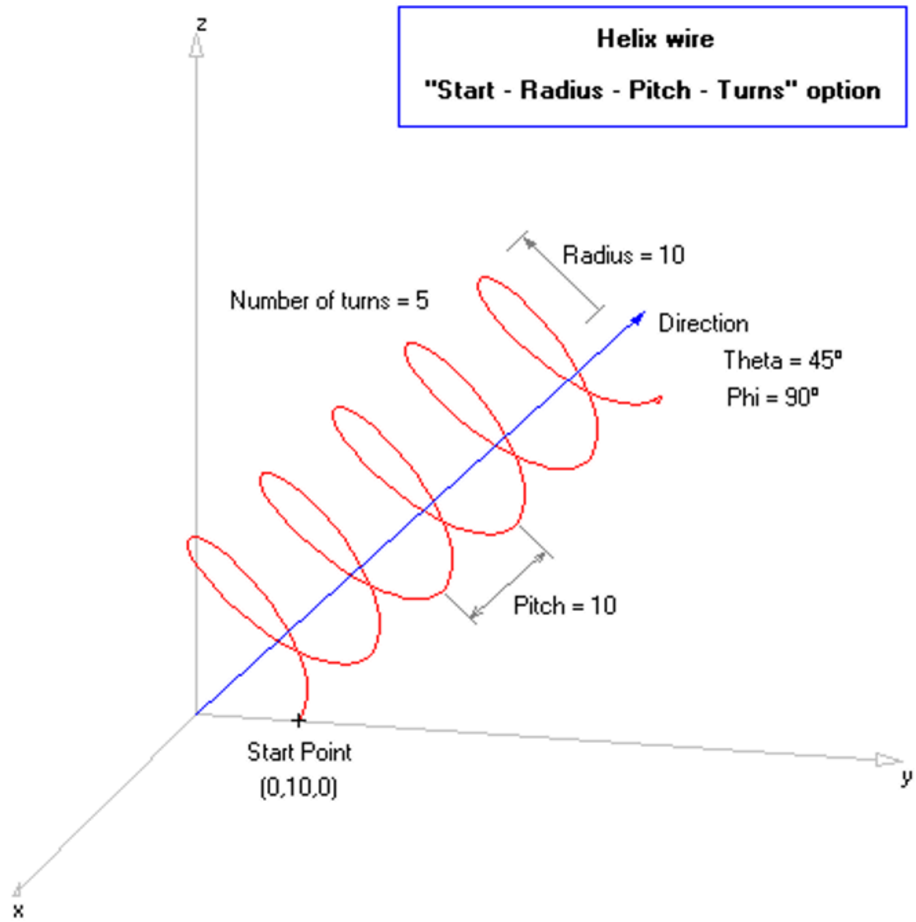


Fig. 2: A Helix drawn using the "Start – Radius – Pitch – Turns" option and with the parameters shown in Fig. 1.

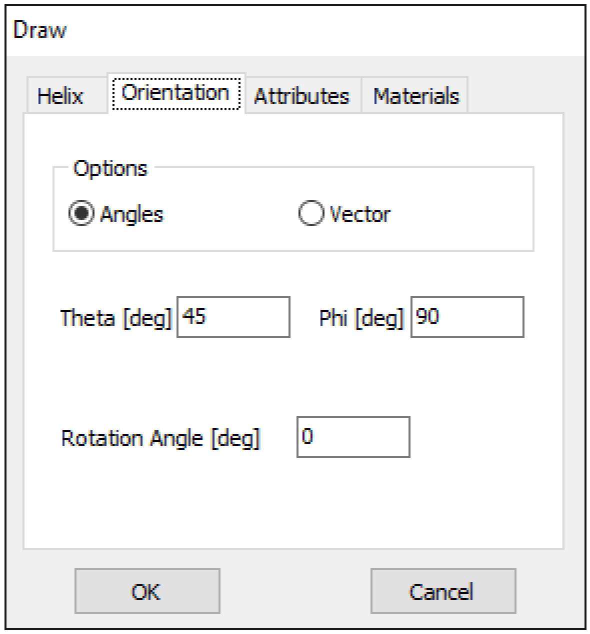


Fig. 3: Orientation page of the Draw dialog box for the Helix with the Theta and Phi angles shown in Figure 2.

If **Start – End – Radius – Turns** is selected, the helix will be drawn starting from the **Start Point** and ending at the **End Point**, with the specified **Radius** and **Number of turns**, as illustrated in Figures 4 and 5. The **Number of turns** must be an integer, and a positive (negative) value results in a right-handed (left-handed) helix. The orientation of the helix axis is determined by the starting and ending points. The helix can be rotated around its axis by specifying a **Rotation Angle**. The **Orientation** page will be hidden when the **Start – End – Radius – Turns** option is chosen, as the helix axis orientation is already defined by the line connecting its start and end points.

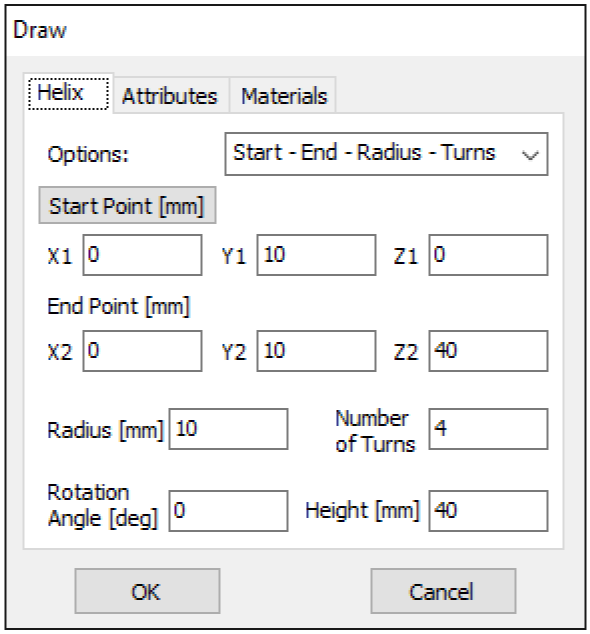


Fig. 4: “Start – End – Radius – Turns” option in the Helix page of the Draw dialog box for the Helix.

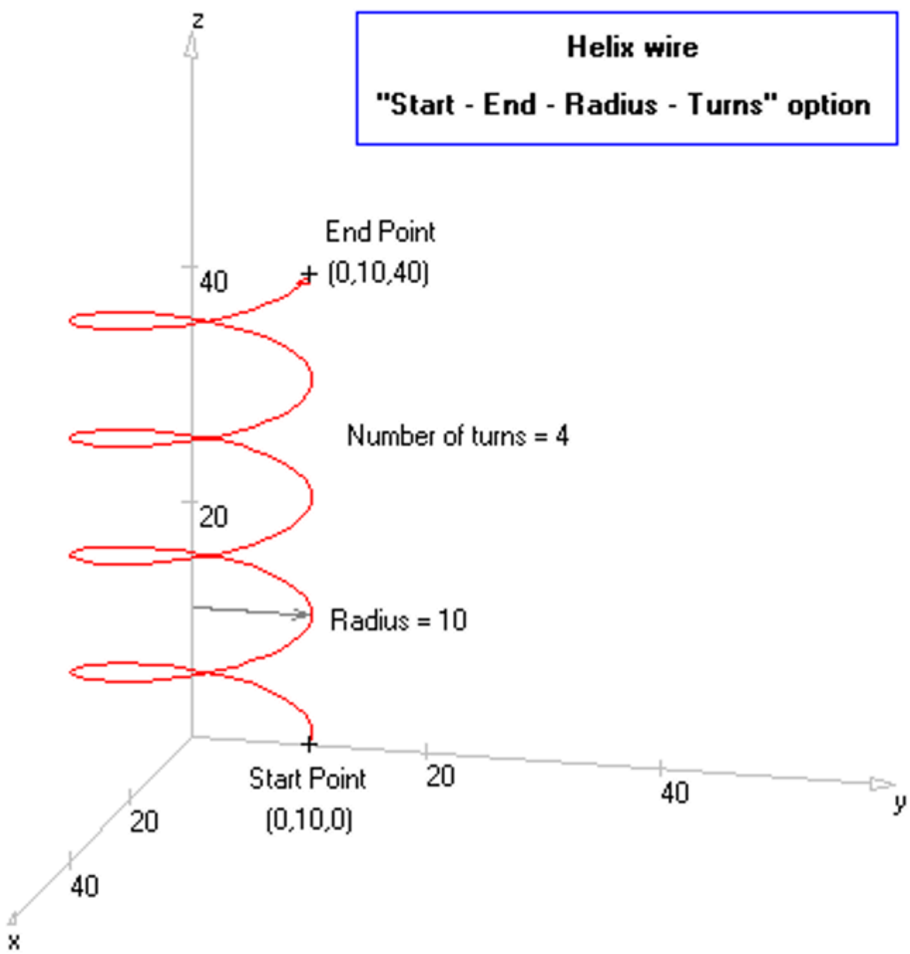


Fig. 5: A Helix drawn using the “Start – End – Radius – Turns” option and with the parameters shown in Fig. 4.

After setting the geometrical parameters on the **Helix** and **Orientation** pages, you can select the **Attributes page** to specify the **Number of Segments** and **Cross-Section**. The **Materials page** allows you to set the wire **Resistivity** and **Coating**.

Orientation Page

The **Orientation** page provides options for setting the helix orientation. A box with two options is available: **Angles** and **Vector** (Fig. 3).

If **Angles** is selected, the helix axis can be defined by specifying its direction in 3D space using the **Theta** and **Phi** angles in spherical coordinates.

If **Vector** is selected, the helix axis can be defined by entering a vector in the axis direction. The $\mathbf{N_x}$, $\mathbf{N_y}$, and $\mathbf{N_z}$ components determine this vector.

The helix can be rotated around its axis by specifying a **Rotation Angle**.

Quadratic

The Quadratic refers to a quadratic wire or parabola.

Go to **Draw > Quadratic** in the main menu to display the Draw dialog box for the Quadratic. This dialog box has three pages: **Quadratic**, **Attributes**, and **Materials**.

The Quadratic page

In the **Quadratic** page the geometrical parameters for the Quadratic can be set, Fig. 1.

The Quadratic is entered by giving three points. A quadratic curve starting from Start Point, passing through Second Point and ending at End Point will be drawn on the workspace, as shown in Figs. 2.

Once the geometrical parameters in the Quadratic page have been set, the **Attributes >** page can be selected, where the number of segments and cross-section can be set. The wire resistivity and coating can be set in the **Materials >** page.

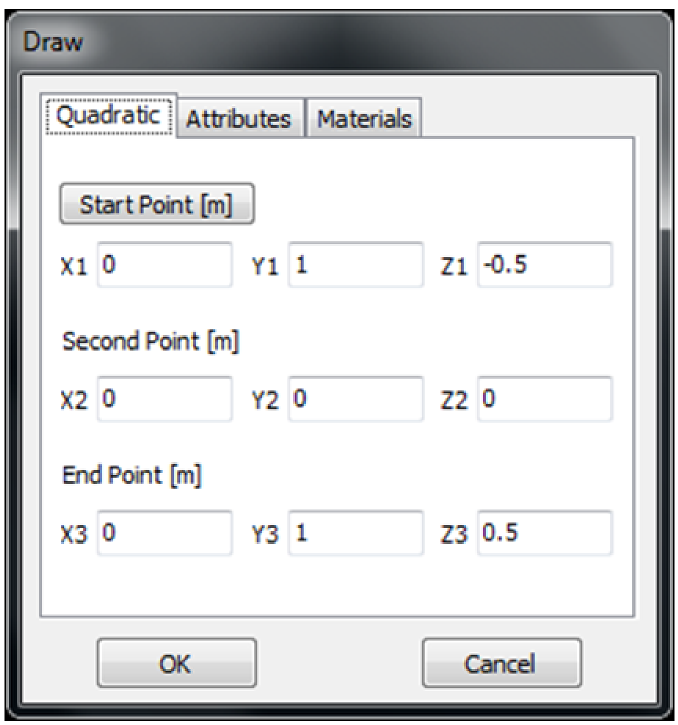


Fig. 1: Quadratic page of the Draw dialog box.

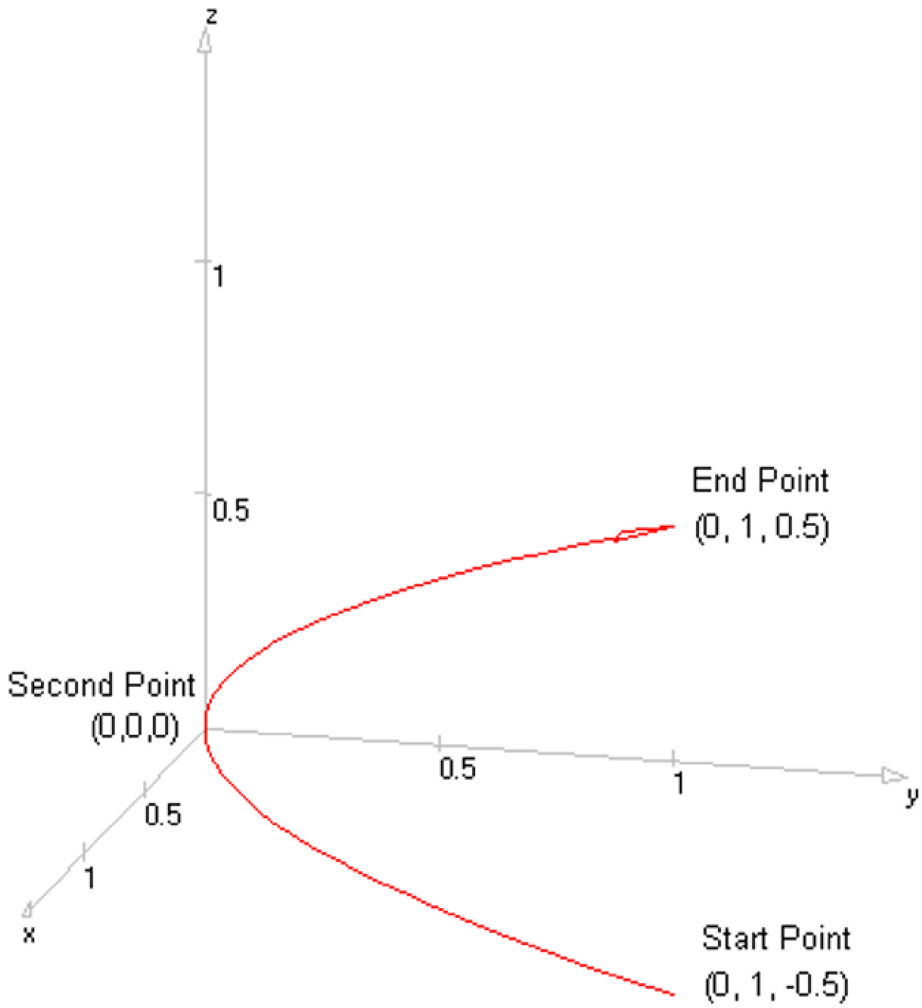


Fig. 2: A Quadratic drawn using the points shown in Fig. 1.

Archimedean Spiral

The Archimedean Spiral refers to the Archimedes' spiral with polar equation $r(\alpha) = r_0 + p/(2\pi)\ \alpha$, where r_0 is the starting radius and p is the pitch. For a spiral with an integer number of turns, M, we have $\alpha = 2\pi M$ at its end point, so $r_{end} = r_0 + pM$, the pitch p being the separation between turns. Besides, we have that the pitch equals the constant growth rate of the spiral radius $r(\alpha)$ per turn, that is $p = 2\pi dr/d\alpha$.

Go to **Draw > Archimedean Spiral** in the main menu to display the Draw dialog box for the Archimedean Spiral. This dialog box has three pages: **Archimedean Spiral**, **Attributes**, and **Materials**.

The Archimedean Spiral page

In the **Archimedean Spiral** page, the geometrical parameters for the Archimedean Spiral can be set, Fig. 1.

The Archimedean spiral is entered by giving the Start Point, Start Radius r_0, Pitch p (positive or negative) and Number of Turns M (complete turns and fractions of a turn can be set). The spiral lies on a plane given by the **Orientation Angles** Theta and Phi (normal to the plane in spherical coordinates) and can be rotated by setting a **Rotation Angle**, Fig. 2.

Once the geometrical parameters in the Archimedean Spiral page have been set, the **Attributes >** page can be selected, where the number of segments and cross-section can be set. The wire resistivity and coating can be set in the **Materials >** page.

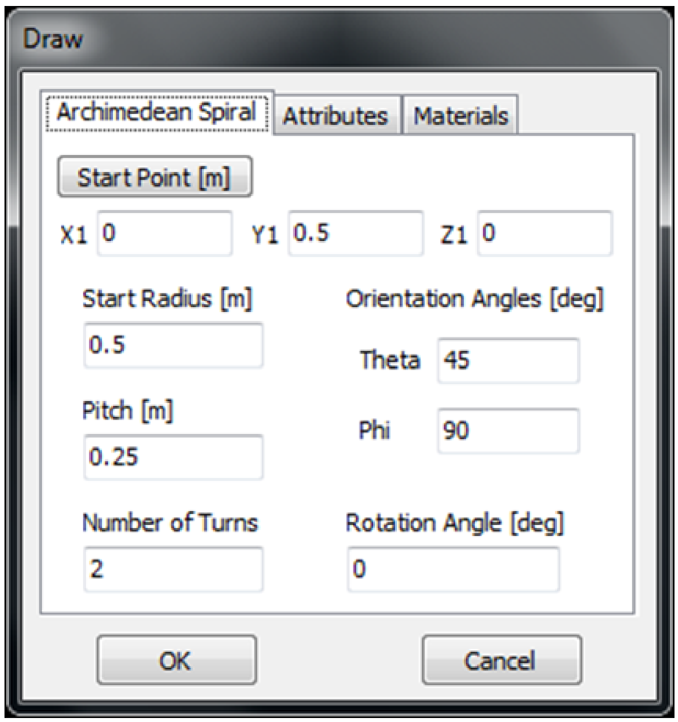


Fig. 1: Archimedean Spiral page of the Draw dialog box.

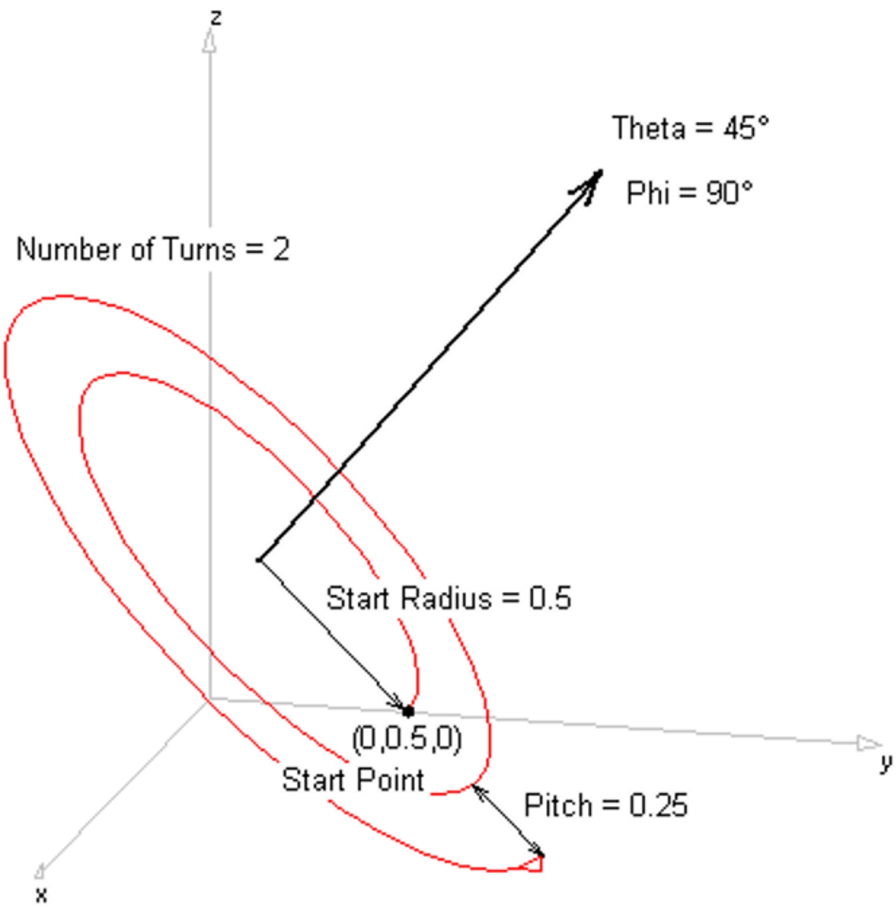


Fig. 2: An Archimedean Spiral drawn using the data shown in Fig. 1.

Logarithmic Spiral

The Logarithmic Spiral refers to a spiral with polar equation $r(\alpha) = r_0 \exp(b\alpha)$, where r_0 is the starting radius (r at $\alpha = 0$), $b = p/(2\pi r_0)$ and p is the starting pitch, that is, the derivative $2\pi dr/d\alpha$ at $\alpha = 0$ (starting growth rate of the spiral radius $r(\alpha)$ per turn). The first two terms of the Taylor expansion $r(\alpha) = r_0 + p/(2\pi)\, \alpha + r_0(b\alpha)^2/2 + \ldots$ give the polar equation of an Archimedean spiral.

Go to **Draw > Logarithmic Spiral** in the main menu to display the Draw dialog box for the Logarithmic Spiral. This dialog box has three pages: **Logarithmic Spiral**, **Attributes**, and **Materials**.

The Logarithmic Spiral page

In the **Logarithmic Spiral** page, the geometrical parameters for the Logarithmic Spiral can be set, Fig. 1.

The logarithmic spiral is entered by giving the Start Point, Start Radius r_0, Start Pitch p (positive or negative) and Number of Turns (complete turns and fractions of a turn can be defined). The spiral lies on a plane given by the **Orientation Angles** Theta and Phi (normal to the plane in spherical coordinates) and can be rotated by setting a **Rotation Angle**, Fig. 2.

Once the geometrical parameters in the Logarithmic Spiral page have been set, the **Attributes >** page can be selected, where the number of segments and cross-section can be set. The wire resistivity and coating can be set in the **Materials >** page.

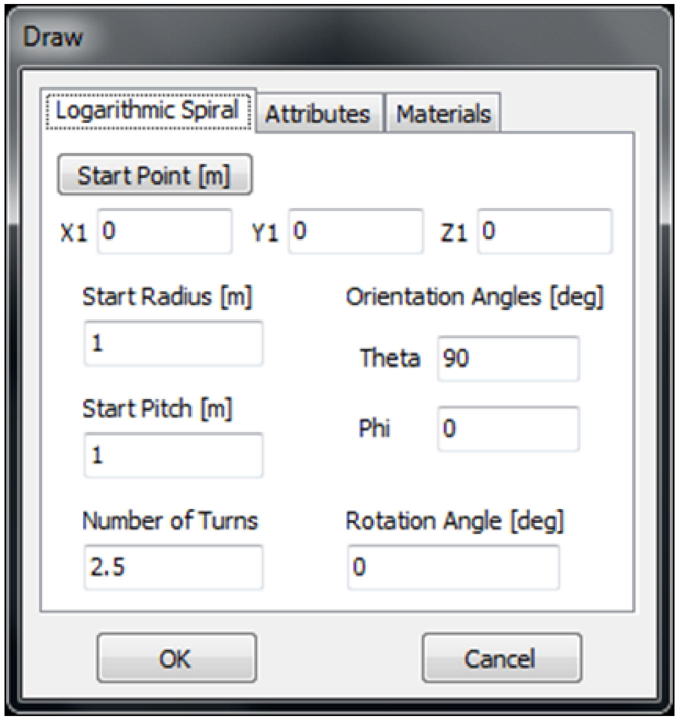


Fig. 1: Logarithmic Spiral page of the Draw dialog box.

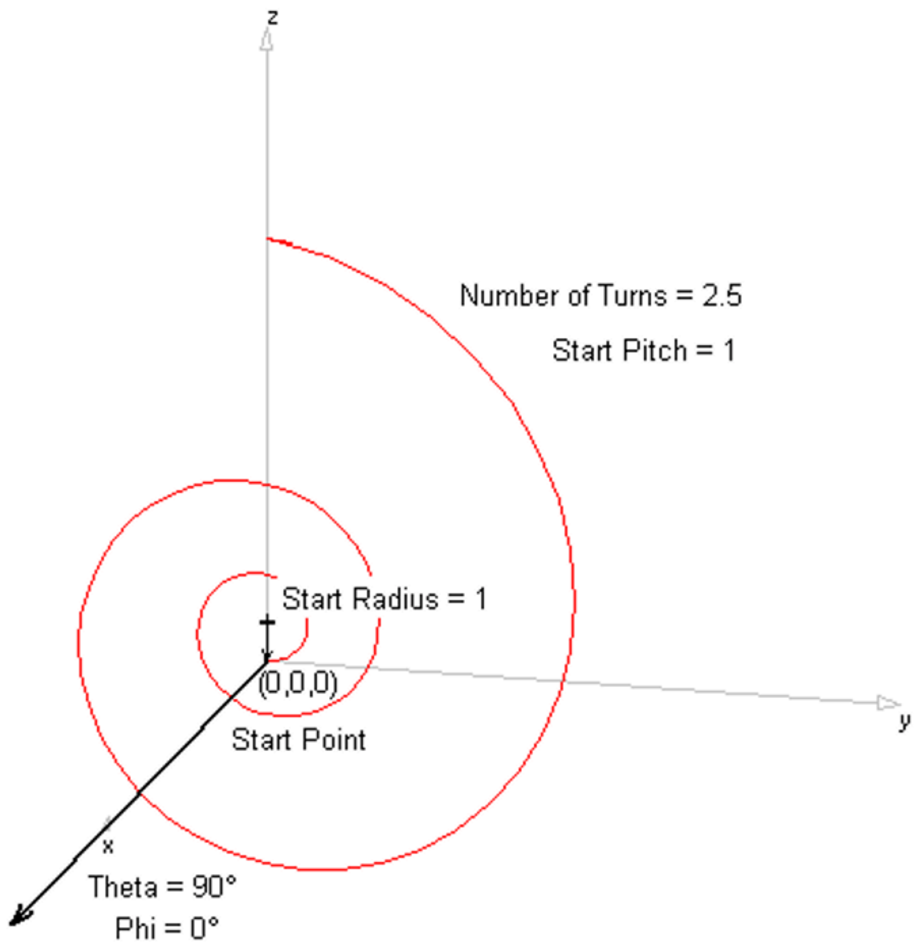


Fig. 2: A Logarithmic Spiral drawn using the data shown in Fig. 1.

Tapered Wires

A tapered wire is a wire with a variable radius along its length. The cross section of tapered wires is always circular. The radius is varied linearly along the wire and in defined steps, then a wire with a stepped radius is obtained, as shown in Fig. 1.

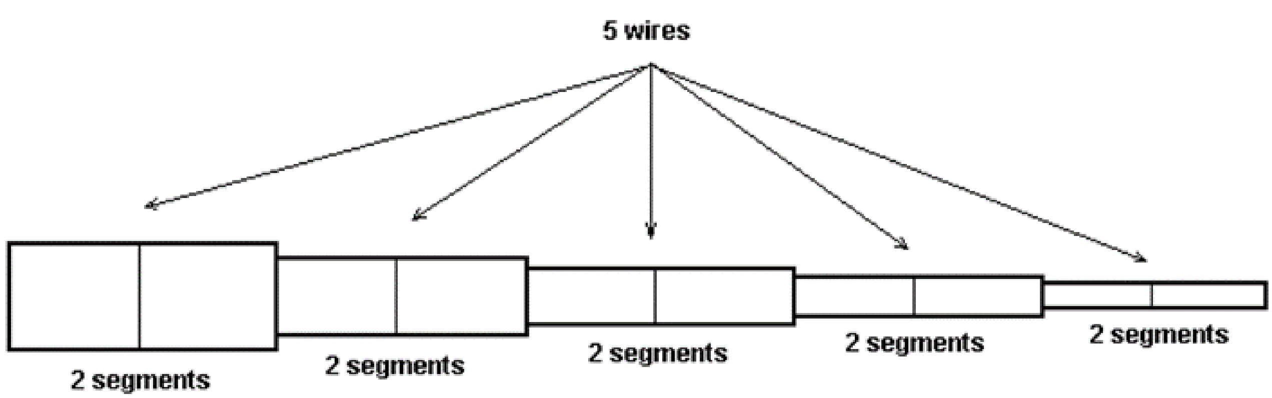


Fig. 1: Example of a tapered wire divided into 5 wire portions. Each portion is divided into 2 segments.

Go to **Draw > Tapered Wire** in the main menu and select a wire type for drawing. The wire types available are the same as in the Draw menu. As an example, Fig. 2 shows the Line page of the Draw dialog box when a linear wire is selected.

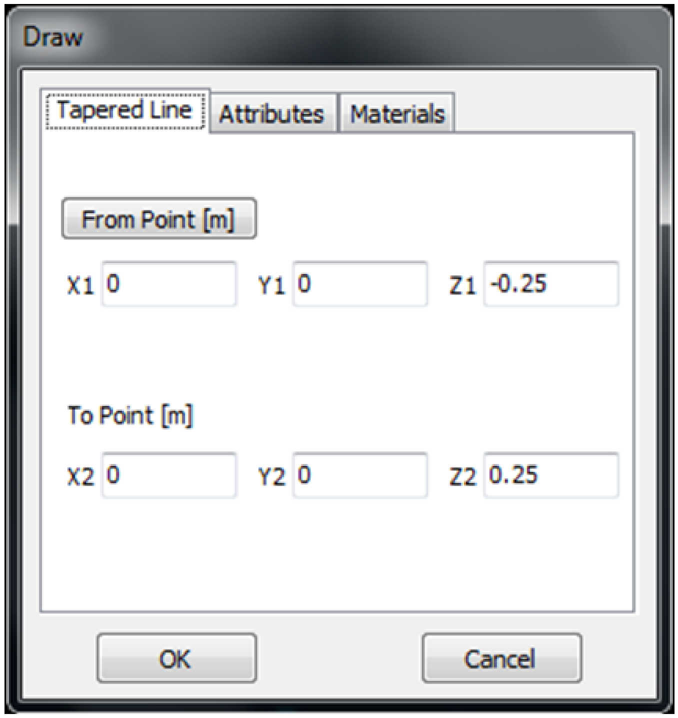


Fig. 2: Tapered Line page in the Draw dialog box. Go to main menu > Draw > Tapered Wire > Tapered Line.

The wire must be divided into wire portions according to the desired steps in radius, as it is indicated in Fig. 1. Also, each wire portion having a uniform radius must be divided into segments as it is required by the Method of Moments used for the simulation.

The number of wire portions and the number of segments per wire can be set by going to the **Attributes** tab, Fig. 3. In this page, the Start and End radii can be set. The resistivity for the conductive wire and its coating material can be set in the **Materials** tab, Fig. 4. In this case, a tapered coating shield can also be set by giving a Start and End thickness.

The wire portions will be displayed in alternating colors for easy identification in the workspace.

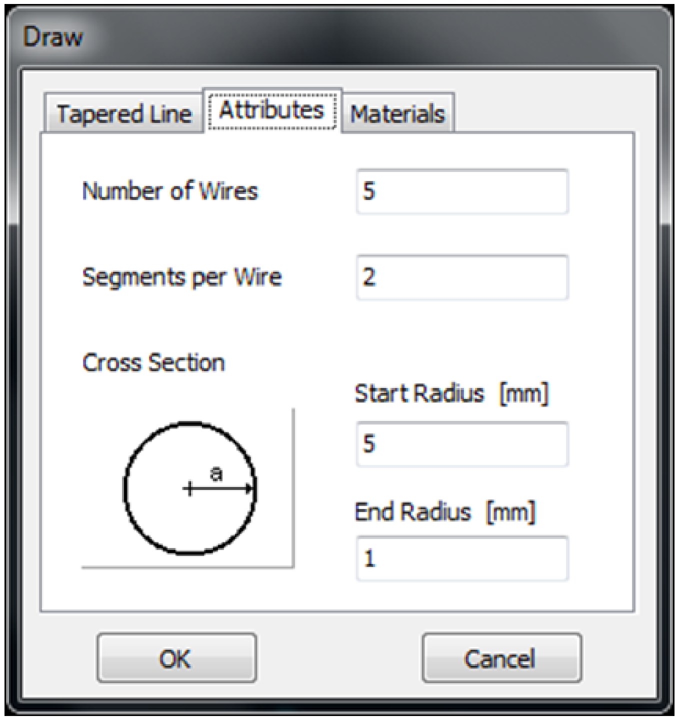


Fig. 3: Attributes page where the number of wire portions and segments per wire can be set, as well as Start and End radii.

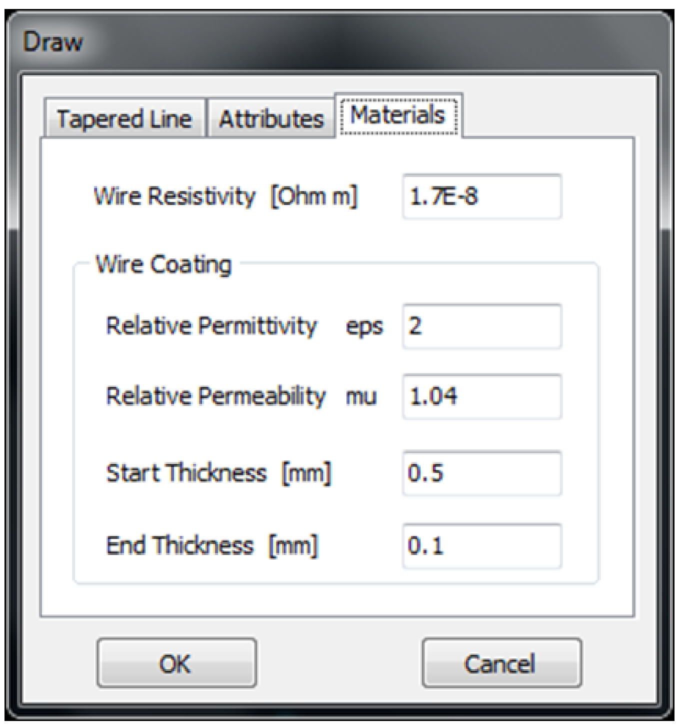


Fig. 4: Materials page where the wire resistivity and coating can be set. A tapered coating can be defined by giving the Start and End thicknesses.

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 four options will be displayed: **AN-SOF**, **NEC**, **DXF**, and **MM** formats.
3. Note that DXF and MM formats 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 language is not supported. To run simulations with variable geometric parameters, you can write scripts to generate the NEC files and then use the **Run Bulk Simulation** command (refer to section “12.8 Running a Bulk Simulation”). **See examples here >**.

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 RLC loads, fixed impedances R+jX, and wire conductivity can be set.

- Set **Type = 0** for a series 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” and “LD 4” lines.**

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 is used to specify frequencies for simulations. The syntax is as follows:

FR Type Num 0 0 Freq Df

[Enter]

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

Num: Number of frequency steps.

Freq: Frequency in MHz or starting frequency in 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.

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.

DXF Format

The DXF file format is a standard format for storing CAD (Computer Aided Design) geometrical data as ASCII text lines.

Only DXF files containing LINE objects can be imported into AN-SOF. The structure of a LINE entity is as follows, where only the (X,Y,Z) coordinates of the starting and ending points are read:

LINE

8 *// Subclass marker. Not read*

0 *// Thickness (default = 0). Not read*

10 *// Starting point – 10, 20, 30 are tags – Not read*

-0.5000 *// X value*

20 *// Not read*

-0.5000 *// Y value*

30 *// Not read*

1.000 *// Z value*

11 *// Ending point – 11, 21, 31 are tags – Not read*

0.5000 *// X value*

21 *// Not read*

-0.5000 *// Y value*

31 *// Not read*

1.000 *// Z value*

0 *// Extrusion direction (default = 0) – Not read*

Since LINE objects have zero thickness, AN-SOF will set a wire radius equal to 0.5% of the wire length. The LINE coordinates in the DXF file are in meters. AN-SOF will also set the number of segments for each wire according to the operating frequency, so it is recommended to set the frequencies before importing the DXF file. Wire radii and the number of segments can be modified after importing the DXF file using the **Modify command >** in the main menu.

Download examples of DXF files to import into AN-SOF >

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, automatic segmentation of a wire can be obtained by entering any number equal or less than zero as the number of segments. The units for the coordinates of the start and end points of any wire must be consistent with the length unit chosen in the AN-SOF Preferences dialog box. Also, the wire radius or diameter of any imported wire must be expressed in the unit chosen in the **Preferences >** dialog box.
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.

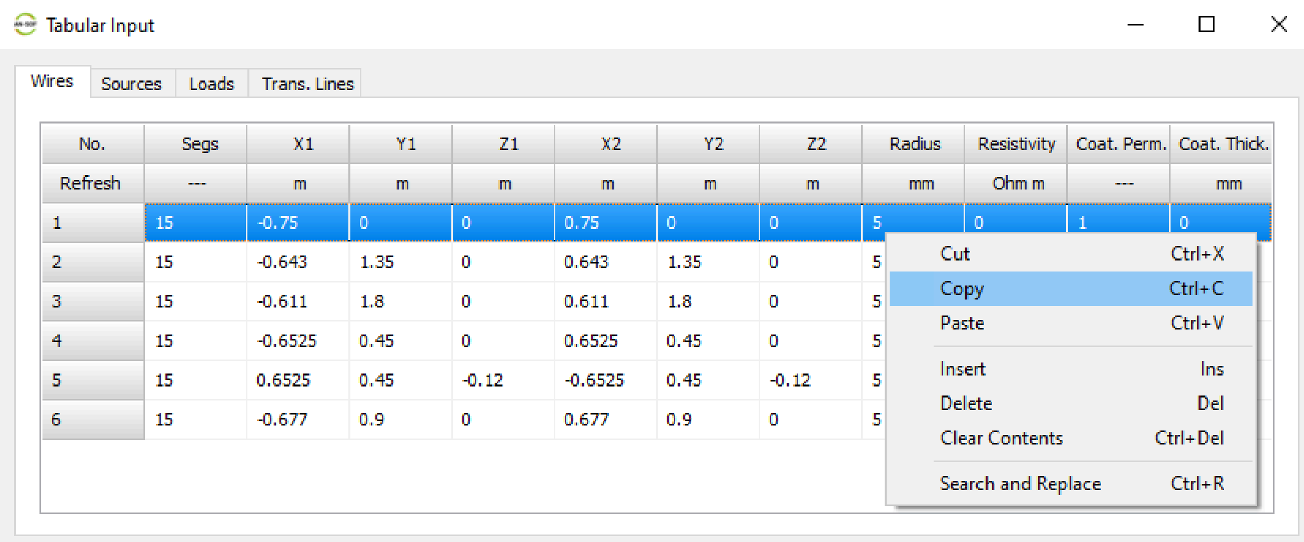


Fig. 1: Tabular Input window – Wires tab, for entering and editing linear wires.

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.

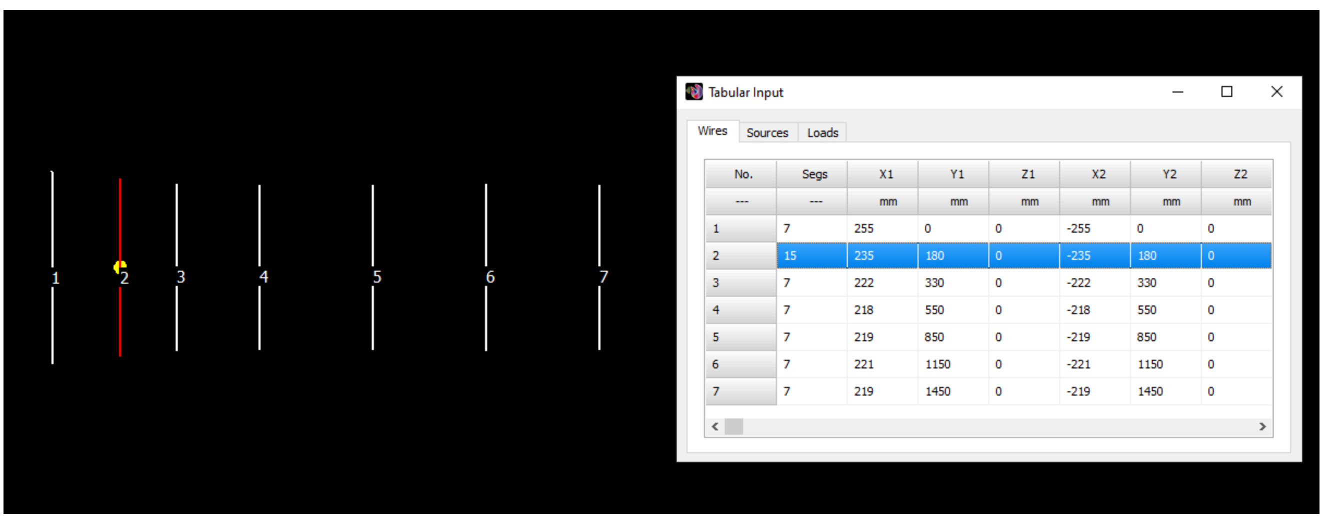


Fig. 2: Tabular Input window, showing wire numbers displayed in the workspace.

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.

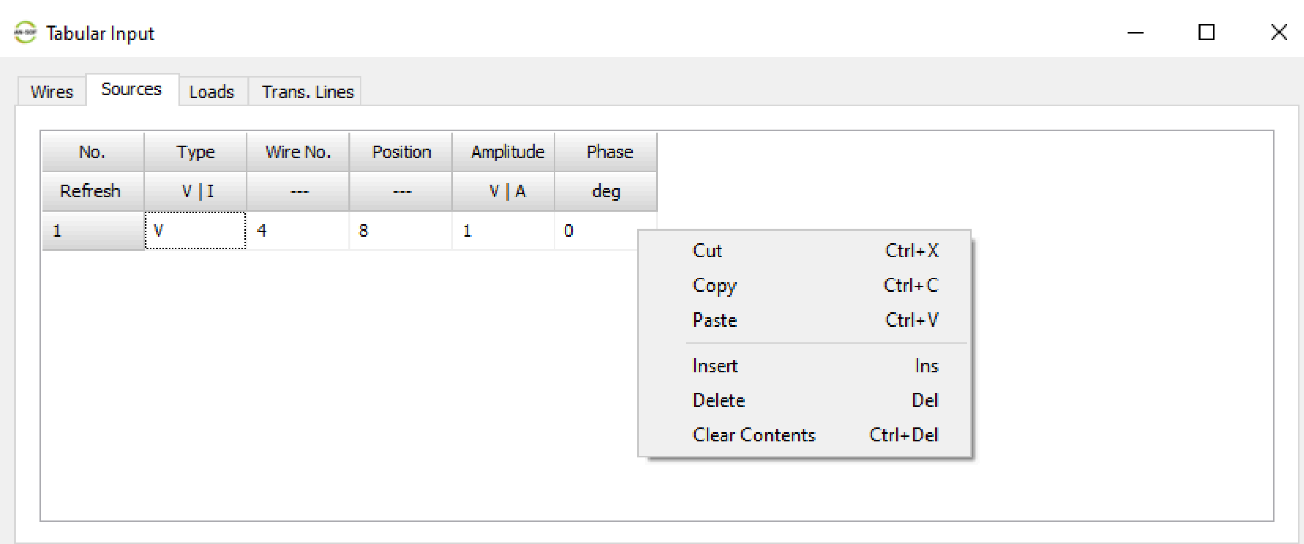


Fig. 3: Tabular Input window – Sources tab, for entering and editing sources.

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.

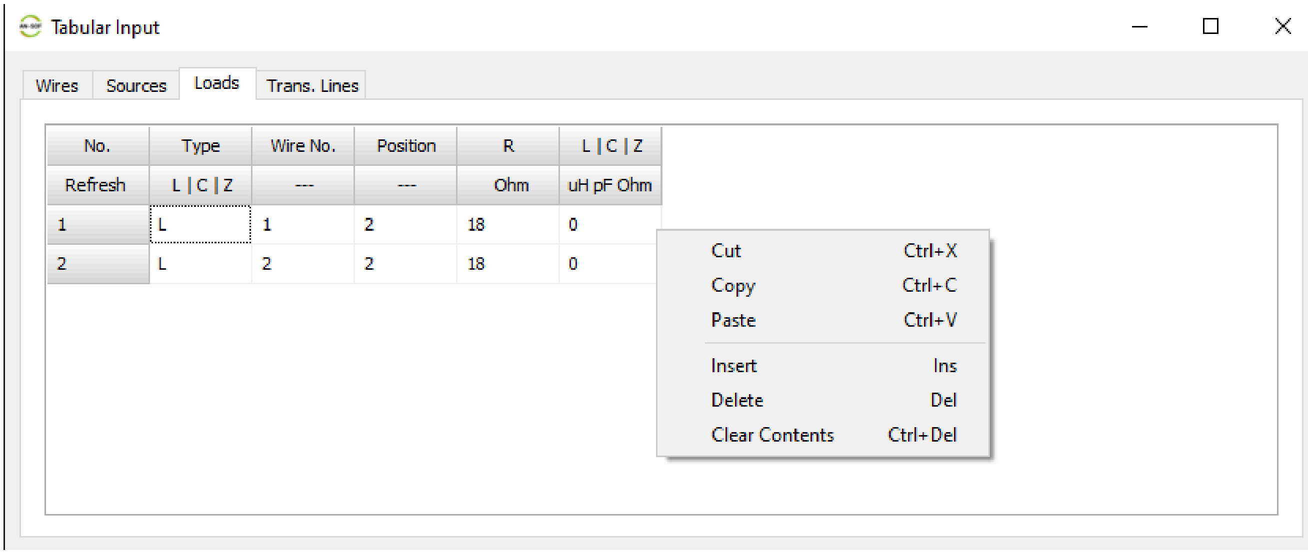


Fig. 4: Tabular Input window – Loads tab, for entering and editing load impedances.

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.

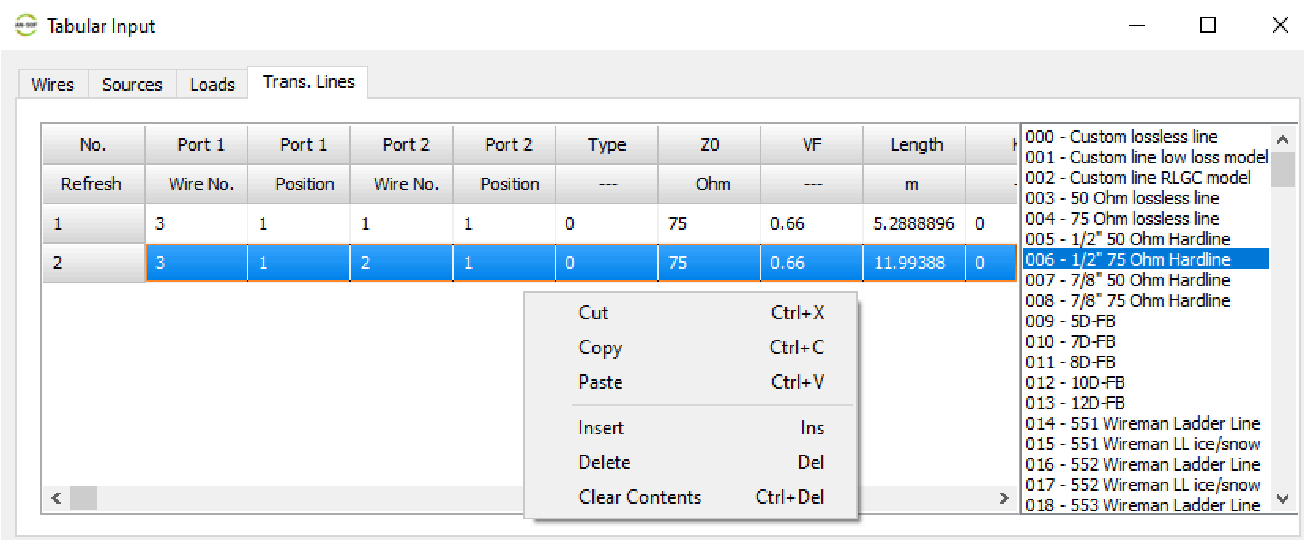


Fig. 5: Tabular Input window – Trans. Lines tab, for entering and editing transmission lines.

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.

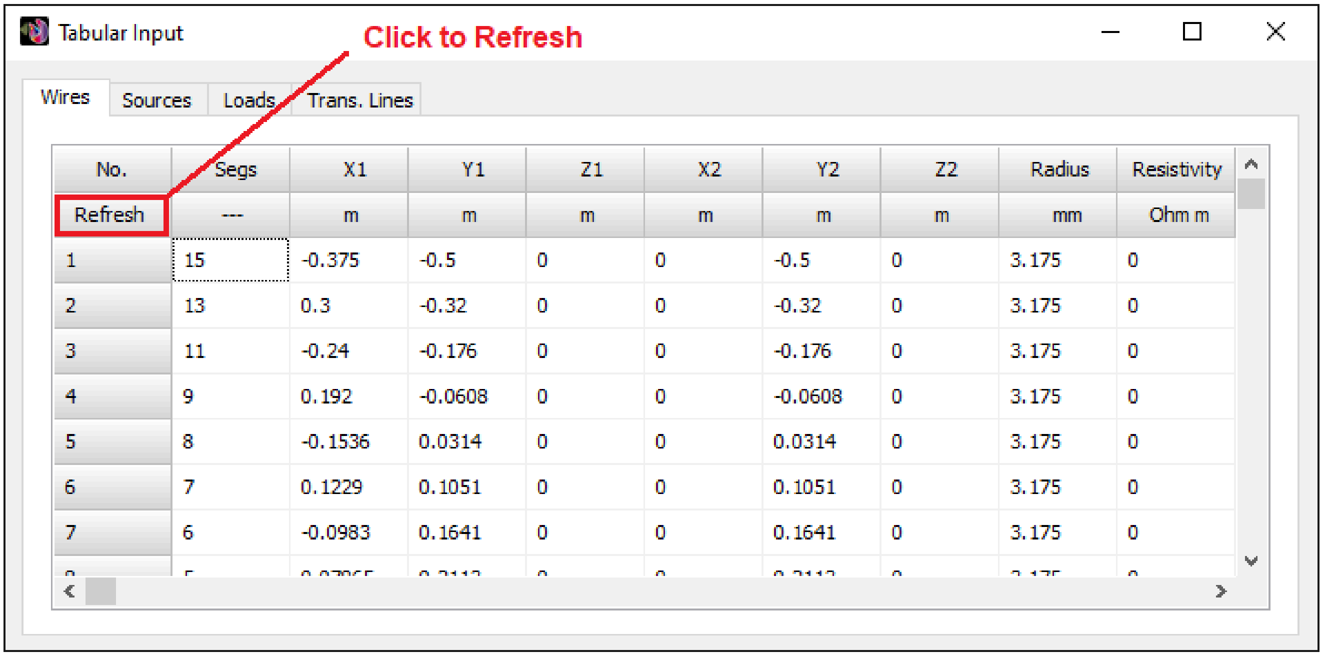


Fig. 6: Refresh button in the Tabular Input window, for applying changes instantly.

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.

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).

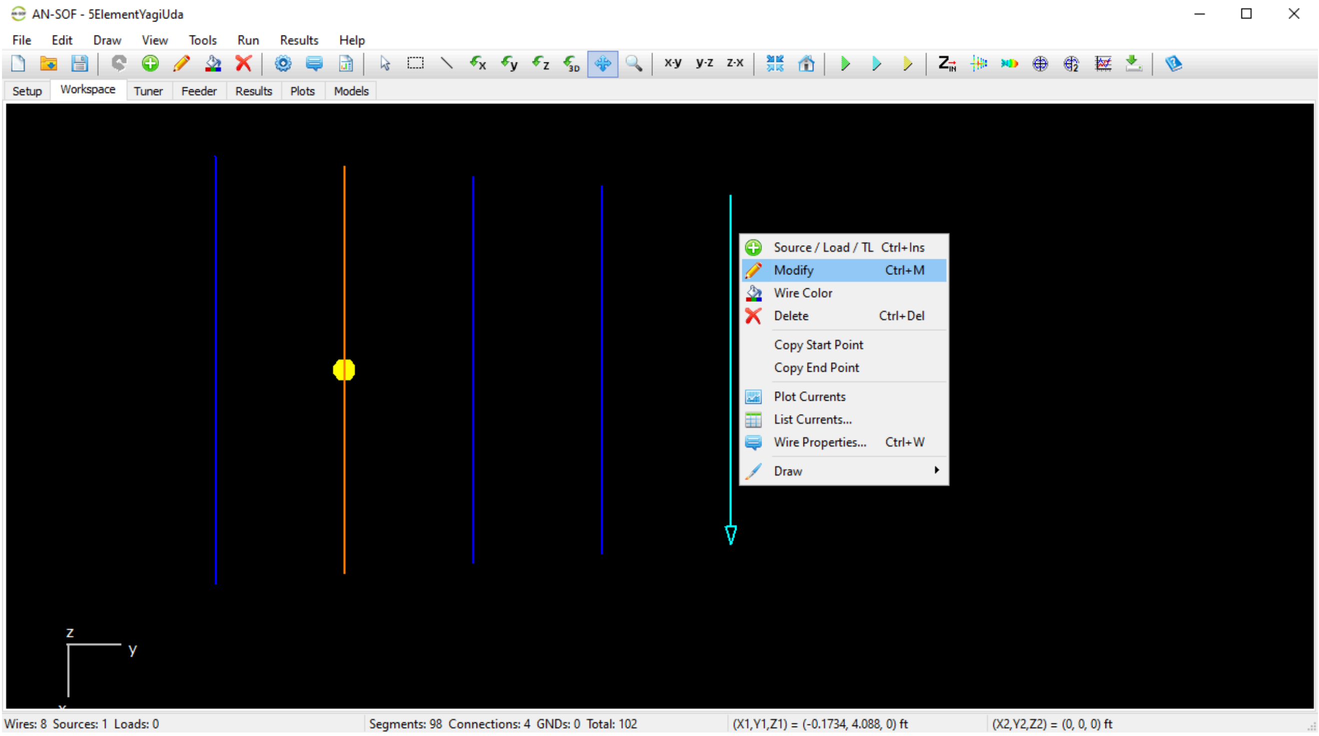


Fig. 1: Pop-up menu displayed when right-clicking on a selected wire.

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 a wire, right-click on it to bring up the **pop-up menu**. From the menu, select the **Modify** command to open the **Modify dialog box**, where you can adjust the geometrical parameters and attributes of the selected wire.

Alternatively, you can modify a wire by first pressing the **Select Wire** button (arrow icon) in the main toolbar and then left-clicking on the wire. Once the wire is selected, navigate to **Edit > Modify** in the main menu. This option is enabled only when a wire is selected.

Deleting a Wire

To delete a wire, right-click on it to bring up the **pop-up menu**. From the menu, select the **Delete** command to remove the selected wire, along with all sources and loads placed on it.

Alternatively, you can delete a wire by first pressing the **Select Wire** button (arrow icon) in the main toolbar and then left-clicking on the wire. Once the wire is selected, navigate to **Edit > Delete** in the main menu. This option is enabled only when a wire is selected.
Modifying a Group of Wires

AN-SOF allows you to simultaneously edit a group of wires. There are three ways to select a group of 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.
3. To deselect all wires, double-click on the screen or click the **Home** button in the toolbar.

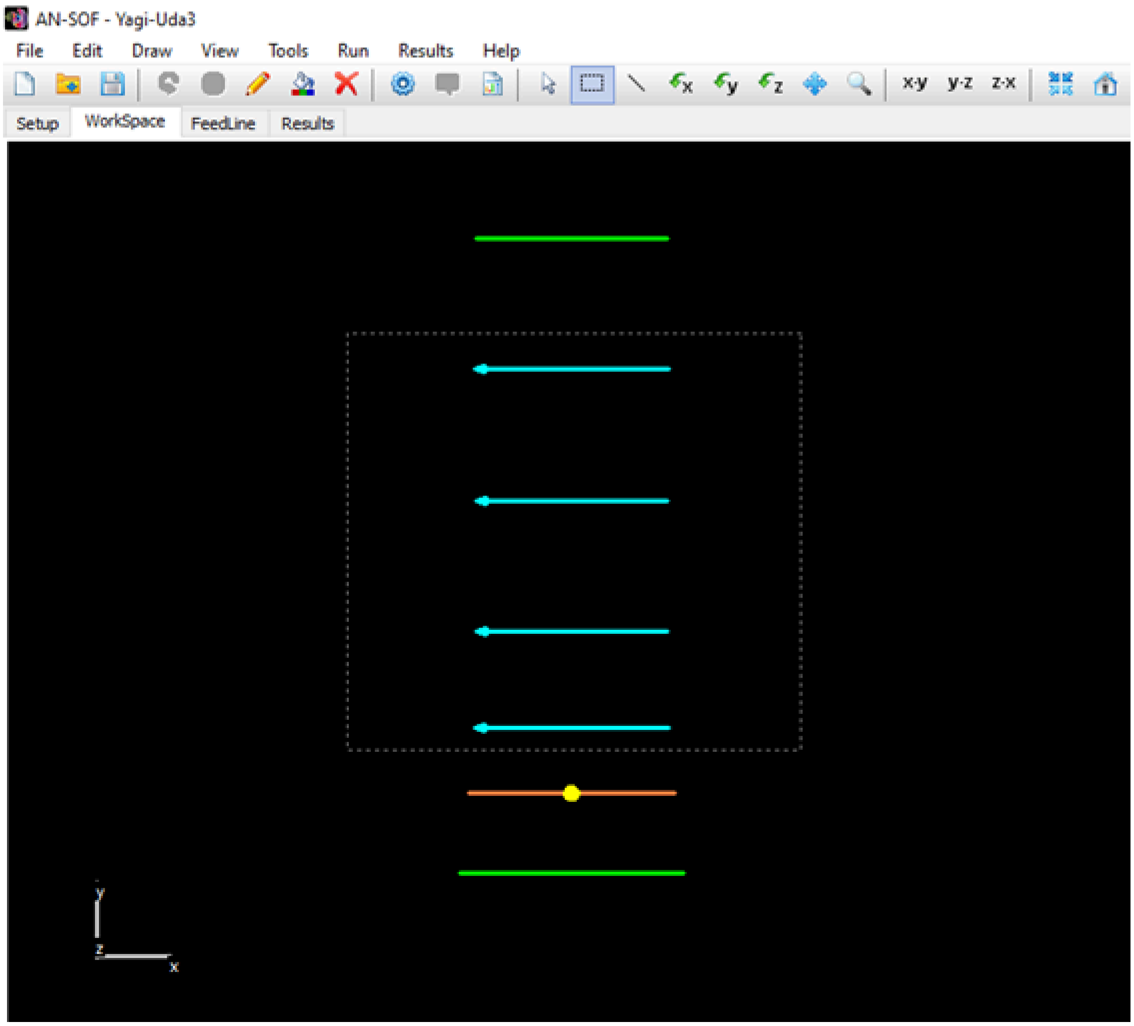
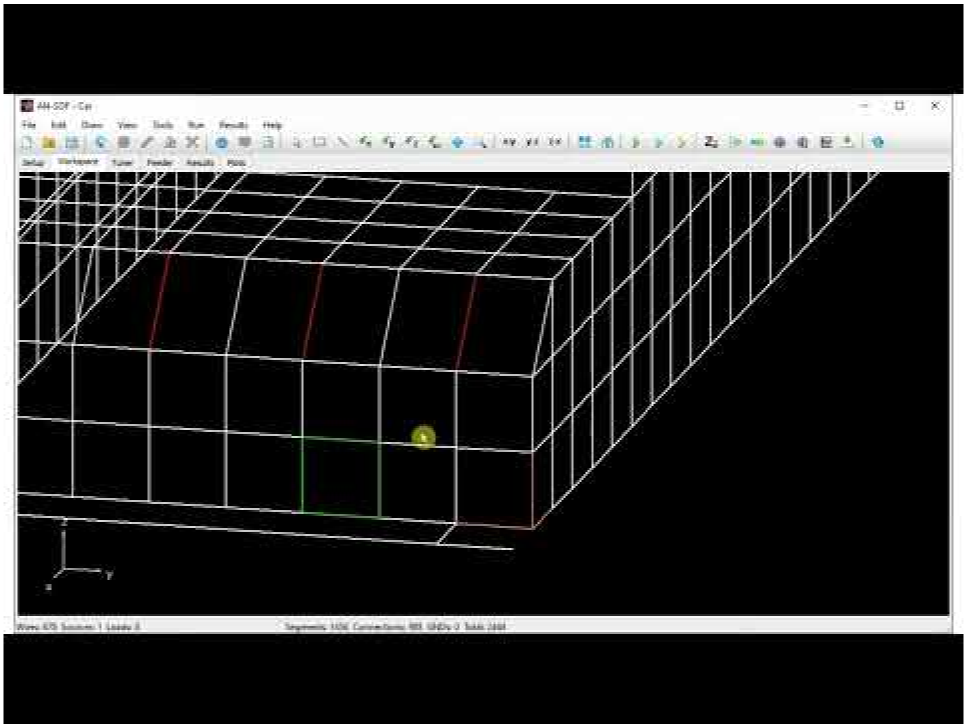


Fig. 1: Selection box for choosing a group of wires.

Selecting Wire by Wire

1. Click the **Select Wire** button (arrow icon) on the main toolbar.
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 click the **Home** button in the toolbar.



Watch Video At: https://youtu.be/BL7tpIoW0So

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. 2) 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"**.

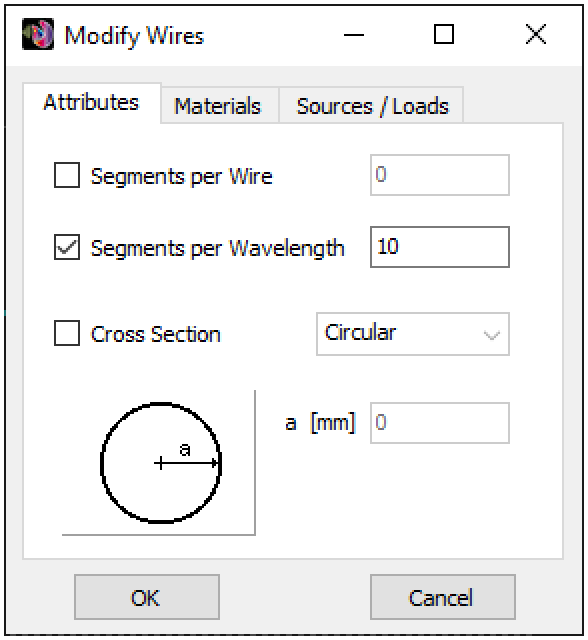


Fig. 2(a): The Attributes tab in the Modify Wires dialog box.

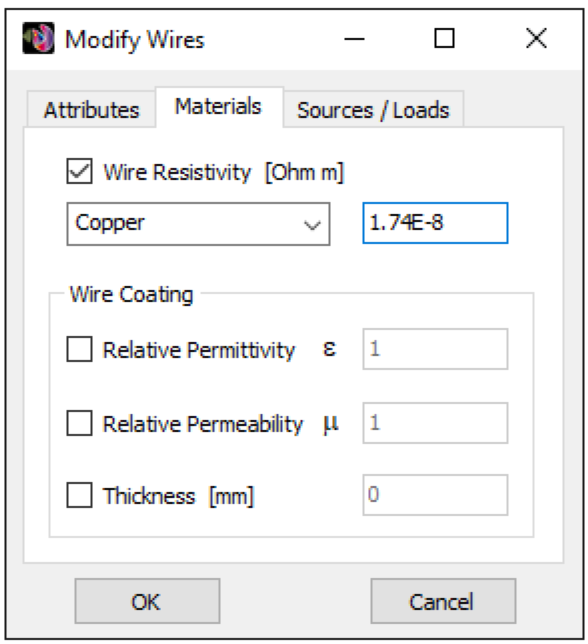


Fig. 2(b): The Materials tab in the Modify Wires dialog box.

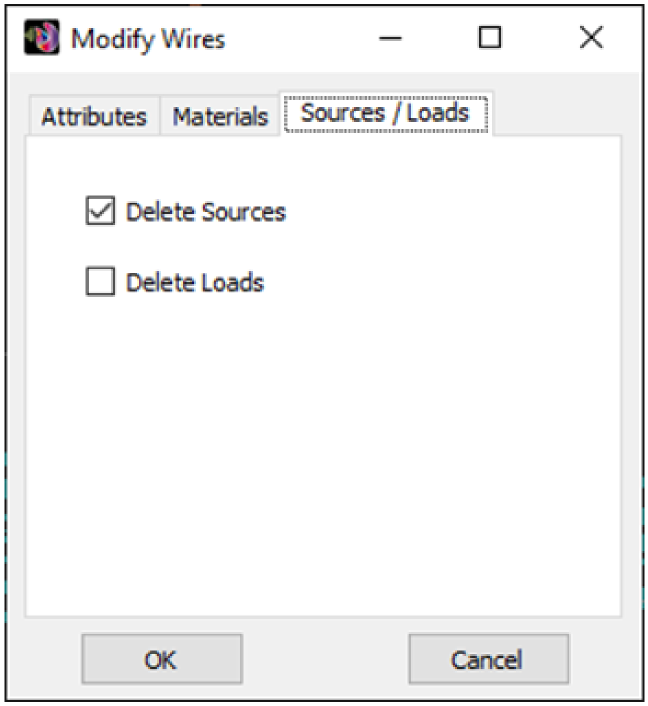


Fig. 2(c): The Sources/Loads tab in the Modify Wires dialog box.

Deleting a Group of Wires

You can delete a selected group of wires in bulk. First, select the group of wires using one of the following methods:

- **Selection Box**: Drag a rectangular box to select multiple wires.
- **Wire by Wire**: Select individual wires by left-clicking on them while holding the **Ctrl** key.
- **Combination of Both**: Use a mix of the Selection Box and wire-by-wire selection.

For detailed instructions, refer to the section: **Modifying a Group of Wires**.

Once the wires are selected, navigate to **Edit > Delete** in the main menu to delete the group. Alternatively, you can use the keyboard shortcut **Ctrl + Del** or click the **Delete** button on the toolbar.

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**:

1. Select the wires using the **Selection Box** or by selecting them individually (as explained in the section **Modifying a Group of Wires**).
2. Navigate to **Edit > Wire Color** in the main menu.

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 Ω = 2 ln(*L*/*a*), where *L* is the wire length and *a* is the wire radius.

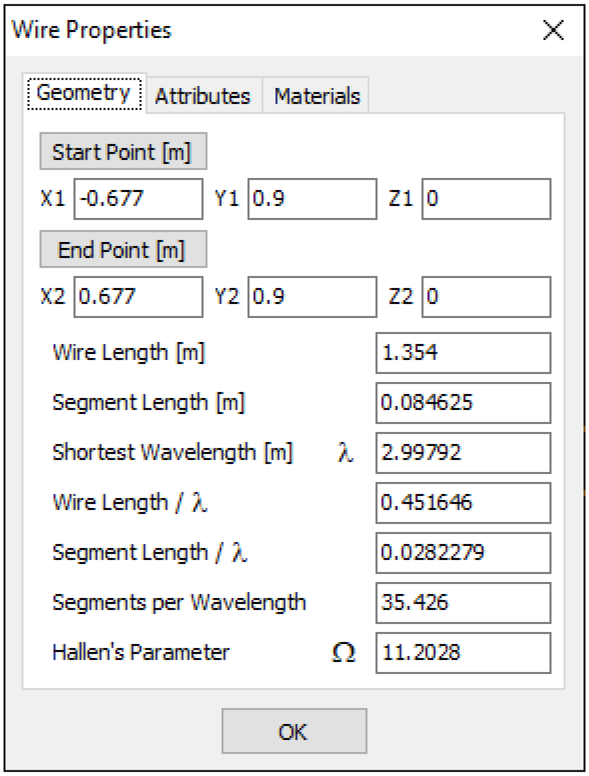


Fig. 1: Wire Properties window – Geometry tab, displaying the geometrical properties of the selected wire.

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.

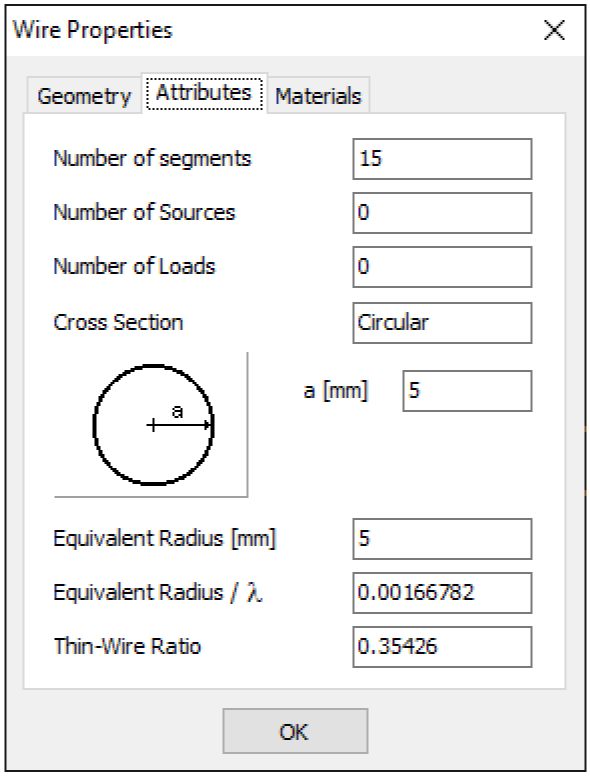


Fig. 2: Wire Properties window – Attributes tab, displaying the segmentation, number of sources and loads, and cross-section type of the selected wire.

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.

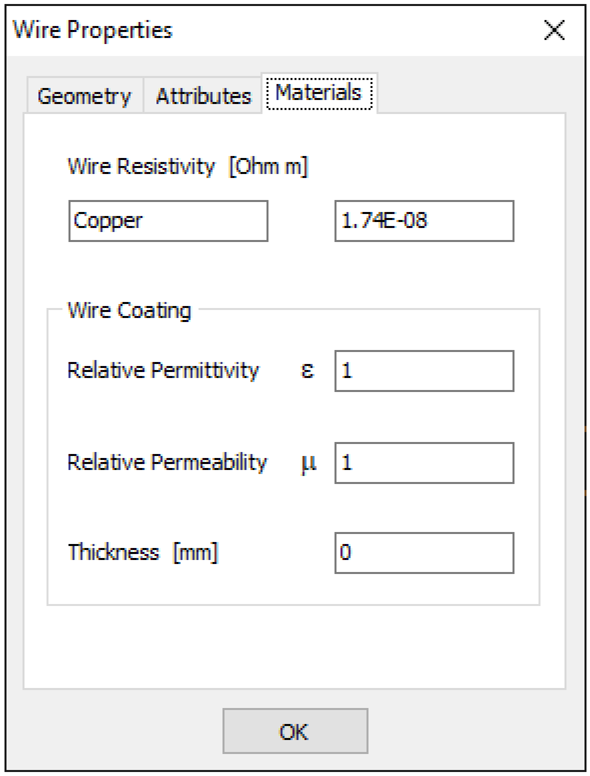


Fig. 3: Wire Properties window – Materials tab, displaying the material parameters of the conductive wire and its coating shield or insulation.

Connecting Wires

A wire junction is automatically established whenever the coordinates of a wire end are identical to the end coordinates of a wire previously specified. However, two wires will be also connected automatically when their ends are spaced one tenth of the wire radius. Wire junctions must be established to satisfy Kirchhoff's current law at the connection point.

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

Fig. 1: Wrong and right ways to connect wires.

Two wires can be connected by copying and pasting their ends. The following procedure will show how to connect the Start Point of a wire #1 to the Start Point of a wire #2.

Procedure for connecting two wires at their ends

1. Right clicking on wire #1 will display a pop-up menu.
2. Choose the **Copy Start Point** or **Copy End Point** command from the pop-up menu. This command is also available in the Wire Properties window of the selected wire, Fig. 1.
3. In this example, wire #2 will be a Line. Then, choose Draw/Line in the main menu to display the Draw dialog box for the Line.
4. Press the **From Point** button to paste the copied point, Fig. 2. Then, complete the definition of wire #2.

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

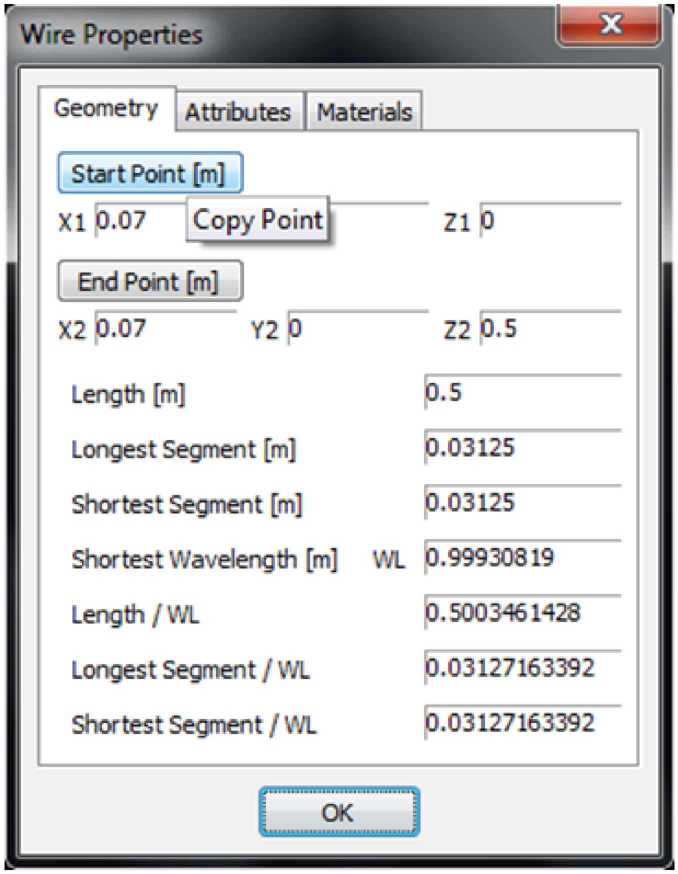


Fig. 1(a): Wire Properties dialog box. Click on the "Start Point" button to copy a wire end.

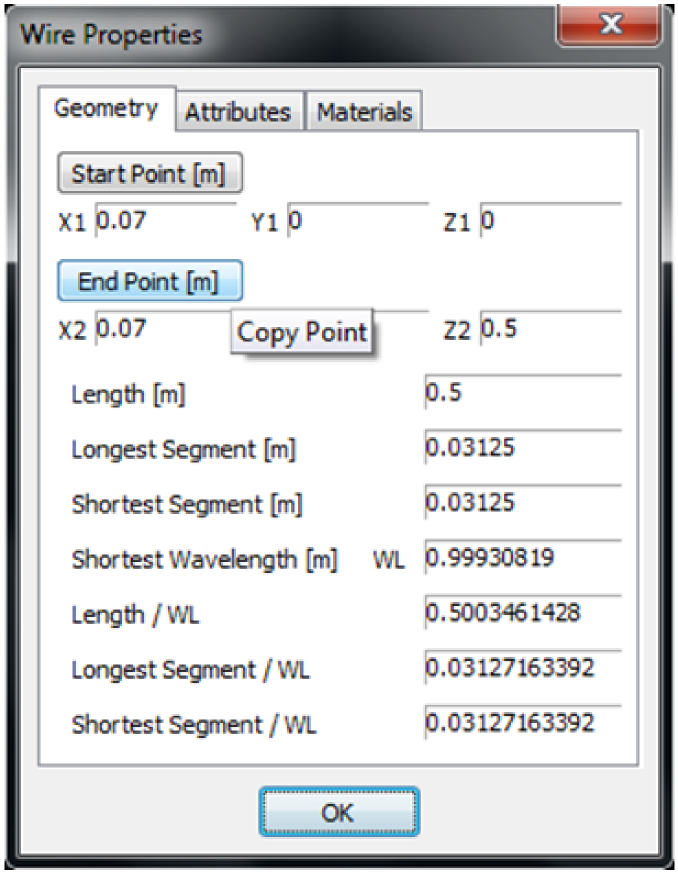


Fig. 1(b): Wire Properties dialog box. Click on the "End Point" button to copy a wire end.

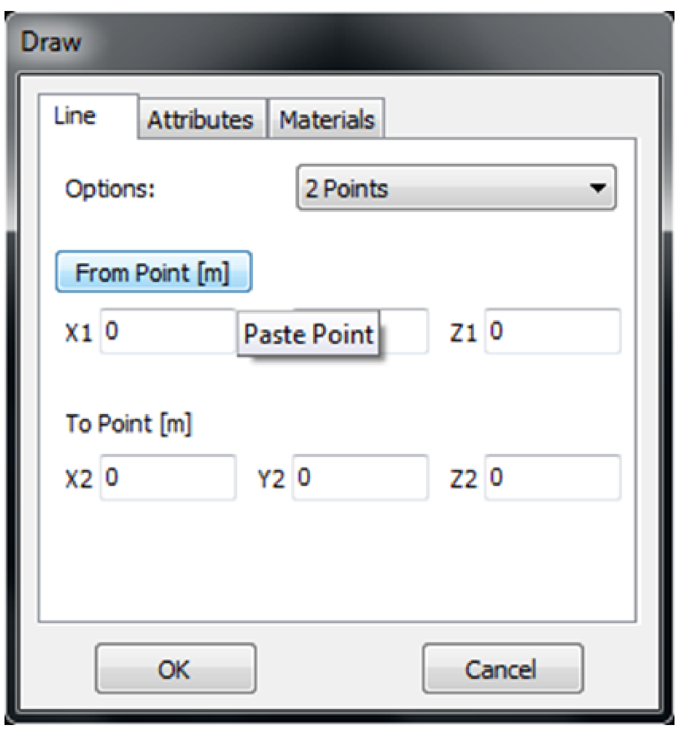


Fig. 2: Draw dialog box for wire #2. Click on the "From Point" button to paste the copied end of wire #1.

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.
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 click the **Home** button on the toolbar.

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

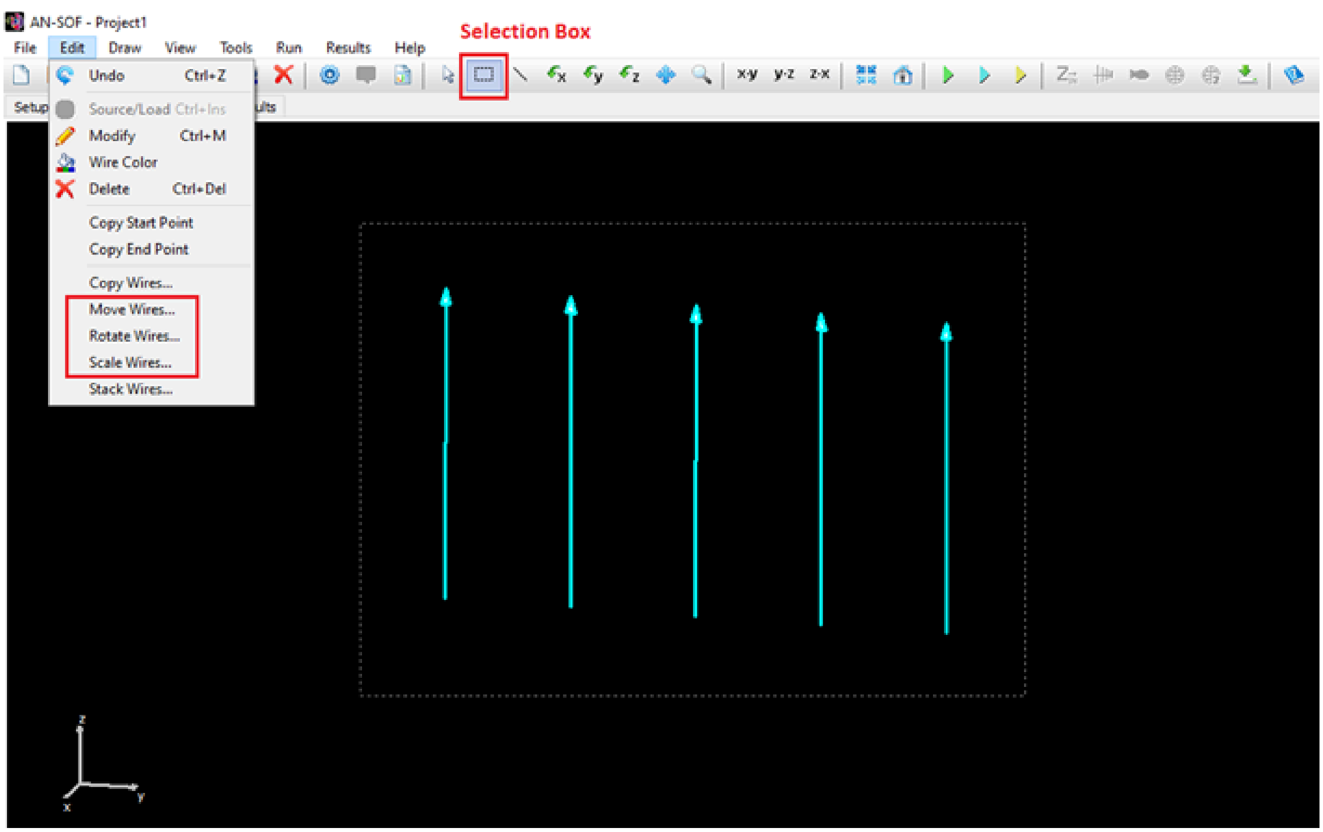


Fig. 1: "Selection Box" button on the toolbar to select a group of wires and commands in the Edit menu to move, rotate, and scale the selected wires.

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.

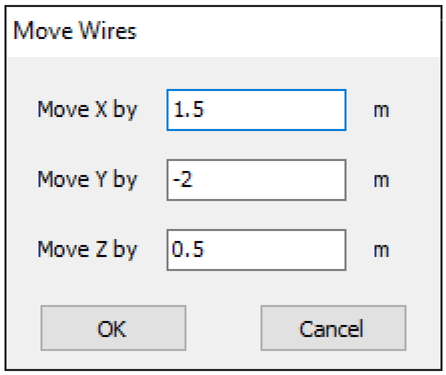


Fig. 2: Move Wires dialog box.

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.

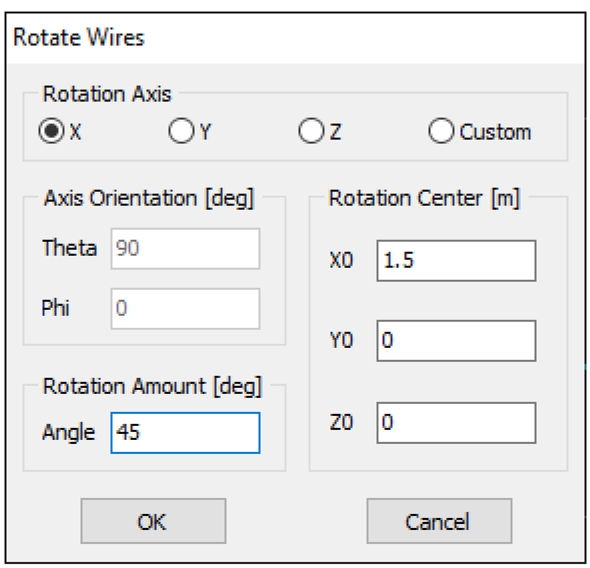


Fig. 3: Rotate Wires dialog box.

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.

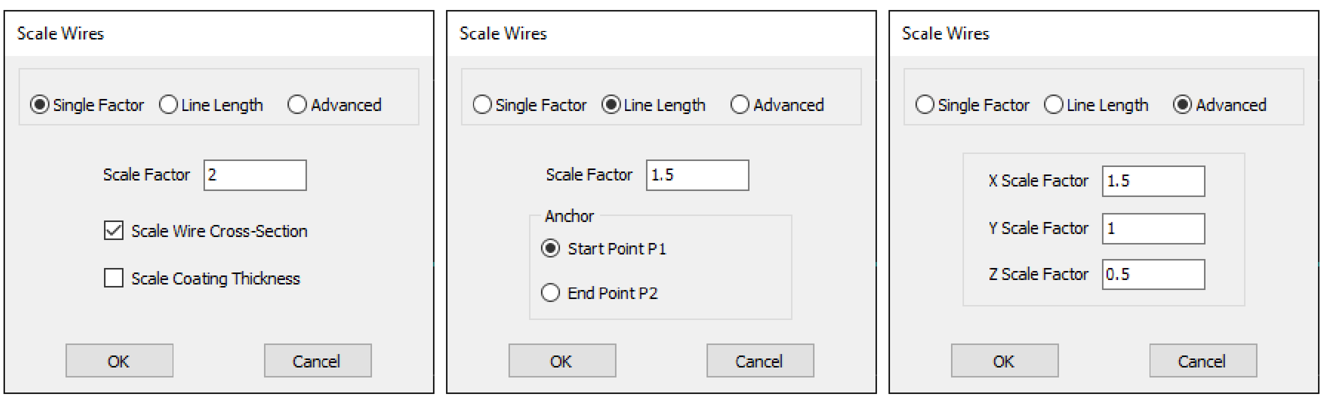


Fig. 4: Scale Wires dialog box. (Left) "Single Factor" option. (Center) "Line Length" option. (Right) "Advanced" option.

> **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 is often necessary to copy wires from one position to another. An antenna array is an example of such a scenario. To copy wires, you must first select them by pressing the **Selection Box** button on the toolbar and then expanding a box using the mouse to enclose the wires you wish to copy, as explained in the **Moving, Rotating and Scaling Wires** section.

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

Copy Wires

Displays the **Copy Wires** dialog box for copying the selected wire or group of wires (Fig. 1). You can specify the **number of copies** of the selected group of wires. The first copy will be offset from the original wire group according to the entered X, Y, and Z offsets and/or rotated around each axis according to the entered angles. Subsequently, each copy will be offset and/or rotated relative to the previous copy.

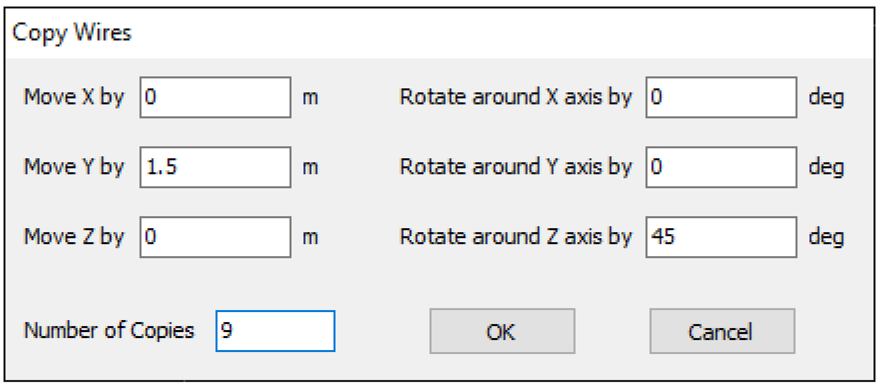


Fig. 1: Copy Wires dialog box.

Stack Wires

Displays the **Stack Wires** dialog box for stacking the selected wire or group of wires along the specified direction and according to the given **number of elements** in the stack (Fig. 2). An “element” in the stack is composed of the selected wires, so an element could be a single wire or a group of wires. You must also specify the **spacing between the elements**.

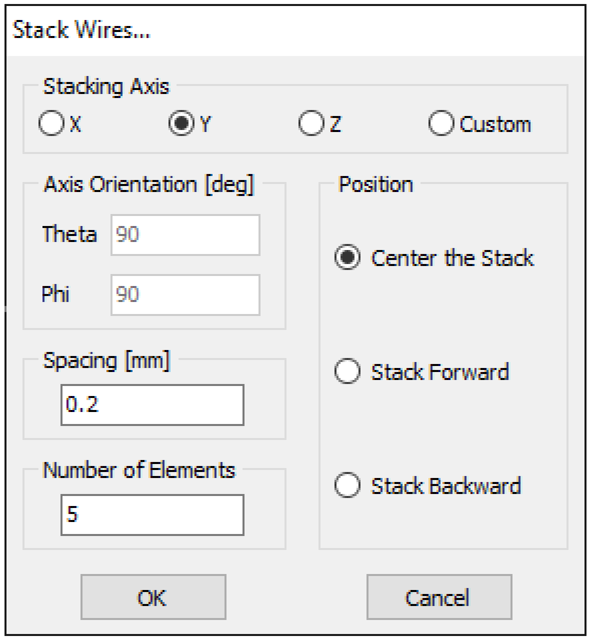


Fig. 2: Stack Wires dialog box.

Grids and Surfaces

Types of Grids and Surfaces

The **grids** are wire frameworks with holes on the surface they depict, whereas the **surfaces** represent solid metal sheets without holes. The wires of a grid do not overlap but are connected to each other. Wires used in grids or surfaces can be **straight** or **curved**.

AN-SOF offers various types of grids and surfaces, each with its unique geometric parameters and attributes that can be configured in dedicated Draw dialog boxes.

To access these options, navigate to **Draw > Wire Grid / Solid Surface** in the main menu, where you will find the following choices:

- **Patch:** Opens the Draw dialog box for creating a rectangular patch parallel to the xy-plane.
- **Plate:** Opens the Draw dialog box for creating a plate or bilinear surface.
- **Disc:** Opens the Draw dialog box for creating a disc.
- **Flat Ring:** Opens the Draw dialog box for creating a flat ring, which is a disc with a hole at its center.
- **Cone:** Opens the Draw dialog box for creating a cone.
- **Truncated Cone:** Opens the Draw dialog box for creating a truncated cone.
- **Cylinder:** Opens the Draw dialog box for creating a cylinder.
- **Sphere:** Opens the Draw dialog box for creating a sphere.
- **Paraboloid:** Opens the Draw dialog box for creating a paraboloid.

> **Tip**
>
> Go to **View > Drawing Panel** in the main menu to quickly access the wire grids and solid surfaces.

Grid/Surface Attributes

The **Attributes** page is part of the **Draw dialog box** for various wire grids and solid surface types. As shown in Fig. 1, this example illustrates the Attributes page for the Plate, but note that all grids and surfaces share the same Attributes page.

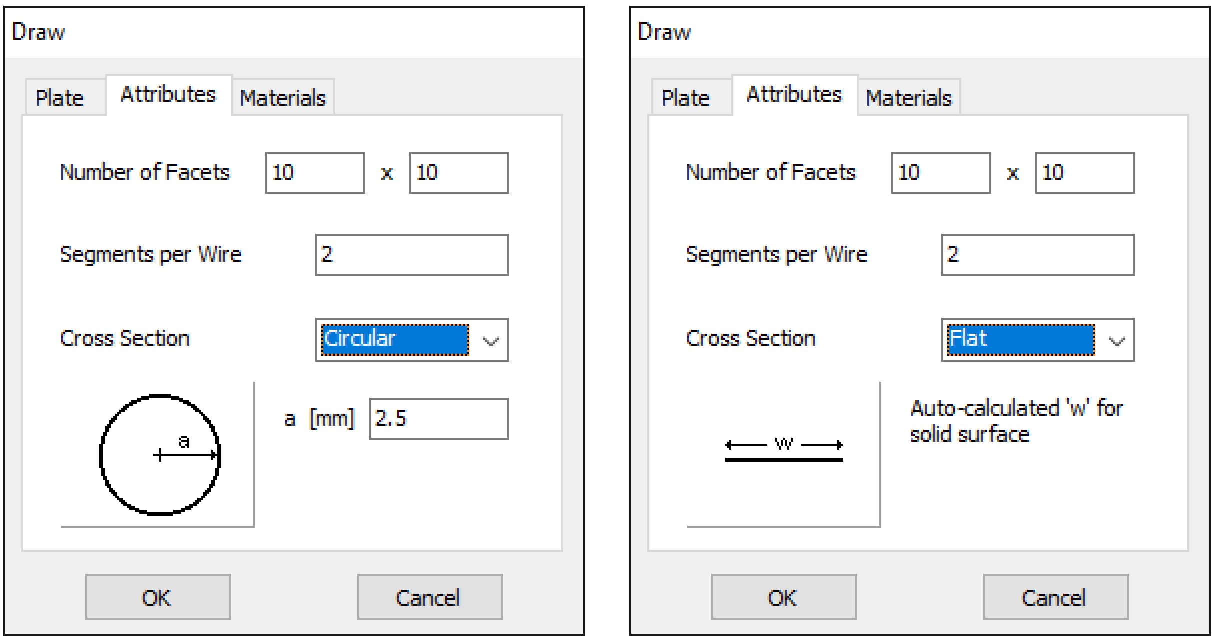


Fig. 1: Attributes page in the Plate Draw dialog box. Selection of Circular cross-section represents a wire grid, while Flat or Rectangular cross-section represents a solid surface.

To select between a **wire grid** or a **solid surface**, refer to the “Cross-Section” field below. Wire grids consist of wires with a specified circular cross-section, leaving gaps between them, while solid surfaces use flat wires whose width is automatically adjusted to cover the surface without gaps.

On the Attributes page, you can set the following parameters:

Number of Facets

Each grid or surface consists of a specific **number of facets**. For instance, **the plate shown here** has a 10×10 arrangement of facets, while **the disc here** has 6×12 facets. Each facet is a quadrilateral formed by **four wires**, with each wire divided into **segments**.

For **solid surfaces**, the wires are essentially **flat strips** that cover the **entire surface**. In the AN-SOF workspace, only the **strip axes are displayed**. During the simulation process, an unknown current is determined for each wire segment.

You have the flexibility to individually edit any curved or straight wire that comprises a grid or surface. Refer to **Modifying a Wire** for details on editing individual wires. If you need to make mass edits to the wires that make up a grid or surface, please refer to **Modifying a Grid/Surface**.

In the case of a **Patch**, setting the number of facets to **0x0** results in an automatic calculation. The calculation considers **10 segments per wavelength** along each side of the patch, with the wavelength corresponding to the highest frequency defined.

Segments per Wire

This parameter determines the **number of segments** for each wire within the grid/surface. If "Segments per Wire" is set to zero, each wire will be automatically divided into segments, with the calculation based on a default value of **10 segments per wavelength**.

Please note that the **Patch** type does not offer the option to specify "Segments per Wire" since its facets are composed of one-segment wires and the number of facets can be automatically computed by setting 0x0 facets.

Cross-Section

To define a **wire grid**, choose a **Circular** cross-section and set the **radius** of the wires comprising the grid, as shown in Fig. 1 on the left. Wire grids cannot have infinitesimally thin wires, so the cross-section radius "a" must be greater than zero.

To define a **solid surface**, select either the **Flat** or **Rectangular** cross-section for the wires that constitute the surface, as shown in Fig. 1 on the right. These wires are essentially **flat strips** that completely cover the surface. With the 'Rectangular' cross-section option, you can specify the **thickness** of the solid surface.

Modifying a Grid/Surface

A **grid** or **surface** can be modified using the procedure described in **Modifying a Group of Wires**. To select multiple wires, wire grids, or solid surfaces, click on the **Selection Box** button on the main toolbar. Left-click on the workspace, drag the mouse to create a selection box, and all wires within it will be highlighted in light blue, as shown in Fig. 1.

To apply modifications to the selected wires, go to **Edit > Modify** (you can also use the shortcut **Ctrl + M**), or use the **Modify** button on the toolbar. This command becomes active when you have a group of wires, a wire grid, or a solid surface selected. For details on the dialog window that allows you to modify selected wires, please refer to **Modifying a Group of Wires**.

If you need to perform actions such as **moving**, **rotating**, **scaling**, **copying**, or **stacking** wire grids and solid surfaces, please consult **Moving, Rotating and Scaling Wires** and **Copying and Stacking Wires** for more information.

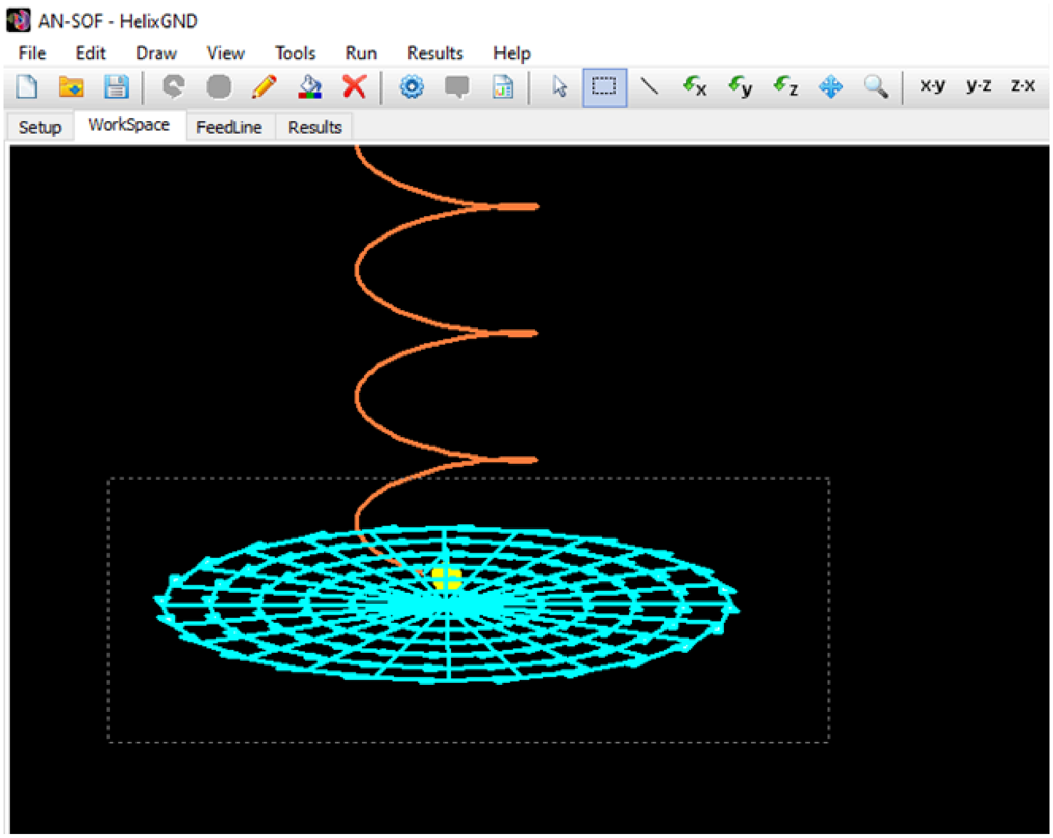


Fig. 1: A wire grid selected by the Selection Box.

Deleting a Grid/Surface

Click on the **Selection Box** button in the main toolbar. By left-clicking on the workspace and dragging a box with the mouse, you can select a wire grid or a solid surface, as explained in **Modifying a Grid/Surface** or **Modifying a Group of Wires**. All wires inside the selection box will be highlighted in light blue.

Go to Edit > **Delete (Ctrl + Del)** in the main menu to delete the selected grid or surface. There is also a button on the toolbar with the **Delete** command. This command is enabled when a group of wires, a wire grid, or a solid surface is selected.

Grid/Surface Color

Click on the **Selection Box** button in the main toolbar. By left-clicking on the workspace and dragging a box with the mouse, you can select a wire grid or a solid surface, as explained in **Modifying a Grid/Surface** or **Modifying a Group of Wires**. All wires inside the selection box will be highlighted in light blue.

Go to Edit > **Wire Color** in the main menu to change the color of the selected grid or surface. A dialog window will be opened where a color can be chosen. There is also a button on the toolbar with the **Wire Color** command. This command is enabled when a group of wires, a wire grid, or a solid surface is selected.

Patch

A **Patch** in AN-SOF represents a **solid rectangular conductive surface** lying on the xy-plane or a plane parallel to it (z = constant). This structure consists of wires with a flat or rectangular cross-section that cover the entire surface of the patch.

You can use this command to model **patch antennas**, where the patch is a **solid rectangular metal sheet**. To do this, you must choose the **Substrate** option as the ground plane by navigating to the **Setup tab > Environment panel > Ground Plane box**.

If you need to model a solid rectangular surface or a rectangular wire grid in free space or above a real ground plane, use the **Plate** command instead of Patch.

To access the Patch command, go to **Draw > Wire Grid / Solid Surface > Patch** in the main menu. The displayed dialog box consists of three pages: **Patch**, **Attributes**, and **Materials**, detailed in Fig. 1.

The Patch page

On the **Patch** page, you can configure the geometric parameters for the Patch. To define the Patch, specify the coordinates of two opposite corner points in a plane z = constant, as illustrated in Fig. 2.

Once you've configured the geometric parameters on the Patch page, you can proceed to the **Attributes** page, where you can specify the **number of facets** for the Patch. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

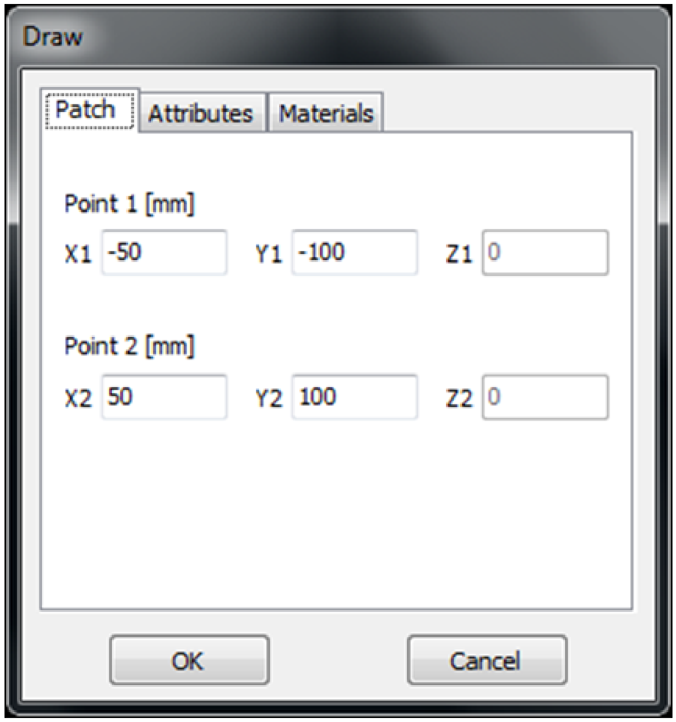


Fig. 1: Patch page of the Draw dialog box.

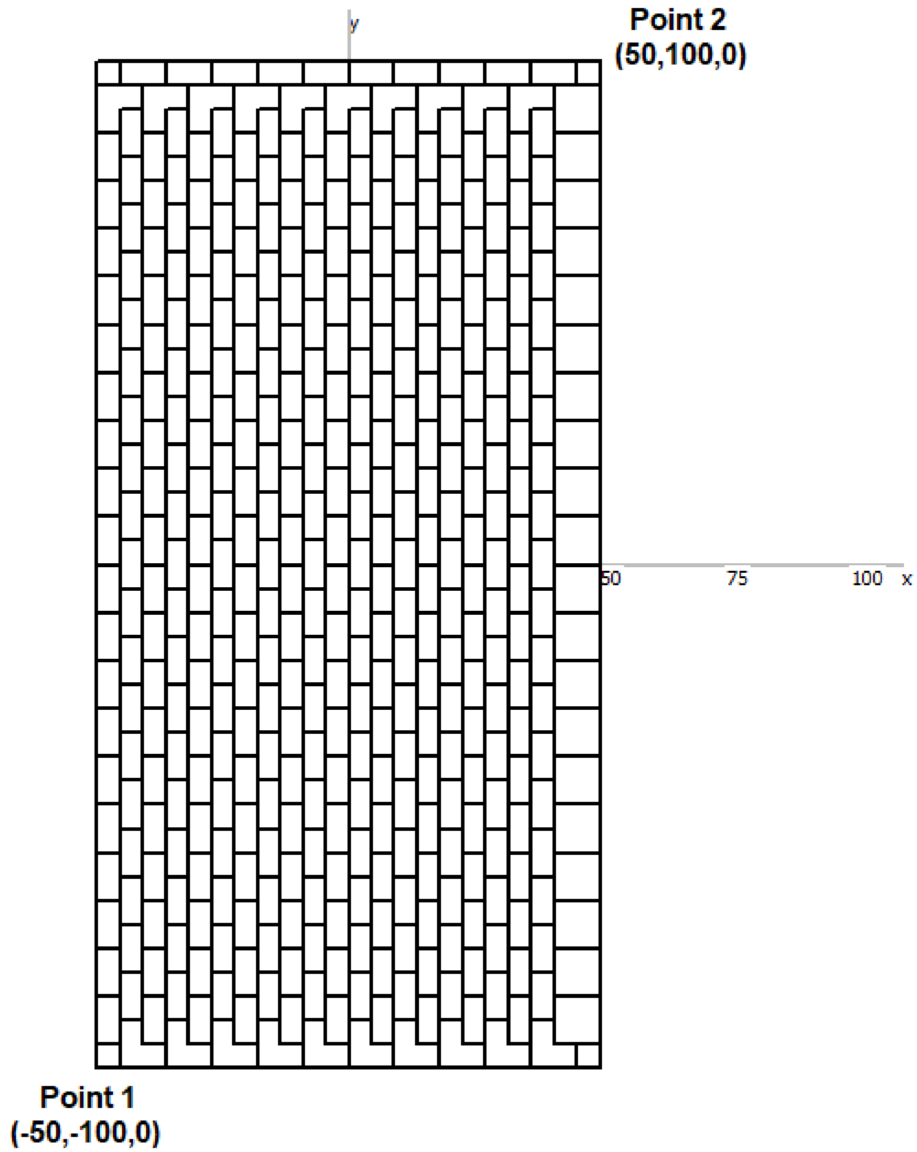


Fig. 2: A Patch drawn using the input data of Fig. 1.

Plate

The **Plate** command refers to a plate or bilinear surface.

To access the Plate command, go to **Draw > Wire Grid / Solid Surface > Plate** in the main menu. The dialog box for the Plate command contains three pages: **Plate**, **Attributes**, and **Materials**, detailed in Fig. 1.

The Plate page

In the **Plate** page, you can set the geometrical parameters for the Plate. The Plate is defined by specifying the coordinates of its four corner points. In general, a plate or bilinear surface is a non-planar quadrilateral, uniquely defined by its four vertices, as shown in Fig. 2. In some cases, the bilinear surface degenerates into a flat quadrilateral.

After setting the geometrical parameters on the Plate page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Plate and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

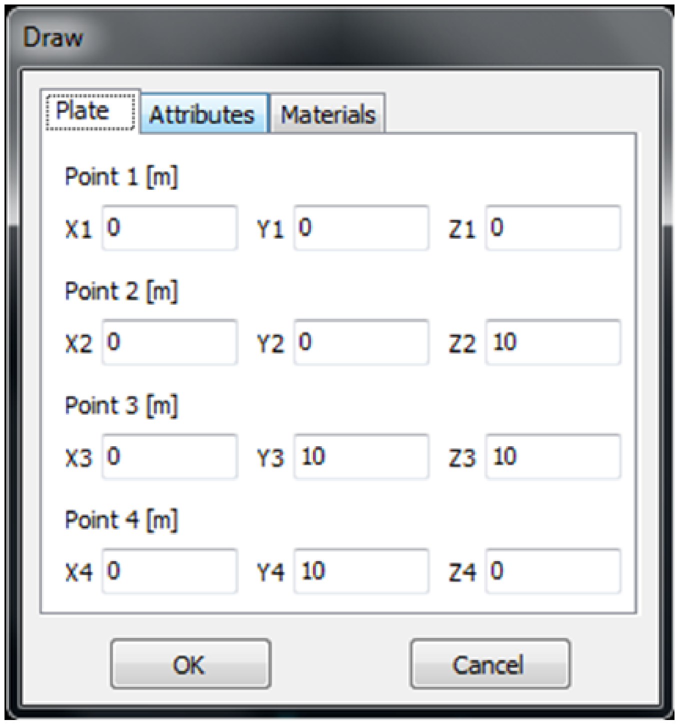


Fig. 1: Plate page of the Draw dialog box.

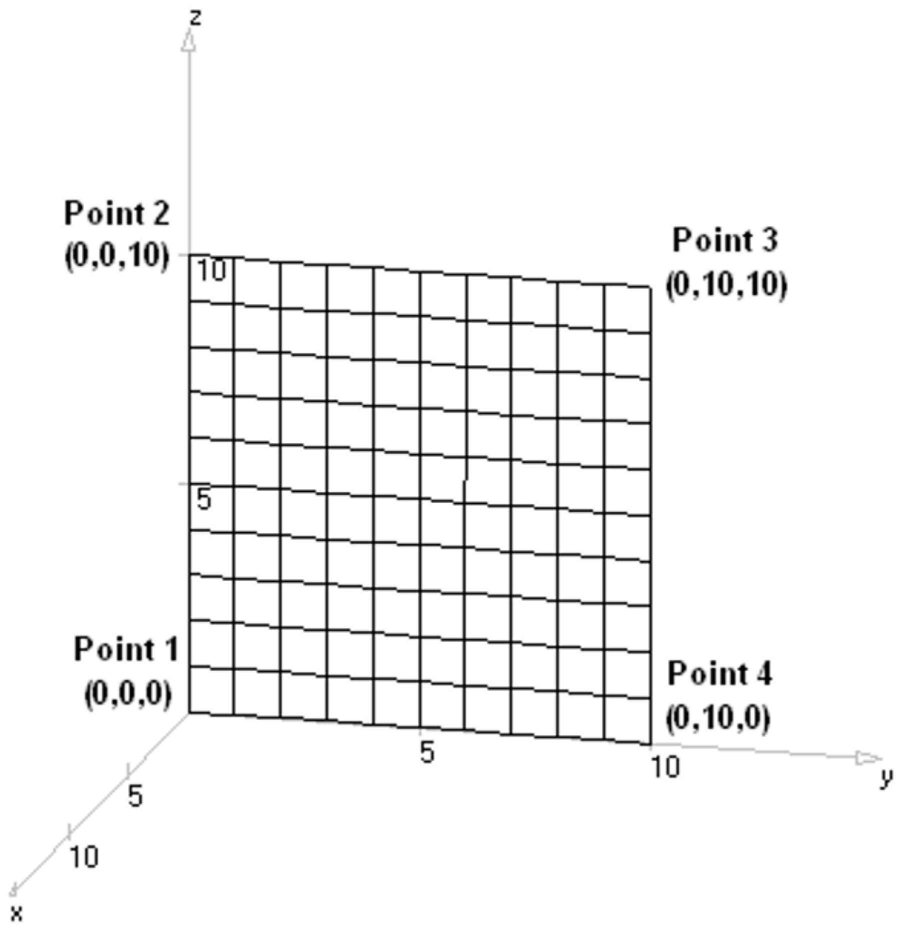


Fig. 2: A Plate drawn using the input data of Fig. 1.

Disc

The **Disc** command is used to create a disc or circular surface.

To access this command, go to **Draw > Wire Grid / Solid Surface > Disc** in the main menu. This action will open the Draw dialog box for the Disc. The dialog box consists of three pages: **Disc**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Disc page

In the **Disc** page, you can configure the geometrical parameters for the Disc. Here, you'll find a combo-box offering two options: **Curved segments** and **Straight segments**. Select **Curved segments** for an exact representation of the disc's curvature. The **Straight segments** option provides an approximation using linear wires.

The Disc is defined by specifying the **Center coordinates**, **Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define a planar disc surface, as illustrated in Fig. 2.

After setting the geometrical parameters on the Disc page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Disc and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

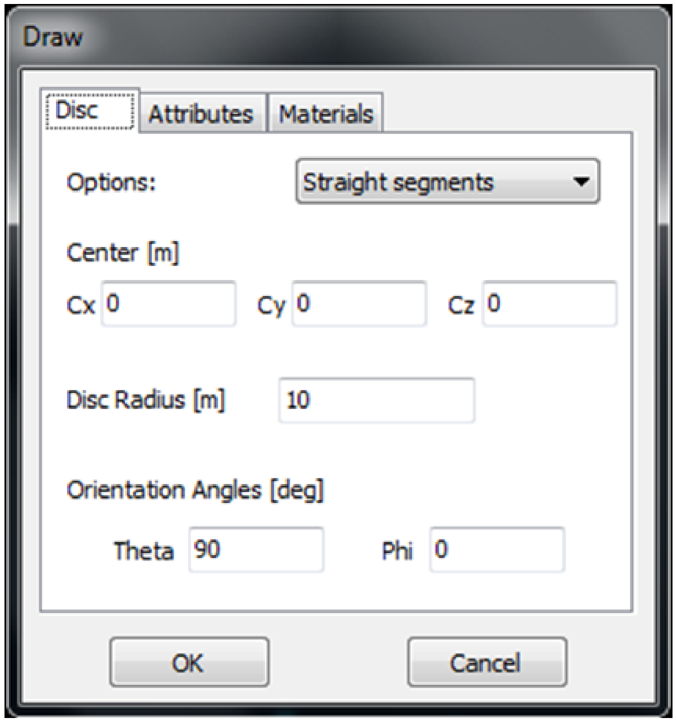


Fig. 1: Disc page of the Draw dialog box.

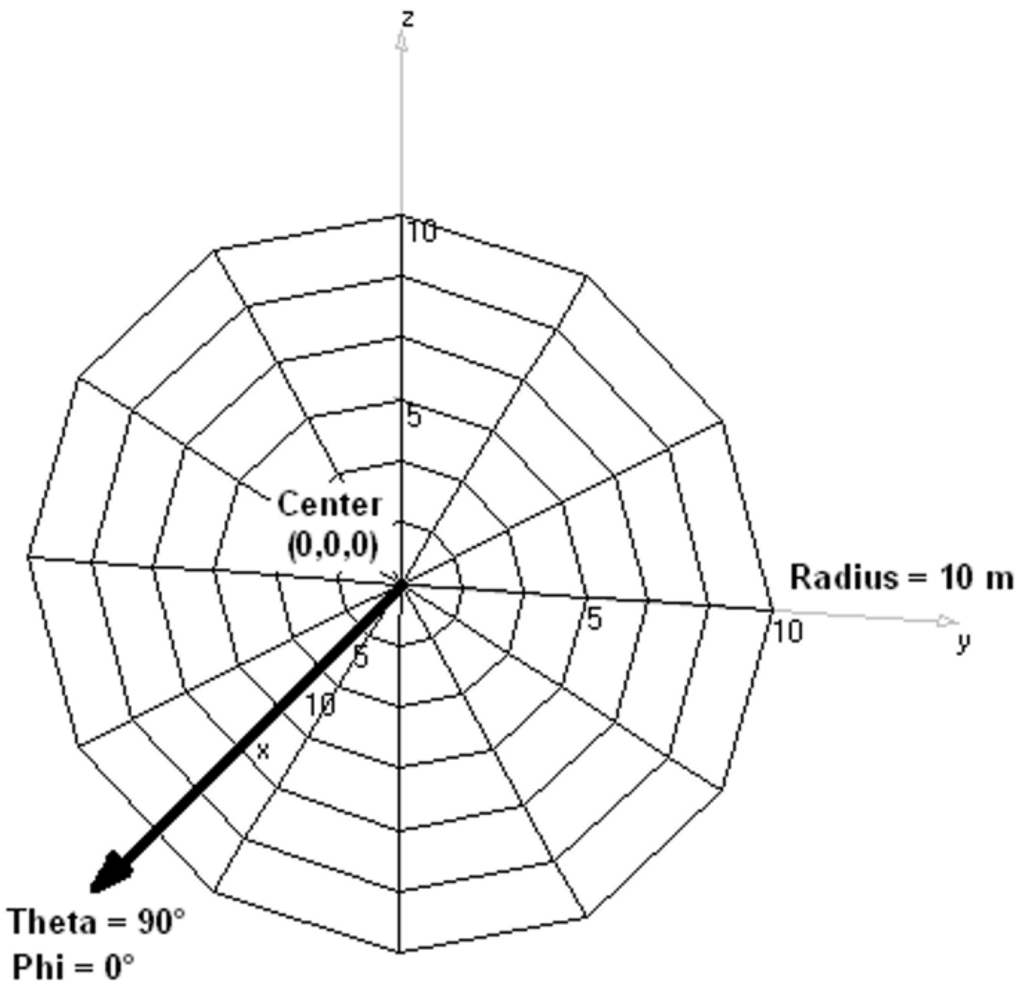


Fig. 2: A Disc drawn using the input data of Fig. 1.

Flat Ring

The **Flat Ring** command creates a disc with a hole at its center.

To access this command, go to **Draw > Wire Grid / Solid Surface > Flat Ring** in the main menu. This action opens the Draw dialog box for the Flat Ring. The dialog box comprises three pages: **Flat Ring**, **Attributes**, and **Materials**, detailed in Fig. 1.

The Flat Ring page

On the **Flat Ring** page, you can specify the geometrical parameters for the Flat Ring. Here, you'll find a combo-box offering two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the flat ring's curvature. The **Straight segments** option provides an approximation using linear wires.

The Flat Ring is defined by providing the **Center coordinates**, **Inner Radius** (hole radius), **Outer Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define a planar flat ring surface, as illustrated in Fig. 2.

After setting the geometrical parameters on the Flat Ring page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Flat Ring and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

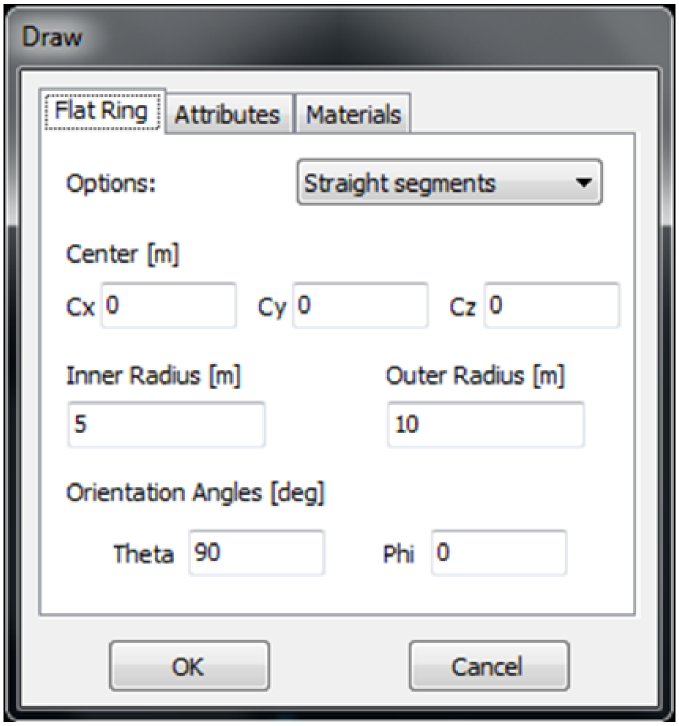


Fig. 1: Flat Ring page of the Draw dialog box.

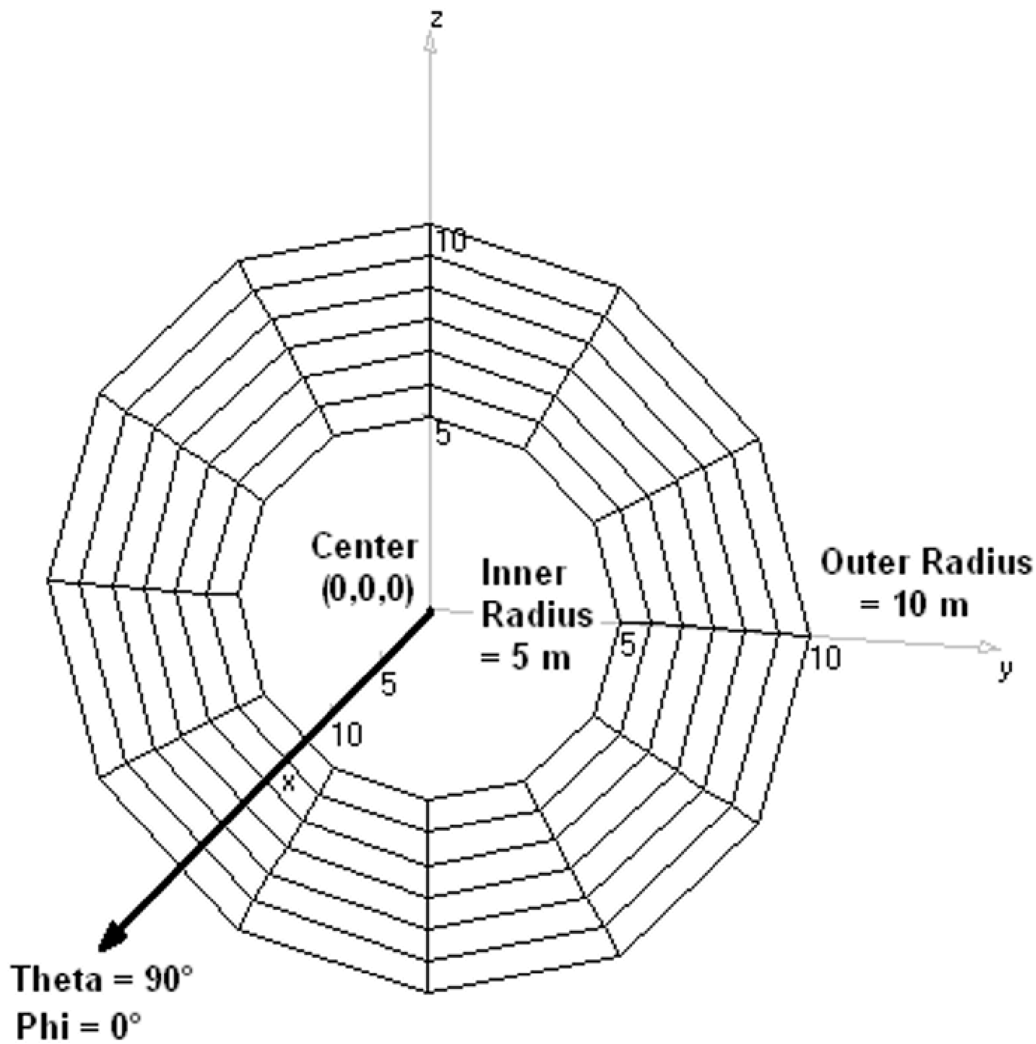


Fig. 2: A Flat Ring drawn using the input data of Fig. 1.

Cone

The **Cone** command creates a cone-shaped structure.

To access this command, go to **Draw > Wire Grid / Solid Surface > Cone** in the main menu, which opens the Draw dialog box for the Cone. The dialog box comprises three pages: **Cone**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Cone page

On the **Cone** page, you can set the geometrical parameters for the Cone. You'll find a combo-box with two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the cone's curvature, while the **Straight segments** option provides an approximation using linear wires.

The Cone is defined by specifying the **Vertex** coordinates, **Aperture Angle**, **Aperture Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define the cone's surface, as illustrated in Fig. 2.

After setting the geometrical parameters on the Cone page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Cone and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

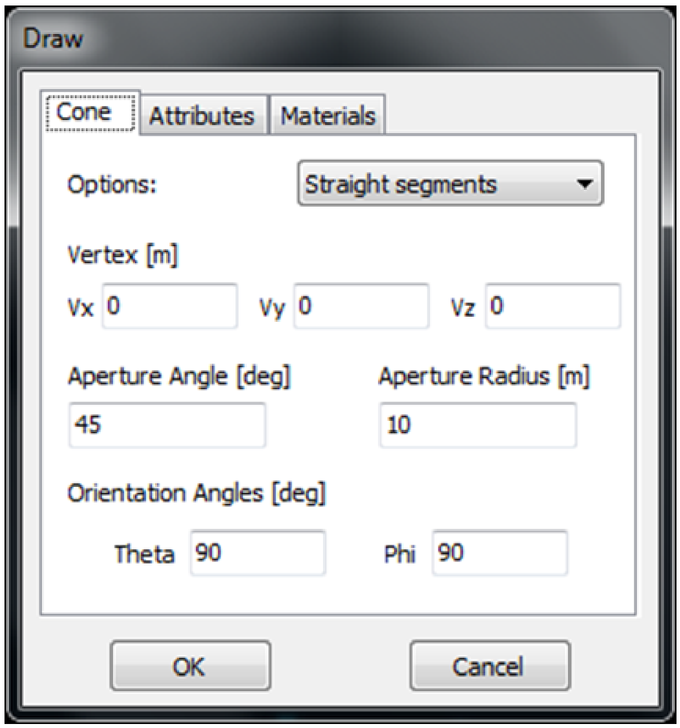


Fig. 1: Cone page of the Draw dialog box.

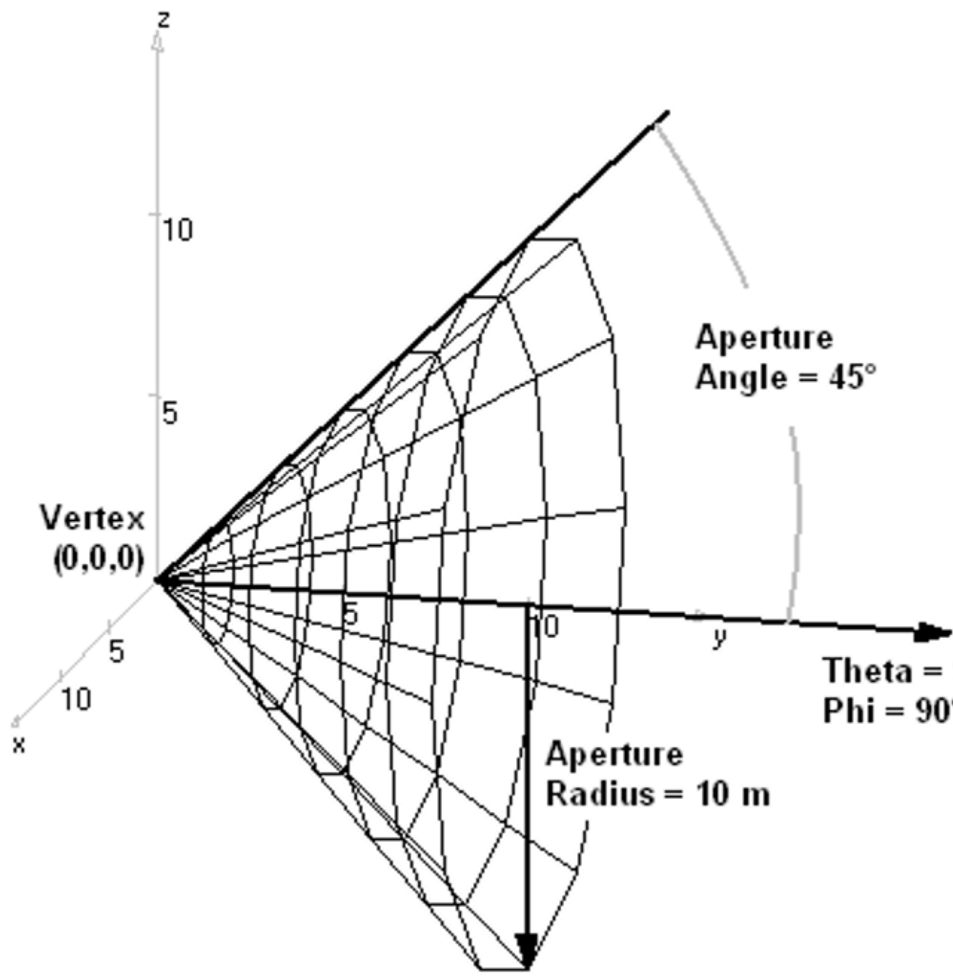


Fig. 2: A Cone drawn using the input data of Fig. 1.

Truncated Cone

The **Truncated Cone** command creates a truncated cone-shaped structure.

To access this command, go to **Draw > Wire Grid / Solid Surface > Truncated Cone** in the main menu, which opens the Draw dialog box for the Truncated Cone. The dialog box comprises three pages: **Truncated Cone**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Truncated Cone page

On the **Truncated Cone** page, you can set the geometrical parameters for the Truncated Cone. You'll find a combo-box with two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the truncated cone's curvature, while the **Straight segments** option provides an approximation using linear wires.

The Truncated Cone is defined by specifying the **Base Point** coordinates, **Base Radius**, **Top Radius**, **Aperture** angle, and orientation angles, **Theta** and **Phi**. These parameters uniquely define the truncated cone's surface, as illustrated in Fig. 2. Depending on its parameters, a truncated cone can take on various shapes, including a cylinder, a cone, a disc, or a flat ring.

After setting the geometrical parameters on the Truncated Cone page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Truncated Cone and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

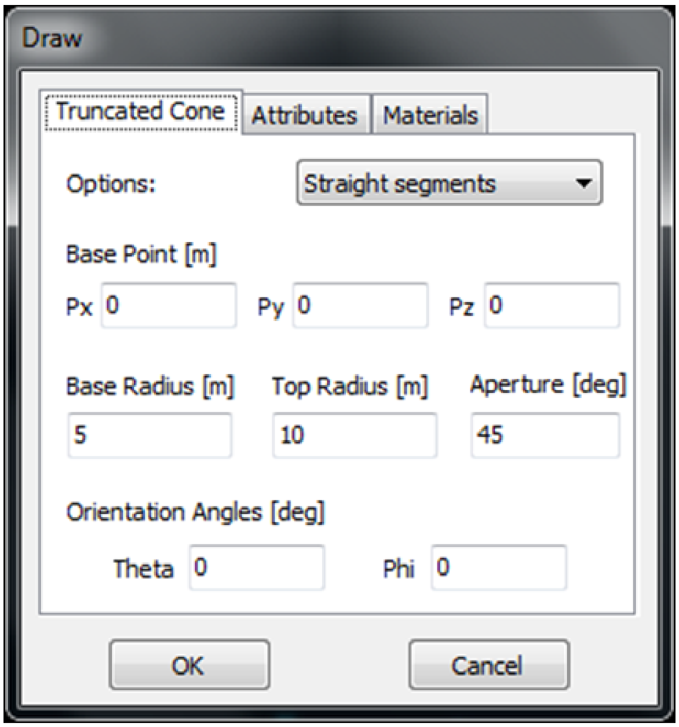


Fig. 1: Truncated Cone page of the Draw dialog box.

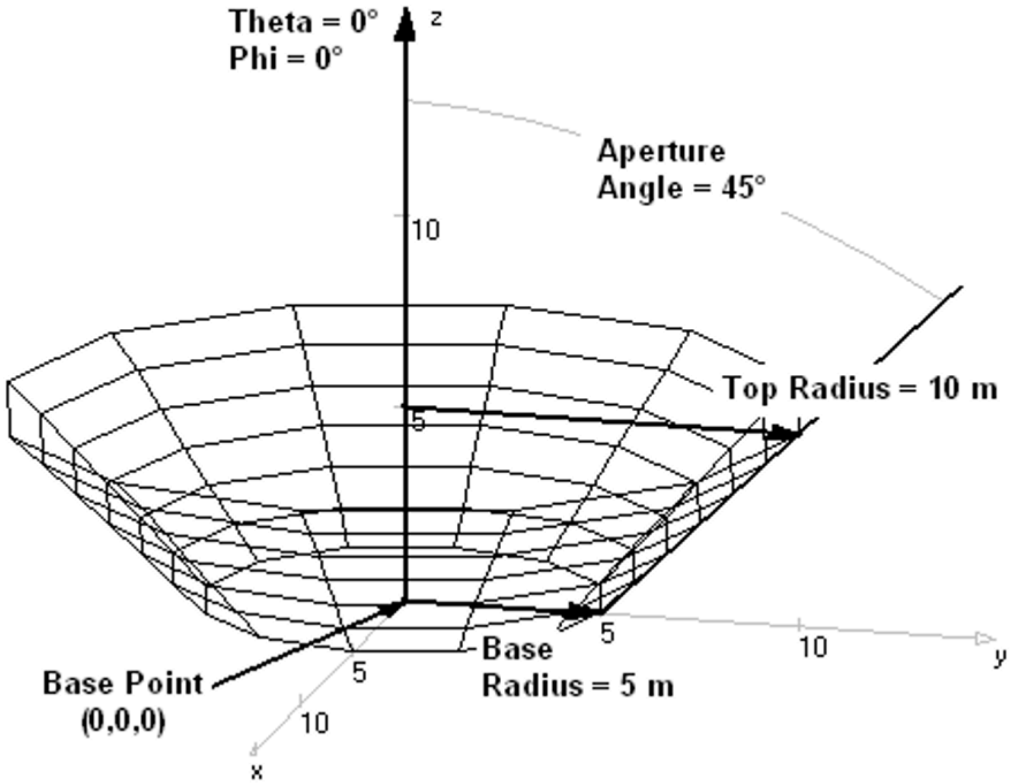


Fig. 2: A Truncated Cone drawn using the input data of Fig. 1.

Cylinder

The **Cylinder** command creates a cylindrical structure.

To access this command, go to **Draw > Wire Grid / Solid Surface > Cylinder** in the main menu, which opens the Draw dialog box for the Cylinder. The dialog box comprises three pages: **Cylinder**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Cylinder page

On the **Cylinder** page, you can set the geometrical parameters for the Cylinder. You'll find a combo-box with two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the cylinder's curvature, while the **Straight segments** option provides an approximation using linear wires.

The Cylinder is defined by specifying the **Base Point** coordinates, **Length**, **Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define the cylinder's surface, as illustrated in Fig. 2.

After setting the geometrical parameters on the Cylinder page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Cylinder and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

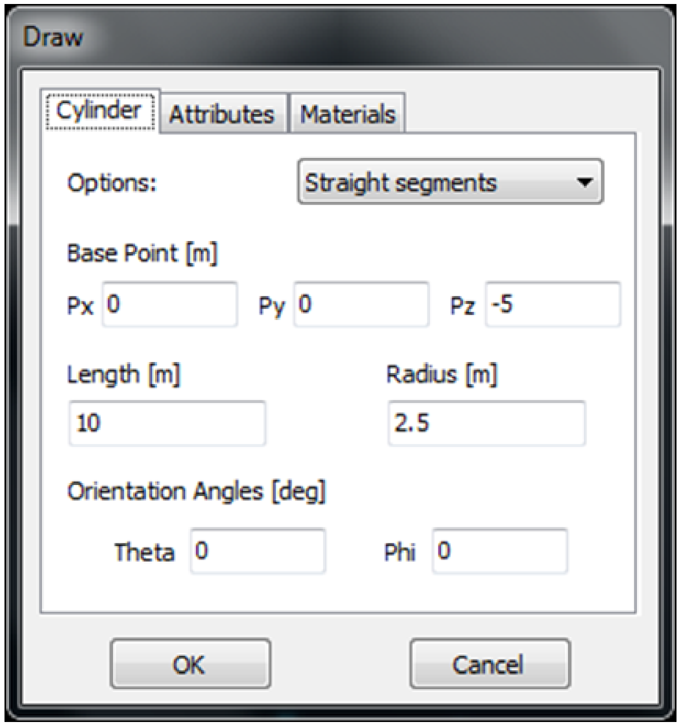


Fig. 1: Cylinder page of the Draw dialog box.

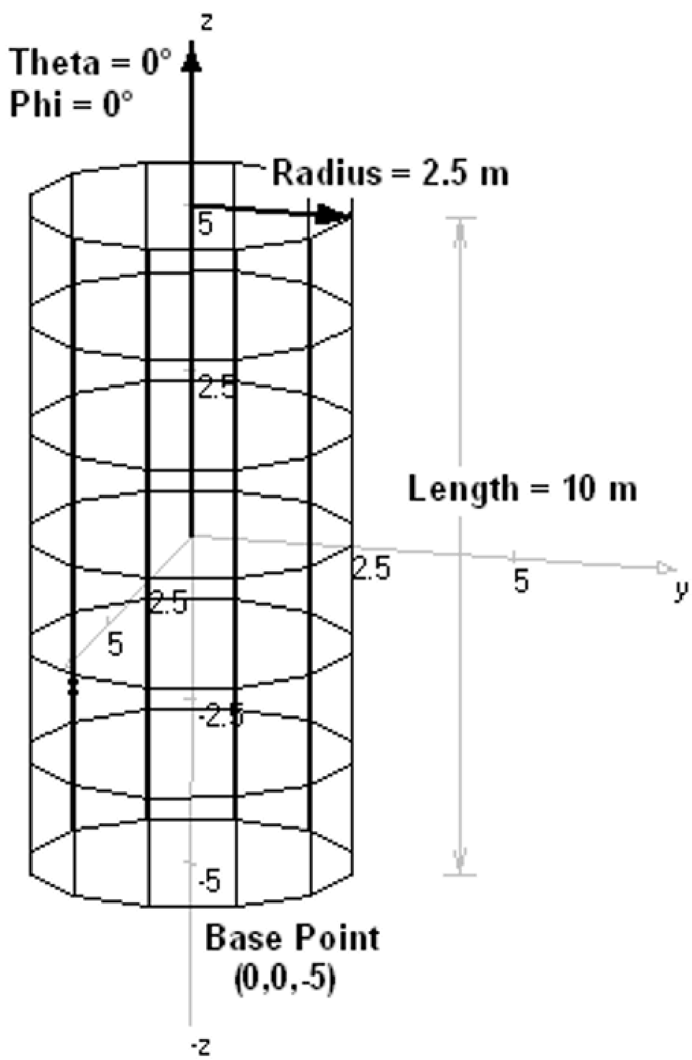


Fig. 2: A Cylinder drawn using the input data of Fig. 1.

Sphere

The **Sphere** command creates a spherical structure.

To access this command, go to **Draw > Wire Grid / Solid Surface > Sphere** in the main menu, which opens the Draw dialog box for the Sphere. The dialog box comprises three pages: **Sphere**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Sphere page

On the **Sphere** page, you can set the geometrical parameters for the Sphere. You'll find a combo-box with two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the sphere's curvature, while the **Straight segments** option provides an approximation using linear wires.

The Sphere is defined by specifying the **Center** coordinates, **Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define the sphere's surface, as shown in Fig. 2.

After setting the geometrical parameters on the Sphere page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Sphere and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

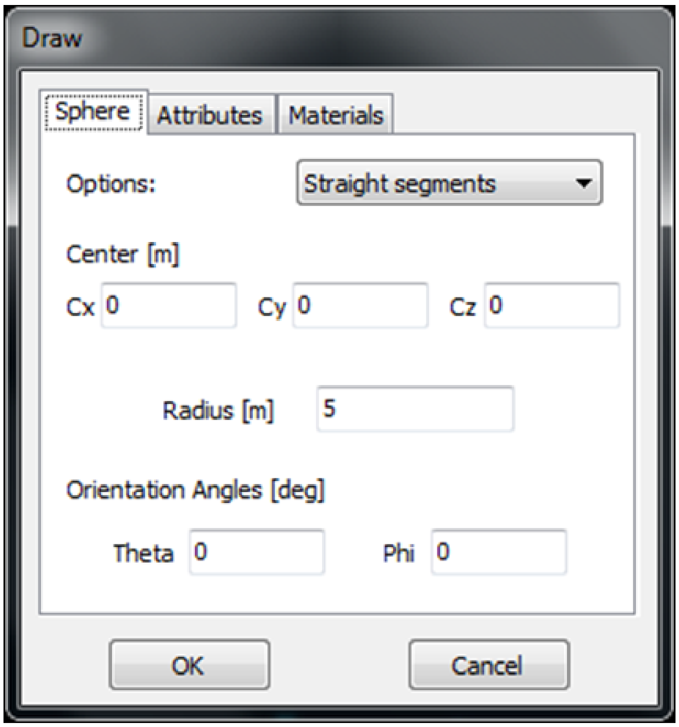


Fig. 1: Sphere page of the Draw dialog box.

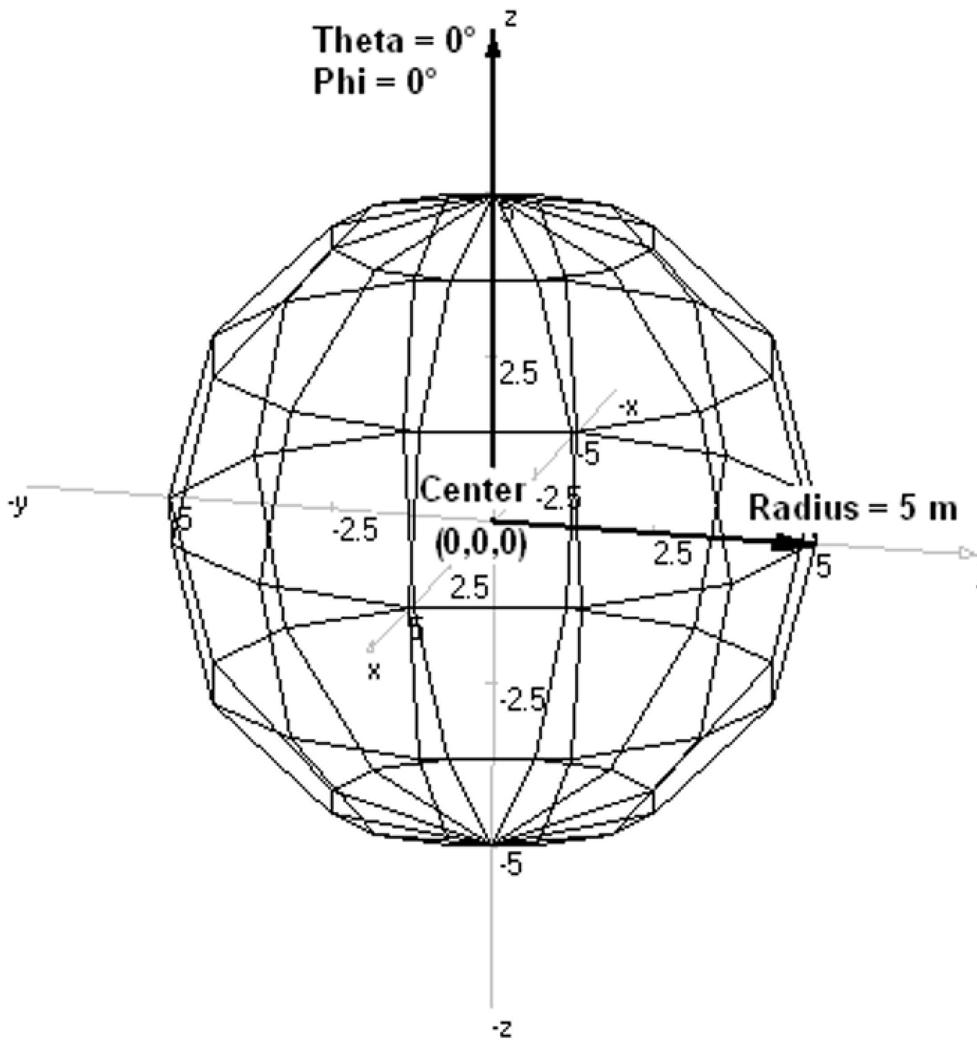


Fig. 2: A Sphere drawn using the input data of Fig. 1.

Paraboloid

The **Paraboloid** command creates a paraboloidal structure.

To access this command, go to **Draw > Wire Grid / Solid Surface > Paraboloid** in the main menu, which opens the Draw dialog box for the Paraboloid. The dialog box comprises three pages: **Paraboloid**, **Attributes**, and **Materials**, as detailed in Fig. 1.

The Paraboloid page

On the **Paraboloid** page, you can set the geometrical parameters for the Paraboloid. You'll find a combo-box with two options: **Curved segments** and **Straight segments**. Choose **Curved segments** for an exact representation of the paraboloid's curvature, while the **Straight segments** option provides an approximation using linear wires.

The Paraboloid is defined by specifying the **Vertex** coordinates, **Focal Distance**, **Aperture Radius**, and orientation angles, **Theta** and **Phi**. These parameters uniquely define the paraboloid's curved surface, as shown in Fig. 2.

After setting the geometrical parameters on the Paraboloid page, you can move on to the **Attributes** page. Here, you can specify the **number of facets** for the Paraboloid and choose whether it should be a **wire grid** or a **solid surface**. See **Grid/Surface Attributes** for additional parameters in the **Attributes** page and **Wire Materials** for parameters in the **Materials** page.

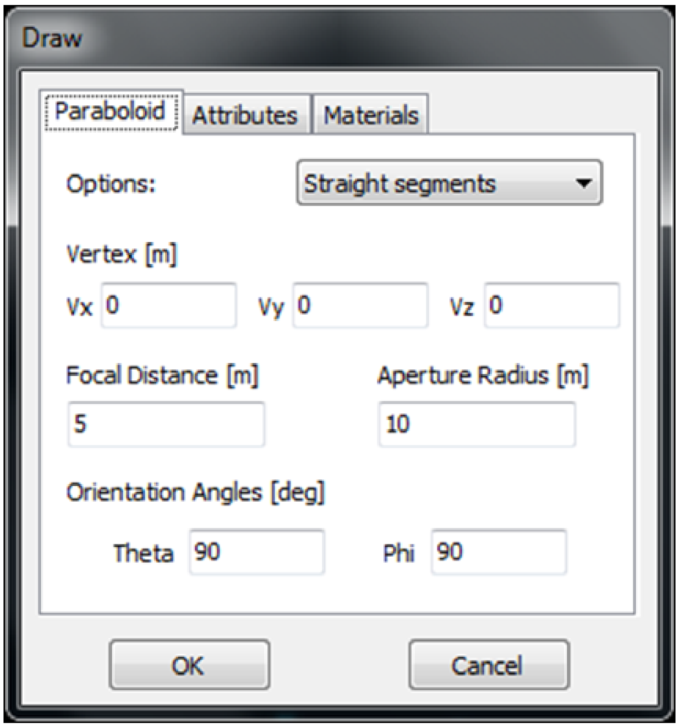


Fig. 1: Paraboloid page of the Draw dialog box.

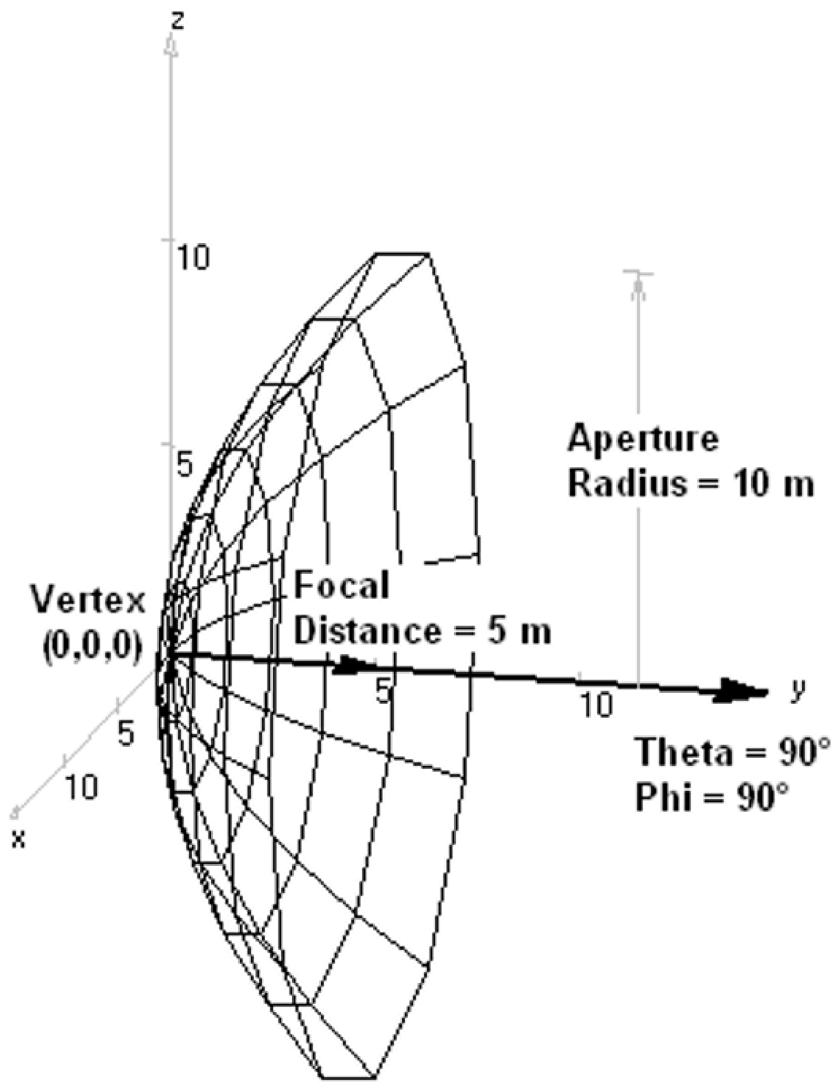


Fig. 2: A Paraboloid drawn using the input data of Fig. 1.

Sources and Loads

Types of Excitations and Loads

Discrete Sources, Incident Field, and Loads

A structure can be excited by discrete sources or an incident field. Refer to **Excitation by an Incident Field >** for the second case. Discrete sources can be located on any wire segment and there can be more than one source, as many as there are segments.

A source is used to model the feed point of a transmitting antenna or generator in an electrical circuit. There are two types of sources:

- **Voltage sources**

- **Current sources**

Current sources can be used to model impressed currents.

For each source, its amplitude and phase must be set. Internal impedances can also be added to model imperfect sources, which can be series RL, series RC, or R+jX impedances.

Lumped loads can also be added to any wire segment, representing resistors, inductors, capacitors, or fixed impedances. There are three types of loads:

- **Series RL impedance (inductive)**
- **Series RC impedance (capacitive)**
- **Fixed R+jX impedance (the reactance X does not scale with frequency)**

To model a pure resistor, add an inductive impedance with L = 0. The unit of inductance can be pH, nH, uH, mH or H, while that of capacitance can be pF, nF, uF, mF or F. These units can be set going to **main menu > Tools > Preferences >**.

> **Tips**
>
> Sources are displayed as a yellow circle in the workspace, while loads are displayed as a green highlighted segment. To change the default colors, go to **main menu > Tools > Preferences > Workspace tab**.
>
> **Voltage sources** have their **internal impedance in series**, so set a null impedance to model a perfect source.
>
> **Current sources** have their **internal impedance in parallel**, so set a very large impedance (1E6 Ohm) to model a nearly perfect source.

Excitation by Sources

To excite the wire structure with discrete sources, go to the **Setup tab > Excitation panel** and select the **Discrete Sources** option, Fig. 1.

If the **Set Input Power** option is checked, you can set the total input power to the structure. In this case, the amplitudes of the voltage and current sources will be adjusted to achieve the specified input power.

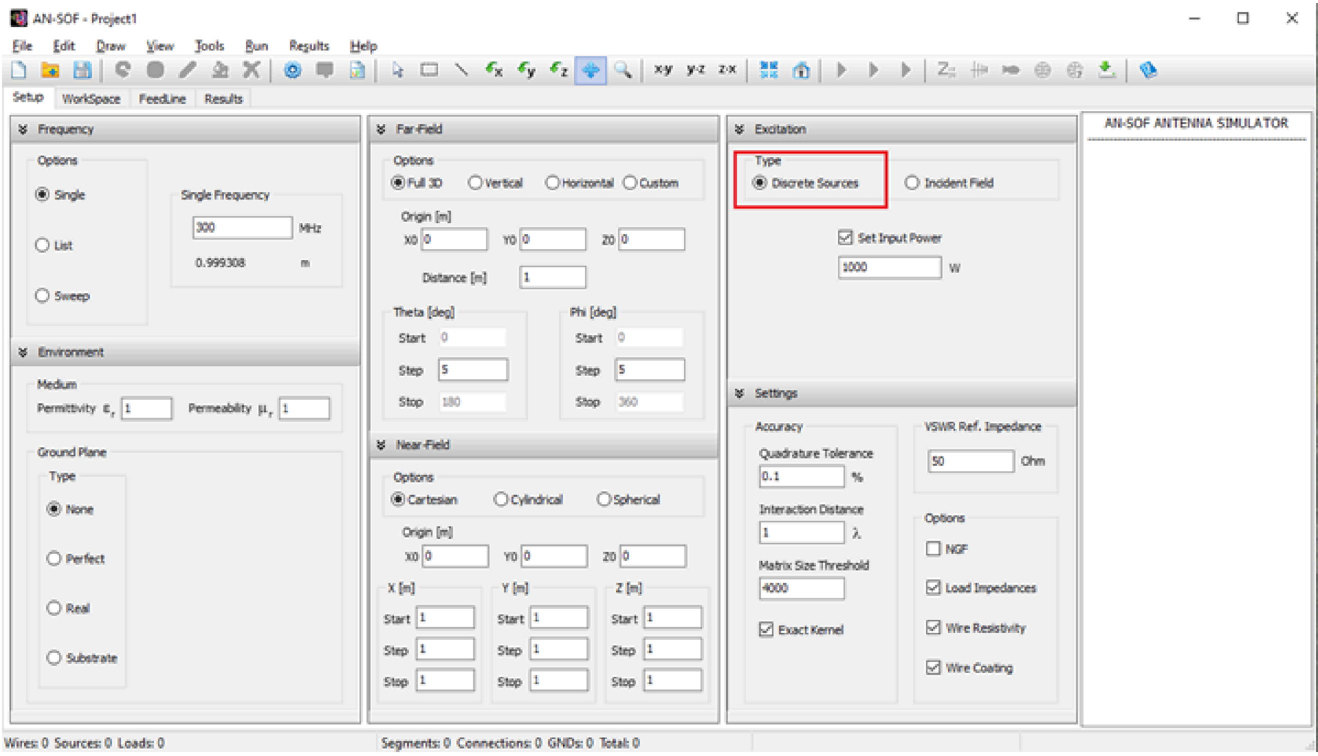


Fig. 1: Discrete Sources option in the Excitation panel of the Setup tabsheet.

The Source/Load/TL Toolbar

The **Source/Load/TL** toolbar is used to connect a **source**, **load**, or **transmission line** to a selected wire **segment**. This toolbar also provides the means to edit sources, loads, and transmission lines.

When you right-click on any part of a wire, a **pop-up menu** will appear. Click on the **Source/Load/TL (Ctrl + Ins)** command from the pop-up menu to open the Source/Load/TL toolbar, Fig. 1.

The **Source/Load/TL** command is also accessible from the main toolbar or by going to the main menu and selecting **Edit > Source/Load/TL (Ctrl + Ins)**. To enable this command, first click on the **Select Wire** button (the arrow icon) on the main toolbar and then left-click on the wire where you want to place the source or load.

The Source/Load toolbar has the following components:

Fig. 1: Source/Load/TL toolbar.

The Slider

Each position of the slider corresponds to the position of a segment in the selected wire. So, the slider allows us to select a particular segment on the wire. At the right corner of this toolbar, the position of the selected segment is shown. The segment's position as a percentage of the wire length is also shown. It is measured from the starting point of the wire to the middle point of the selected segment and is defined as follows:

% position = 100 (position / wire length)

The 50% button

The **50% button** is used to position the slider in the middle of the wire. Discrete sources and loads are often added at the center of wires, so you can click this button to quickly select the segment at the wire's center. Please note that the wire must have an **odd number of segments** for it to have a segment at its center.

The Add Source button

Click the **Add Source** button to display a dialog box for adding a source to the selected wire segment, as shown in Fig. 2. This dialog box allows you to set the type of source, its amplitude, phase, and internal impedance.

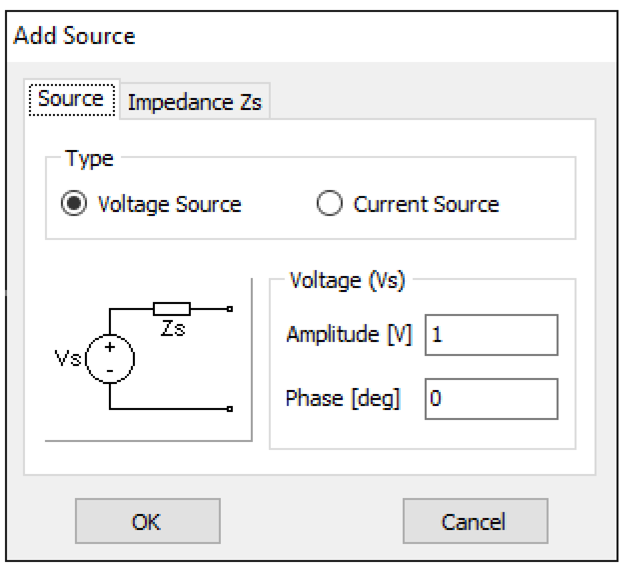


Fig. 2: Add Source dialog box.

The Add Load button

Click the **Add Load** button to display a dialog box for adding a load to the selected wire segment, as shown in Fig. 3. A load can represent either a resistor in series with an inductor (RL), a resistor in series with a capacitor (RC), or a fixed impedance (R+jX) where the reactance X does not scale with frequency.

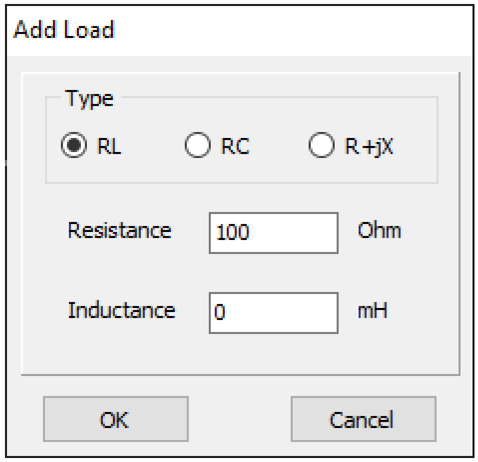


Fig. 3: Add Load dialog box.

The Transmission Lines button

Click on the **Transmission Lines** button to display a dialog box for connecting a transmission line to the selected wire segment. Refer to **Adding Transmission Lines** for further details.

The Delete button

If the selected segment has a source or a load on it, you can click the **Delete** button to remove the source or load from the segment.

The Modify button

If the selected segment has a source or a load on it, you can click the **Modify** button to open the Modify dialog box, allowing you to edit the source or load.

The Exit button

Click the **Exit** button to close the Source/Load/TL toolbar.
Adding Sources

A source can be added to a selected wire segment by means of the following steps:

1. Right click on any part of a wire to display the **pop-up menu**.
2. Choose the **Source/Load/TL** command from the pop-up menu to display the **Source/Load/TL toolbar**.
3. Move the slider to select the desired segment.
4. Click on the **Add Source** button to display the Add Source dialog box.
5. Set the type of source, its amplitude (rms value), phase and internal impedance. Then, press the OK button.
6. Click on the Exit button to close the Source/Load/TL toolbar.

Editing Sources

A source can be edited by means of the following steps:

1. Right click on any part of a wire to display the **pop-up menu**.
2. Choose the **Source/Load/TL** command from the pop-up menu to display the **Source/Load/TL toolbar**.
3. Move the slider to select the segment where the source is placed.
4. Click on the **Modify** button to display a dialog box where the source can be edited. The source can be deleted by clicking on the **Delete** button.
5. Click on the Exit button to close the Source/Load/TL toolbar.

Adding Loads

A load can be added to a selected wire segment by means of the following steps:

1. Right click on any part of a wire to display the **pop-up menu**.
2. Choose the **Source/Load/TL** command from the pop-up menu to display the **Source/Load/TL toolbar**.
3. Move the slider to select the desired segment.
4. Click on the **Add Load** button to display the Add Load dialog box.
5. Set the type of load and the values of resistance and inductance or capacitance. Then, press the OK button.
6. Click on the Exit button to close the Source/Load/TL toolbar.

Editing Loads

A load can be edited by means of the following steps:

1. Right click on any part of a wire to display the **pop-up menu**.
2. Choose the **Source/Load/TL** command from the pop-up menu to display the **Source/Load/TL toolbar**.
3. Move the slider to select the segment where the load is placed.
4. Click on the **Modify** button to display a dialog box where the load can be edited. The load can be deleted by clicking on the **Delete** button.
5. Click on the Exit button to close the Source/Load/TL toolbar.

Enabling/Disabling Loads

All the loads can be enabled or disabled at the same time. This option avoids deleting the load impedances when loads must not be considered in a simulation.

Go to **Setup tab > Settings panel >** in the main window. If the option **Load Impedances** is checked, the loads are enabled, otherwise they are disabled, Fig. 1.

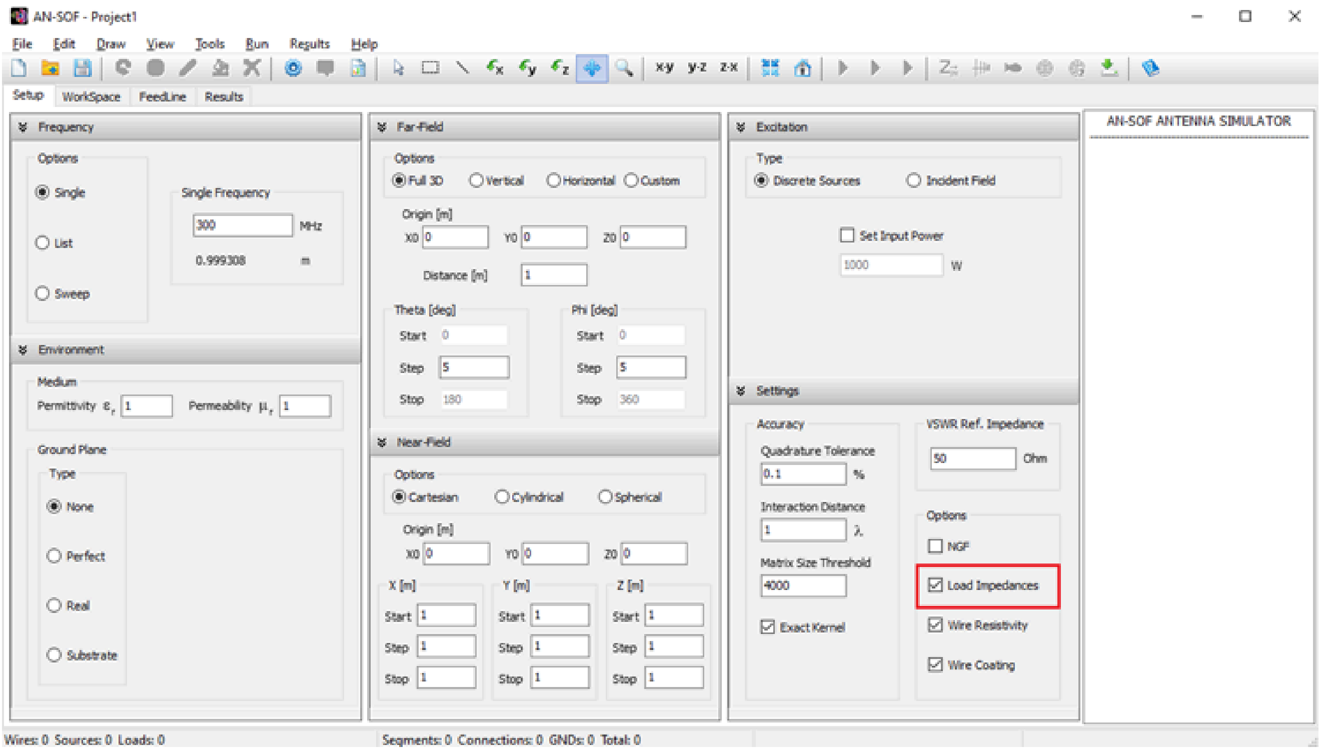


Fig. 1: Load impedances option in the Settings panel of the Setup tabsheet.

Incident Field

Excitation by an Incident Field

To choose an incident plane wave as excitation of the structure, go to the **Setup tab > Excitation panel >** and select the **Incident Field** option, Fig. 1. When this option is selected, if there are discrete sources on the structure, none will be considered in the simulation.

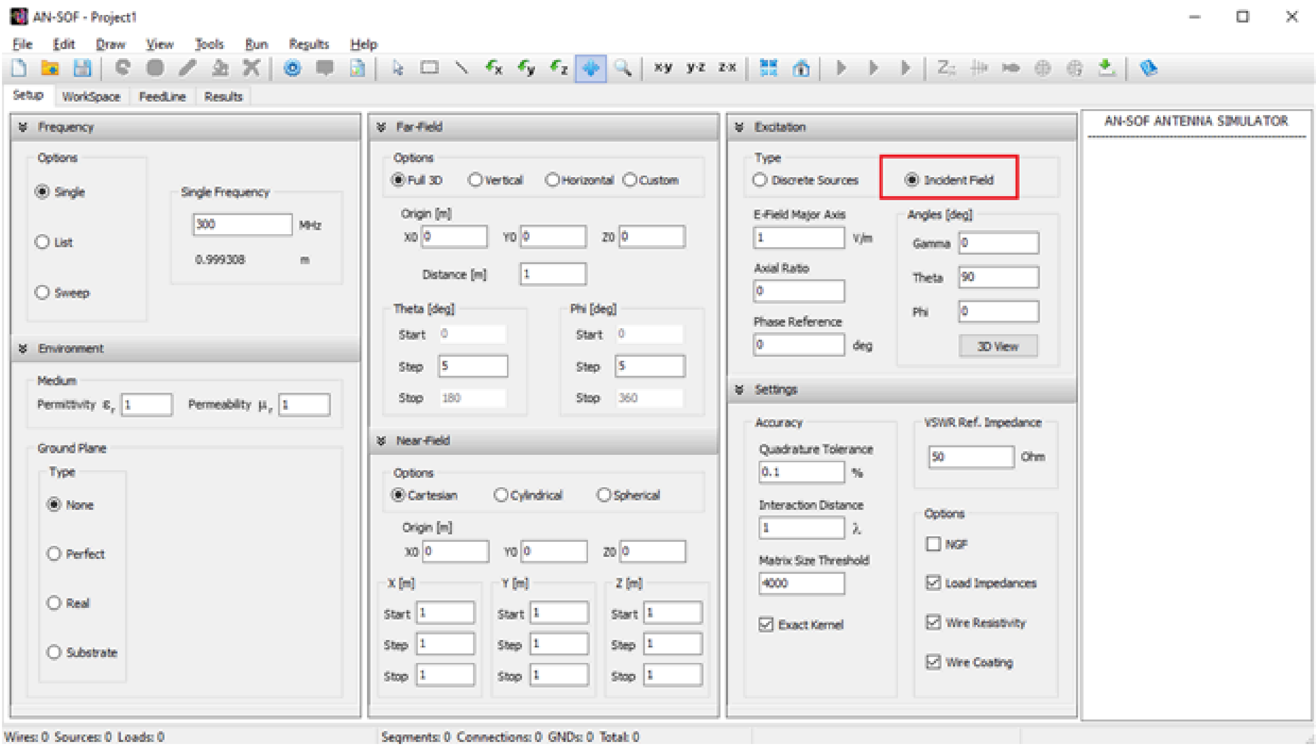


Fig. 1: Incident Field option in the Excitation panel of the Setup tabsheet.

Incident Field Parameters

The following incident field parameters can be set in the **Excitation panel >** of the Setup tabsheet after clicking on the **Incident Field** option:

E-Field Major Axis: Amplitude, in V/m (Volts rms per meter), of the linearly polarized incoming electric field. For elliptical polarization, it is the length of the major ellipse axis.

Axial Ratio: For an elliptically polarized plane wave, it is the ratio of the minor axis to the major axis of the ellipse. A positive (negative) axial ratio defines a right-handed (left-handed) ellipse. If the axial ratio is set to zero, a linearly polarized plane wave is defined.

Phase Reference: Phase, in degrees, of the incident plane wave at the origin of coordinates. It can be used to change the phase reference in the calculation. Its value only shifts all phases in the structure by the given amount.

Gamma: Polarization angle of the incident electric field in degrees. For a linearly polarized wave, Gamma is measured from the plane of incidence to the direction of the electric field vector, Fig. 1. For an elliptically polarized wave, Gamma is the angle between the plane of incidence and the major ellipse axis.

Theta: Zenith angle of the incident direction in degrees, Fig. 1.

Phi: Azimuth angle of the incident direction in degrees, Fig. 1.

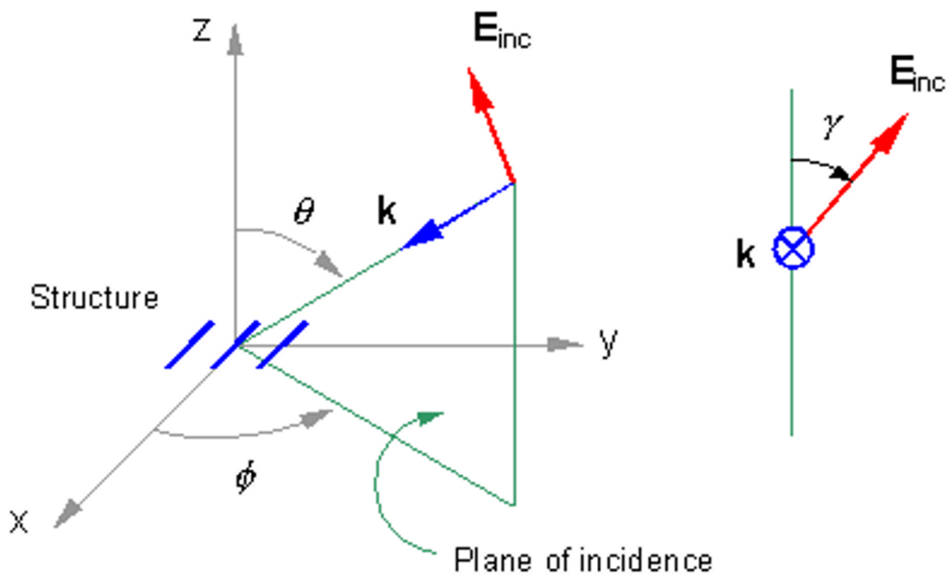


Fig. 1: Parameters of an incident field.

> **Note**
>
> When an incident plane wave is used as excitation, all discrete sources, if any, will not be considered in the simulation.

The 3D-View Interface

The 3D-View interface allows us entering the parameters of the incident field in a graphical way. Follow these steps:

1. Go to the Setup tabsheet and select the **Incident Field** option in the **Excitation panel >**.
2. Click on the **3D View** button to open the interface and display the Incident Wave dialog box, Fig. 1.
3. Set the **Gamma**, **Theta** and **Phi** angles and press ENTER. You can also use the small arrows to change these angles.
4. Close the Incident Wave dialog box. The angles that have been entered in the dialog box will appear in the Excitation panel, Fig. 2.

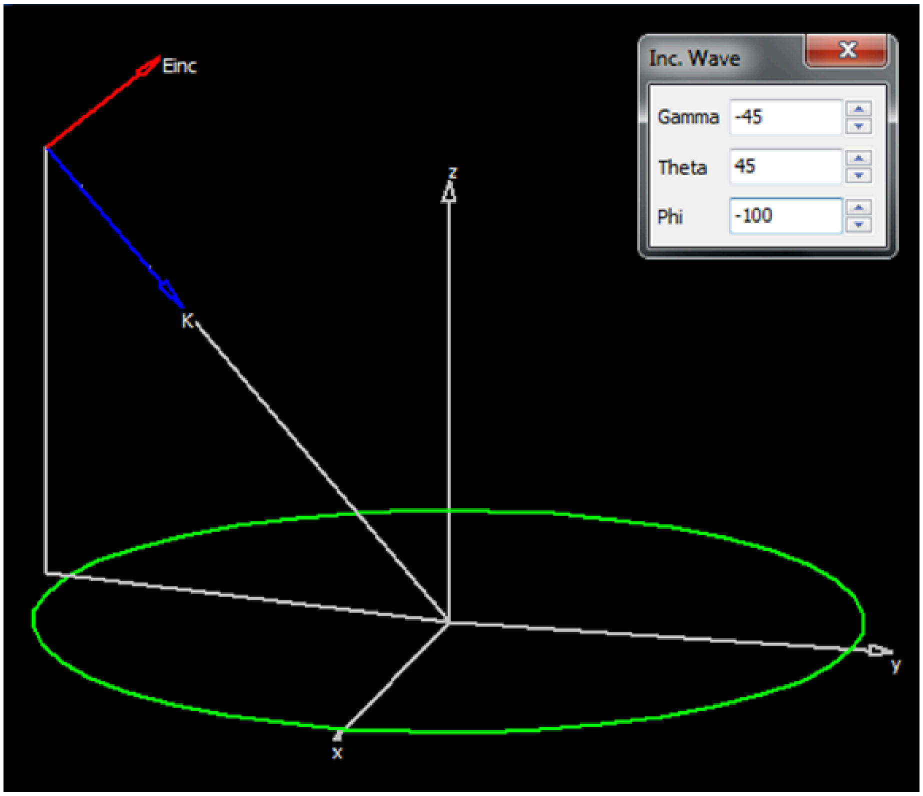


Fig. 1: 3D-View interface for the definition of the incident field. The Incident Wave dialog box is also shown. Gamma, Theta, and Phi are set to –45, 45 and –100 deg., respectively.

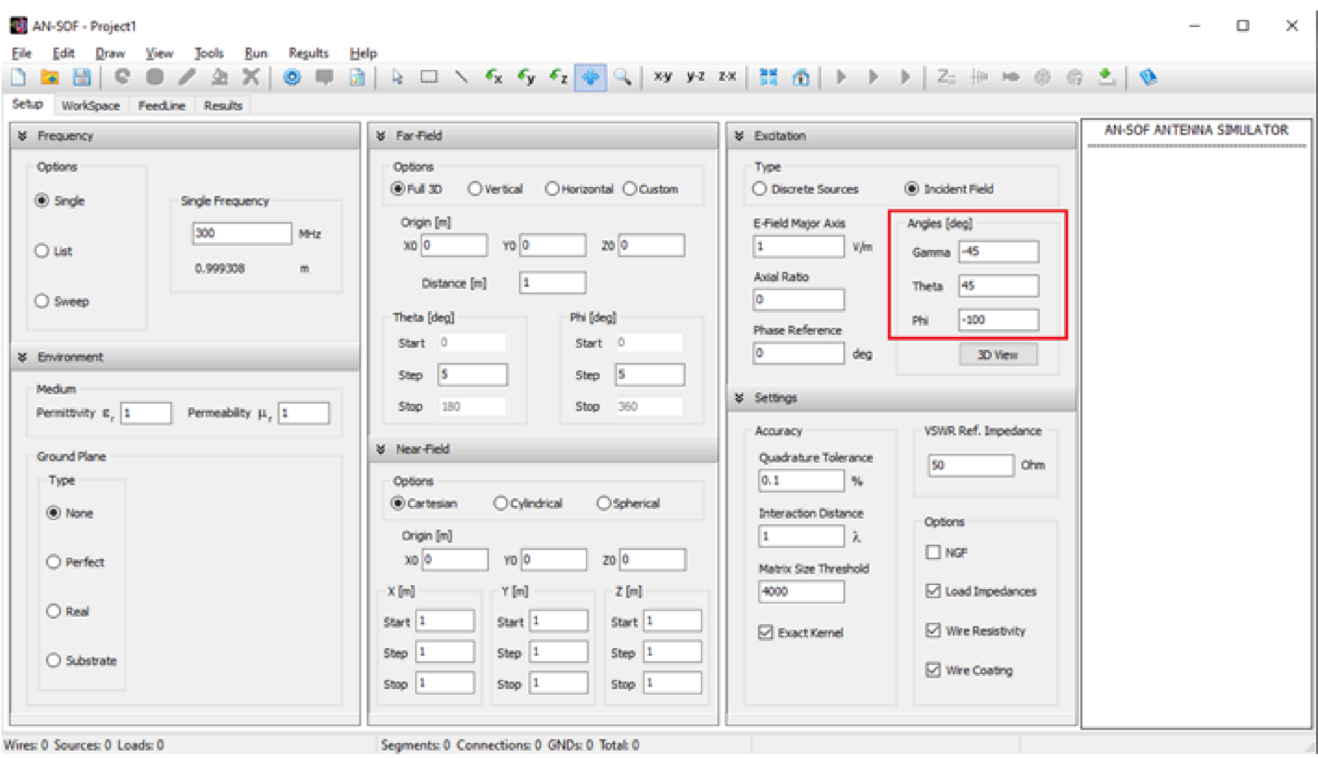


Fig. 2: The Gamma, Theta and Phi angles entered in the Incident Wave dialog box will appear in the Excitation panel of the Setup tabsheet.

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).

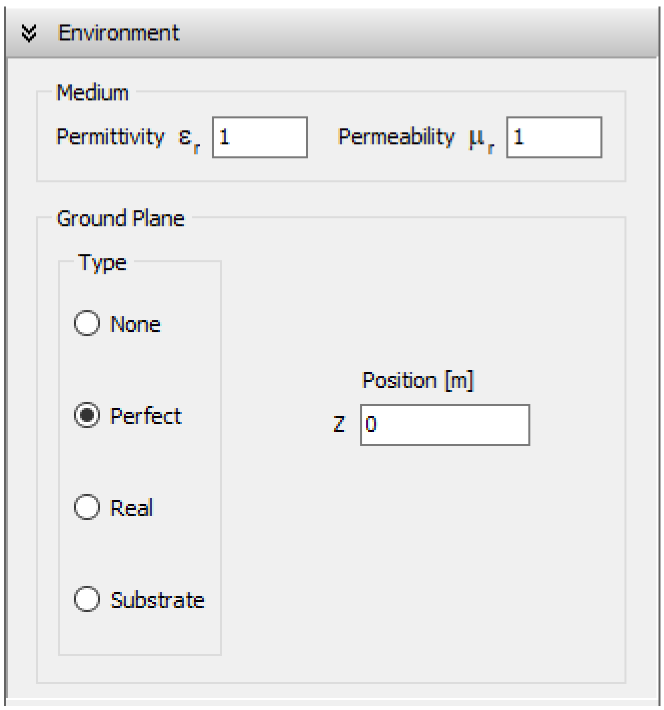


Fig. 1: Perfect option in the Ground Plane box of the Environment panel.

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.

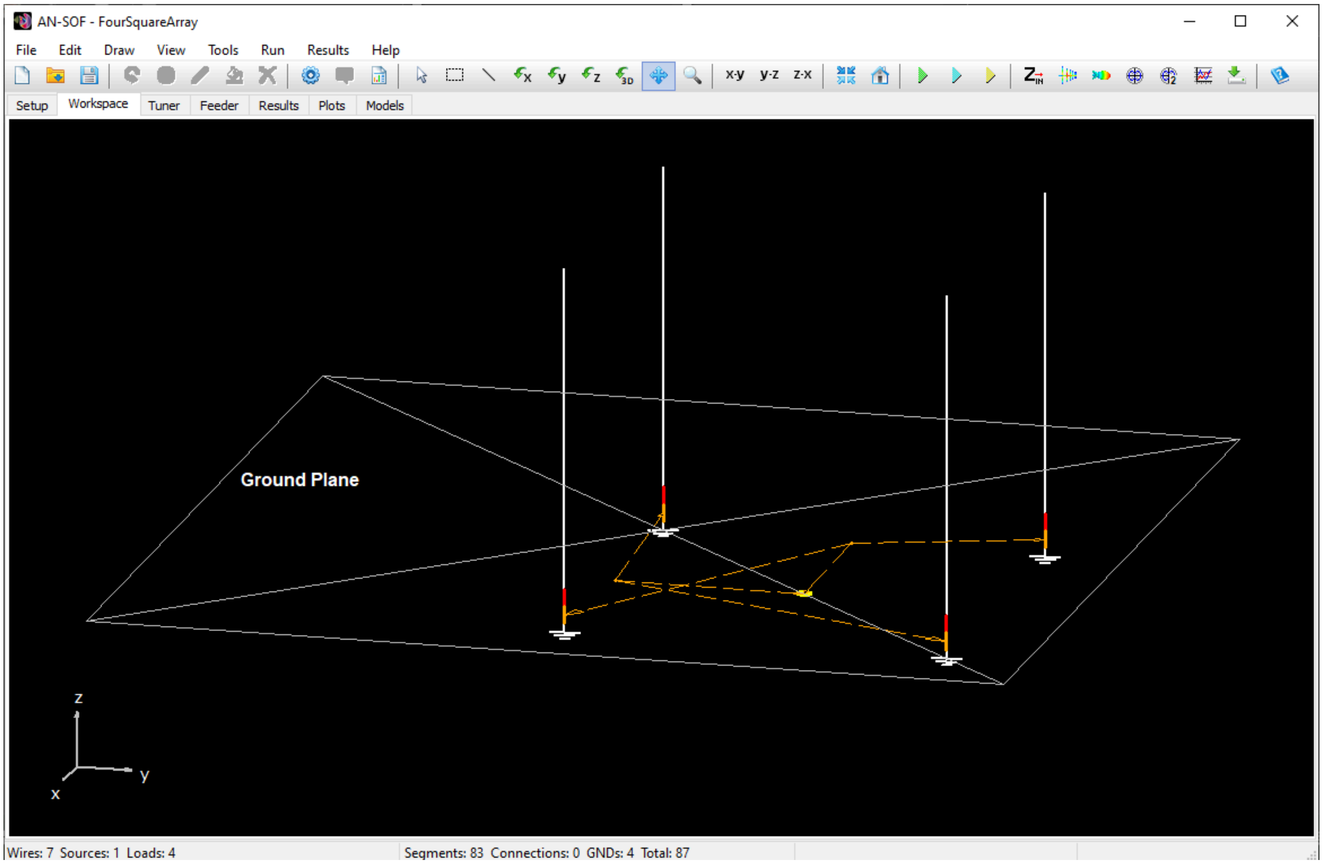


Fig. 2: Ground plane symbol in the workspace, indicating the position of the ground plane.

Adding a Real Ground Plane

A real (imperfect) 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/Asymptotic**
    - **Radial Wire Ground Screen**
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**.

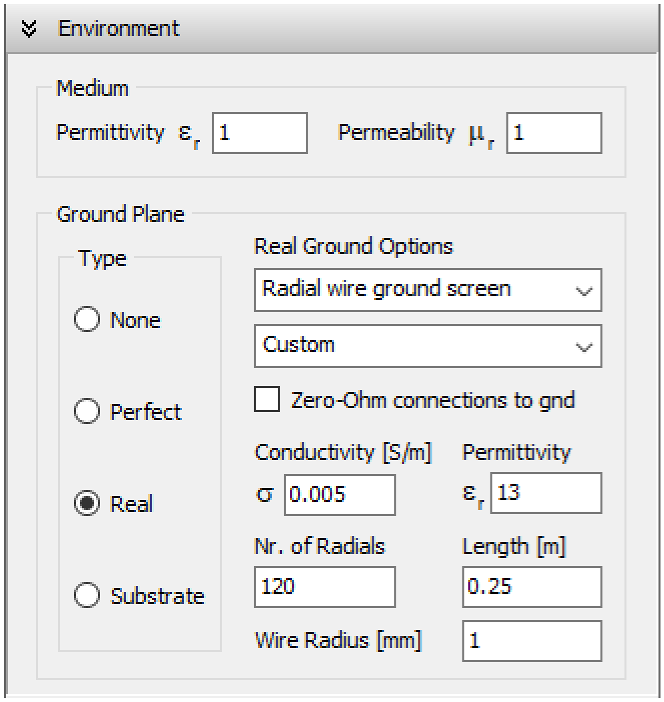


Fig. 1: Real option in the Ground Plane box of the Environment panel.

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).

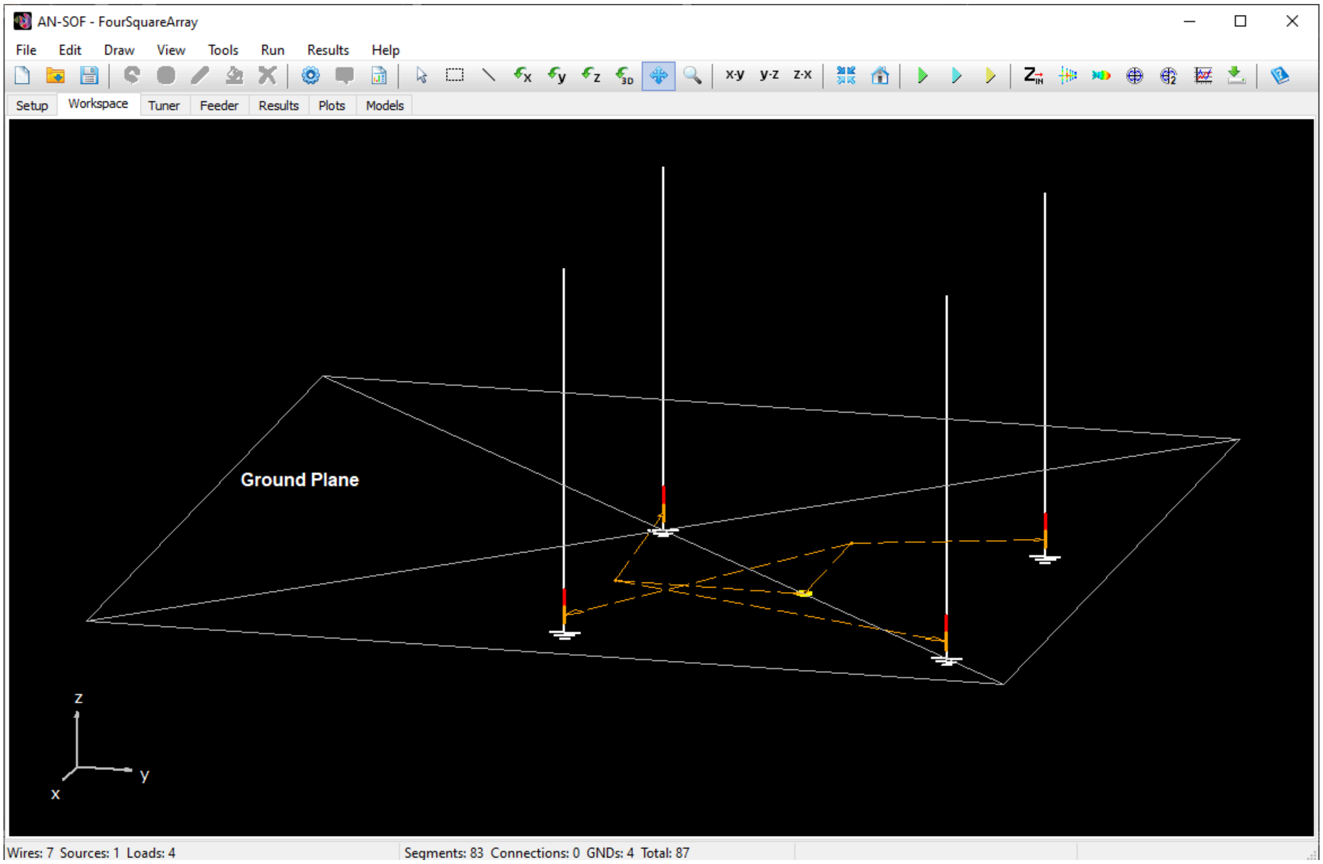


Fig. 2: Ground plane symbol in the workspace, indicating the position of the ground plane.

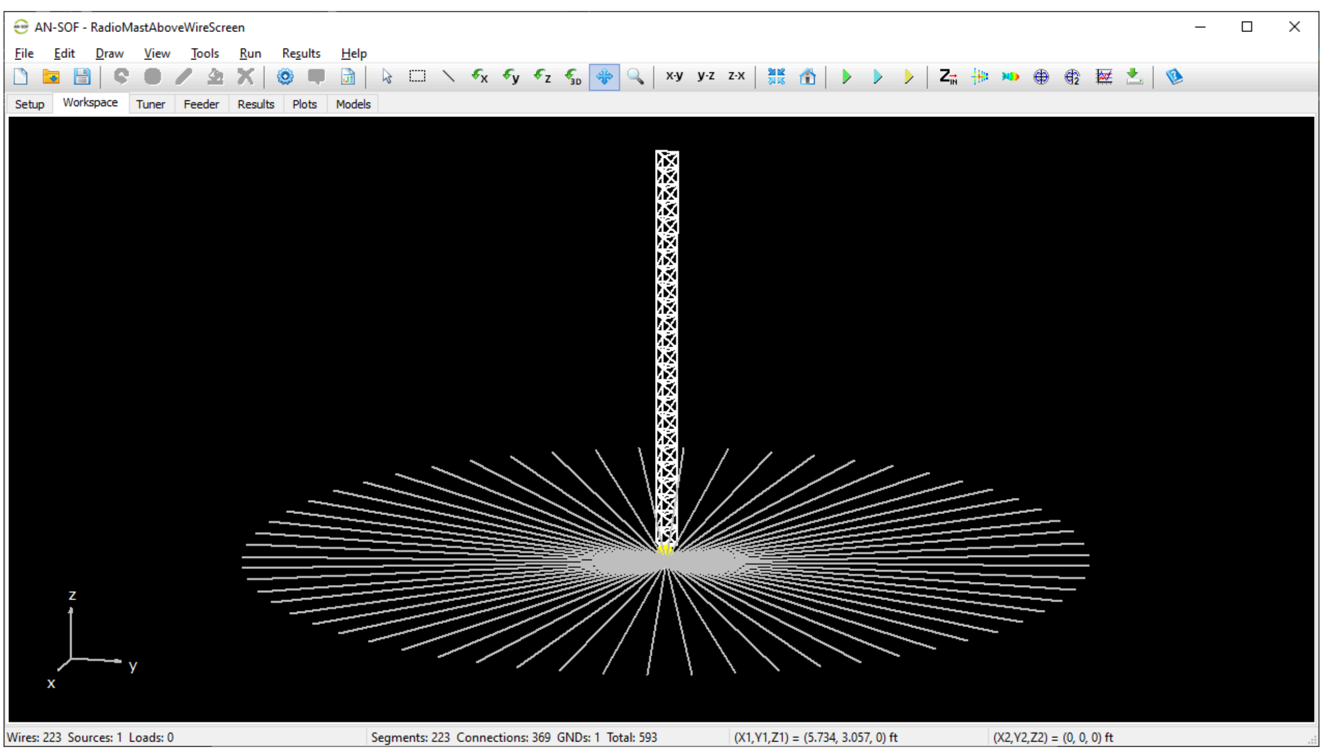


Fig. 3: Radial wire ground screen in the workspace, showing the position of the ground plane and ground screen.

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.

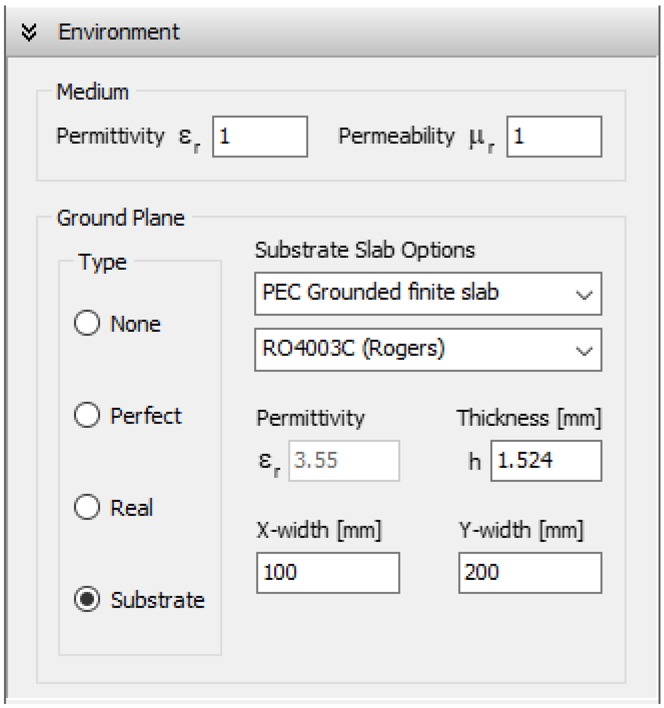


Fig. 1: Substrate option in the Ground Plane box of the Environment panel.

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).

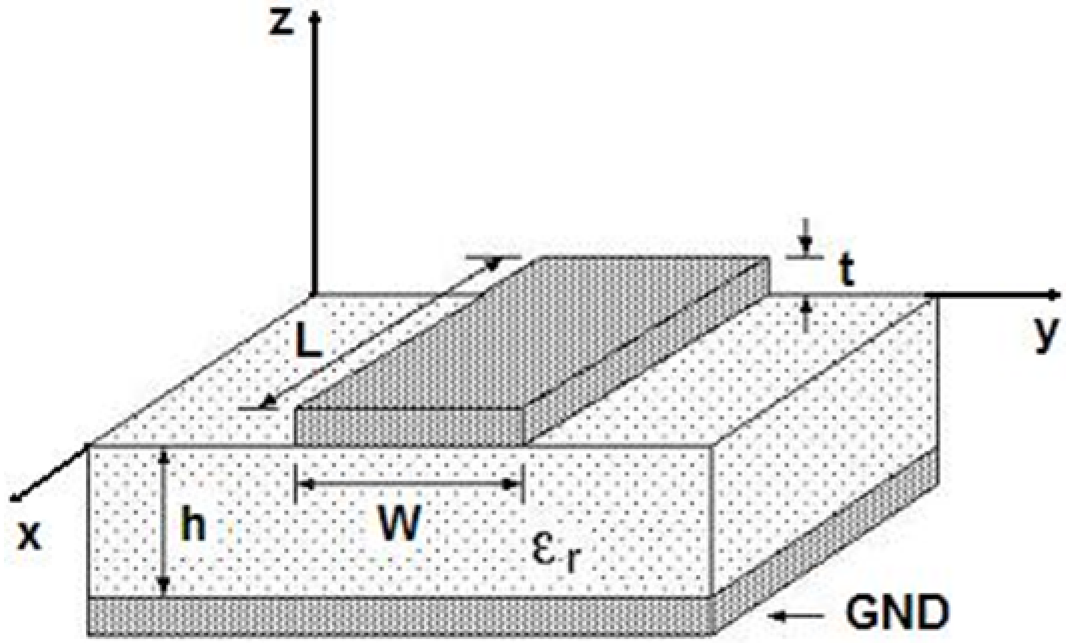


Fig. 2: Dielectric substrate positioned below the xy-plane, with a microstrip line placed above the xy-plane.

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).

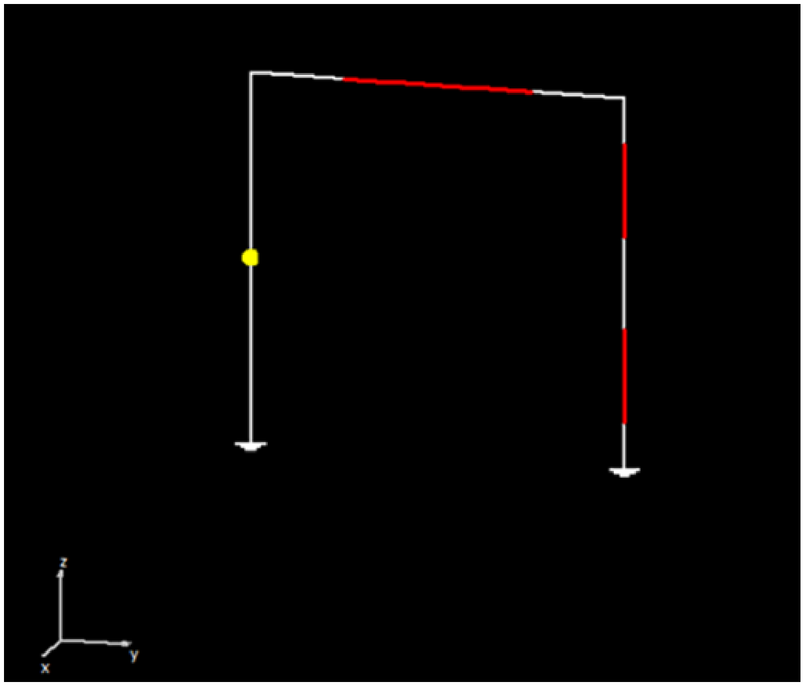


Fig. 1: 3D symbols indicating ground connections.

> **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** option in the **Ground Plane** box (see Fig. 1).

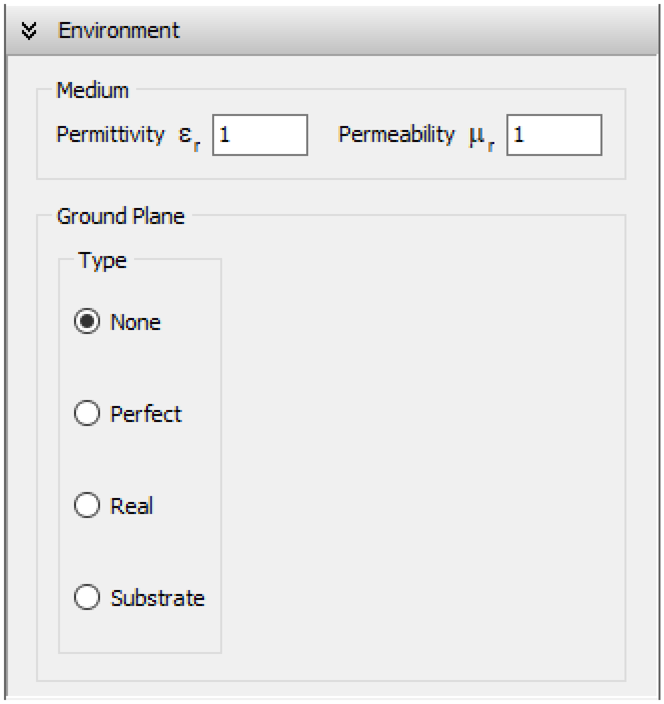


Fig. 1: None option in the Ground Plane box of the Environment panel.

Running Calculations

The Run ALL Command

Once the frequencies, the environment, the geometry of the structure, the excitation, and the points of observation of the radiated field have been set, AN-SOF is ready to execute the calculations. First, the current distribution on the wire segments will be calculated, which allows us to obtain the input impedance when we have a transmitting antenna. Later, the far and near fields can be calculated from the currents in the segments.

The **Run ALL (F10)** command allows us to run the calculation of the current distribution and the near and far fields sequentially and automatically. Go to **main menu > Run > Run ALL** to run this command, Fig. 1, or click on the Run ALL button on the toolbar.

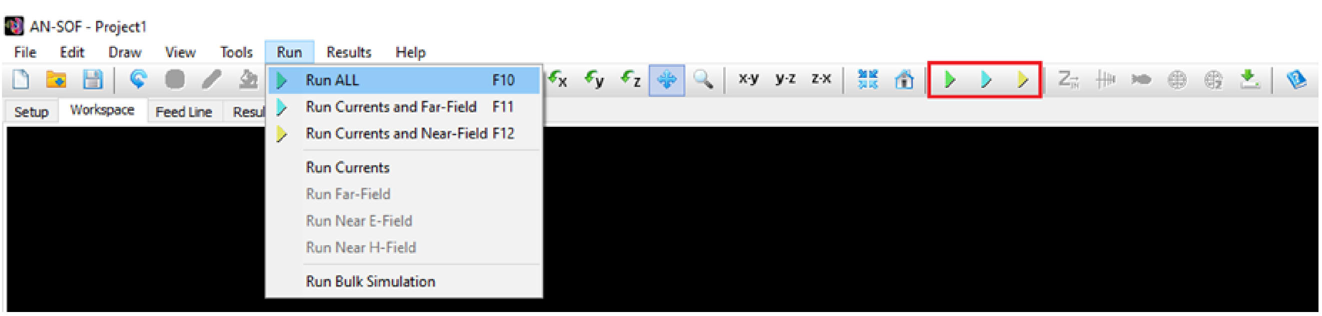


Fig. 1: The Run ALL command in the main menu. There are also buttons on the toolbar to run the calculations.

If the near field is not required, the calculation can only be run for currents and far fields by clicking on the Run > **Run Currents and Far-Field (F11)** command. This command is also available on the toolbar.

If the far field is not required, the calculation can only be run for currents and near fields by clicking on the Run > **Run Currents and Near-Field (F12)** command. This command is also available on the toolbar.

The currents, far and near fields can be computed separately as it is explained in the **next articles >**.

Calculating the Current Distribution

When the frequencies, the environment, the geometry, and the excitation are set, AN-SOF is ready to compute the currents flowing on the wire segments.

Go to **Run > Run Currents** in the main menu to run the calculation of the current distribution, Fig. 1.

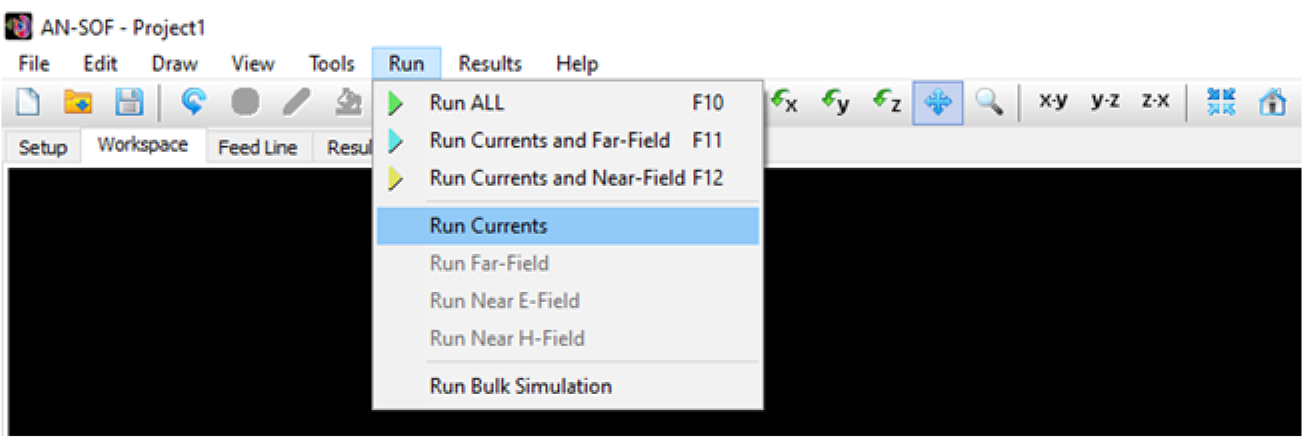


Fig. 1: The Run Currents command in the main menu.

> **Tip**
>
> When we are modeling a transmitting antenna and we only need the **input impedance**, this command allows us to save time since **the radiated field is not calculated**.

Calculating the Far Field

Once the current distribution on the structure has been obtained, the far-field in the angular ranges set in the **Far-Field panel >** of the Setup tabsheet can be computed.

Go to **Run > Run Far-Field** in the main menu to run the calculation of the far-field, Fig. 1. This command is only enabled when the current distribution has already been calculated.

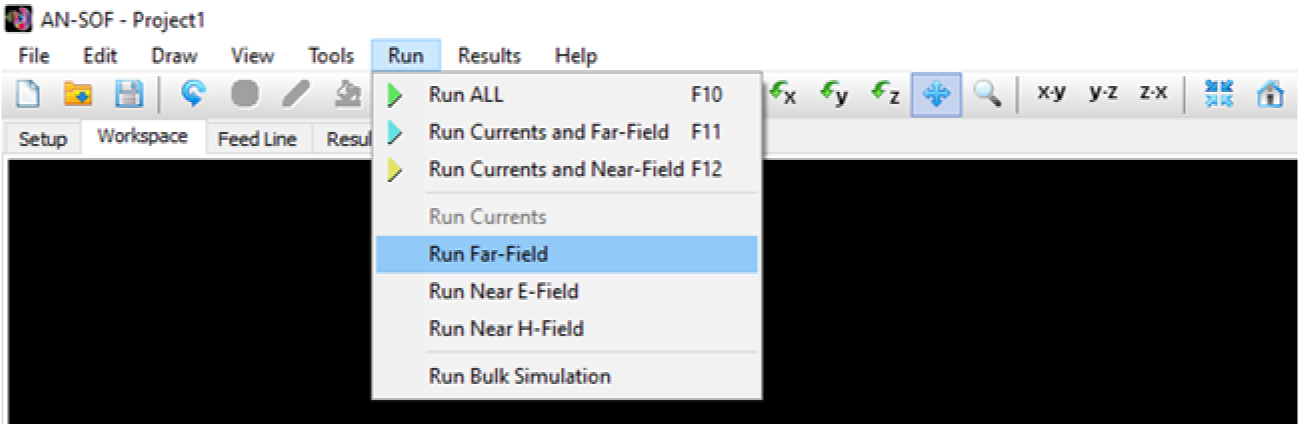


Fig. 1: The Run Far-Field command in the main menu.

> **Tip**
>
> To run the calculation of the current distribution and the far field sequentially and automatically, click on the **Run Currents and Far-Field (F11)** button on the toolbar.

Calculating the Near E-Field

Once the current distribution on the structure has been obtained, the near electric field at those points in space set in the **Near-Field panel >** of the Setup tabsheet can be computed.

Go to **Run > Run Near E-Field** in the main menu to run the calculation of the near electric field, Fig. 1. This command is only enabled when the current distribution has already been calculated.

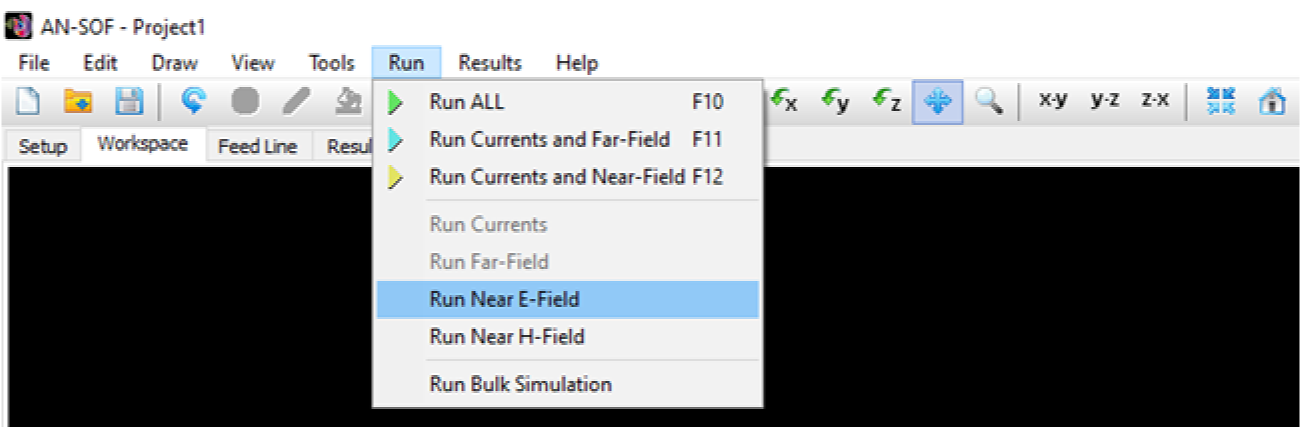


Fig. 1: The Run Near E-Field command in the main menu.

> **Tips**
>
> To run the calculation of the current distribution and the near fields sequentially and automatically, click on the **Run Currents and Near-Field (F12)** button on the toolbar. This command also runs the calculation of the near H-Field.
>
> To avoid the calculation of the H-Field, go to **main menu > Tools > Preferences > Options >** and uncheck the **“Run ALL” also calculates the H-Field** option.

Calculating the Near H-Field

Once the current distribution on the structure has been obtained, the near magnetic field at those points in space set in the **Near-Field panel >** of the Setup tabsheet can be computed.

Go to **Run > Run Near H-Field** in the main menu to run the calculation of the near magnetic field, Fig. 1. This command is only enabled when the current distribution has already been calculated.

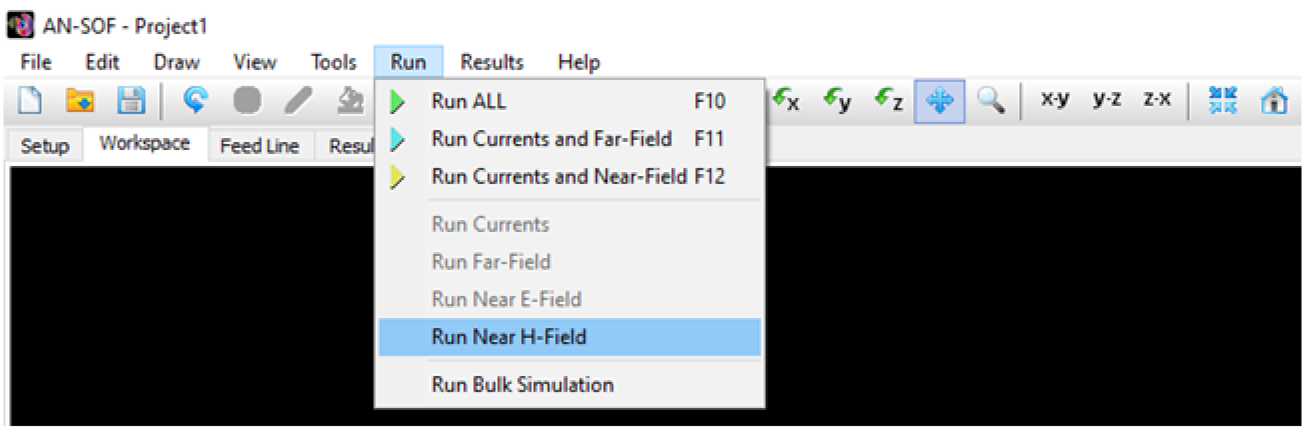


Fig. 1: The Run Near H-Field command in the main menu.

> **Tips**
>
> To run the calculation of the current distribution and the near fields sequentially and automatically, click on the **Run Currents and Near-Field (F12)** button on the toolbar. This command also runs the calculation of the near electric field.
>
> Go to **Tools > Preferences > Options >** in the main menu and check the **“Run ALL” also calculates the H-Field** option to enable the calculation of the H-field.

Aborting the Calculations

When a calculation is executed using the commands under the **Run menu >**, the **Processing** window will be displayed, Fig. 1. There is a button to abort the calculation at any time. Note that you will be prompted to save the project before aborting, as AN-SOF will restart.

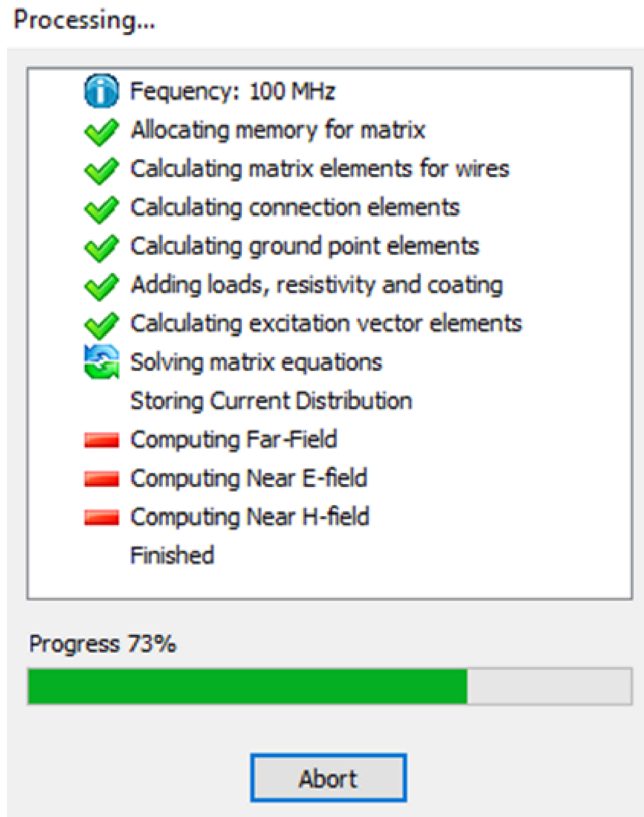


Fig. 1: The Processing window.

Numerical Green's Function

There are simulations in which we need to change the excitation of the structure frequently. For example, when we must often adjust the amplitudes of discrete sources or alter the direction of arrival of an incident field. In these cases, we can save a significant amount of time by enabling the **NGF (Numerical Green's Function)** option in the **Settings panel** of the **Setup** tab, as shown in Fig. 1.

When an NGF calculation is performed, the LU-decomposed matrix of the system is stored in a file after the initial calculation. Subsequently, by reusing this stored matrix, new calculations can be performed more quickly than the initial one.

When **transmission lines** are included in the model, the NGF option will be automatically enabled.

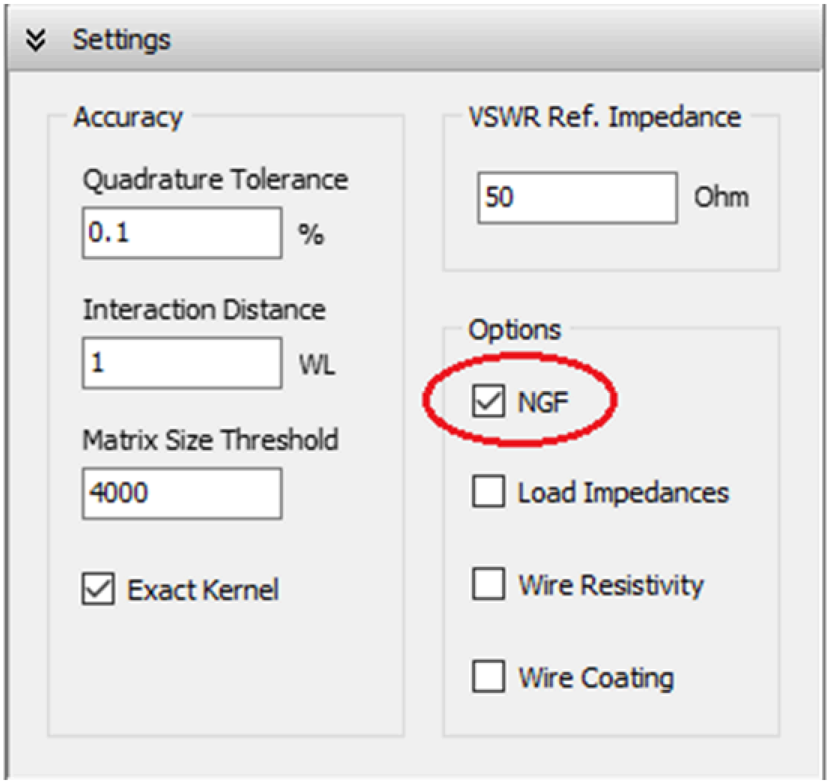


Fig. 1: NGF option in the Settings panel of the Setup tabsheet.

Running a Bulk Simulation

AN-SOF is capable of importing a sequence of **input files** to obtain a corresponding sequence of **output files**, all without requiring any 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 here: **Importing Wires**.

The output data consists of power budget or RCS (Radar Cross Section), input impedances, far field, and near fields, all provided in **CSV format**. For each NEC input file, AN-SOF generates an individual project containing **.emm** and **.wre** files (see **File Formats**). This way, each project can be opened separately once the bulk simulation is completed.

To initiate a bulk simulation, navigate to the main menu and choose **Run > Run Bulk Simulation**. A prompt will appear, asking whether you want to save the changes in the current project, as the bulk simulation requires closing the currently open project. Subsequently, a dialog box will be displayed, allowing you to select a directory and the input .nec files. Upon selecting the desired files and clicking the “Open” button, the bulk simulation will commence, with the input files being imported and computed one after another in alphabetic order.

For instance, if we consider an input file named “InputFile.nec,” the following files will be generated:

Files of the AN-SOF project

InputFile.emm > main file of the project (it 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

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

where “X” represents the frequency in Hz (e.g., X = 300000000 for a frequency of 300 MHz). Consequently, a FarField, EField, and HField file will be generated for each frequency if a frequency sweep simulation has been configured.

Bulk simulations serve the purpose of automating the calculation process for multiple NEC files, even if they are not directly related, eliminating the need for manual calculations file by file. Conversely, they are also useful for sequentially running calculations on NEC files generated with **varying geometric parameters** in an antenna. Subsequently, the results can be analyzed by reading data from the generated CSV files.

For instance, 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 understand how to accomplish this and read the output data from the CSV files, you can refer to the following link: **Element Spacing Simulation Script for Yagi-Uda Antennas**.

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.

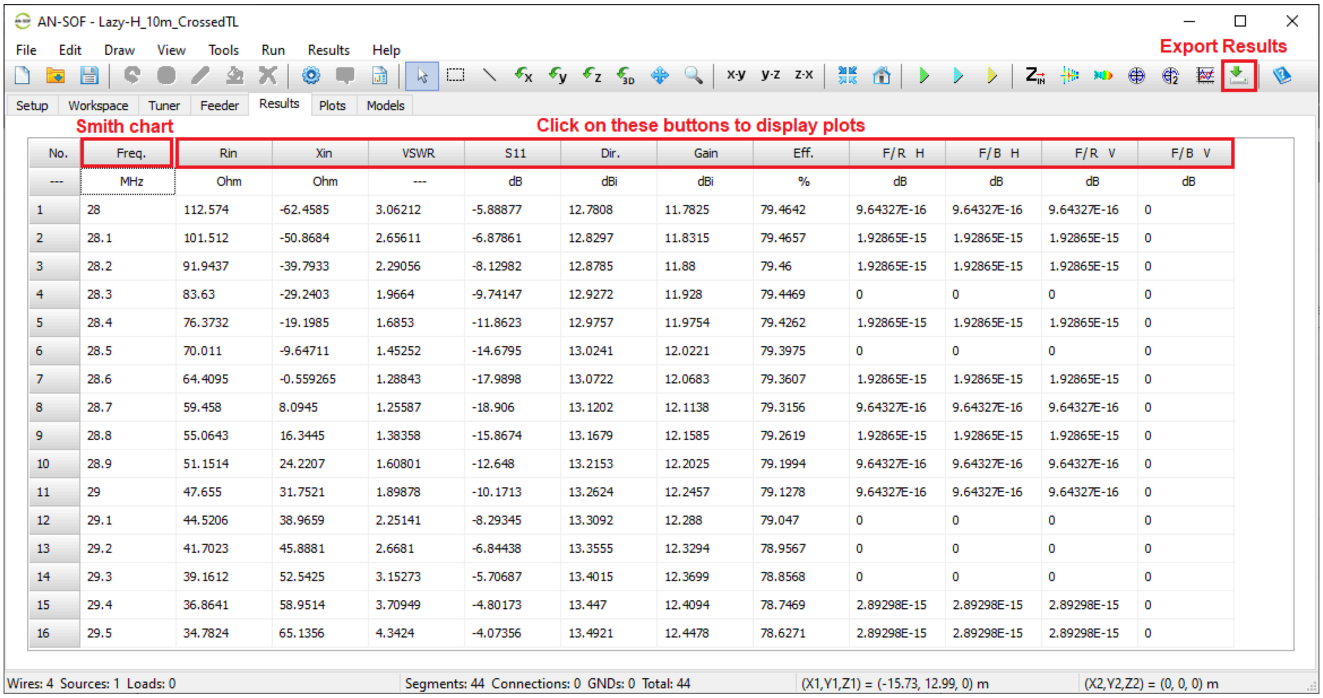
In the case of fields, they can be listed at a given point versus the frequency **(Spectrum)** or at a given frequency versus the observation point **(Pattern)**.

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**
- **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.

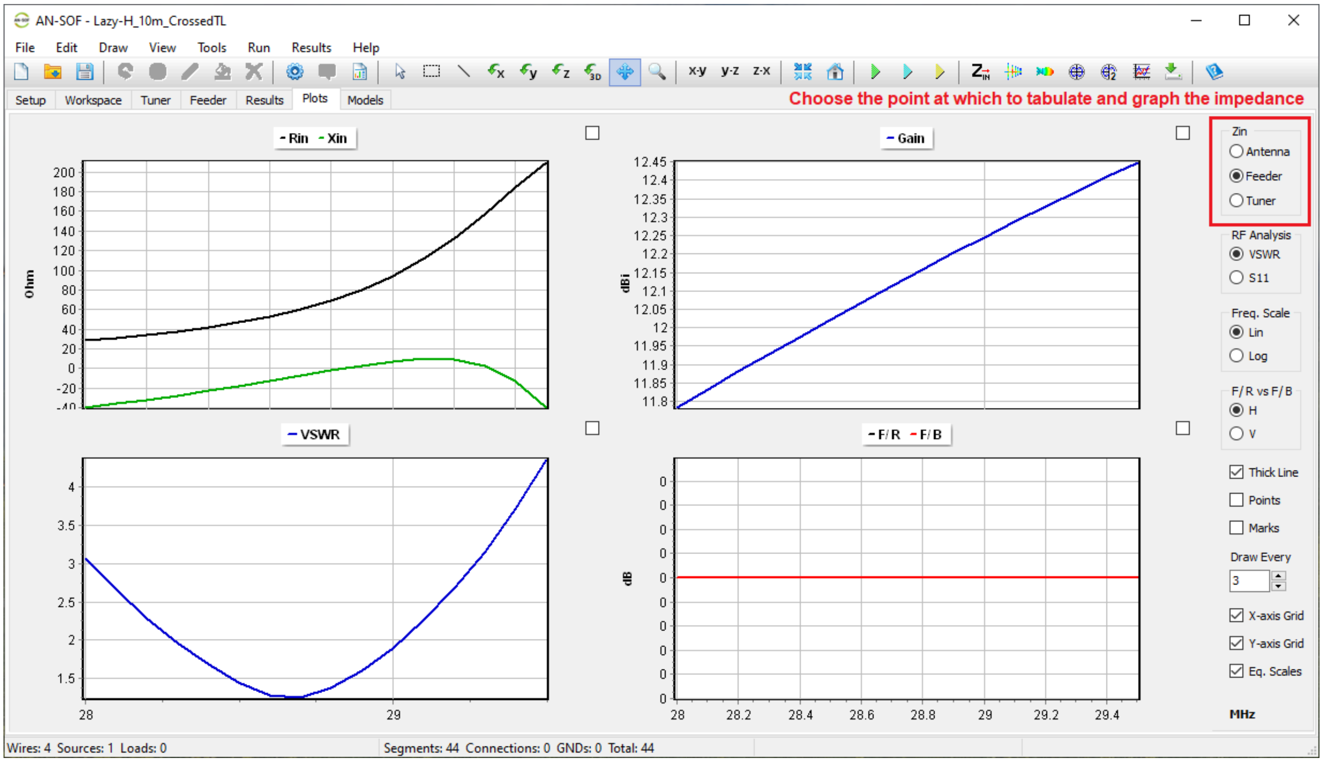


Fig. 2: *Plots tab in the main window, showing the Antenna, Feeder, and Tuner options in the Zin box for selecting the point to tabulate and graph impedance.*

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 obtained from the table in the **Results** tab.

The left column in the Plots tab presents the real and imaginary parts of the **input impedance** and **VSWR**. On the right column are the **antenna gain** in dBi and the **front-to-rear** (F/R) and **front-to-back** (F/B) **ratios** in dB. These plots are aligned vertically to make it easy to compare.

Use the controls on the right side of the Plots tab to change different aspects of the graphics, including line thickness, visualization of points and marks, scales, axes, and also to choose between **VSWR** or **S_{11}** and **horizontal (H) or vertical (V) F/R vs. F/B ratios**. Each plot can be maximized by clicking on the “Maximize” checkbox located at its upper right corner.

The input impedance and VSWR/S_{11} plots can represent the **antenna input impedance**, the **feeder + antenna input impedance**, or the **tuner input impedance**. The **tuner** and **feeder** can be configured in their corresponding tabs next to the Results tab. Every time a tuner or feeder parameter is changed, the recalculated results in the **Results** and **Plots** tabs can be **refreshed** by clicking the desired option under the “Zin” box highlighted in Fig. 1 below.

If the **Tuner** option is chosen to display the plots and results, note that the **input impedance of the tuner** will be displayed, and 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.

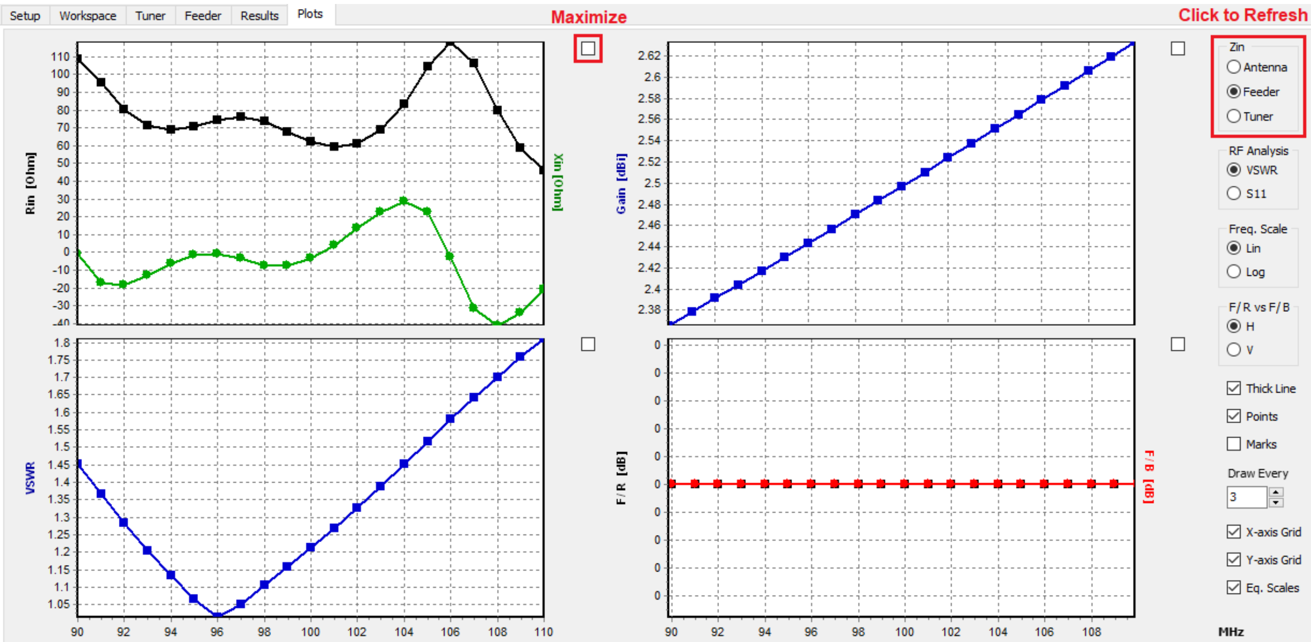


Fig. 1: Plots tab in the AN-SOF main window.

Current Distribution

Plotting the Current Distribution

Go to **Results > Plot Current Distribution** in the main menu to display a 3D graph of the current distribution on the structure. This command executes the **AN-3D Pattern >** application where the amplitude of the currents is displayed on the structure using a color scale. Additionally, the currents in phase, real, and imaginary parts can be plotted selecting these options in the **Plot** menu of AN-3D Pattern, Fig. 1.

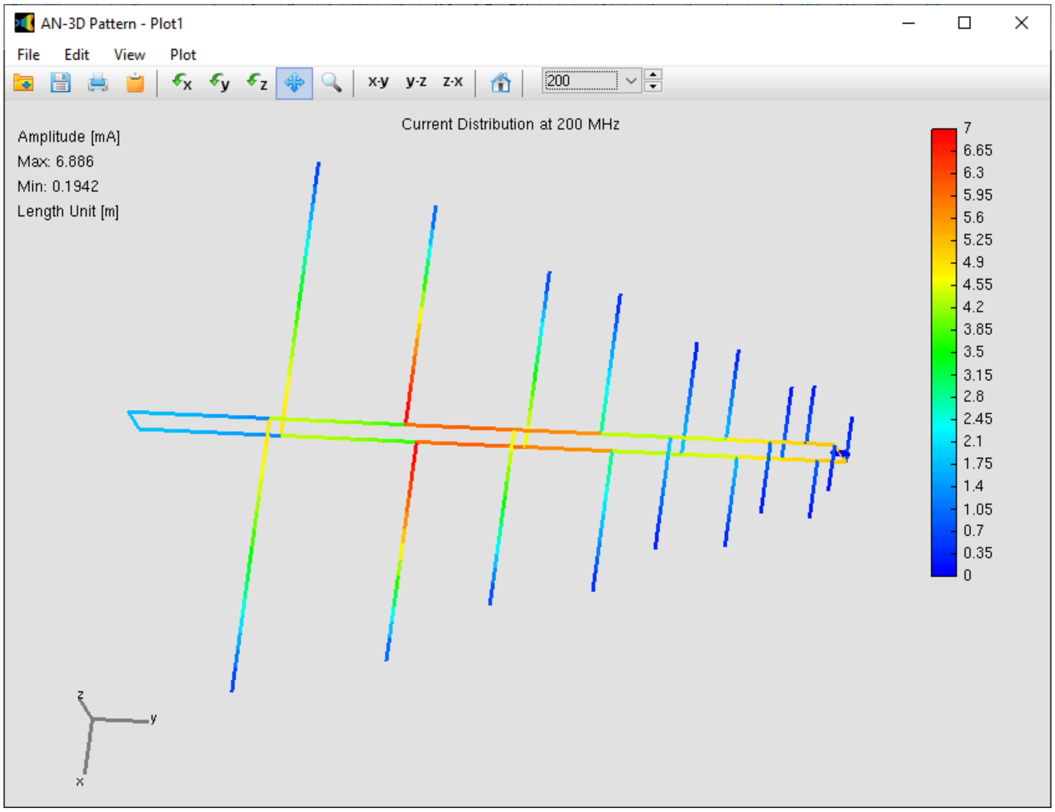


Fig. 1: Current distribution in amplitude plotted by AN-3D Pattern.

A 2D plot of the current distribution along a selected wire can be shown by right clicking on the wire and choosing **Plot Currents** from the pop-up menu, Fig. 2. The Plot Currents command executes the **AN-XY Chart >** application, where the current is plotted in amplitude vs. position along the selected wire. The current distribution can also be plotted in phase, real and imaginary parts by choosing these commands under **View** in the AN-XY Chart main menu.

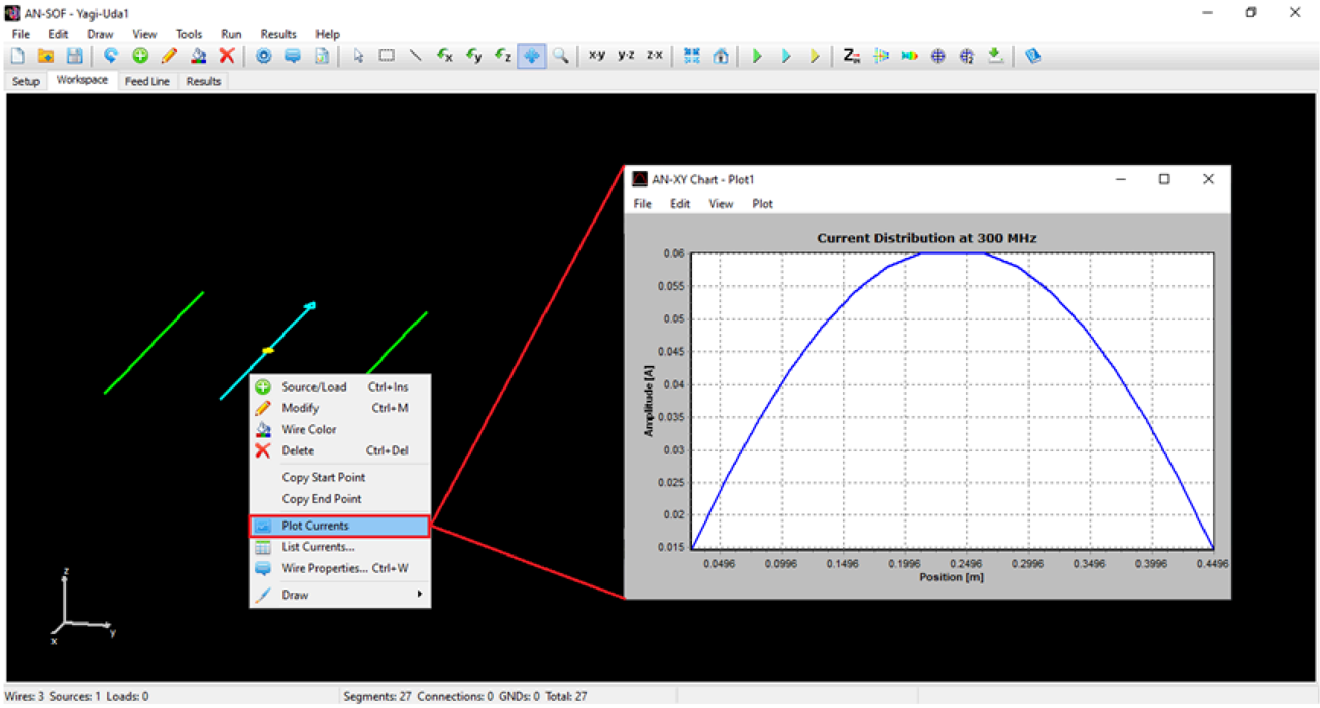


Fig. 2: The Plot Currents command in the pop-up menu and the current distribution in amplitude plotted by AN-XY Chart.

A wire can also be selected by first clicking on the **Select Wire** button (arrow icon) on the toolbar and then left clicking on the wire. Once the wire is selected, go to Results > **Plot Currents** in the main menu to plot the current along that wire. This command is enabled when the current distribution has been calculated.

> **Tips**
>
> The graph plotted by **AN-XY Chart** can be zoomed by **expanding a box** with the left mouse button pressed on the plot.
>
> Right click on the graph and **drag the mouse** to move it.
>
> Left click and **expand a rectangle up** to return to the original view.
>
> There are options to **change the units** of the plotted magnitudes and to **export data** in the AN-XY Chart main menu.

The List Currents Toolbar

Right clicking on a wire shows a **pop-up menu >**. Click on the **List Currents** command to display the **List Currents toolbar**, Fig. 1. This toolbar allows us to select a wire segment to see the current flowing through that segment versus frequency. If the segment has a source or load, the list of input impedances, admittances, voltages, powers, reflection coefficient, VSWR, return and transmission losses can also be displayed.

A wire can also be selected by first clicking on the **Select Wire** button (arrow icon) on the toolbar and then left clicking on the wire. Once the wire is selected, go to **Results > List Currents** in the main menu. This command is enabled when the current distribution has been calculated.

The List Currents toolbar has the following components:

Fig. 1: The List Currents toolbar.

The Slider

Each position of the slider corresponds to the position of a segment along the selected wire. Thus, the slider allows us selecting the desired wire segment. The position of the selected segment is shown at the right corner of this toolbar. The segment position is shown as a number and as a percentage of the wire length. The percentage position is measured from the starting point of the wire to the middle point of the segment, namely,

% position = 100 (position / wire length)

The 50% button

Moves the slider towards the center of the wire. Note that there must be **an odd number of segments** for there to be a segment at the midpoint of the wire.

The Current on Segment button

Displays the **Current on Segment** dialog box, Fig. 2, showing a list of the current in the selected segment versus frequency. Click the **Plot** button to plot the current in the segment as a function of frequency.

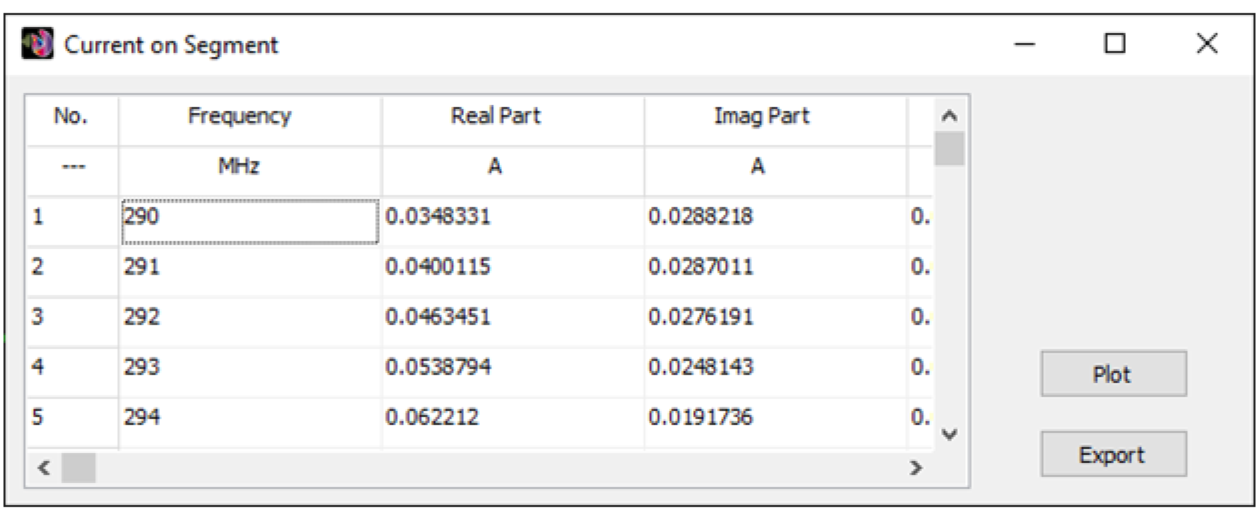


Fig. 2: The Current on Segment dialog box.

The Input List button

If the selected segment has a source on it, the **Input List** button will be enabled. Click this button to display the Input List dialog box, Fig. 3, where the list of input impedances, admittances, currents, voltages, and powers is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency. The input impedance can be plotted in a Smith chart by pressing the **Smith** button. Click the **Export** button to save the list in CSV format.

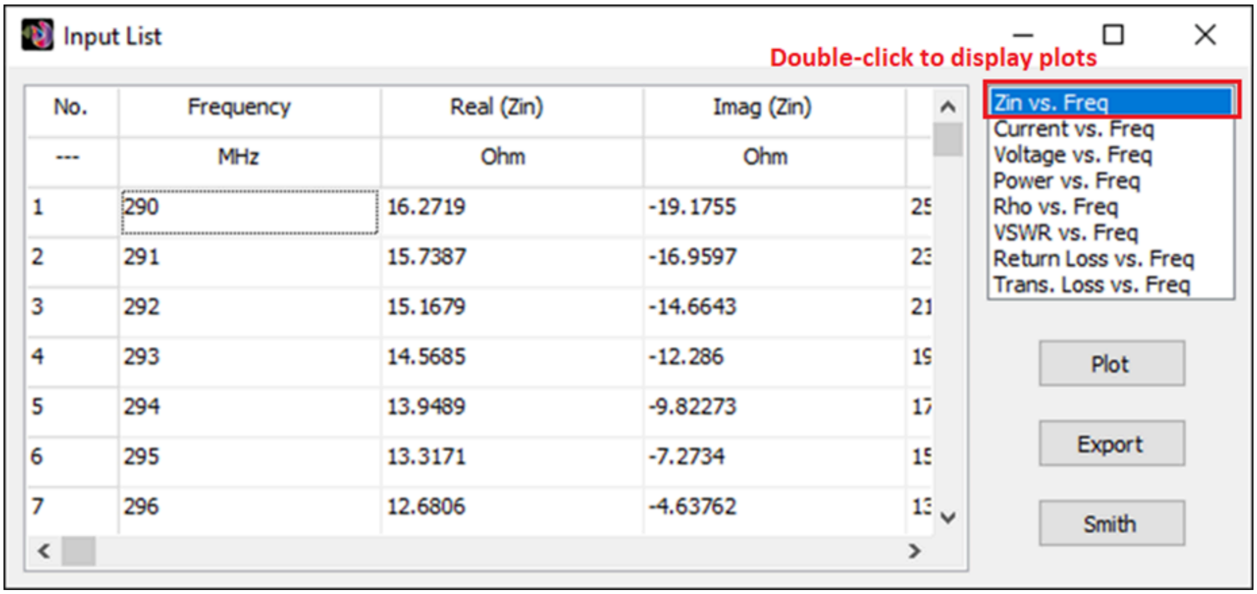


Fig. 3: The Input List dialog box.

The Source List button

If the selected segment has a source on it, the **Source List** button will be enabled. Click this button to display the Source List dialog box, Fig. 4, where the list of currents, voltages, and powers in the source internal impedance is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency. Click the **Export** button to save the list in CSV format.

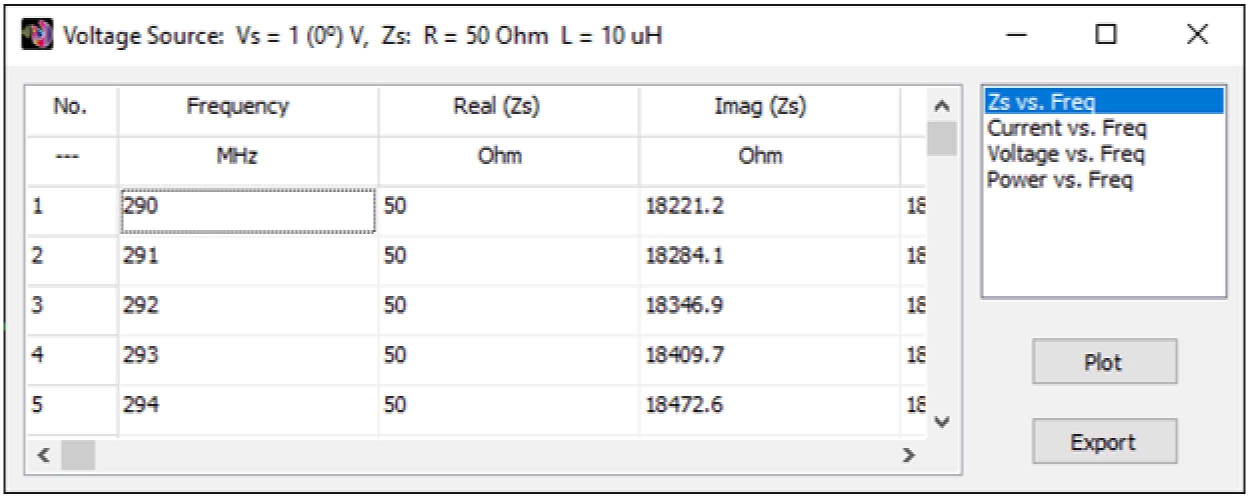


Fig. 4: The Source List dialog box.

The Load List button

If the selected segment has a load on it, the **Load List** button will be enabled. Click this button to display the Load List dialog box, Fig. 5, where the list of load impedances, currents, voltages, and powers in the segment is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency. Click the **Export** button to save the list in CSV format.

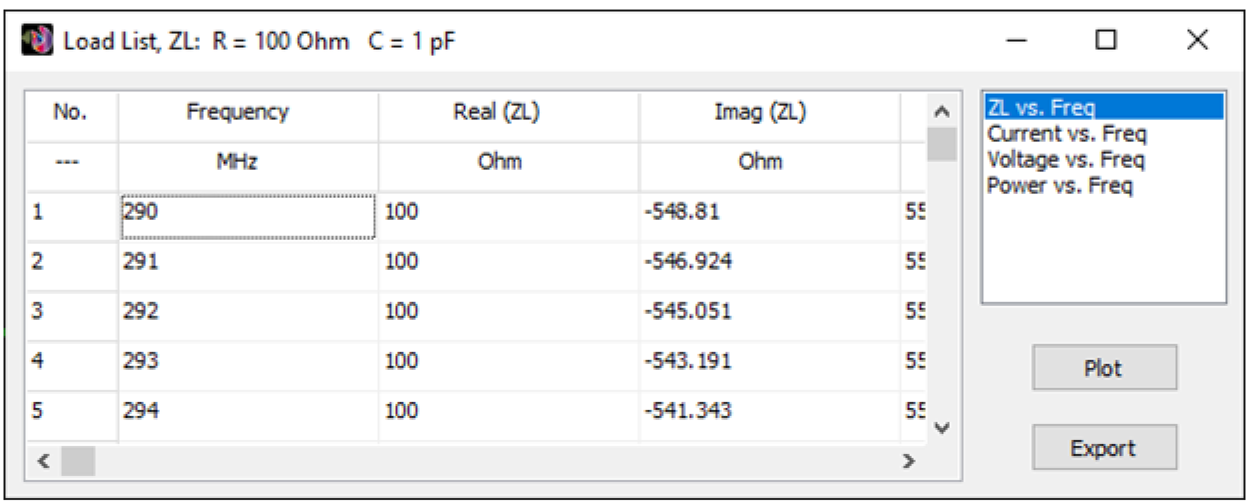


Fig. 5: The Load List dialog box.

The Exit button

Closes the List Currents toolbar.
Listing the Currents in a Segment

The following procedure allows us to select a wire segment to tabulate currents versus frequency:

1. Right click on the wire to display the **pop-up menu >**.
2. Click on the **List Currents** command to display the **List Currents toolbar >**.

3. Move the slider and select the desired segment on the wire.
4. Click on the **Current on Segment** button to display the Current on Segment dialog box, where a list of the currents versus frequency is shown. Currents are shown in amplitude, phase, real and imaginary parts. Click the **Plot** button to plot the current in the selected 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).

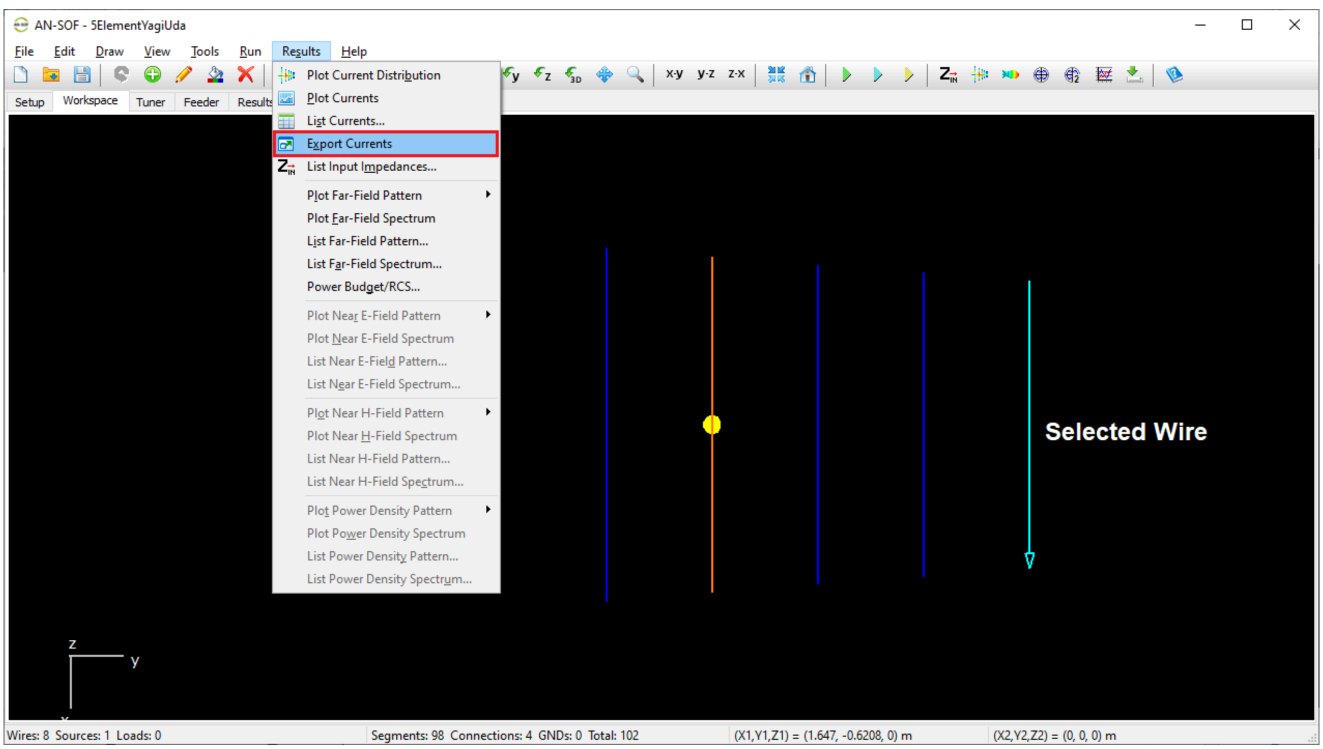


Fig. 1: Export Currents option in the Results menu, enabled when an individual wire is selected.

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

The following procedure allows us to select a segment that has a source to tabulate input impedance versus frequency:

1. Right click on a wire that has a source to display the **pop-up menu**.
2. Click on the **List Currents** command to display the **List Currents toolbar**.
3. Move the slider and select the segment where the source is placed.
4. Click on the **Input List** button to display the **Input List dialog box**, where the list of **input impedances**, admittances, currents, voltages, powers, reflection coefficient, **VSWR**, **S_{11} in decibels**, return and transmission losses is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency. Click the **Smith** button to plot the input impedance in a Smith chart.

> **Tips**
>
> The **reference impedance** for **reflection calculations (VSWR, S_{11}, and Return Loss)** can be set in the **Settings panel** of the **Setup** tabsheet.
>
> When there is a single source on the structure, you can quickly access the input impedance by going to the main menu > Results > **List Input Impedances** or by clicking on the 'List Input Impedances' button on the toolbar.

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.

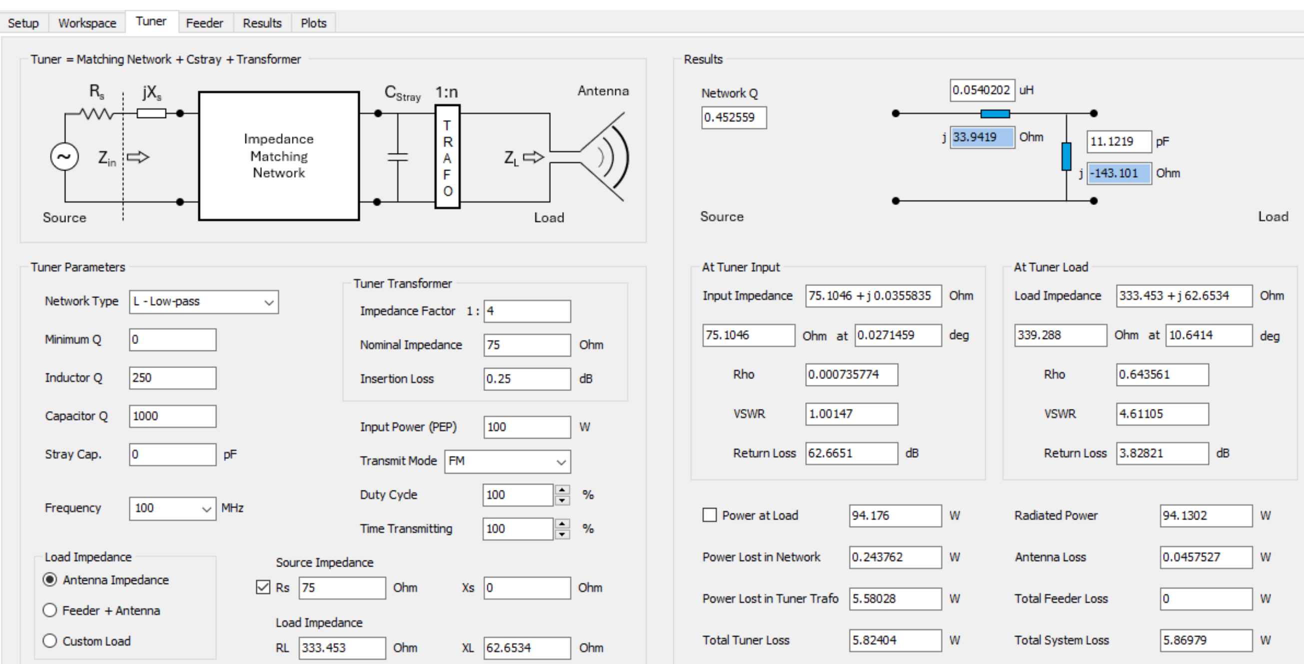


Fig. 1: Tuner tab in the AN-SOF main window where an antenna tuner can be configured.

Impedance Matching Network

In the **Tuner Parameters** box, you can configure the impedance matching network, as shown in Fig. 2.

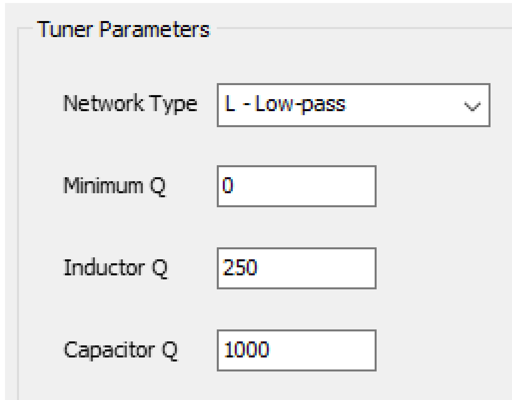


Fig. 2: Tuner Parameters box in the Tuner tab, showing options to specify the network type, minimum Q, and inductor and capacitor Q values.

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.

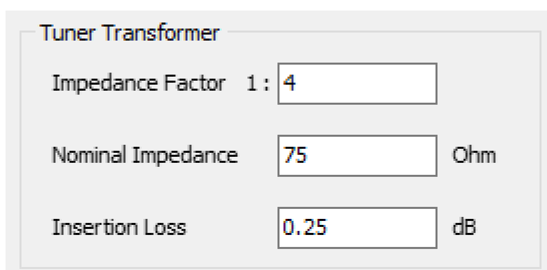


Fig. 3: Specification of a tuner transformer giving its impedance transformation factor, nominal impedance, and insertion loss.

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.

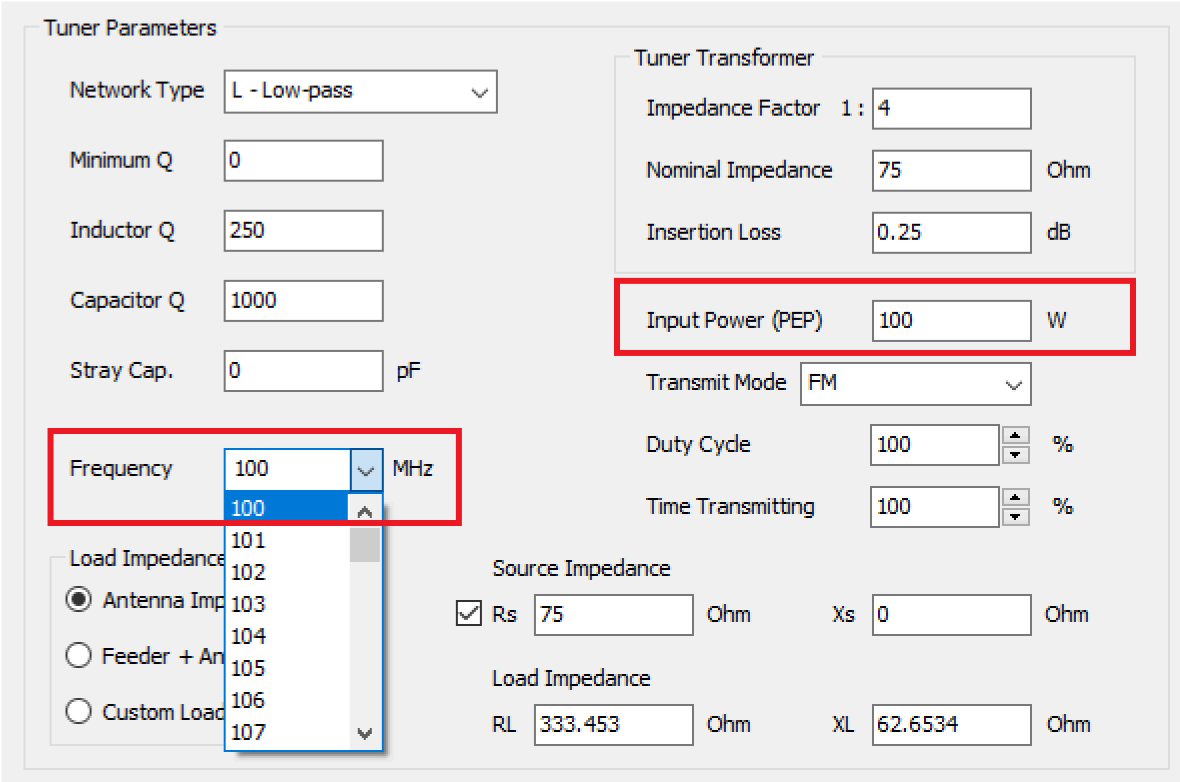


Fig. 4: The tuner design frequency and input power can be set in the Tuner Parameters box.

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.

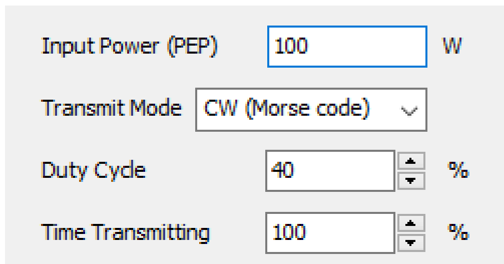


Fig. 5: Transmit Mode, Duty Cycle, and Time Transmitting settings will affect the entered Input Power (PEP).

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.

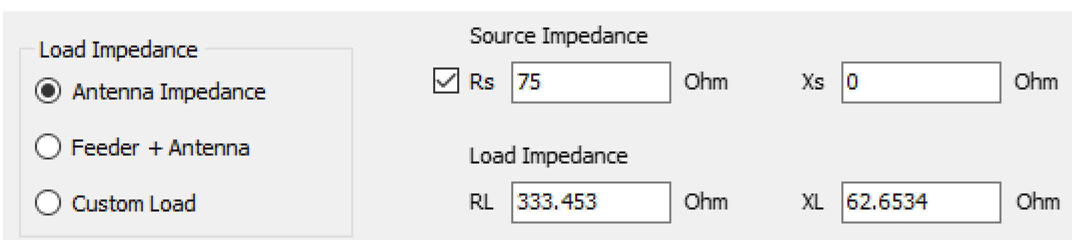


Fig. 6: Specification of the tuner source and load impedances.

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.

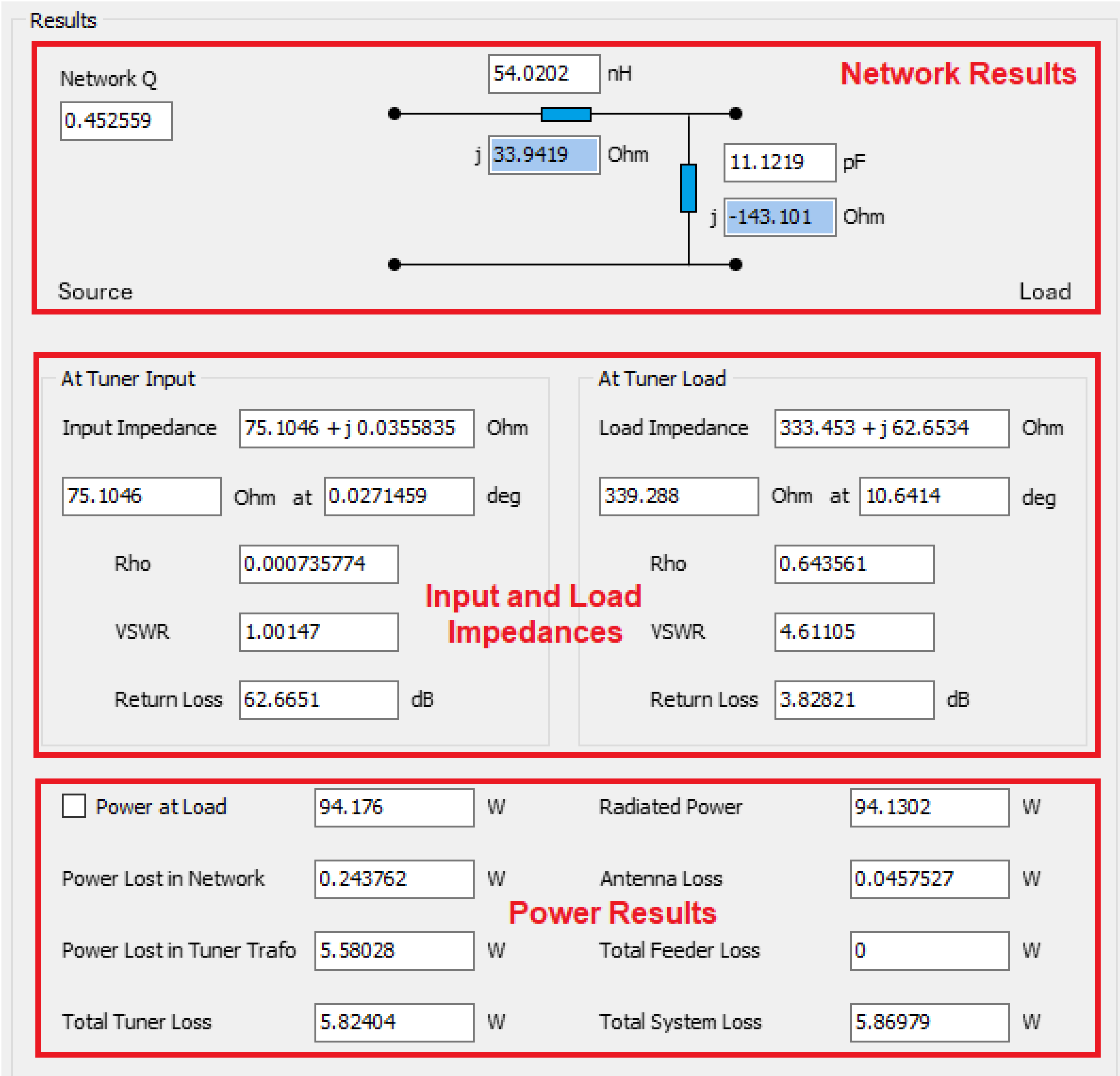


Fig. 7: Tuner Results box on the right side of the Tuner tab.

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).

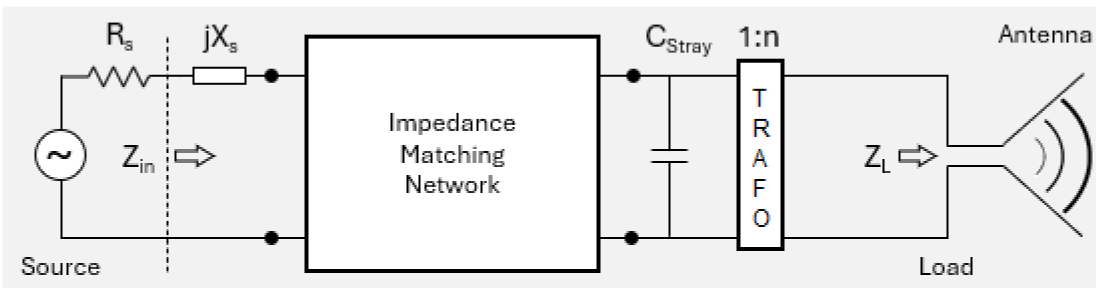


Fig. 8: Tuner diagram showing that the input impedance, Z_{in}, is seen towards the tuner after the source resistance, R_s, since the source reactance, jX_s, is absorbed by the impedance matching network.

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.

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 input impedance as a function of frequency can be plotted in a Smith chart by clicking the **Smith** button in the **Input List >** dialog box. Follow the procedure described in **Listing the Input Impedances >** for listing the input impedances versus frequency, and then click the **Smith button** in the opened dialog box.

Left click on the impedance curve in the Smith chart to see the frequency, input impedance (Zin), reflection coefficient (Rho) and VSWR in a hint message, Fig 1. Go to the AN-Smith **main menu > Plot > Admittance** to plot the input admittance curve. Go to Edit > **Preferences** to change the visualization options in AN-Smith.

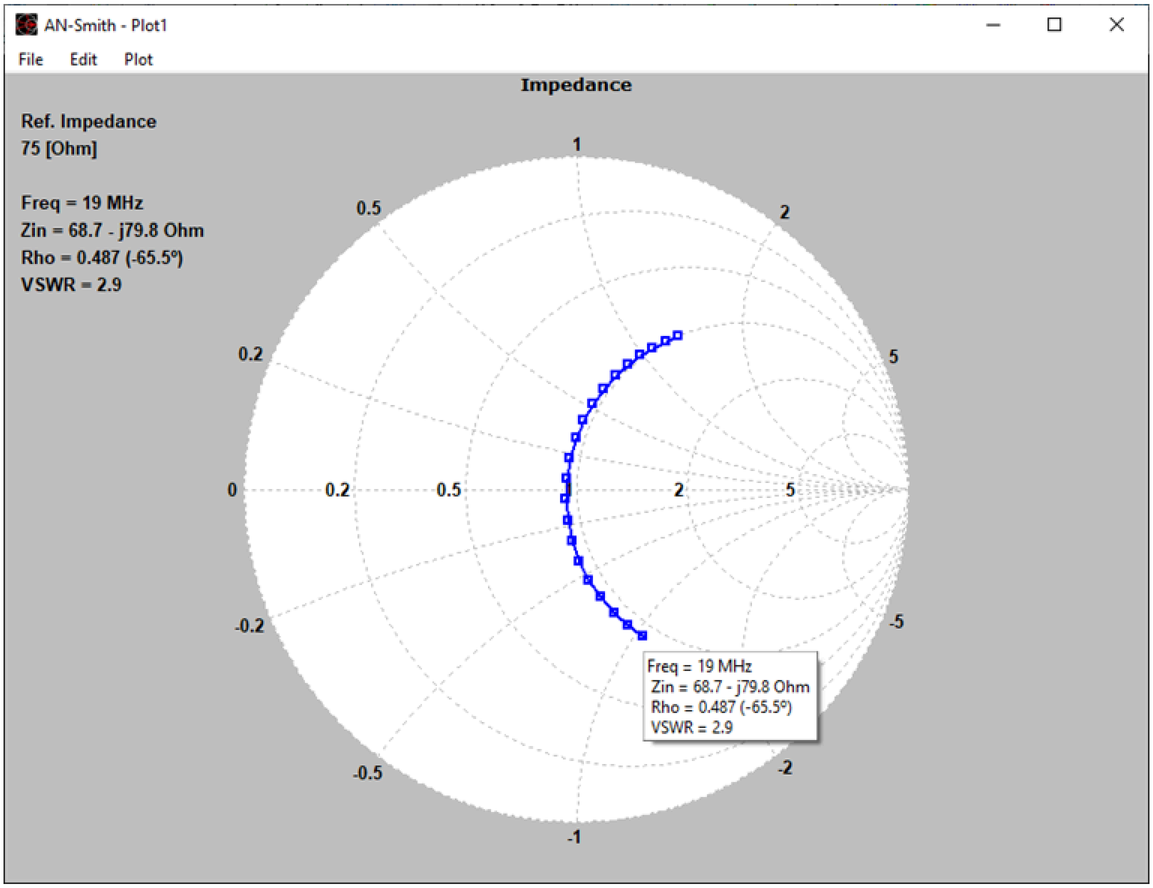


Fig. 1: Input impedance curve in the Smith chart plotted by AN-Smith.

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**.

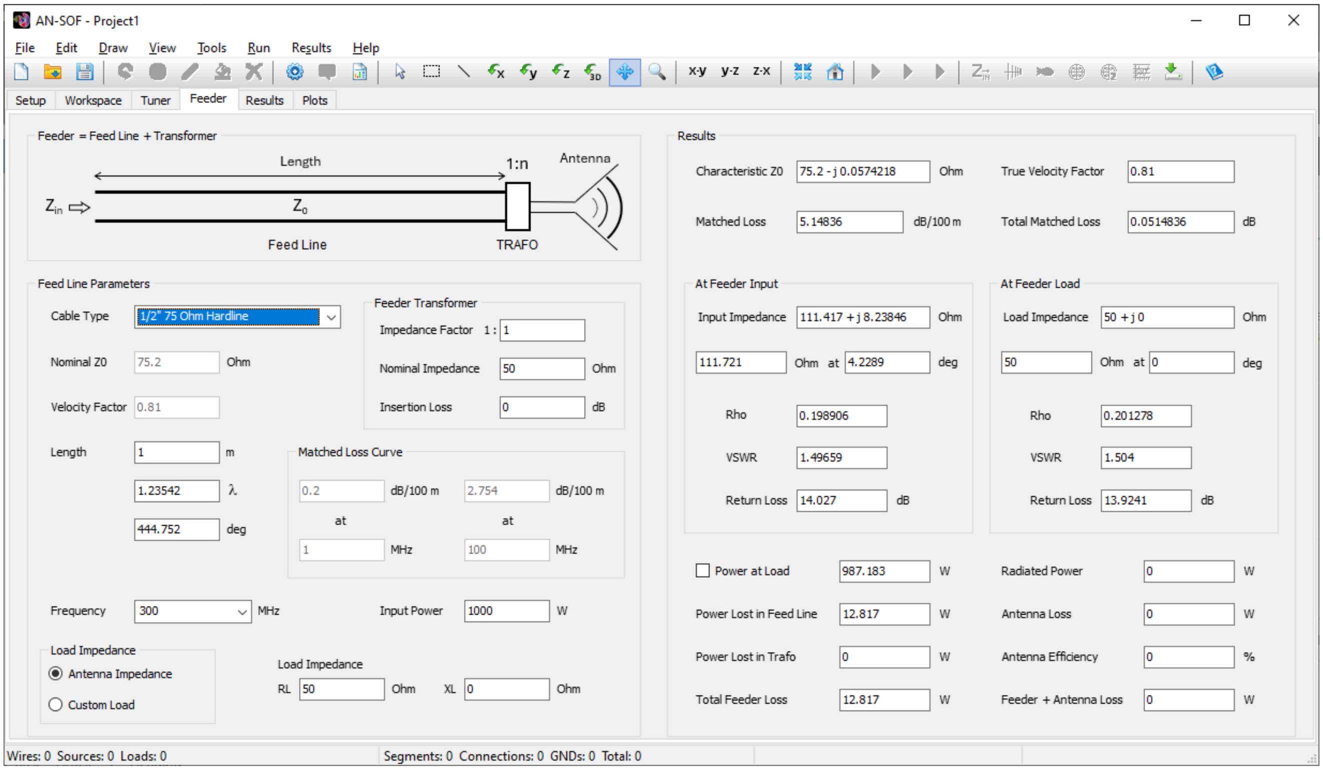


Fig. 1: Feeder tab where the feed line and transformer used to feed a transmitting antenna can be configured.

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**.

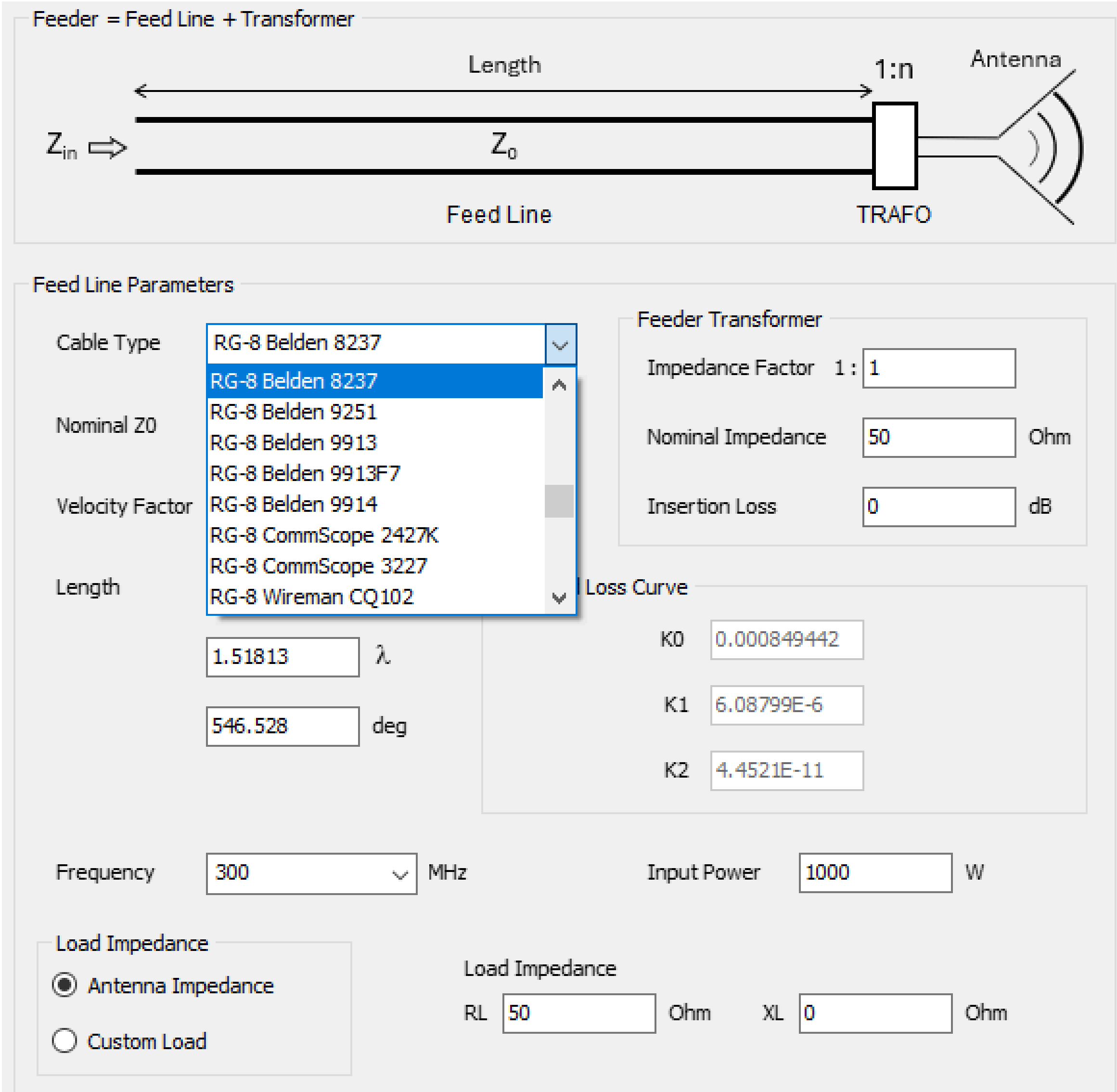


Fig. 2: Cable Type option where the type of transmission line can be chosen.

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):

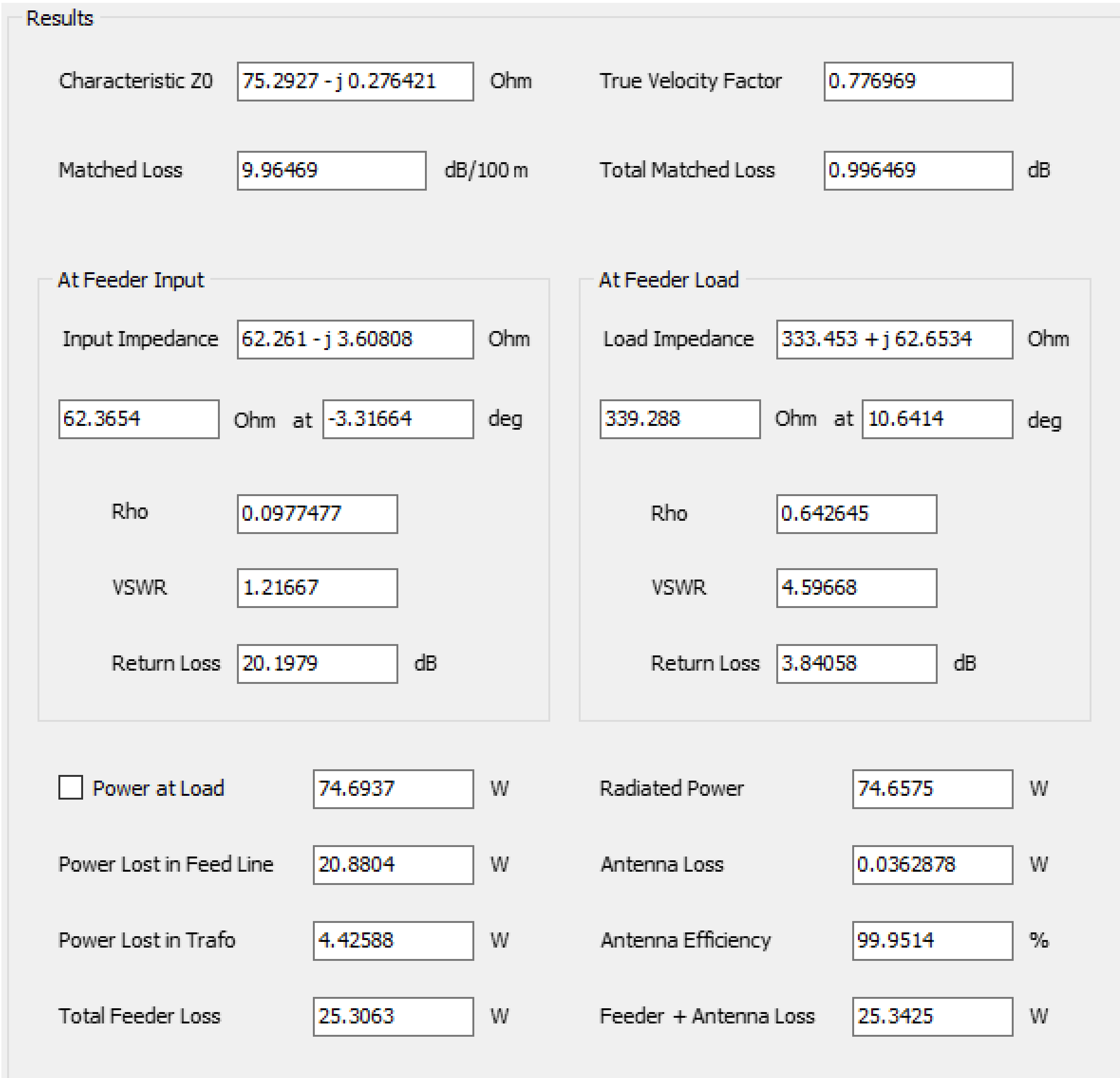


Fig. 1: Results panel on the right side of the Feeder tab. All results are automatically calculated as parameters in the left side of the Feeder tab are modified.

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:

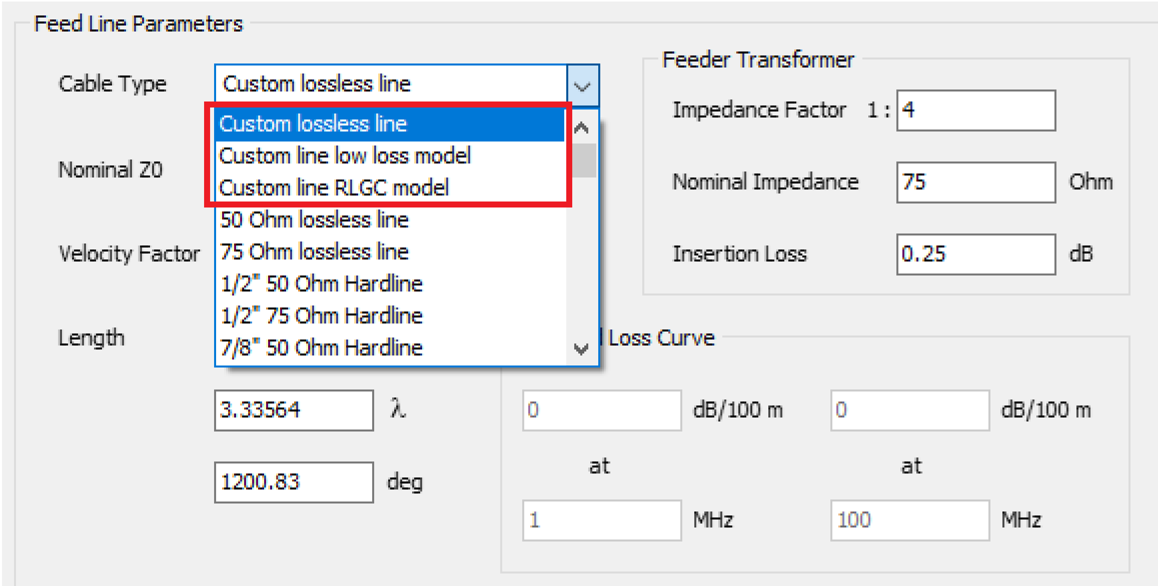


Fig. 1: Custom line options.

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

Follow these steps to select a wire segment that has a load and to tabulate the load impedance versus frequency,

1. Right click on a wire that has a load to display the **pop-up menu >**.
2. Click on the **List Currents** command to display the **List Currents toolbar >**.
3. Move the slider and select the segment where the load is placed.
4. Click on the **Load List** button to display the Load List dialog box, where the list of currents, voltages, and powers in the load impedance versus frequency is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency.

Internal Impedance of a Source

Follow these steps to select a wire segment that has a source and to tabulate the source internal impedance versus frequency,

1. Right click on a wire that has a source to display the **pop-up menu >**.
2. Click on the **List Currents** command to display the **List Currents toolbar >**.
3. Move the slider and select the segment where the source is placed.
4. Click on the **Source List** button to display the Source List dialog box, where the list of currents, voltages, and powers in the internal impedance of the source versus frequency is shown. Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency.

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.

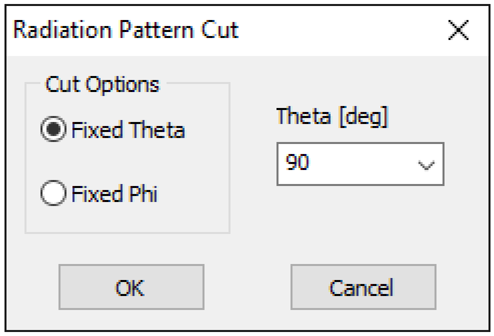


Fig. 1: The Radiation Pattern Cut dialog box.

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).

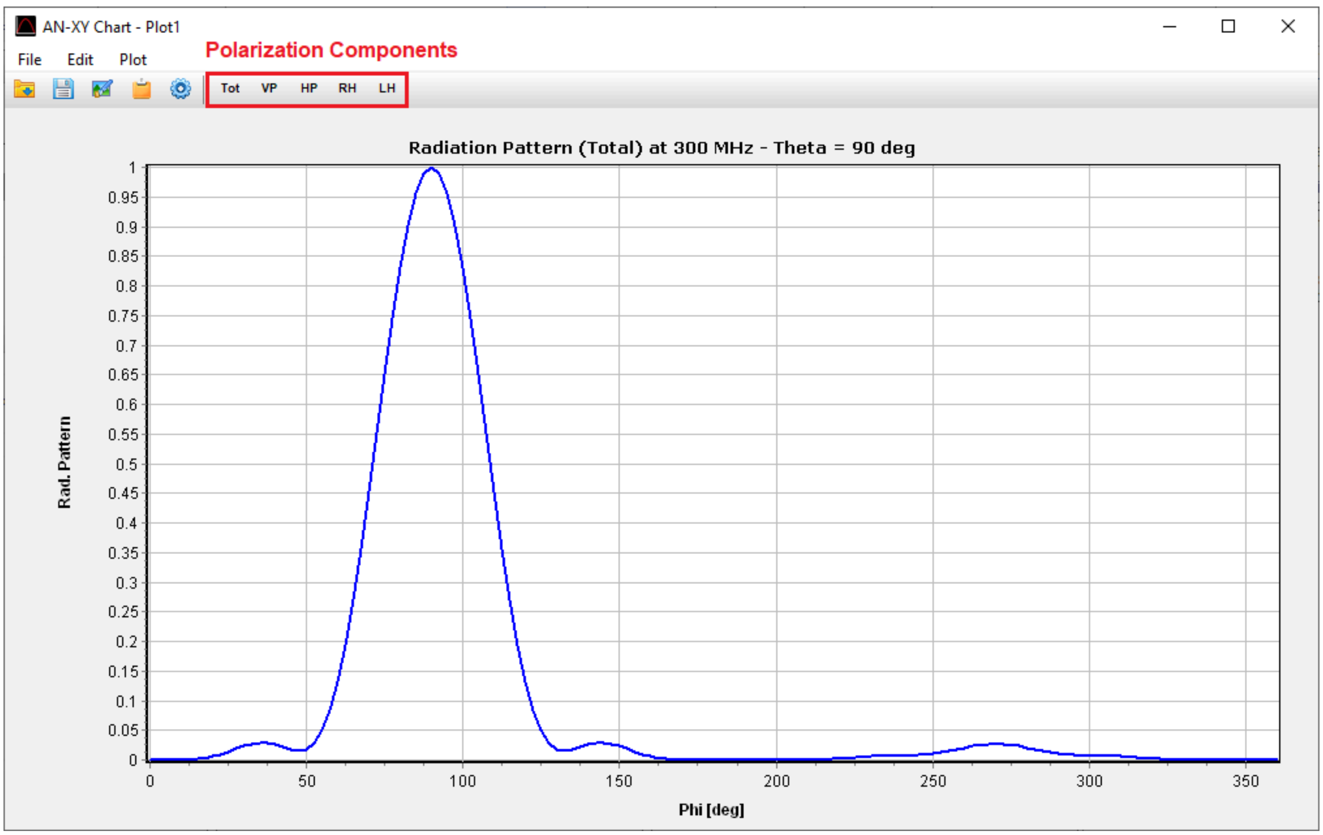


Fig. 2: A Radiation Pattern Cut plotted in AN-XY Chart in a rectangular chart.

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 **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.

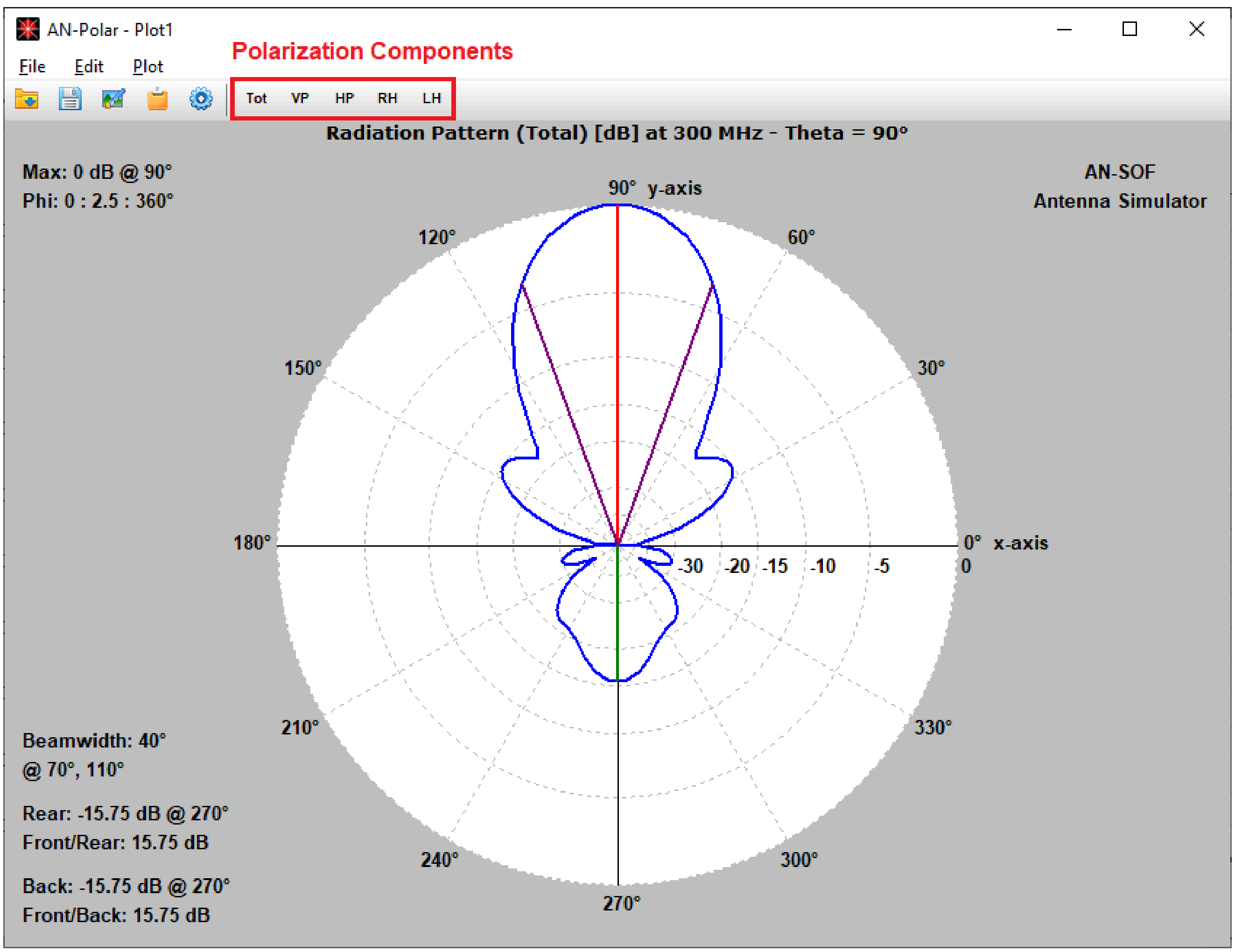


Fig. 3: A radiation pattern cut plotted in AN-Polar.

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).

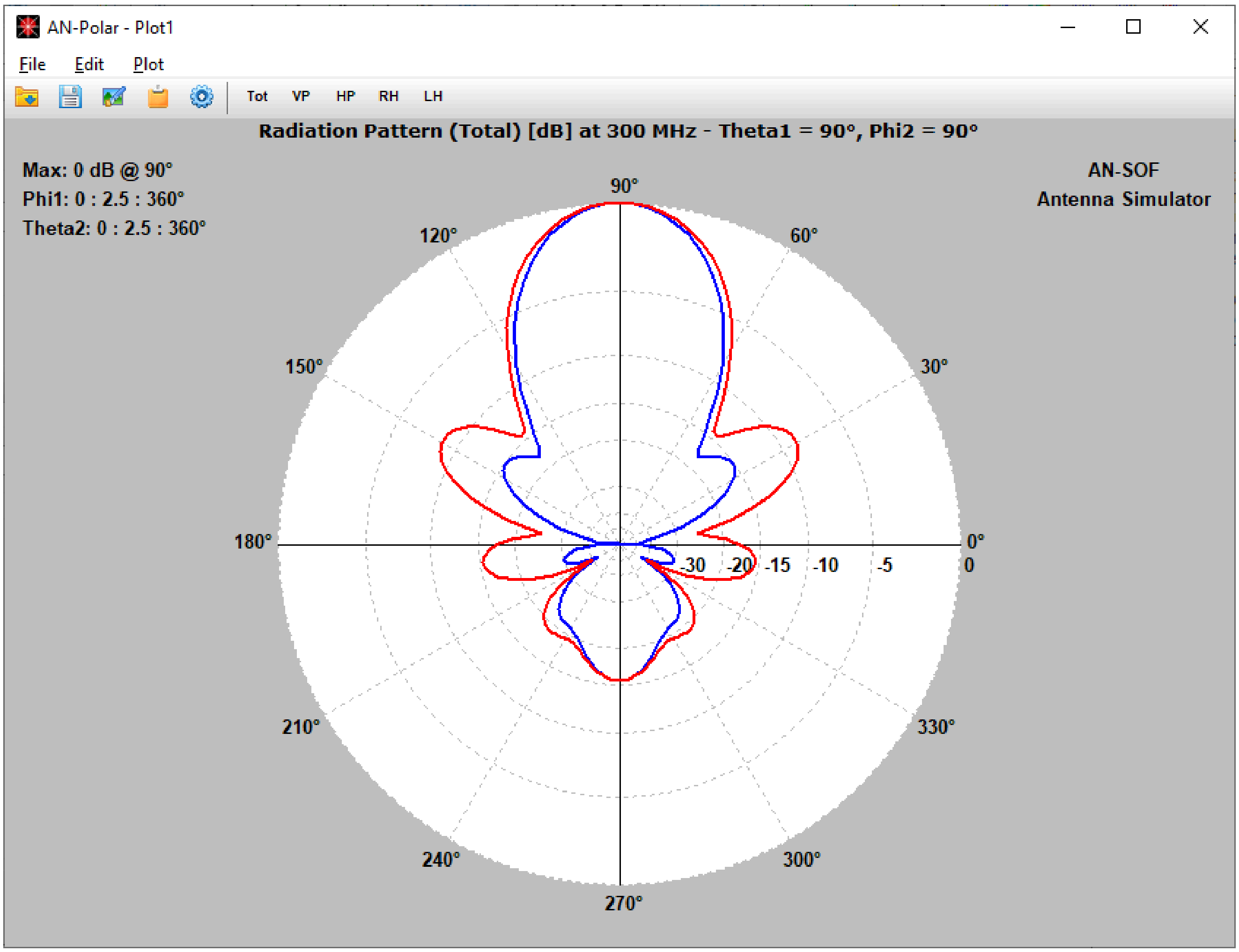


Fig. 4: Two slices of the radiation pattern plotted in AN-Polar.

Clicking on a point in the polar curve will display the corresponding value of the represented metric and the polar angle.

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 **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.

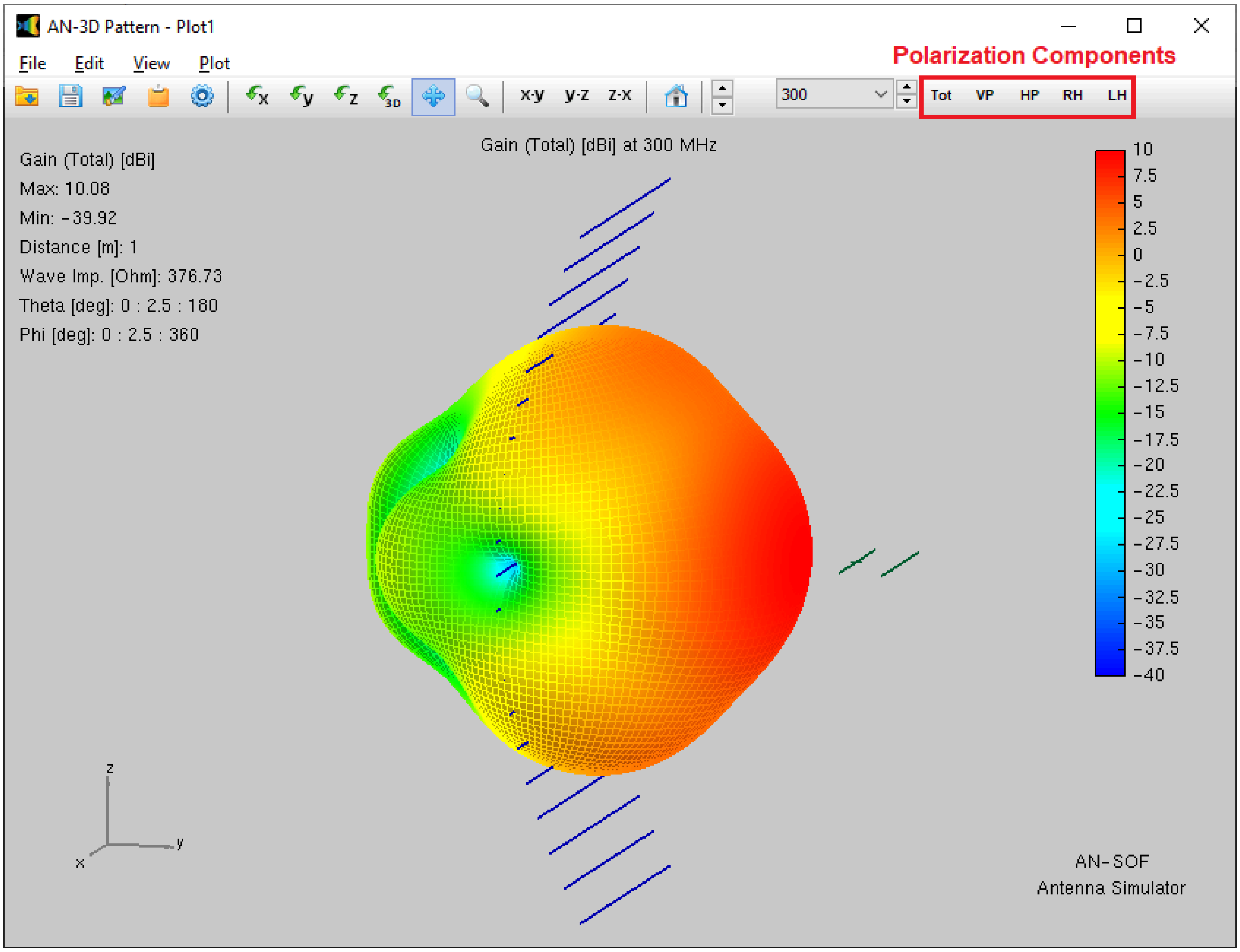


Fig. 1: 3D far-field pattern (Gain in dBi) plotted in AN-3D Pattern.

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.

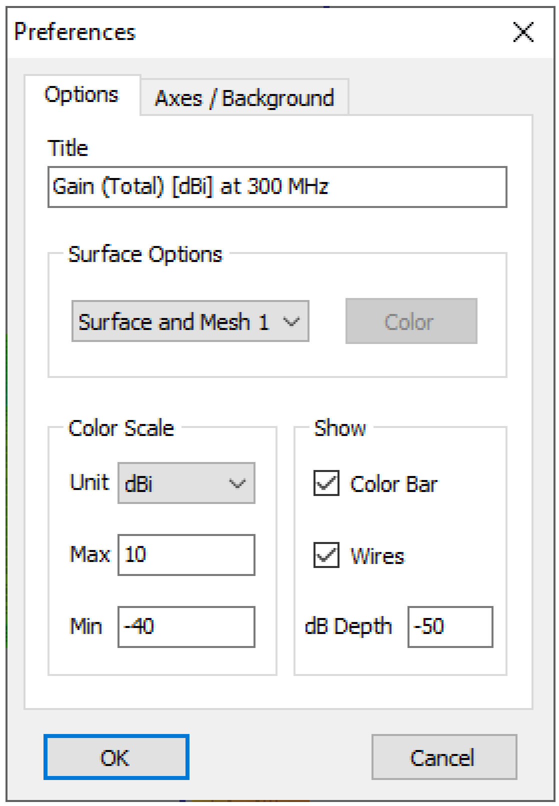


Fig. 2: Preferences dialog box of the AN-3D Pattern application.

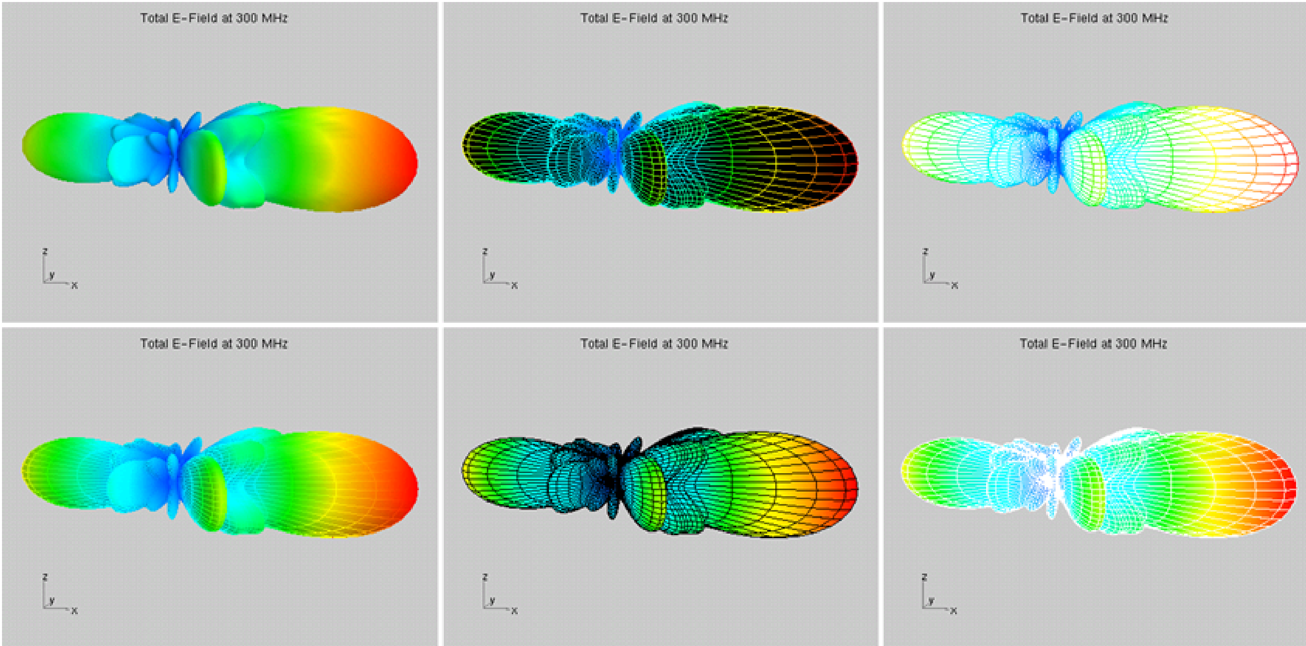


Fig. 3: Different options available for plotting radiation lobes.

Plotting the Far Field Spectrum

Far-field frequency spectra are obtained when a simulation is performed by specifying a list of frequencies or conducting a frequency sweep. For each frequency, the far-field is calculated at various directions determined by the zenith (Theta) and azimuth (Phi) angular ranges, and the distance specified in the Far-Field panel of the Setup tabsheet. Therefore, you must select a fixed direction (Theta, Phi) to plot the far-field versus frequency.

Go to Results > **Plot Far-Field Spectrum** in the main menu to plot the far-field spectrum. This command will display the **Select Far-Field Point** dialog box (see Fig. 1), where you can select the fixed Theta and Phi angles. After clicking the OK button, the **AN-XY Chart** application will display the frequency spectrum of the **total E-field** (refer to Fig. 2).

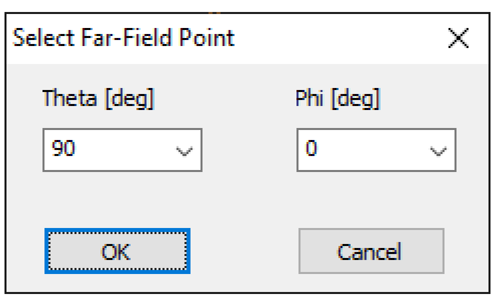


Fig. 1: Select Far-Field Point dialog box for selecting a fixed direction (Theta, Phi).

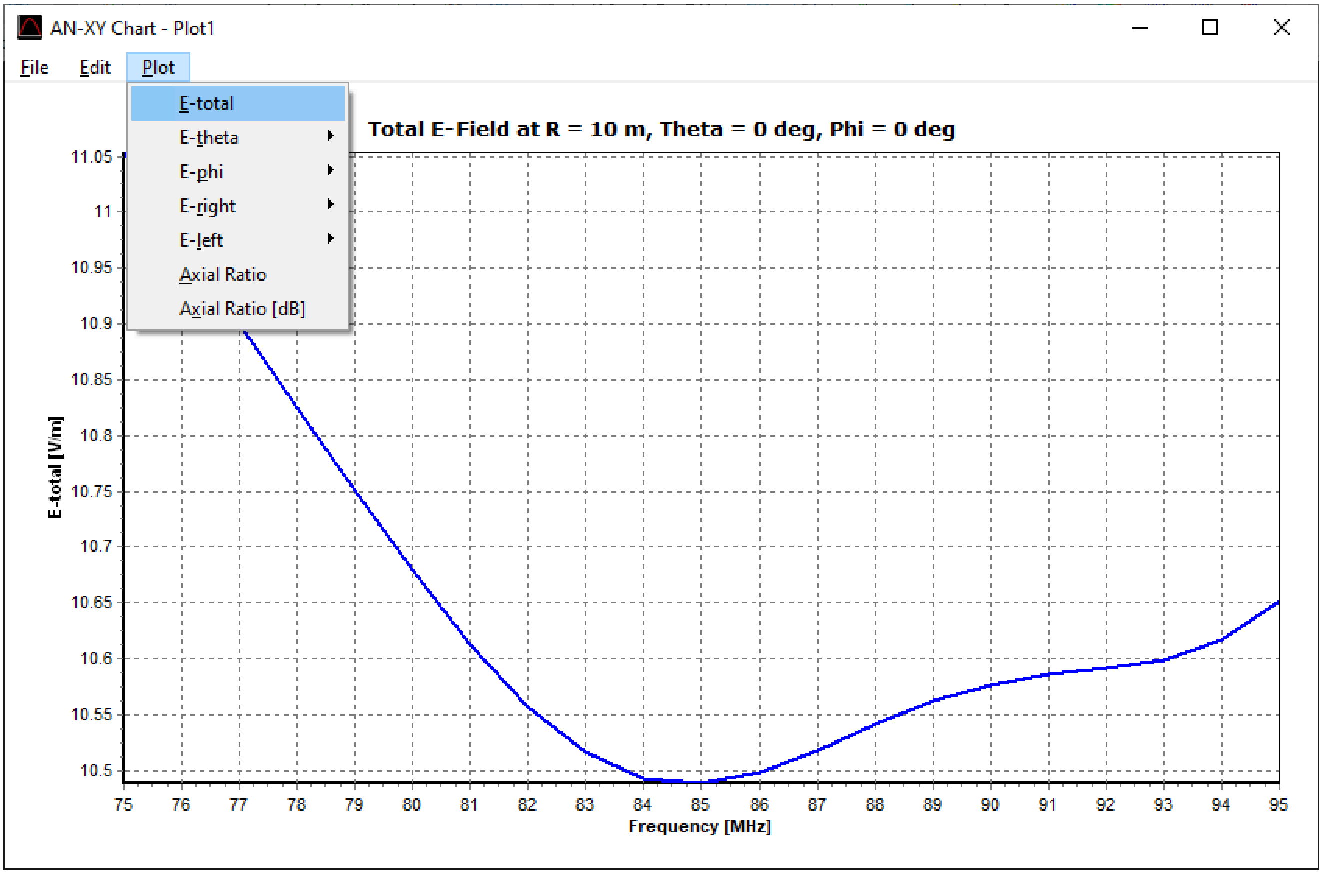


Fig. 2: Far-field frequency spectrum plotted by AN-XY Chart.

You can also plot the linearly polarized field components, **E-theta** and **E-phi**, as well as the circularly polarized components, **E-right** and **E-left**, in amplitude, phase, real, and imaginary parts by selecting these options under the **Plot** menu in the AN-XY Chart application. Additionally, you can plot the **Axial Ratio**, defined as the minor to the major axis ratio of the polarization ellipse, as a function of frequency.

The far-field spectrum for a selected far-field point can also be tabulated. To do this, go to Results > **List Far-Field Spectrum** in the AN-SOF main menu. This action will display the **Select Far-Field Point** dialog box where you can select fixed values for Phi and Theta. Afterward, a list of the far-field components versus frequency will be shown, and you can plot it by clicking the Plot button (as shown in Fig. 3).

Far-Field at Theta = 0 deg, Phi = 0 deg (Distance = 10 m)

No.	Frequency	E-total	Real (E-theta)	Imag (E-theta)	Abs (E-theta)
---	MHz	V/m	V/m	V/m	V/m
1	75	11.0537	2.01656	-7.19126	7.46864
2	76	10.9809	-0.926379	-7.5581	7.61466
3	77	10.9008	-3.67827	-6.73196	7.67131
4	78	10.8243	-5.82895	-4.9971	7.67774
5	79	10.7505	-7.16215	-2.70034	7.6543
6	80	10.679	-7.61123	-0.186291	7.61351
7	81	10.6126	-7.22544	2.24267	7.56548
8	82	10.5566	-6.1321	4.35106	7.51894
9	83	10.5158	-4.49955	5.9761	7.48062
10	84	10.4931	-2.50992	7.0194	7.45464
11	85	10.4883	-0.344887	7.43419	7.44219
12	86	10.4985	1.81943	7.21602	7.44186
13	87	10.5181	3.81495	6.39944	7.45028
14	88	10.5411	5.48688	5.05888	7.46312

Fig 3: Far-Field List showing the far-field components vs. frequency.

Power Budget

To access the **Power Budget** dialog box (see Fig. 1), go to Results > **Power Budget/RCS** in the main menu. The following list of parameters versus frequency is displayed when discrete sources are used for excitation:

The **Input Power** column shows the total input power provided by the discrete sources in the structure.

The **Radiated Power** column shows the total radiated power from the structure.

The **Structure Loss** column shows the total consumed power, representing ohmic losses in the structure.

The **Efficiency** column displays the radiated power-to-input power ratio. When the structure is lossless, it results in an efficiency of 100%.

The **Directivity** columns display the peak directivity, dimensionless and in decibels (dBi) with reference to an isotropic source.

The **Gain** columns display the peak gain, dimensionless and in decibels (dBi) with reference to an isotropic source.

The **Av. EIRP** (Effective Isotropic Radiated Power) columns display the **time-averaged EIRP** in Watts and dBW. This value is calculated by factoring in the **duty cycle** of the selected **transmit mode** in the **Tuner tab**, as well as the **Time Transmitting** percentage.

The **Peak EIRP** (Effective Isotropic Radiated Power) columns display the peak EIRP in Watts and dBW, calculated directly from the **Peak Envelope Power (PEP)**, without factoring in the duty cycle or time transmitting percentage.

The **Av. Power Density** column is the average power density. This value is calculated averaging the power density over all directions in space.

The **Peak Power Density** column is the maximum value of the radiated power density.

The **Theta (max)** and **Phi (max)** columns are the zenith and azimuth angles, respectively, in the direction of maximum radiation.

The **F/R H** and **F/B H** columns are the **front-to-rear** and **front-to-back ratios**, respectively, in a **horizontal slice** of the radiation pattern given by **Theta = Theta (max)**.

The **F/R V** and **F/B V** columns are the **front-to-rear** and **front-to-back ratios**, respectively, in a **vertical slice** of the radiation pattern given by **Phi = Phi (max)**.

The **Error** column is the error in the power balance of the system. A necessary, but not sufficient, condition for a model to be valid is that the input power must be equal to the sum of the radiated and lost powers, so the Error is defined as follows:

Error % = 100 x (Input – Lost – Radiated) Power / (Input – Lost) Power

The **Average Gain Test (AGT)** column represents a similar indicator to the **Error** column. To validate a model, AGT should be close to 1, as it is calculated using the formula:

AGT = (Radiated + Lost) Power / Input Power

Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency. Click on the **Export** button to export the list to a CSV file.

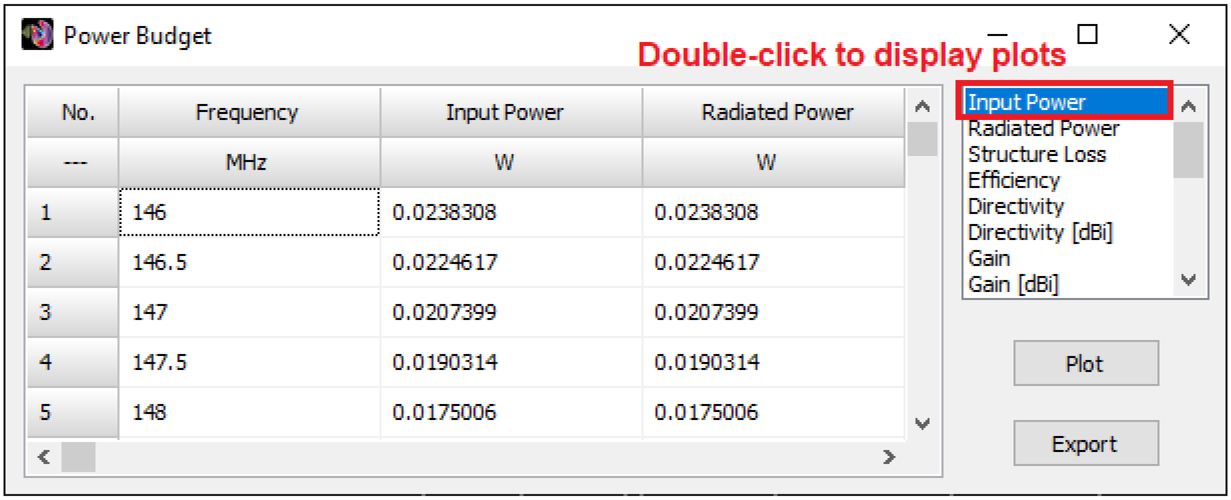


Fig. 1: The Power Budget dialog box.

> **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.
>
> **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

To access the **Radar Cross Section** dialog box (see Fig. 1), go to Results > **Power Budget/RCS** in the main menu. The following list of parameters versus frequency is displayed when an **incident field** is used for excitation:

The **RCS [m^2]** column shows the Radar Cross Section in square meters.

The **RCS [lambda^2]** column shows the Radar Cross Section in square wavelengths.

The **RCS [dBsw]** column shows the Radar Cross Section in decibels with reference to a square wavelength.

The **Radiated Power** column shows the **total scattered power** from the structure.

The **Structure Loss** column shows the total consumed power, representing ohmic losses in the structure.

The **Av. Power Density** column displays the average power density scattered from the structure. This value is computed by averaging the scattered power density over all directions in space.

The **Peak Power Density** column shows the maximum value of the scattered power density.

The **Theta (max)** and **Phi (max)** columns represent the zenith and azimuth angles, respectively, in the direction of maximum radiation.

Select an item from the list in the upper right corner of the window and then press the **Plot** button to plot the selected item versus frequency.

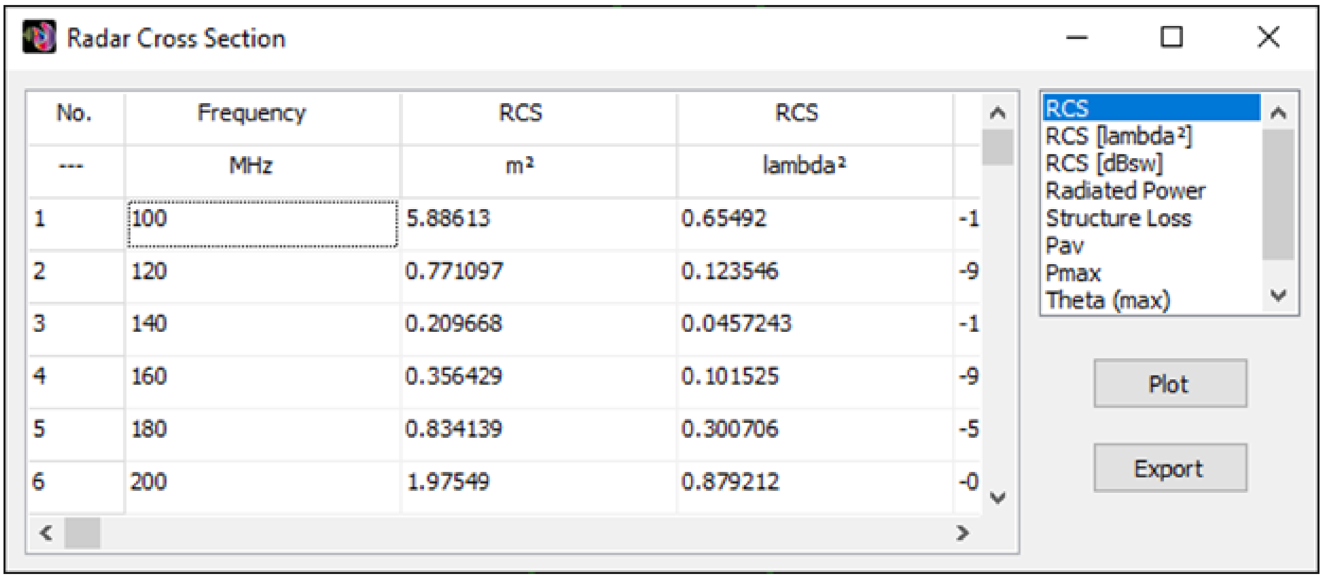


Fig. 1: The Radar Cross Section dialog box.

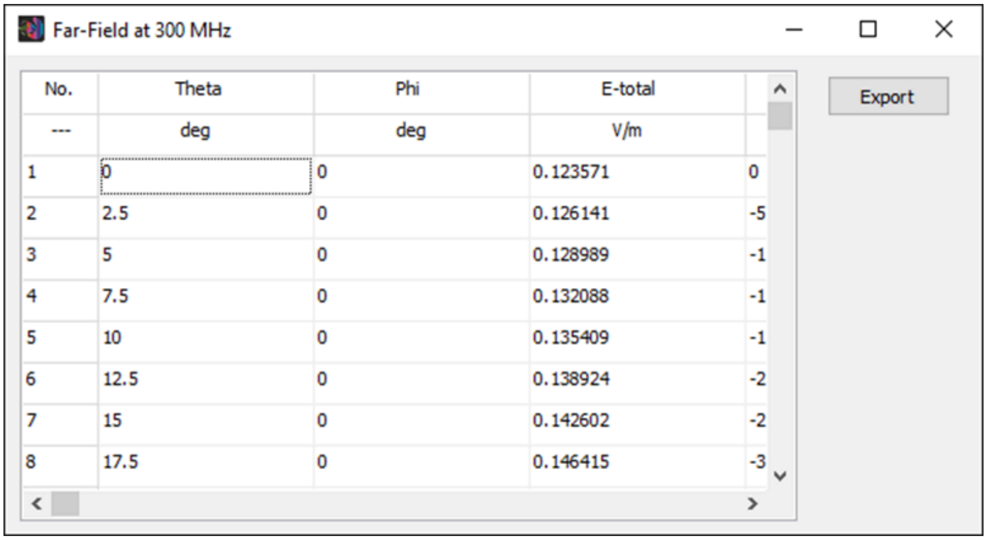
Exporting the Far Field

The far field patterns and spectra can be tabulated and exported by going to the following commands in the **Results menu >**:

List Far-Field Pattern

List Far-Field Spectrum

A table with the results will be displayed after executing any of these commands, Fig. 1. The tabulated values can be exported to a CSV (Comma Separated Values) file by clicking the **Export** button.



Far-Field at 300 MHz

Export

No.	Theta	Phi	E-total	
---	deg	deg	V/m	
1	0	0	0.123571	0
2	2.5	0	0.126141	-5
3	5	0	0.128989	-1
4	7.5	0	0.132088	-1
5	10	0	0.135409	-1
6	12.5	0	0.138924	-2
7	15	0	0.142602	-2
8	17.5	0	0.146415	-3

Fig. 1: Tabulated values of the far-field pattern. Click on the Export button to export the list to a CSV file.

Front-to-Rear and Front-to-Back Ratios: Applying Key Antenna Directivity Metrics

*T*wo commonly used metrics for quantifying the directional properties of an antenna radiation pattern are the **front-to-rear ratio (F/R)** and the **front-to-back ratio (F/B)**. Both F/R and F/B are crucial parameters for evaluating antenna performance, especially in applications requiring high directivity and low interference, such as point-to-point communication links and satellite systems.

> **F/R** is the ratio of the maximum power radiated by the antenna in the **forward direction** to the maximum power radiated in the **backward direction**. It indicates the antenna’s directional gain in the forward direction relative to its backward radiation. A high F/R signifies strong forward radiation and low backward radiation.
>
> **F/B** is the ratio of the maximum power radiated by the antenna in the **forward direction** to the power radiated in the **opposite direction**. It measures the power difference between the front and the directly opposing side of the antenna. A high F/B also implies strong forward radiation and low radiation in the opposite direction.

Both F/R and F/B are typically expressed in decibels (dB).

Metric	Definition
F/R (Worst-case Front-to-Back)	Ratio of maximum forward power to maximum backward power
F/B (180°-Front-to-Back)	Ratio of maximum forward power to power at 180 degrees

Definitions of Front-To-Rear and Front-To-Back Ratios.

Figure 1 illustrates the difference between F/R and F/B, assuming a 360-degree radiation pattern slice.

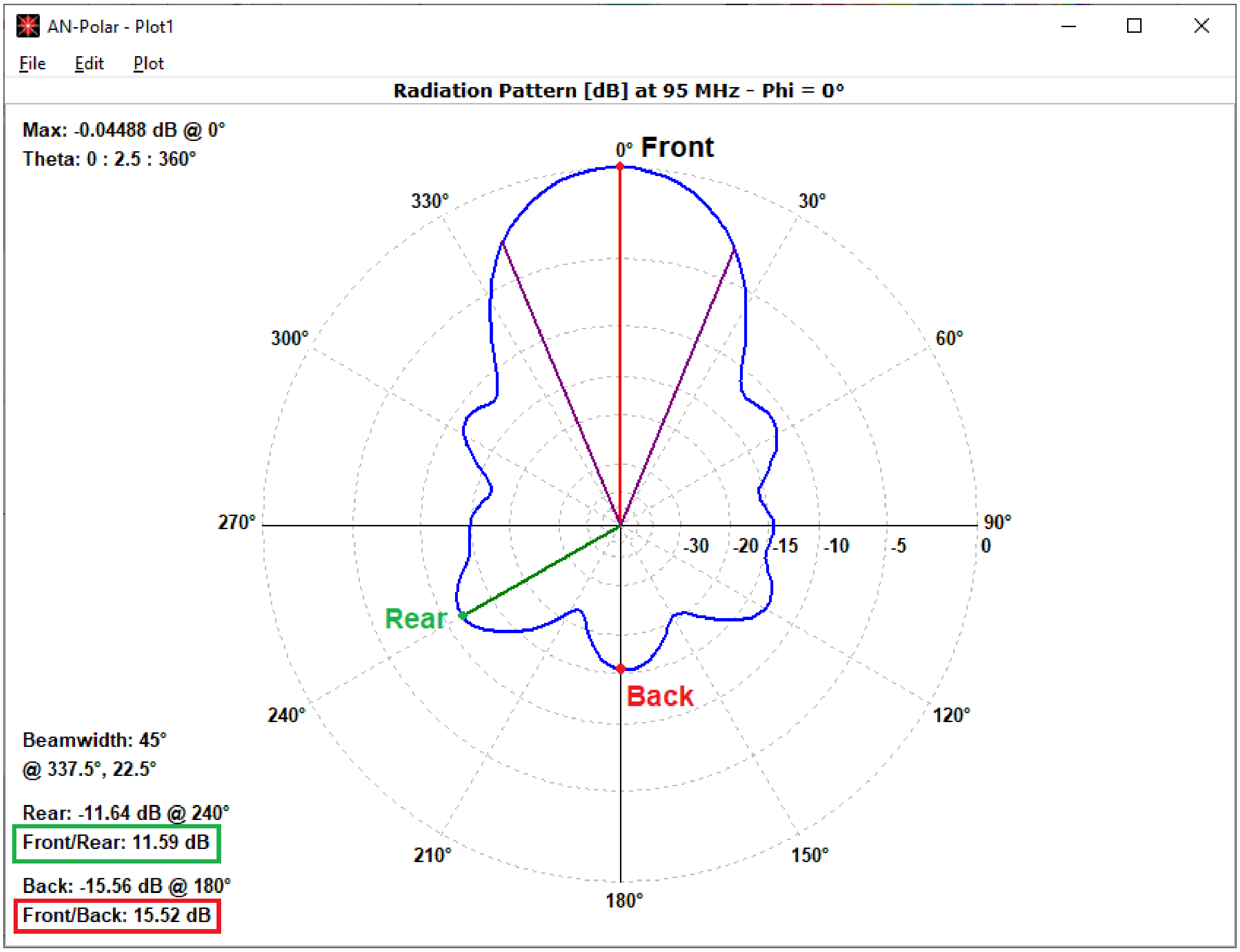


Fig. 1: Vertical slice of a radiation pattern in a polar diagram, illustrating the Front-to-Rear (F/R) and Front-to-Back (F/B) ratios.

In summary, the primary distinction between F/R and F/B lies in **the direction of backward radiation**. F/R compares the maximum forward power to the maximum backward power, while F/B compares the maximum forward power to the power radiated in the opposite direction.

These definitions are applicable to both **horizontal (θ = const.)** and **vertical (φ = const.)** radiation patterns in **free space**. However, the presence of a **ground plane** introduces complexities. For **horizontal** patterns, F/R and F/B calculations remain unchanged as the angular range spans **360 degrees**. Conversely, for **vertical** patterns, the angular range is limited to **180 degrees**. In this case, F/R is redefined as the **front-to-side ratio**, comparing the maximum signal to the maximum signal in the opposite quadrant (as depicted in Fig. 2). F/B becomes irrelevant due to the absence of a 'back' direction for an infinite ground plane, resulting in a zero value from AN-SOF.

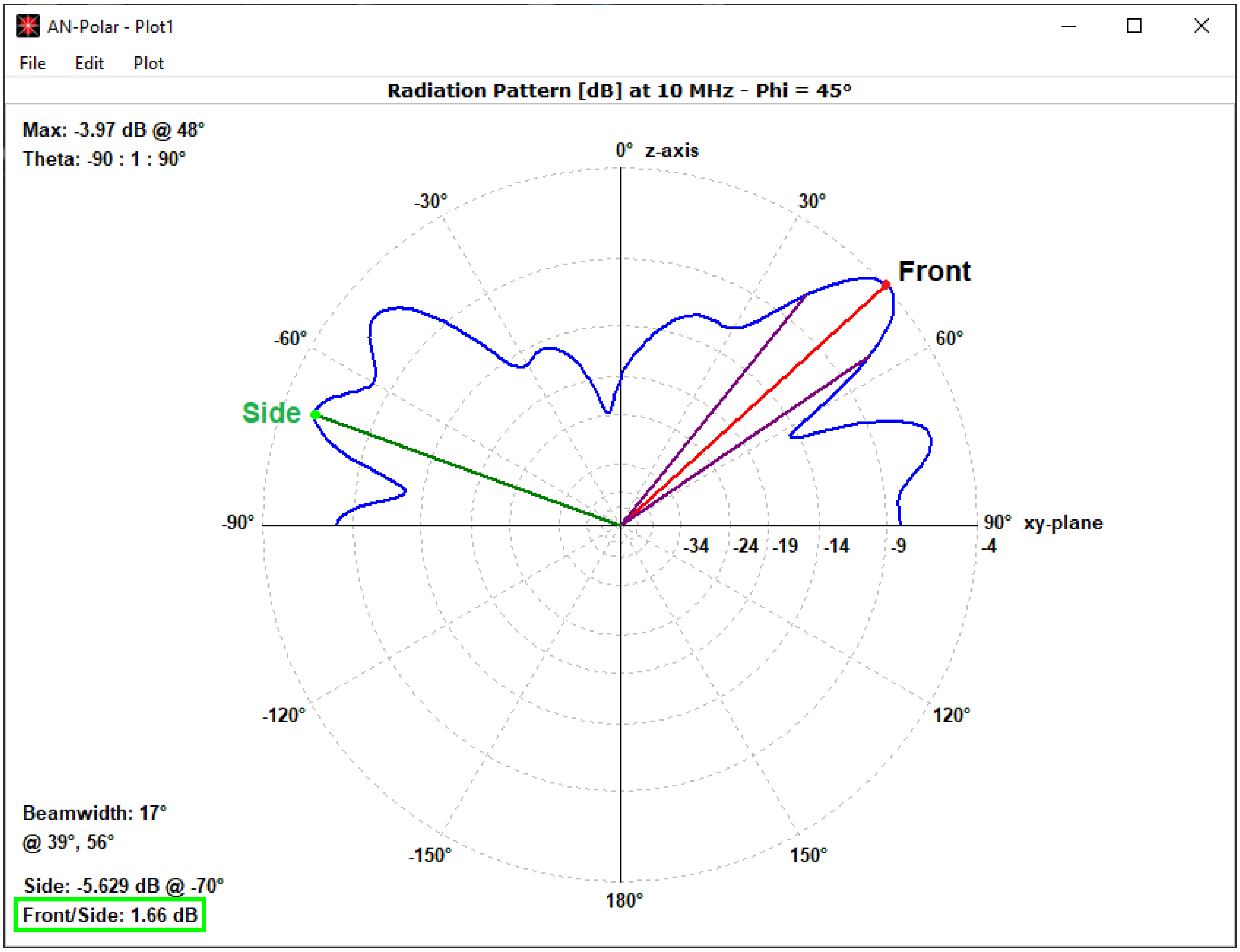


Fig. 2: Definition of Front-to-Side ratio for a vertical pattern in a polar plot when there is a ground plane.

Understanding F/R and F/B is crucial for effective antenna design. The **Results tab** in the AN-SOF main window presents F/R and F/B values in dB as a function of frequency for both vertical (V) and horizontal (H) radiation pattern slices. The **Plots tab** offers a visual comparison of F/R and F/B over the frequency range.

> **Note**:
>
> To ensure proper calculations of F/R and F/B, select the **Full 3D**, **Vertical** or **Horizontal** options in the **Far-Field panel**.
>
> Selecting the **Custom** option in the Far-Field panel will lead to variations in the calculation of F/R and F/B as they will depend on the specific angular ranges that have been configured.
>
> *Golden Engineering*

Near Field

Plotting Near Field Patterns

The grid of points where the near field is calculated can be specified in the **Near-Field panel** of the **Setup tab**. There, the points can be entered in **Cartesian**, **Cylindrical**, or **Spherical Coordinates**. The near electric (**E**) and magnetic (**H**) fields **can be calculated separately**. Of course, the near fields can be calculated in any region of an antenna, very close to it or far away. In the far-field region, the near fields will tend to the known behavior of far-fields: **E** and **H** are perpendicular to each other and perpendicular to the radial direction from the antenna, they oscillate in phase, and their magnitudes have a constant ratio: $E/H \approx 377$ Ohms (often also approximated as 120π Ohms) in free space. This behavior can be verified by performing calculations of the "near" E and H fields far from an antenna.

When both E and H fields have been calculated, the power density (S) will also be available in tables and plots. The total rms power density is calculated as $S = |\mathbf{E} \times \mathbf{H}^*|$. This metric is particularly important for assessments to evaluate **electromagnetic field compliance** with radiation exposure limits published by regulatory authorities.

To plot the near electric field as a 3D graph with a color scale, go to **Results > Plot Near E-Field Pattern > 3D Plot** in the main menu. This command executes the **AN-3D Pattern** application (Fig. 1). To display a 3D plot of the near magnetic field or power density, respectively, go to **Results > Plot Near H-Field Pattern > 3D Plot** or **Results > Plot Power Density Pattern > 3D Plot**.

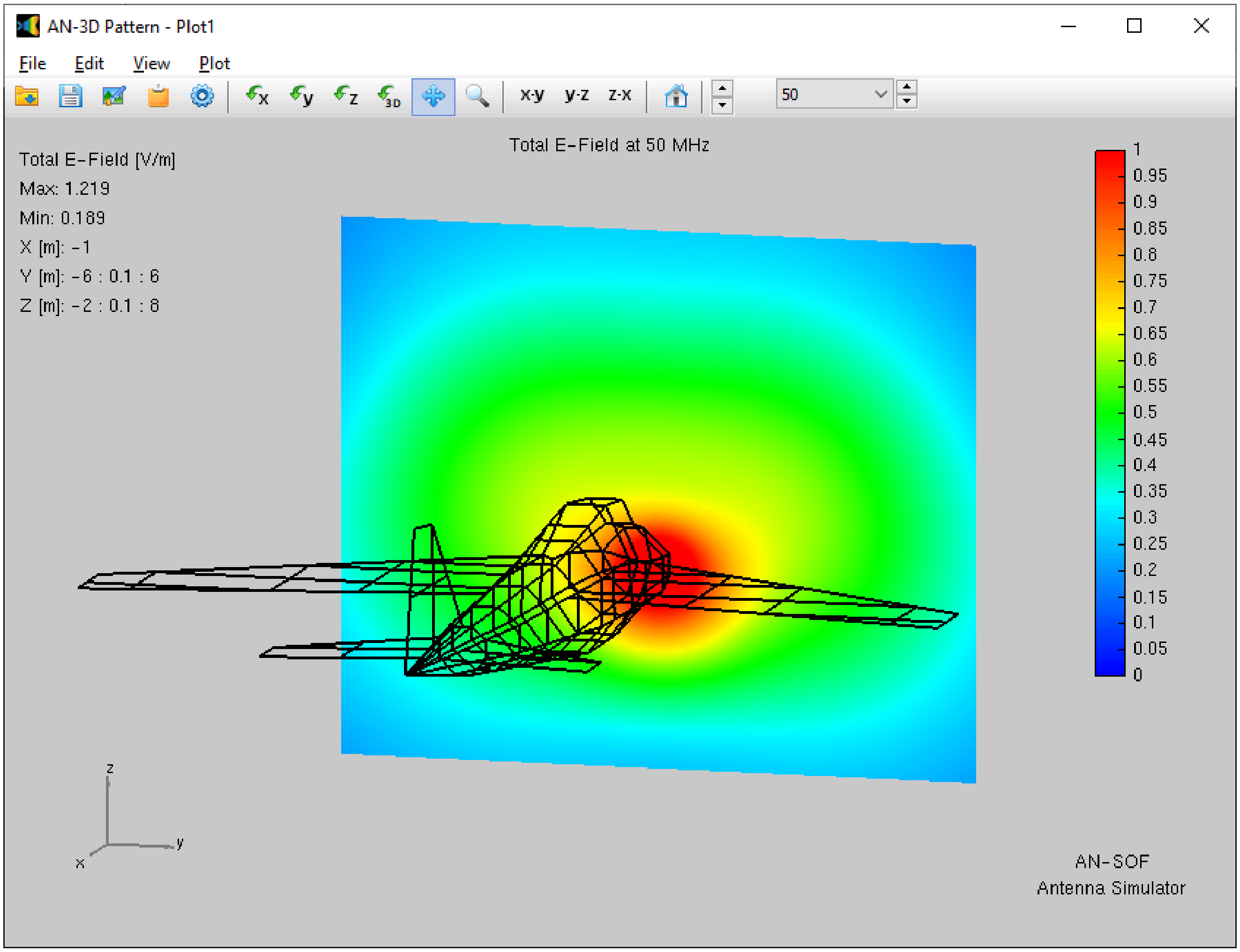


Fig. 1: 3D plot of the near E-field in the AN-3D Pattern application just in front of an aircraft receiving a vertically polarized plane wave from behind.

Near-field 3D plots will be shown according to the type of coordinate system chosen in the **Near-Field panel** of the **Setup** tab: **Cartesian**, **Cylindrical**, or **Spherical**. If near-fields were calculated for more than one frequency, a dialog box asking for a fixed frequency will be shown before plotting the near-field pattern.

The near electric field can also be plotted as a 2D rectangular plot by going to **Results > Plot Near E-Field Pattern > 2D Plot** in the main menu. The near magnetic field can be plotted by going to **Results > Plot Near H-Field Pattern > 2D Plot**, and the power density by going to **Results > Plot Power Density Pattern > 2D Plot**. These commands execute the **AN-XY Chart** application, where the total rms electric field, magnetic field, or power density is plotted in a 2D chart (Fig. 2). The components of the near E and H fields can be plotted individually by going to the **Plot** menu in the **AN-XY Chart** and selecting the desired component.

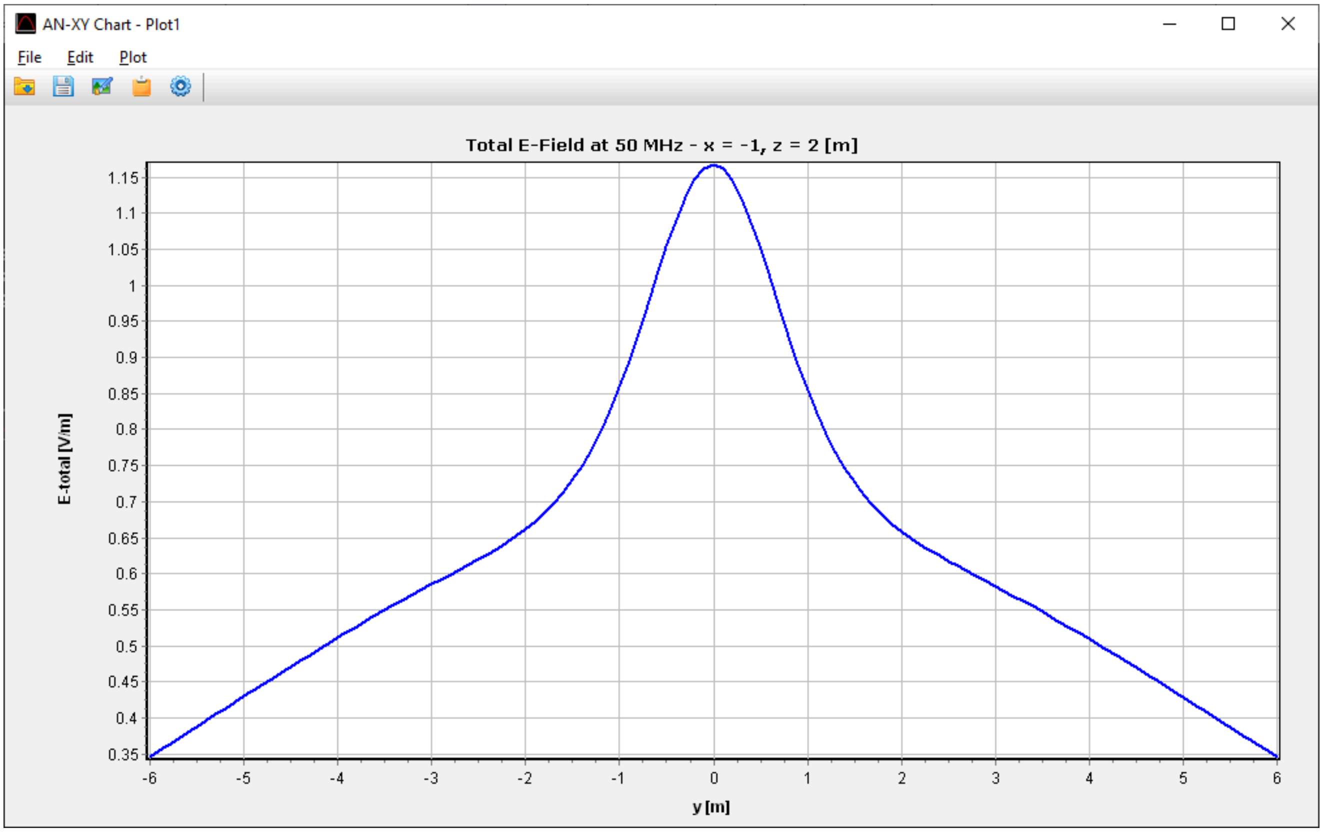


Fig. 2: Near E-field plotted in AN-XY Chart as a function of the y-coordinate corresponding to the horizontal line that passes just in front of the nose of the aircraft in Fig. 1.

The near-field patterns for a given frequency can also be tabulated by going to **Results > List Near E-Field Pattern**, **Results > List Near H-Field Pattern**, or **Results > List Power Density Pattern** in the AN-SOF main menu.

Regarding the E and H Field Components

> If **Cartesian coordinates** have been set in the **Near-Field** panel of the **Setup** tab, the $\mathbf{E_x}$, $\mathbf{E_y}$, and $\mathbf{E_z}$ electric field components and the $\mathbf{H_x}$, $\mathbf{H_y}$, and $\mathbf{H_z}$ magnetic field components will be calculated in a rectangular grid of points in space with coordinates (x, y, z).
>
> If **Cylindrical coordinates** have been set in the **Near-Field** panel of the **Setup** tab, the $\mathbf{E_r}$, $\mathbf{E_{phi}}$, and $\mathbf{E_z}$ electric field components and the $\mathbf{H_r}$, $\mathbf{H_{phi}}$, and $\mathbf{H_z}$ magnetic field components will be calculated in a cylindrical grid of points in space with coordinates (r, phi, z).
>
> If **Spherical coordinates** have been set in the **Near-Field** panel of the **Setup** tab, the $\mathbf{E_r}$, $\mathbf{E_{theta}}$, and $\mathbf{E_{phi}}$ electric field components and the $\mathbf{H_r}$, $\mathbf{H_{theta}}$, and $\mathbf{H_{phi}}$ magnetic field components will be calculated in a spherical grid of points in space with coordinates (r, theta, phi).

Plotting the Near Field Spectrum

Near-field frequency spectra are obtained when a simulation is performed by specifying a list of frequencies or a frequency sweep. For each frequency, the near field is calculated at the points specified in the **Near-Field panel** of the **Setup** tab. Therefore, a fixed point in space must be selected to plot the near field versus frequency (the near field spectrum).

To plot the near E-field, near H-field, or power density spectrum, go to **Results > Plot Near E-Field Spectrum**, **Results > Plot Near H-Field Spectrum**, or **Results > Plot Power Density Spectrum** in the main menu. These commands display the **Select Near-Field Point** dialog box, where a fixed observation point can be selected (Fig. 1). The **AN-XY Chart** application will then show the frequency spectrum of the selected field (Fig. 2). The E and H field components can be plotted in amplitude, phase, real, and imaginary parts by choosing these options under **Plot** in the **AN-XY Chart** main menu.

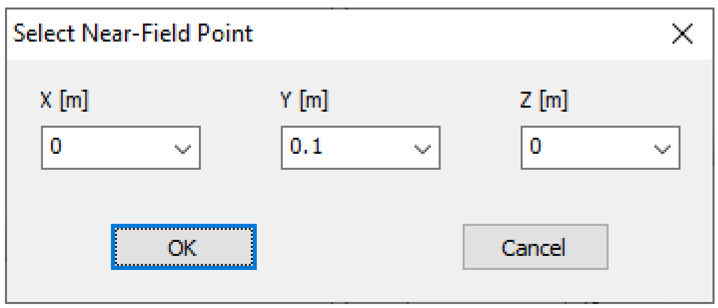


Fig. 1: Select Near-Field Point dialog box for selecting a fixed observation point.

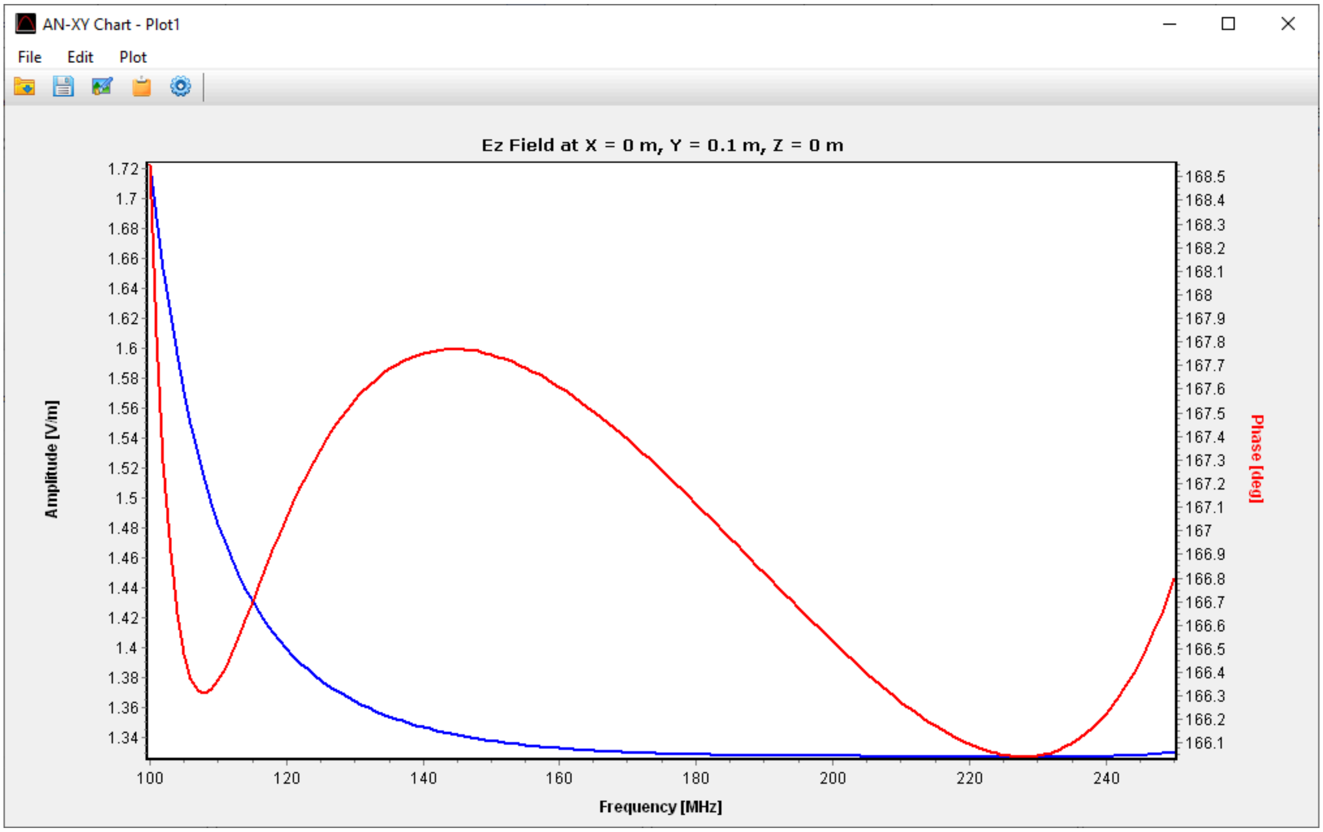


Fig. 2: Near E-field spectrum in amplitude and phase plotted in AN-XY Chart.

Exporting the Near Field

Near field patterns and spectra can be tabulated and exported by going to the following commands in the **Results menu >**:

List Near E-Field Pattern

List Near E-Field Spectrum

List Near H-Field Pattern

List Near H-Field Spectrum

A table with the results will be displayed after executing any of these commands. The tabulated values can be exported to a CSV (Comma Separated Values) file by clicking the **Export** button.

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.

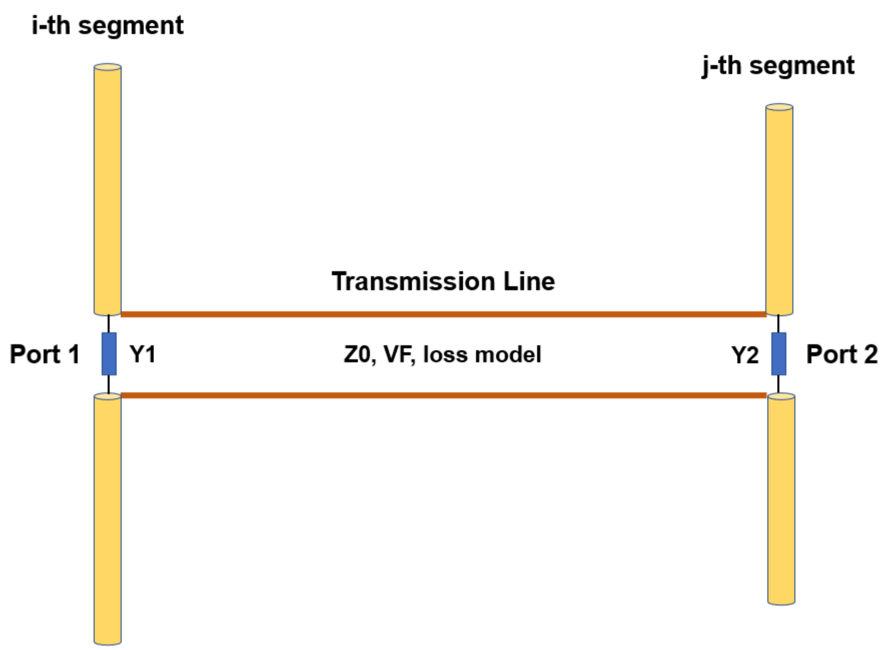


Fig. 1: A transmission line connected between two wire segments. It is defined by its characteristic impedance, Z0, velocity factor, VF, a loss model, and shunt admittances Y1 and Y2.

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.

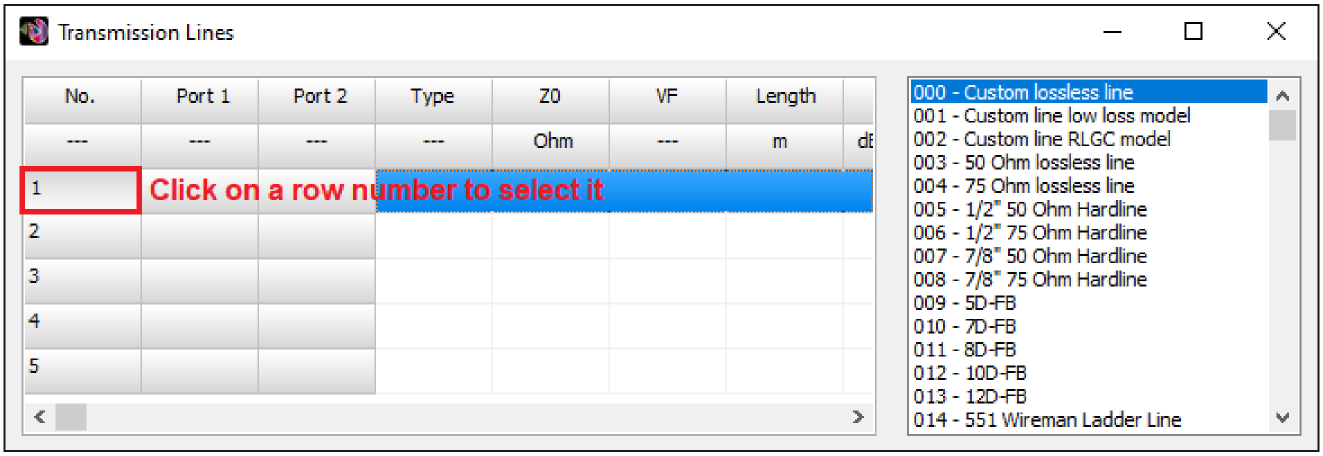


Fig. 2: Table for entering transmission lines. Rows are numbered. Click on a row number to select the entire row.

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.

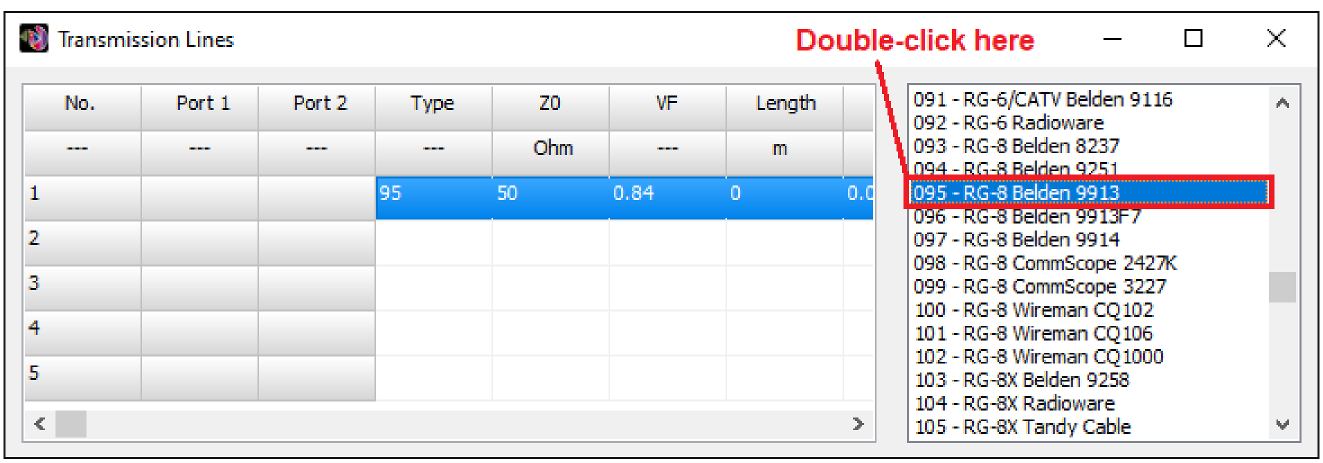


Fig. 3: On the right panel, double-click on the chosen line type to automatically complete the selected row.

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**.

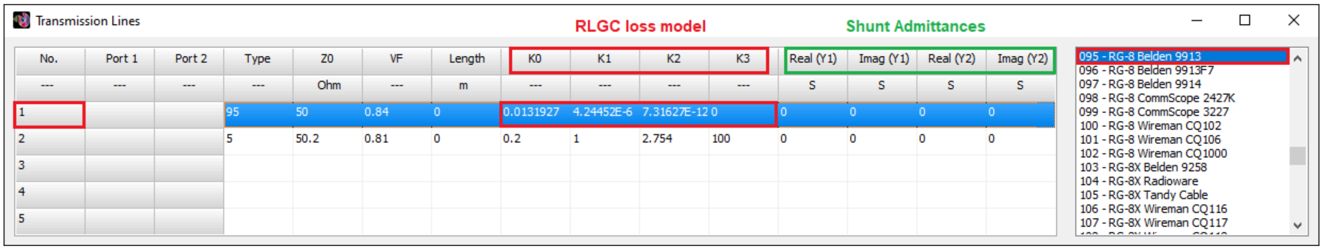


Fig. 4: Enlarge the Transmission Lines window to view the loss model parameters and shunt admittances. In this example, the K0, K1, K2, and K3 columns of the RLGC model are displayed since a line has been chosen whose attenuation curve is adjusted to this model.

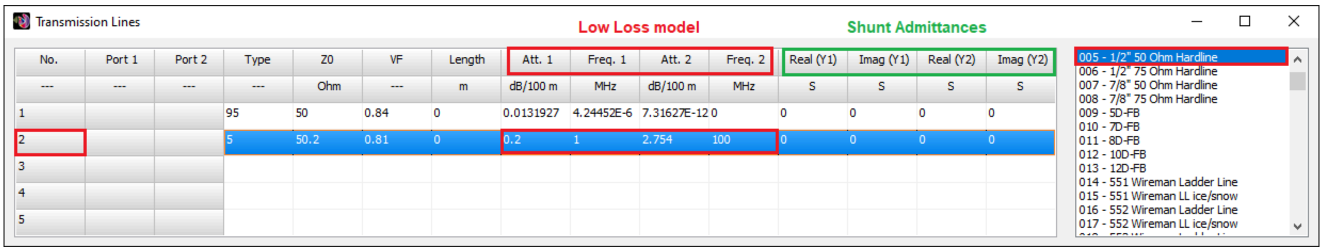


Fig. 5: Enlarge the Transmission Lines window to view the loss model parameters and shunt admittances. In this example, the Att. 1, Freq. 1, Att. 2, and Freq. 2 columns of the low loss model are displayed since a line has been chosen whose attenuation curve is adjusted to this model.

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.

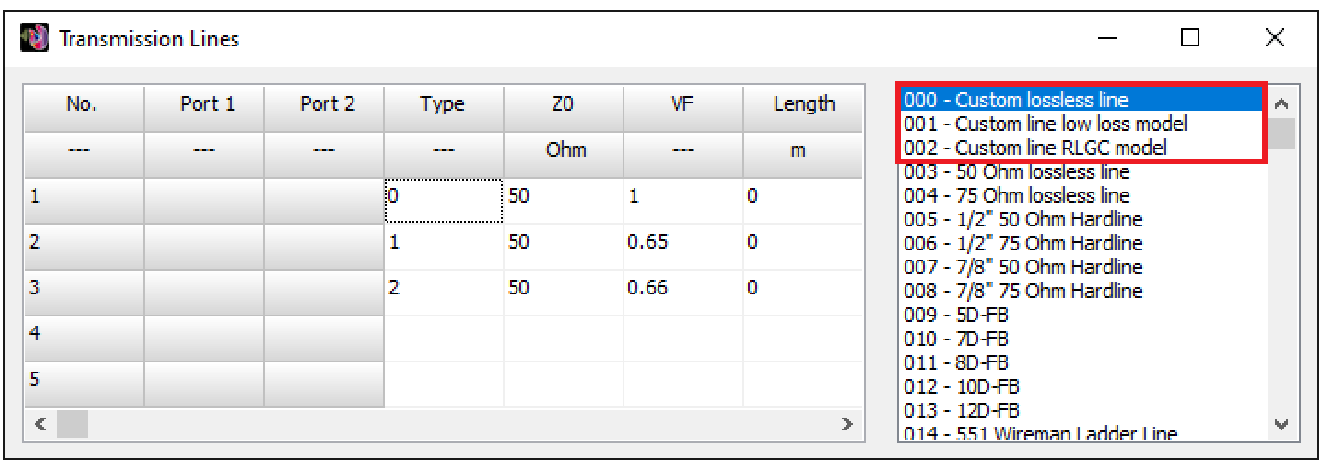


Fig. 1: The first three types of transmission lines, types 0, 1, and 2, are customizable lines.

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.

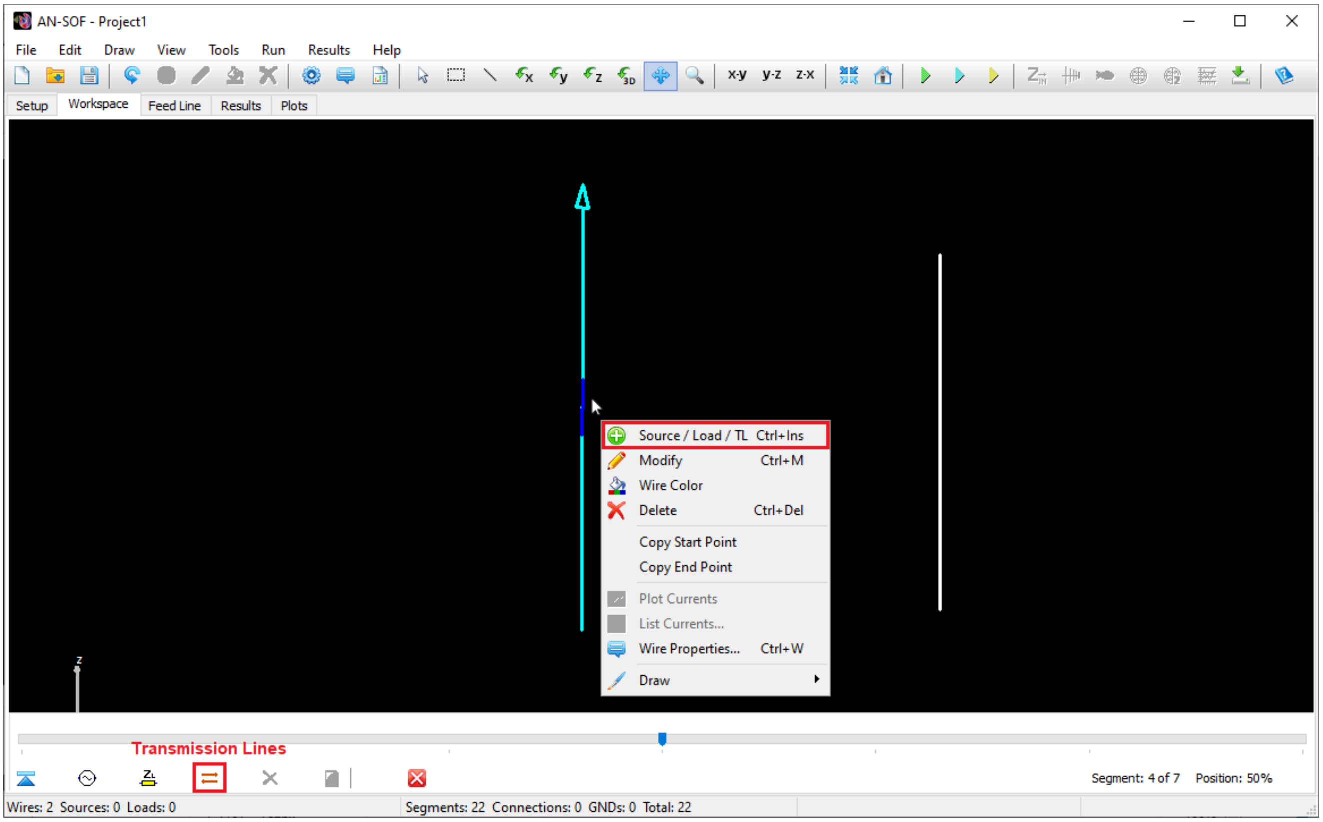


Fig. 1: To display the horizontal toolbar at the bottom of the workspace window, right-click on a wire and choose the "Source / Load / TL" command from the pop-up menu that appears. Then, move the slider to select a segment.

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.

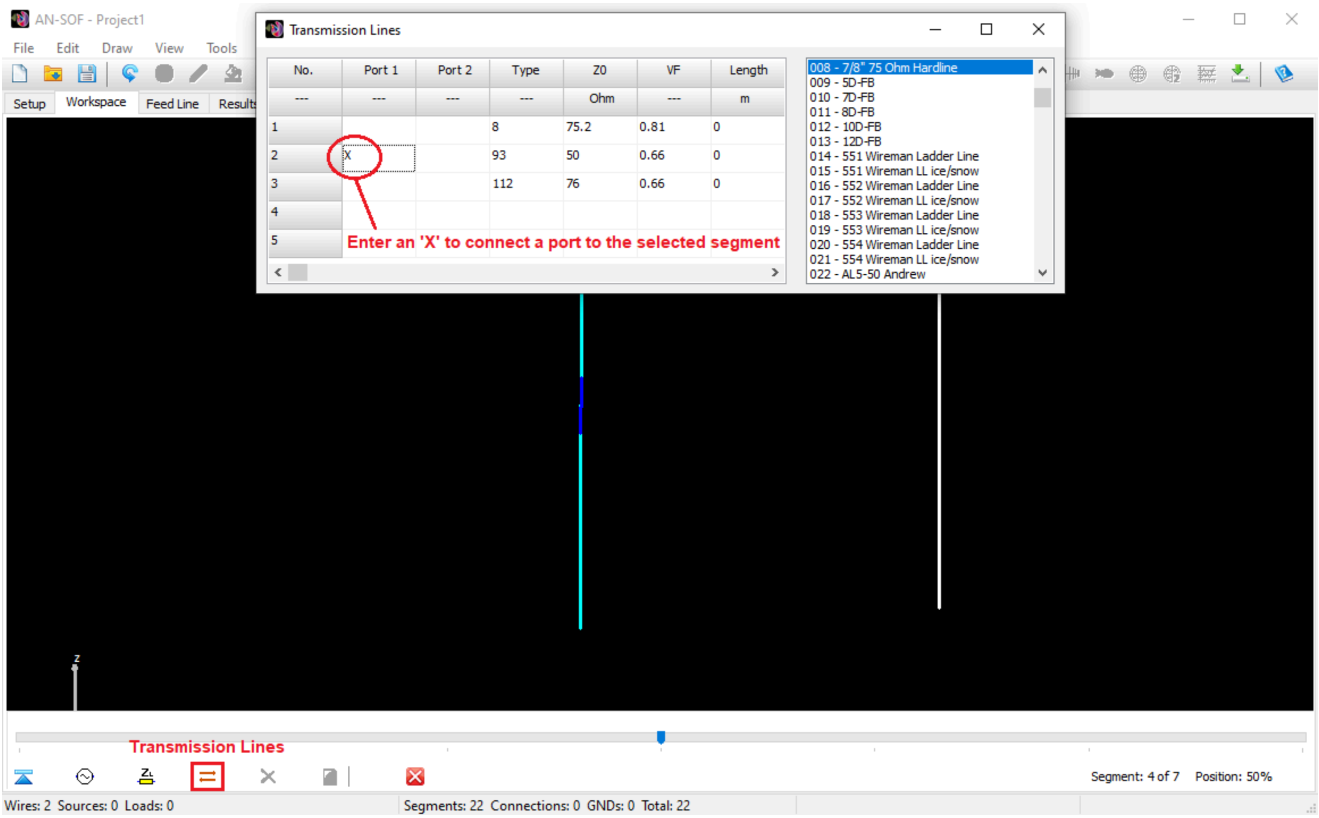


Fig. 2: To display the Transmission Lines window, click on the “Transmission Lines” button located on the horizontal toolbar. Then, enter the letter “X” (in either lowercase or uppercase, without quotes) in the cells of the ports that you wish to connect to the selected segment.

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.

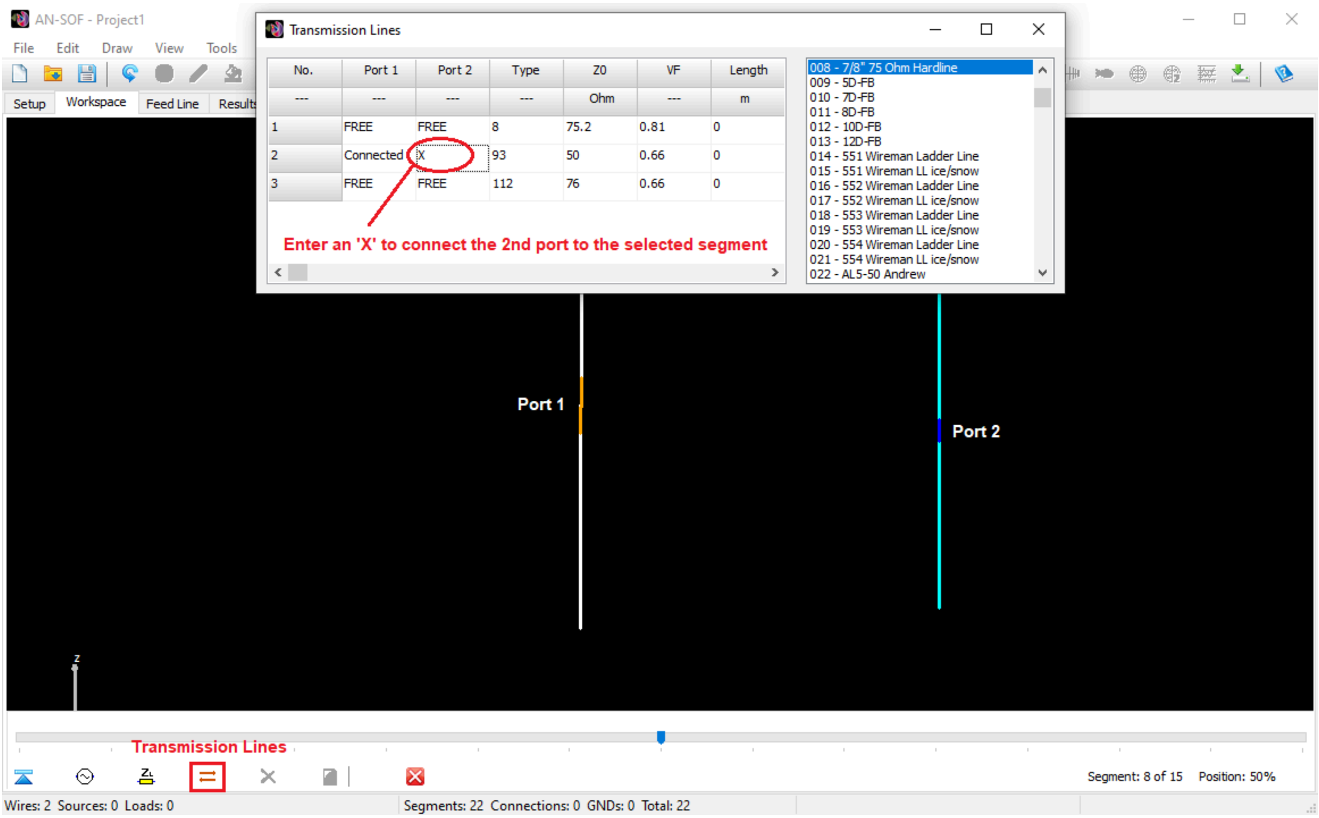


Fig. 3: Select the second segment and enter an 'X' in the ports that you want to connect there.

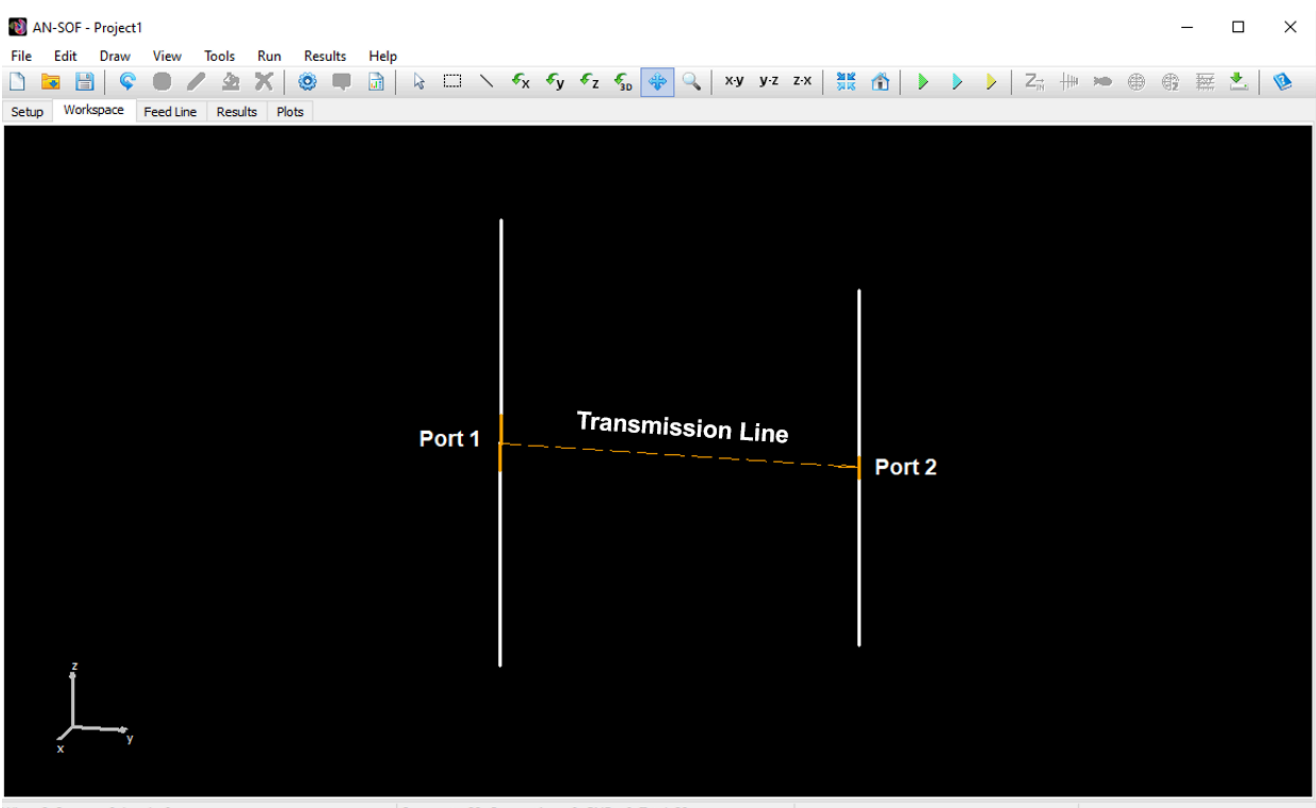


Fig. 4: Transmission line in the workspace, connecting two segments located on different wires.

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.

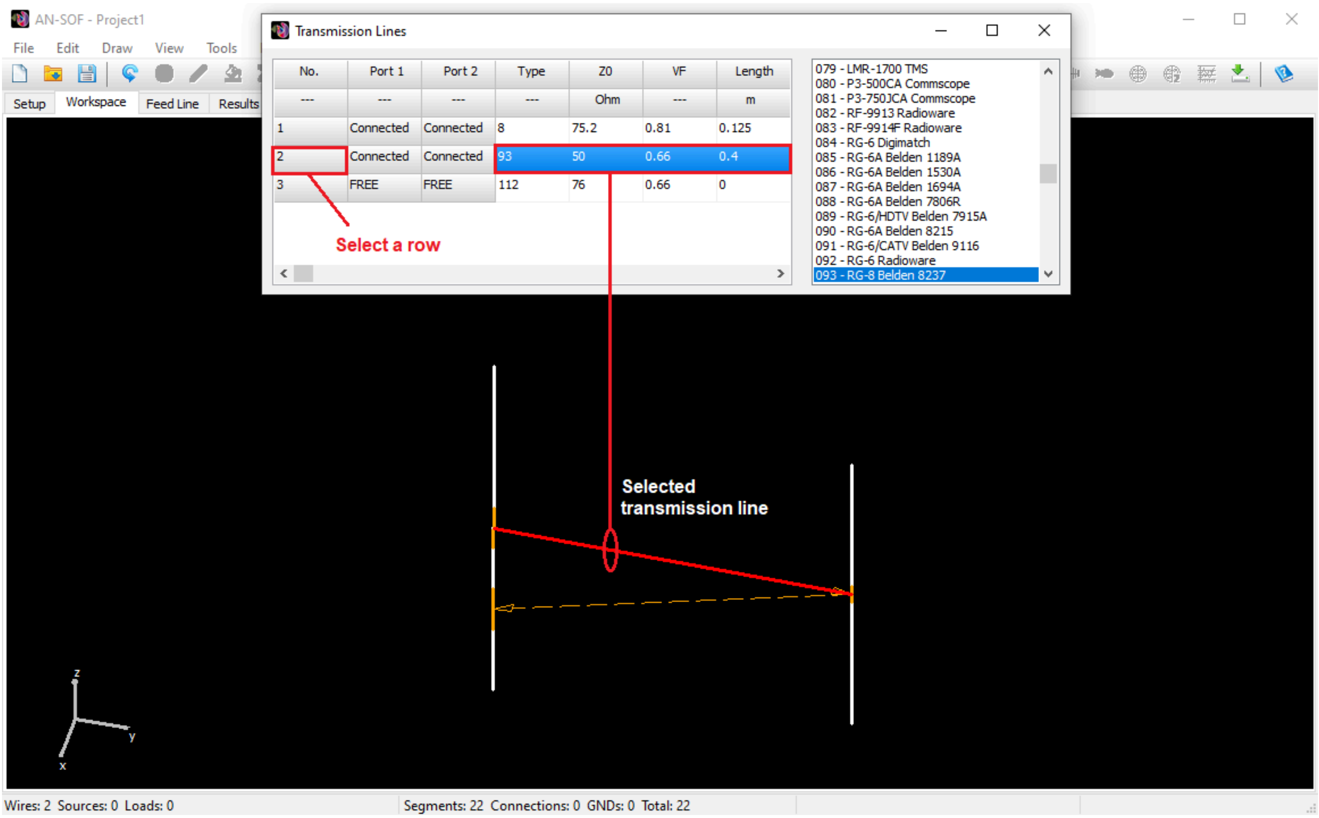


Fig. 5: By selecting a row in the Transmission Lines table, you can easily identify the corresponding transmission line in the workspace as it will be highlighted in red.

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.

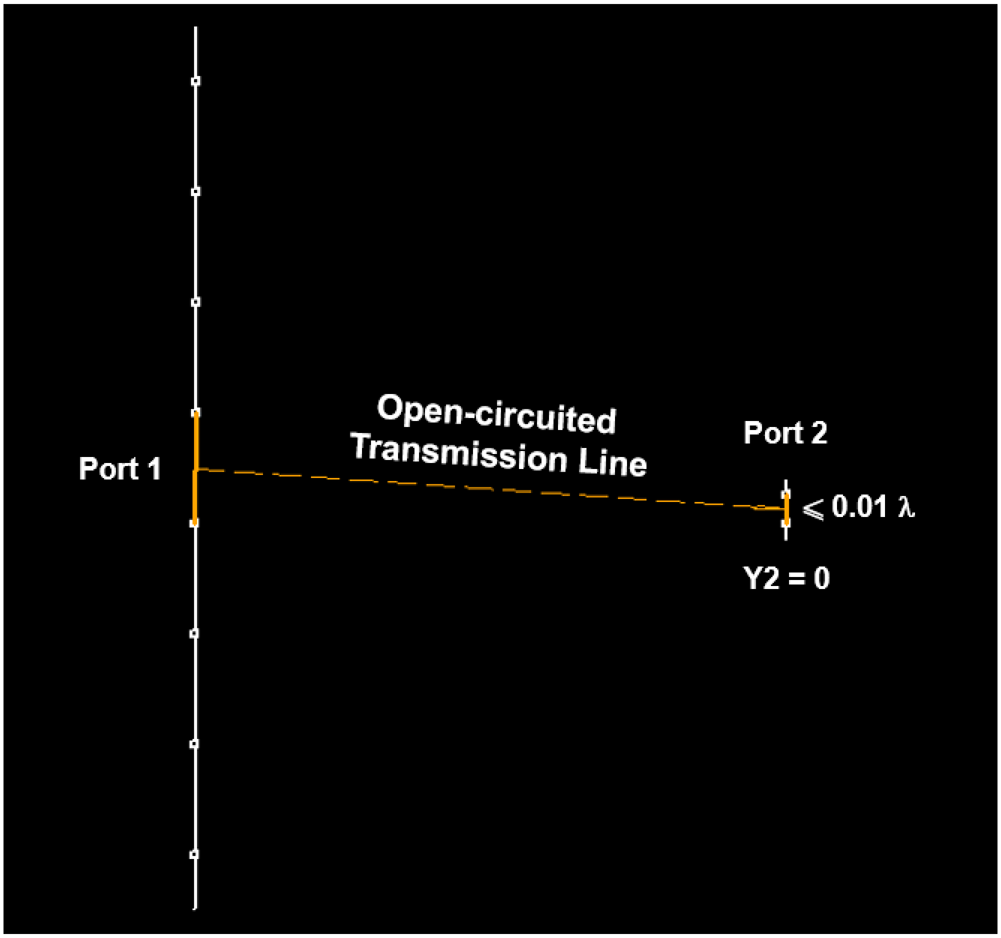


Fig. 1: Open circuit transmission line. The port that is open circuited is connected to a short wire that has only one segment and has a null shunt admittance.

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.

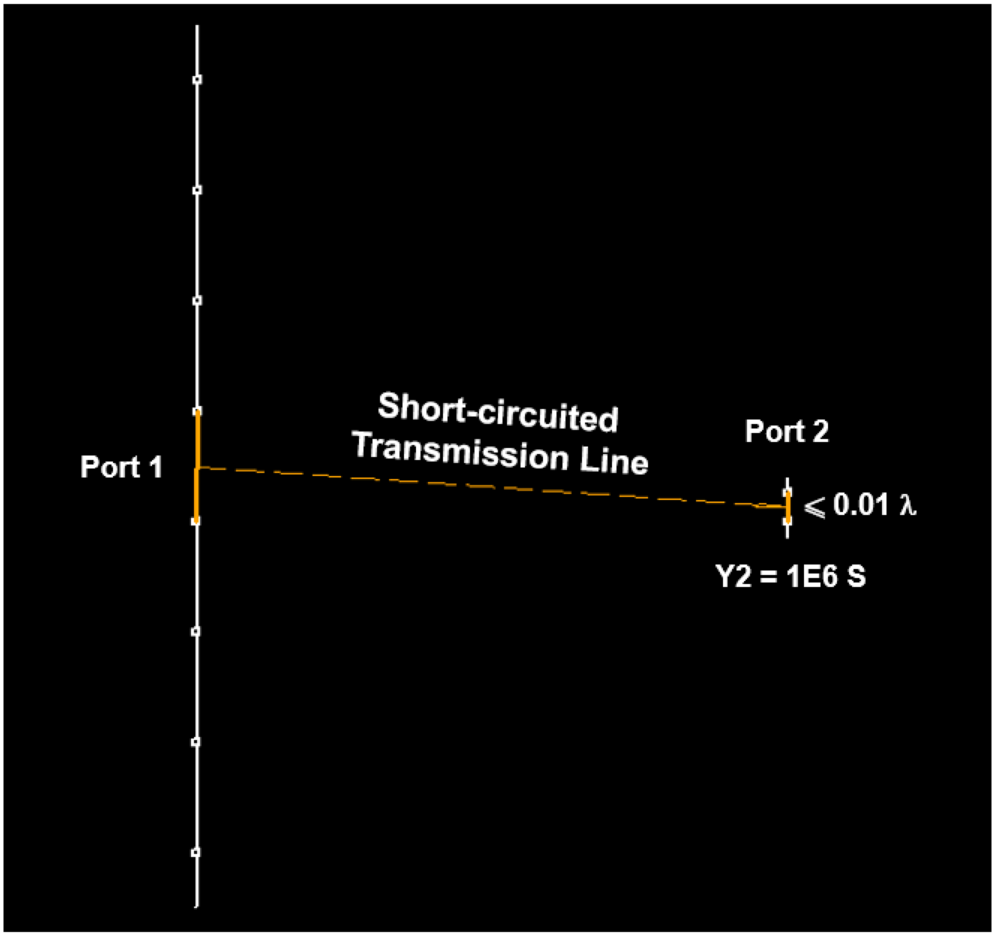


Fig. 2: Short circuited transmission line. The port that is short circuited is connected to a short wire that has only one segment and has a large shunt admittance.

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:

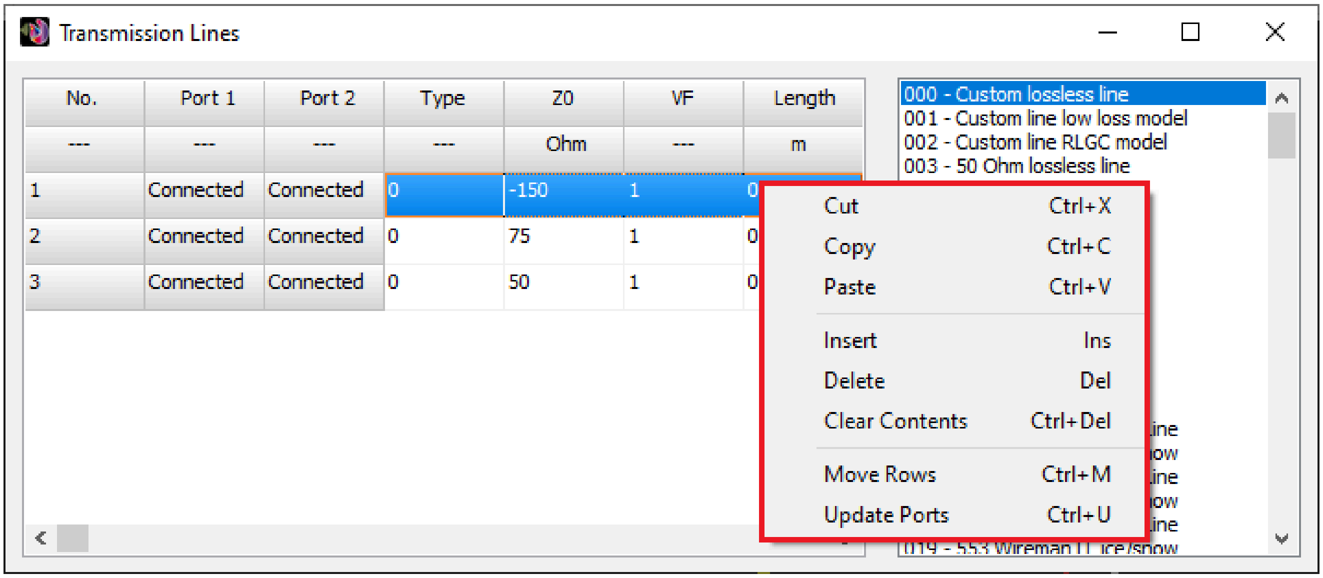


Fig. 1: Pop-up menu in the Transmission Lines table.

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.

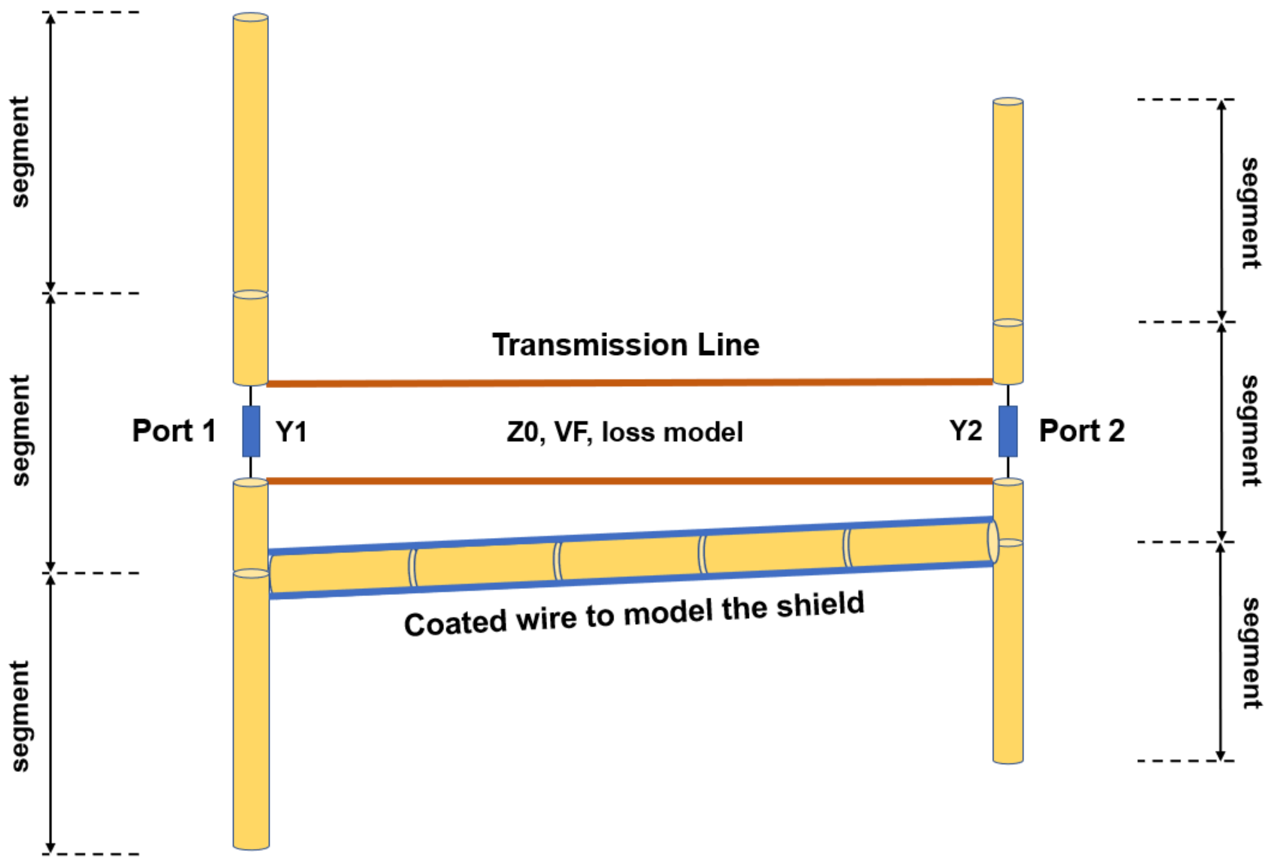


Fig. 1: An additional wire is connected between two ends of the ports of a coaxial transmission line to model the cable shield. This wire needs to be divided into segments just like the rest of the structure.

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.

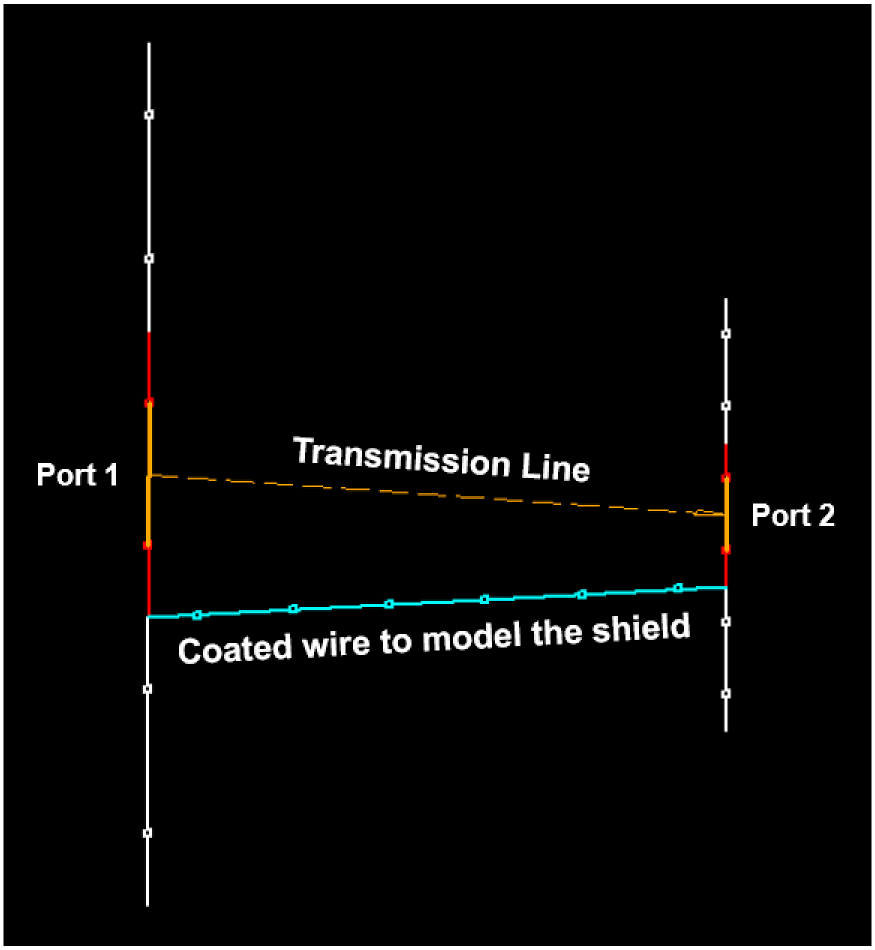


Fig. 2: Visualization of the sketch in Fig. 1 in the AN-SOF workspace. The short wires, each consisting of one segment, that are required to connect the wire representing the coaxial cable shield are highlighted in red.

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.

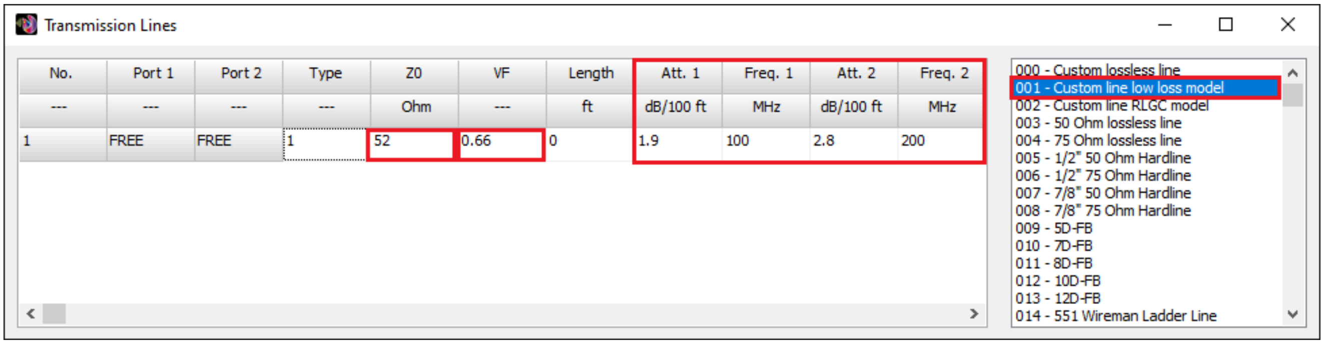


Fig. 1: Entering the values of nominal characteristic impedance, velocity factor, and attenuation for a Belden 8237 cable, type RG-8/U, when the frequency sweep of the simulation is within the range of 100 to 200 MHz.

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 Examples

In the directory where AN-SOF was installed there is a folder called “Examples” which contains many examples of antennas and wire structures. The default directory is

```
C:\AN-SOF X\Examples
```

where X is the AN-SOF version.

You can also download the examples from **here >**.

We constantly upload files with examples on our website. You will find downloadable examples on our **Resources** and **Blog** pages.

At the bottom of our website there are **Categories** and a **Search** bar to facilitate the search for information.

We also invite you to subscribe to our **Newsletter here >** and to follow us on our social media channels.

From **this link** you can download 5 examples of antenna models that have less than 50 segments, so the calculations can be run with the trial version of AN-SOF:

- 2 Element Quad
- 2 Element Delta Loop
- HF Skeleton Slot
- Inverted V

- 5 Element Yagi-Uda

Download 5 Examples
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.

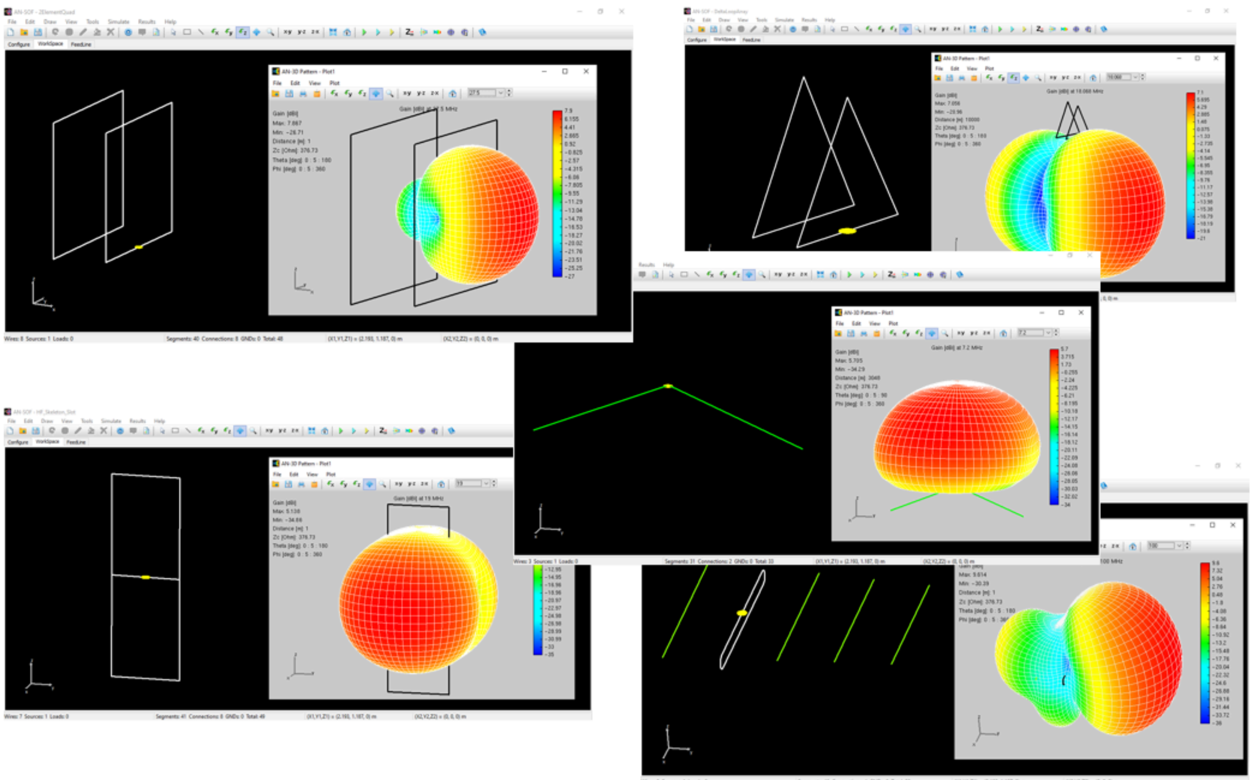


Illustration of 5 antenna models with less than 50 segments in AN-SOF Trial Version.

The trial version of **AN-SOF** is **fully-featured** and **never expires**. It allows users to open all **pre-calculated example files** to view tables and display various graphs and plots. The only limitation is that it can run calculations with up to **50 “unknowns”**. An unknown refers to the electric current value to be determined by the **AN-SOF calculation engine** in each segment, segment-to-segment **connection**, and a connection to a **ground plane**, if any. Therefore, the total number of unknown currents equals the number of segments + number of connections + number of connections to ground. This number must not exceed 50 to run a calculation in AN-SOF Trial version.

The purpose of the trial version is to evaluate the AN-SOF features and capabilities for antenna modeling or design projects. The pre-calculated models can be found in the AN-SOF “Examples” folder typically located in the installation directory, such as **C:\AN-SOF**

X\Examples, where "X" represents the version of the program. Additionally, many model examples with descriptive articles can be found in the **Models** section of our Knowledge Base. These models are categorized according to the antenna type, ranging from simple wire antennas to antennas in complex environments.

For more complex antennas, the 50 unknowns limit may be quickly exceeded. Modifications to pre-calculated examples with more than 50 segments + connections + ground connections cannot be re-run with the trial version of AN-SOF. However, for simple antenna projects or **small antenna sizes in terms of the wavelength**, the trial version can be a useful tool for simulations.

Download the following 5 examples 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 achieve reliable results, **at least 10 segments per wavelength of wire** should be used in a model. For antennas sensitive to element lengths, like Yagis, about 50 segments per wavelength should provide results comparable to VSWR measurements.

Explore more examples and articles in the **Validation** section of our Knowledge Base. Additionally, AN-SOF trial version includes embedded **tuner** and **feeder** calculators, allowing users to synthesize impedance matching networks, add transformers, and calculate tuner and feed line parameters for measured or given load impedance.

In conclusion, AN-SOF Trial Version offers a comprehensive platform for antenna simulation, enabling users to evaluate its features and capabilities for their projects. With access to pre-calculated examples and embedded tools like tuner and feeder calculators, users can explore antenna designs with ease.

See Also:

Complete Workflow: Modeling, Feeding, and Tuning a 20m Band Dipole Antenna
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**.

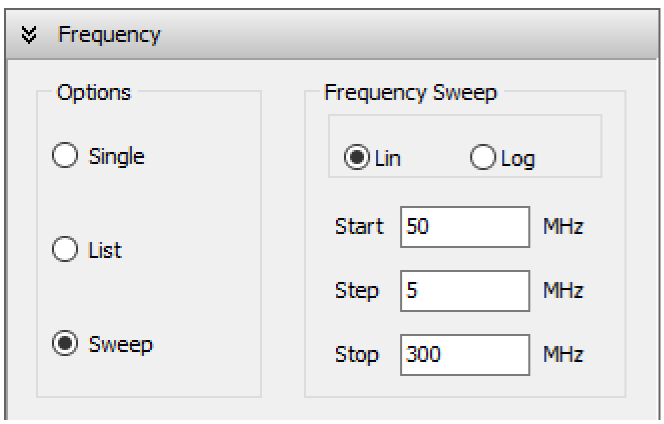


Fig. 1: Frequency sweep parameters setup.

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.

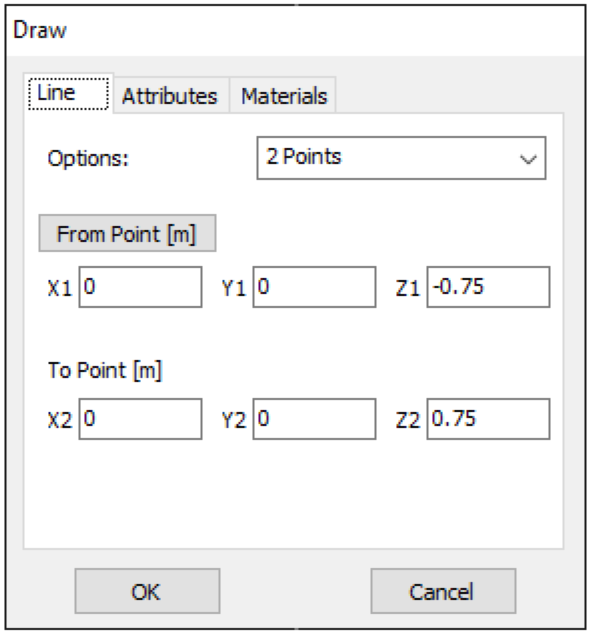


Fig. 2: Line dialog box for defining the antenna geometry.

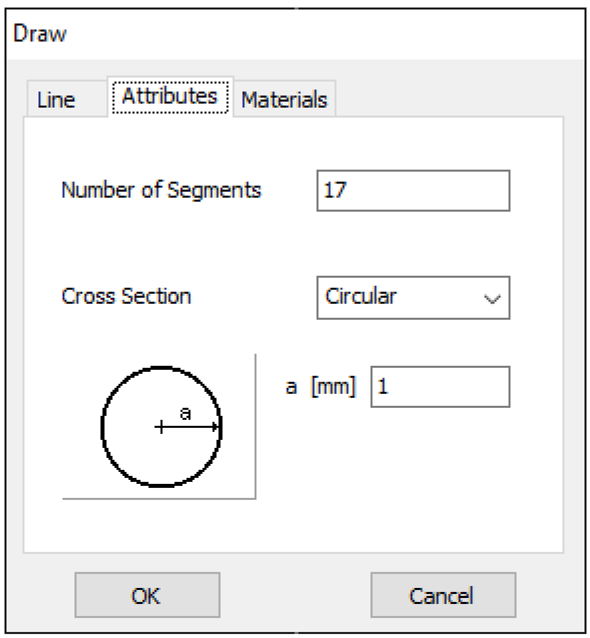


Fig. 3: Line attributes configuration.

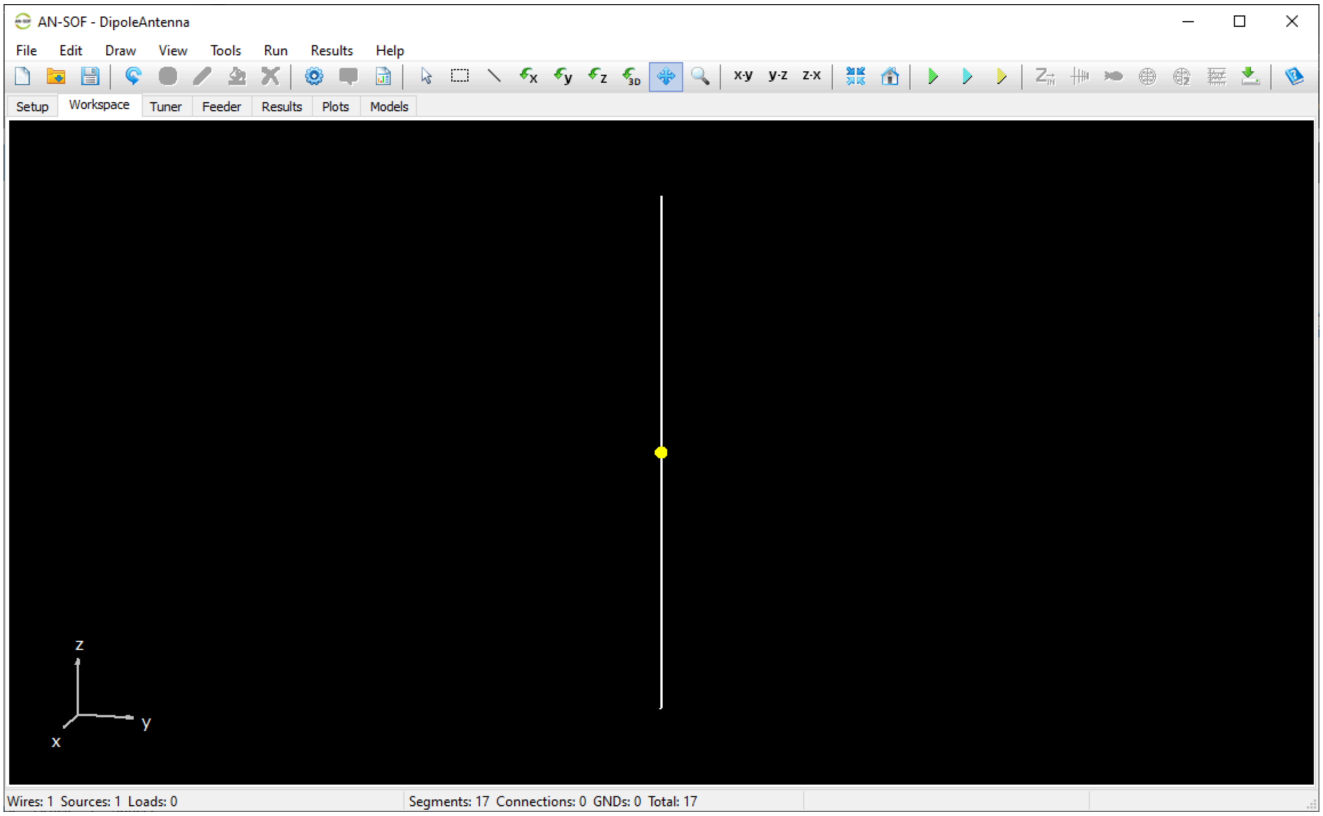


Fig. 4: Center-fed cylindrical antenna geometry.

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.

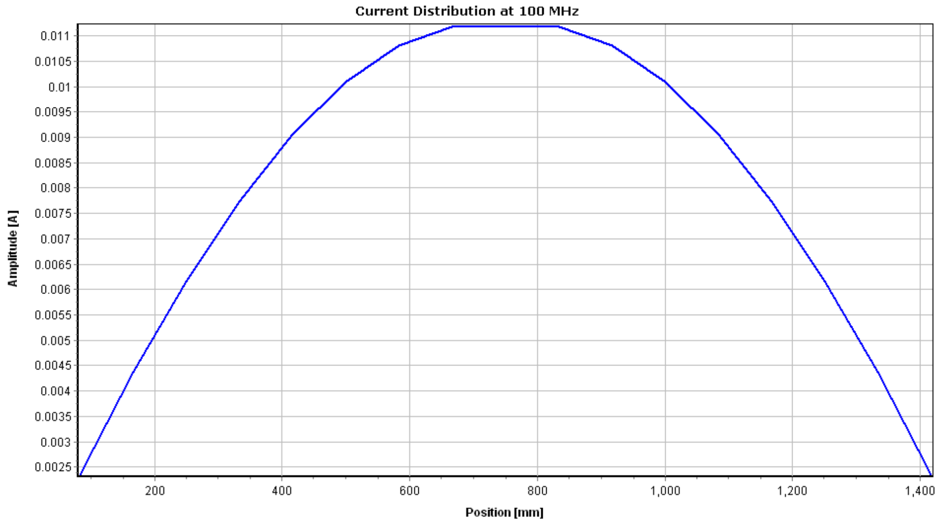


Fig. 5(a): Amplitude of the current distribution along the cylindrical antenna at 100 MHz.

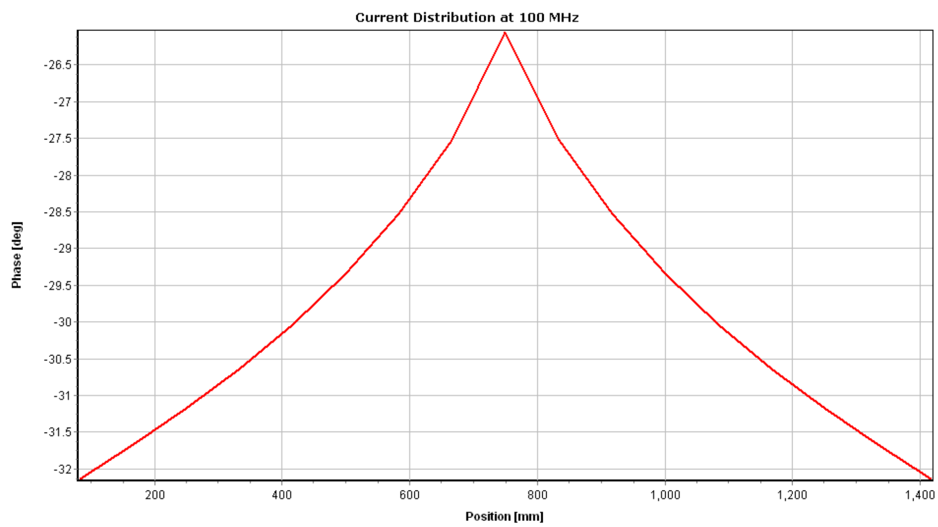


Fig. 5(b): Phase of the current distribution along the cylindrical antenna at 100 MHz.

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.

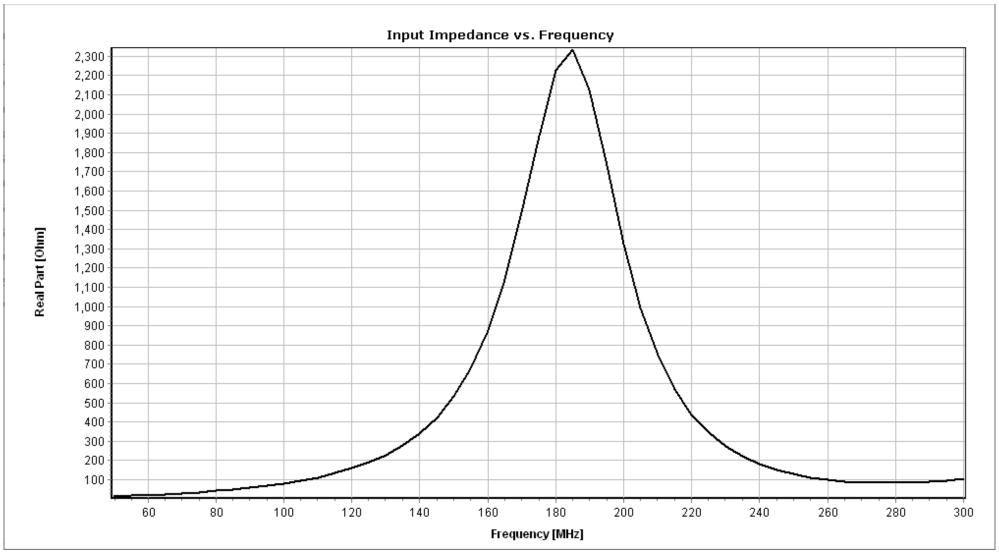


Fig. 6(a): Real part of the input impedance vs. frequency.

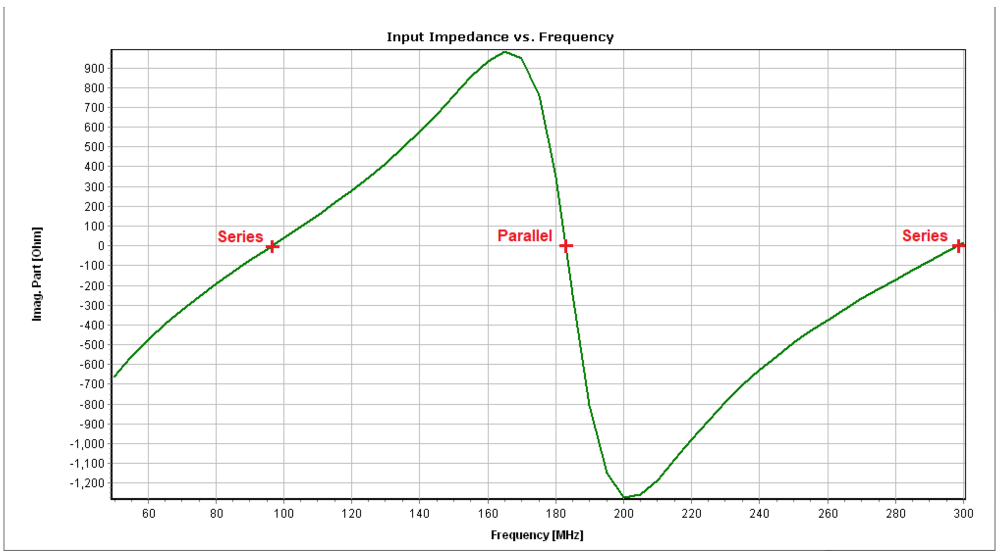


Fig. 6(b): Imaginary part of the input impedance vs. frequency.

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.

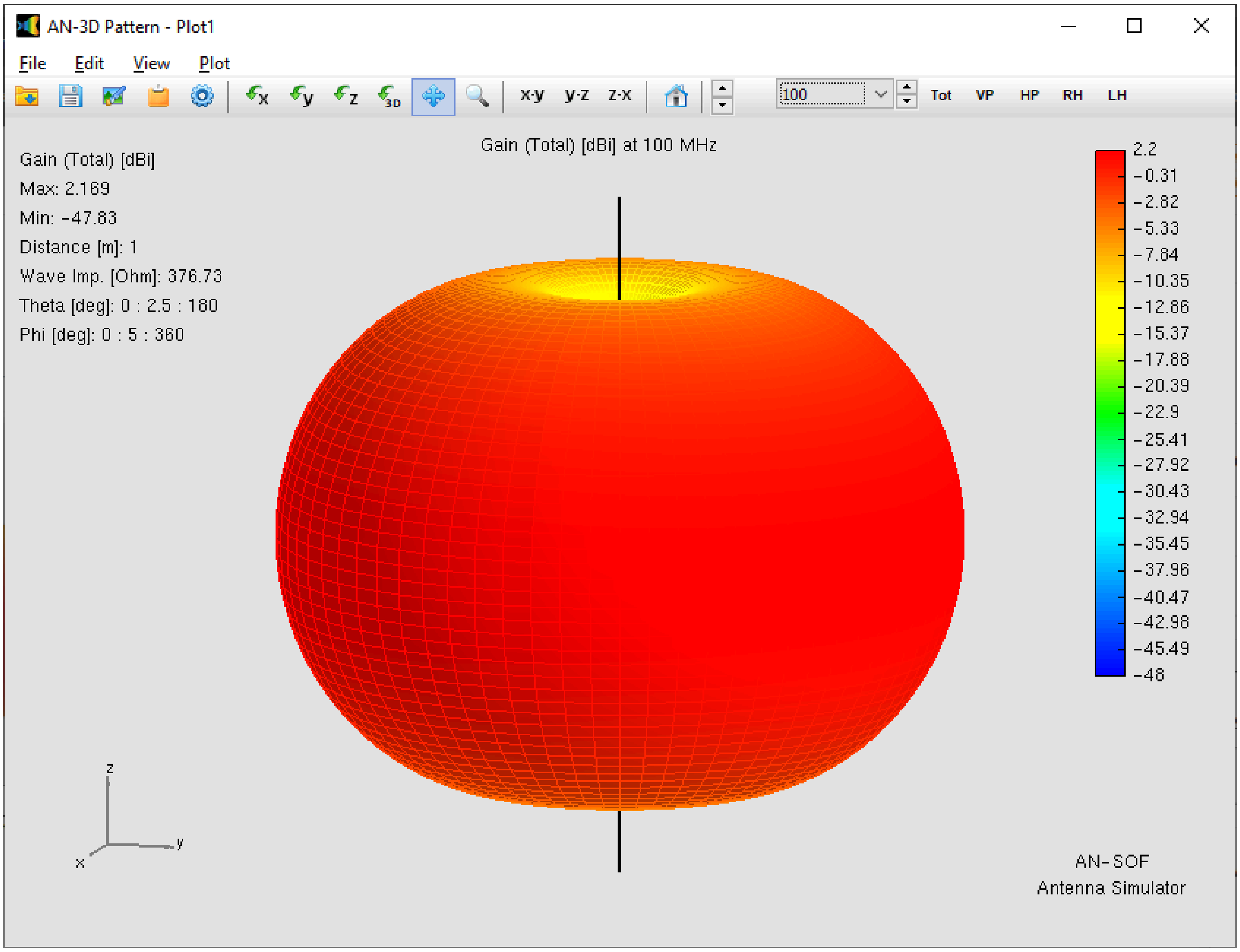


Fig. 7: Gain pattern (dBi) at 100 MHz.

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.

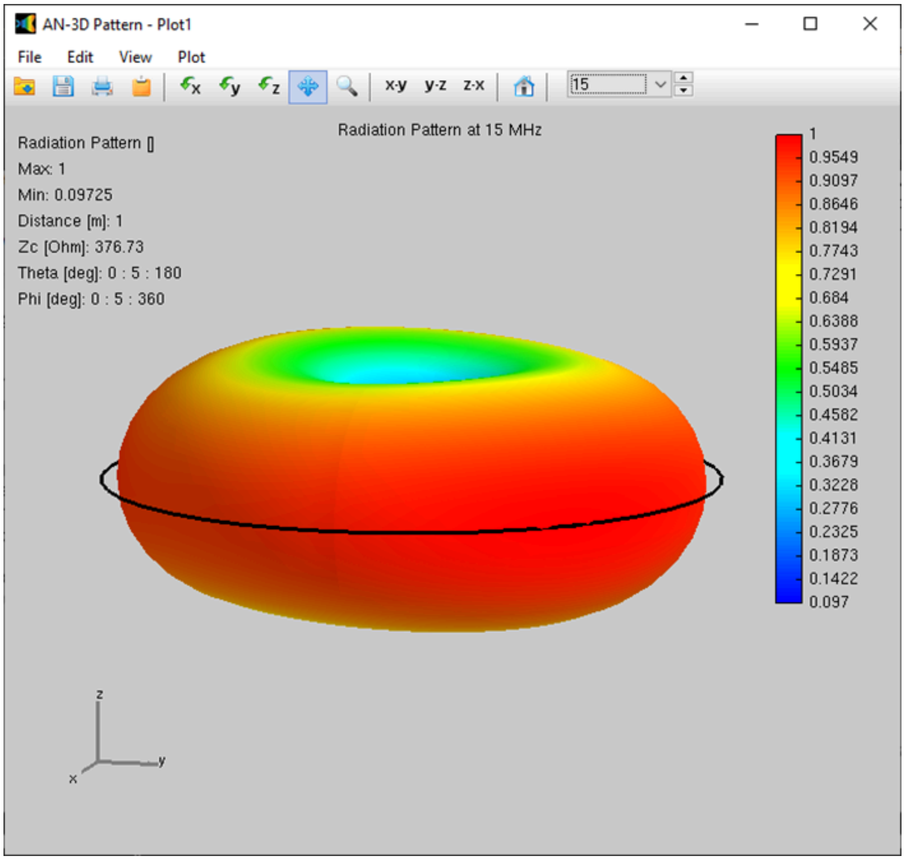
This example serves as a foundation for more complex antenna designs. By understanding the simulation process for this simple geometry, users can apply similar principles to model and analyze a wide range of antenna structures.

See Also:

Linear Antenna Theory: Historical Approximations and Numerical Validation

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. Choose **Lin** for a linear frequency sweep. This allows for evenly spaced data points across the desired range.
2. 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).

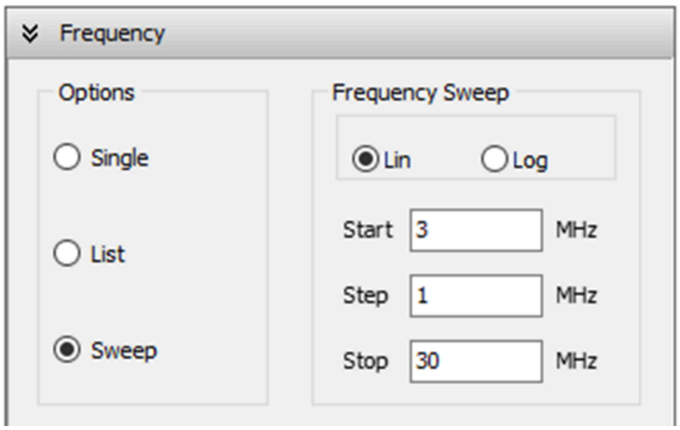


Fig. 1: Configuring the Frequency Sweep in AN-SOF.

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. **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**)

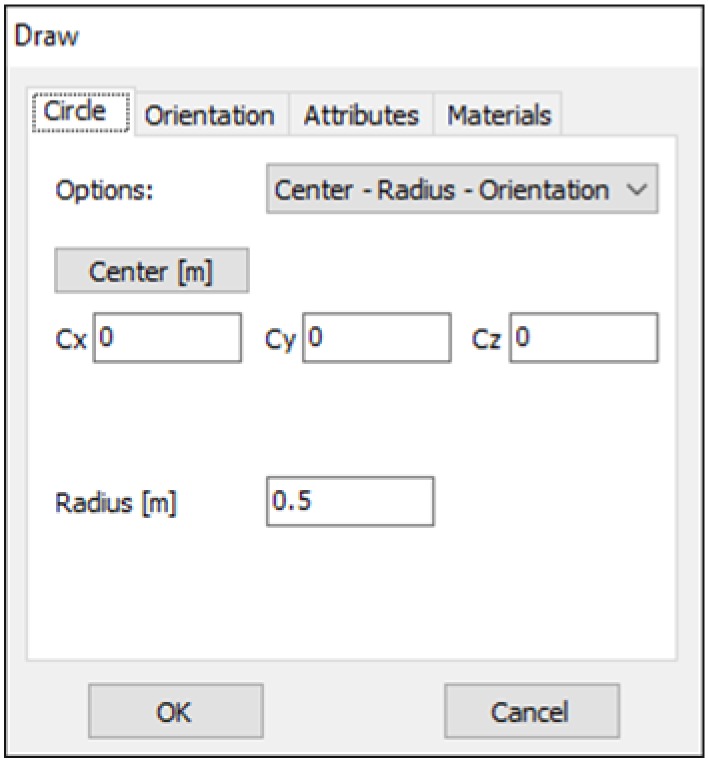


Fig. 2(a): Setting loop dimensions in AN-SOF Draw dialog (Circle tab).

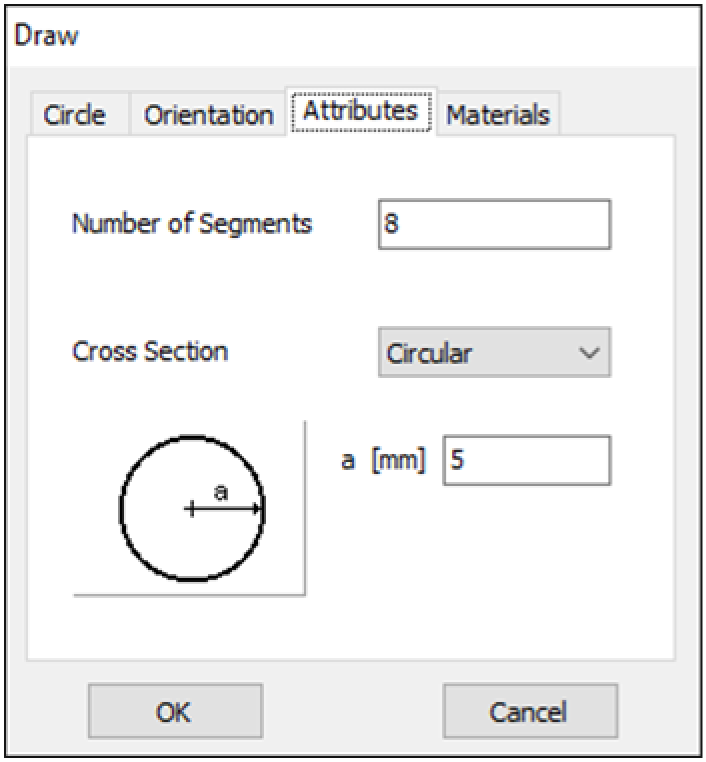


Fig. 2(b): Defining loop segmentation and wire cross-section in AN-SOF Draw dialog (Attributes tab).

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.

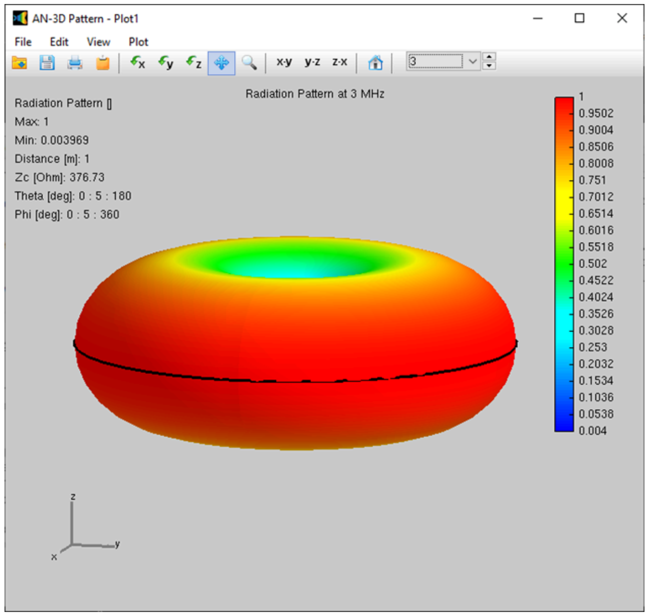


Fig. 3(a): Doughnut-shaped radiation pattern of the loop antenna at 3 MHz.

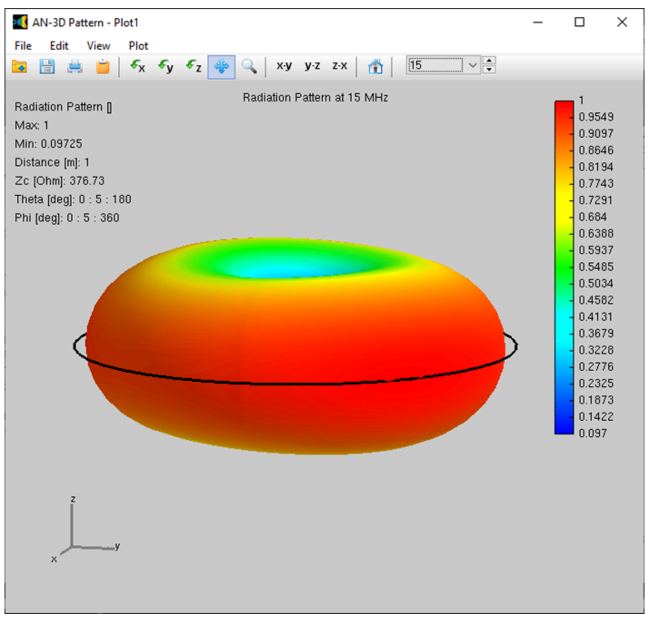


Fig. 3(b): Radiation pattern of the loop antenna at 15 MHz, showing a transition from the low-frequency pattern.

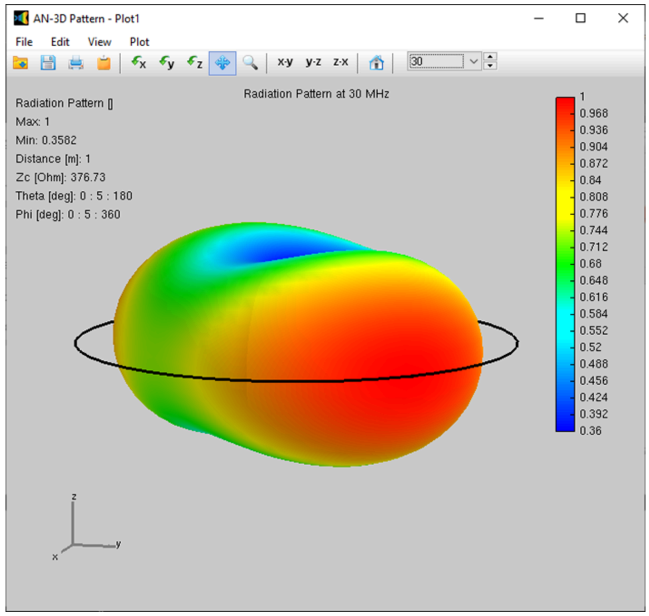
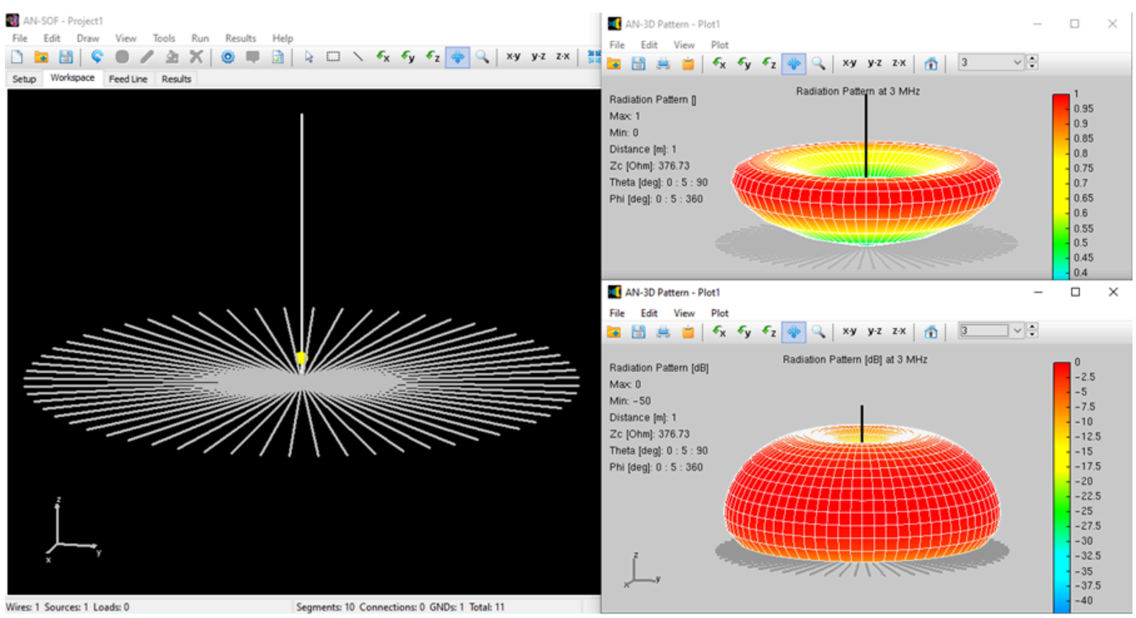


Fig. 3(c): Radiation pattern of the loop antenna at 30 MHz.

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**.

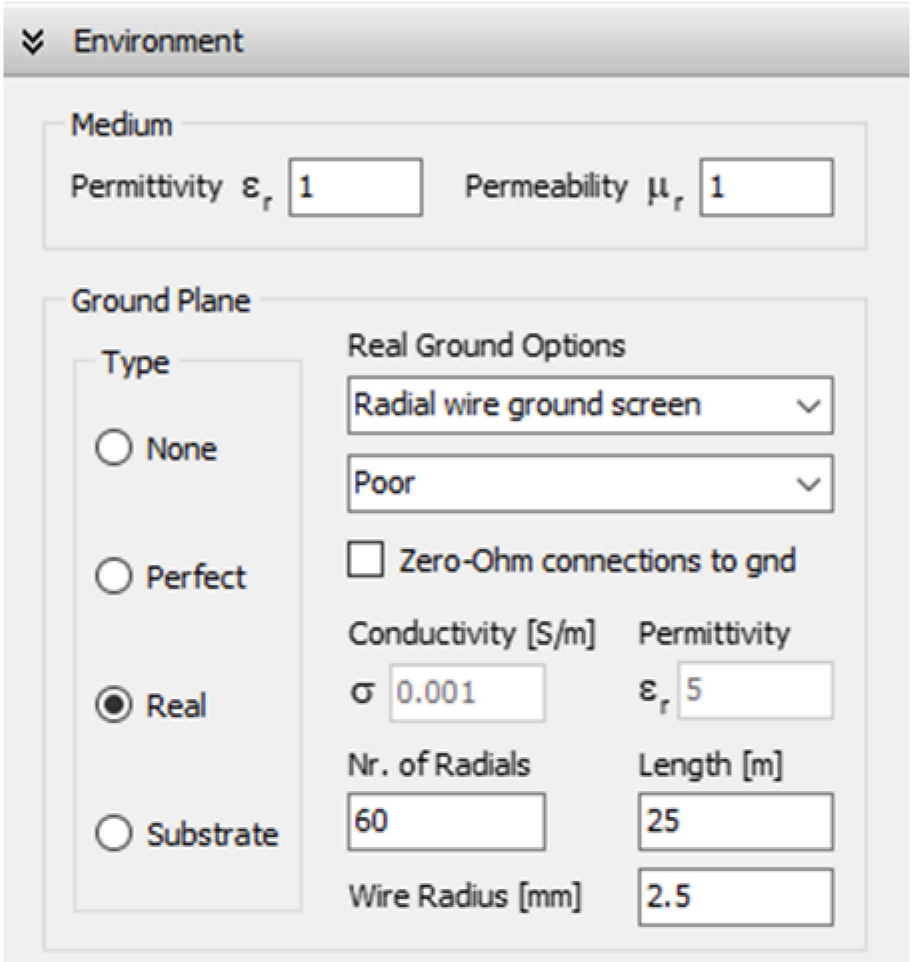


Fig. 1: Configuring a radial wire ground screen in the Environment panel of AN-SOF.

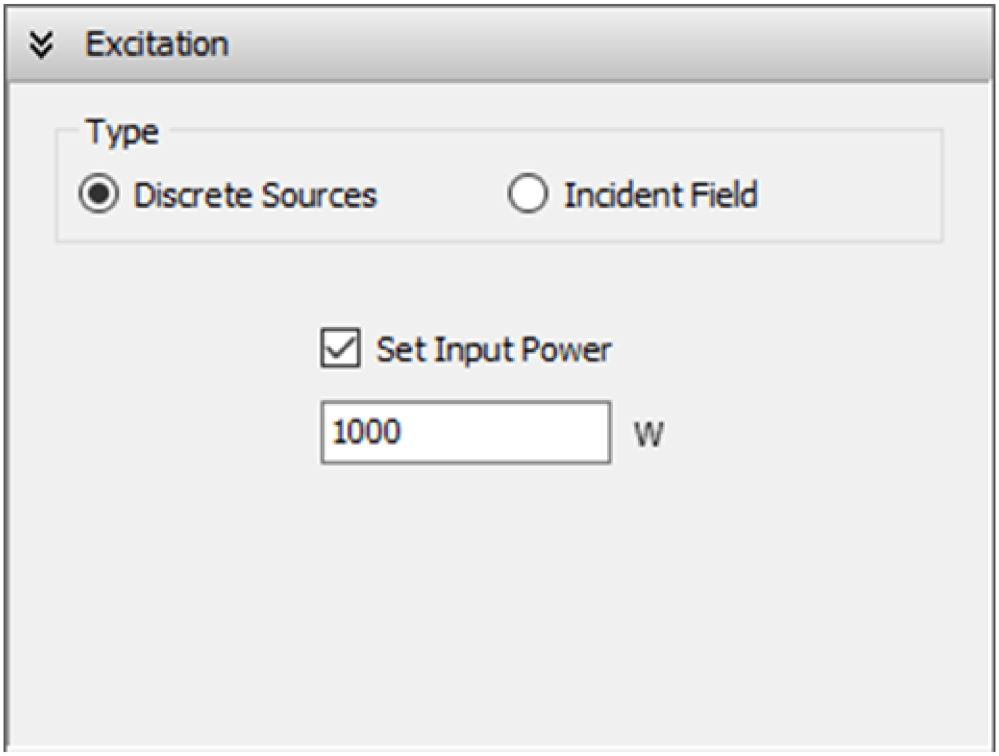


Fig. 2: Configuring discrete sources in the Excitation panel with an input power of 1,000 W.

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.

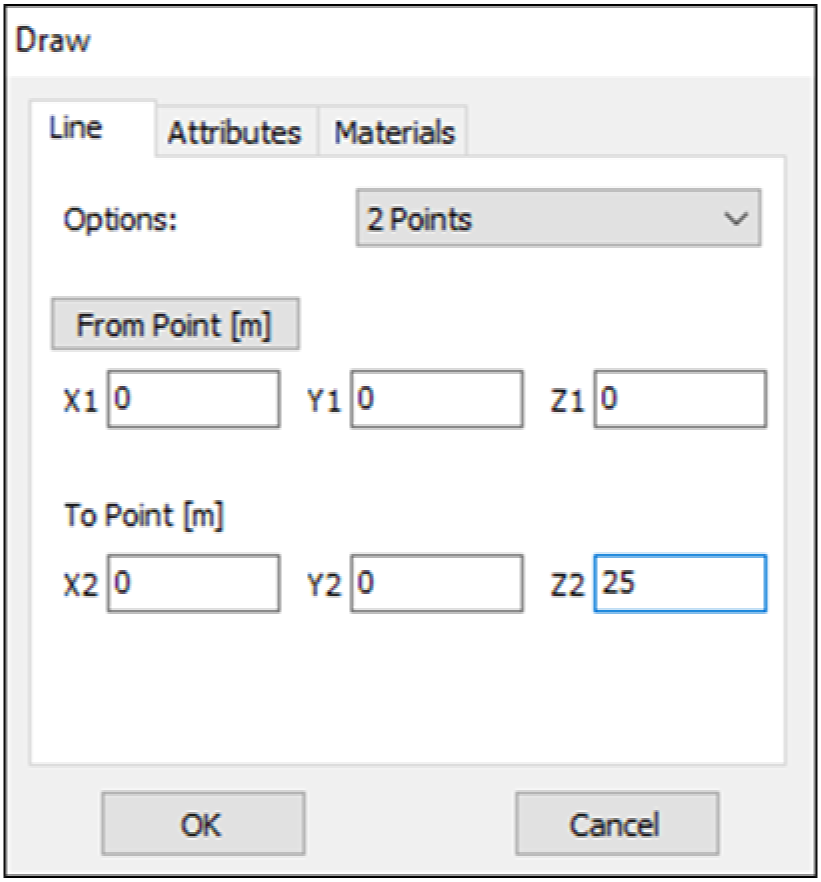


Fig. 3(a): Defining a vertical wire in the Line configuration page.

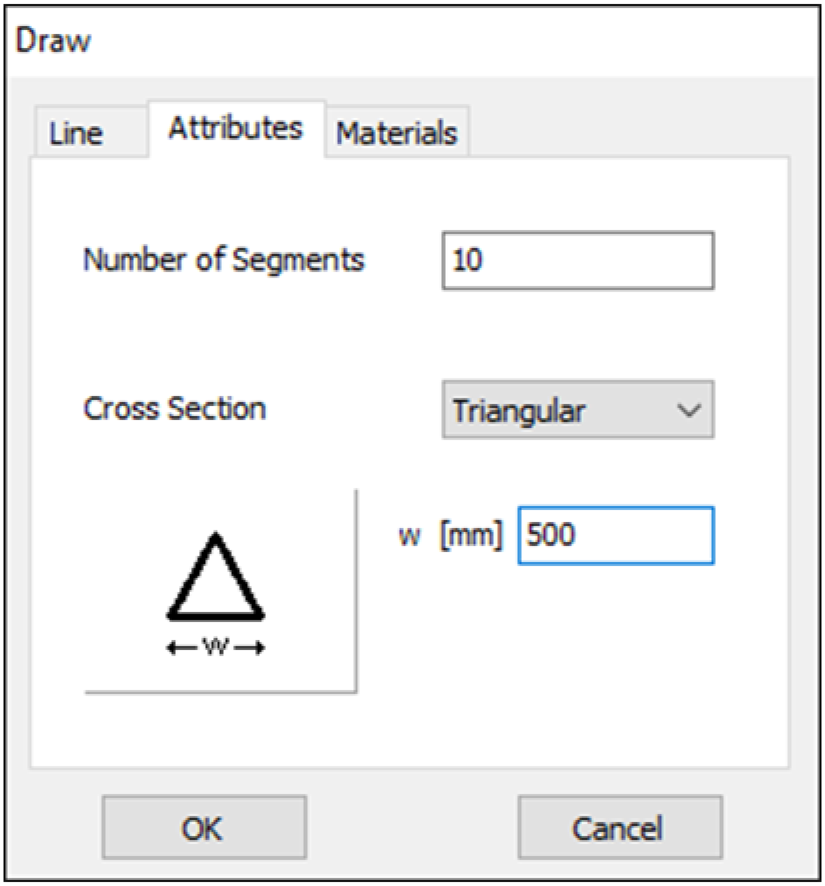


Fig. 3(b): Configuring a triangular cross-section in the Attributes page.

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.

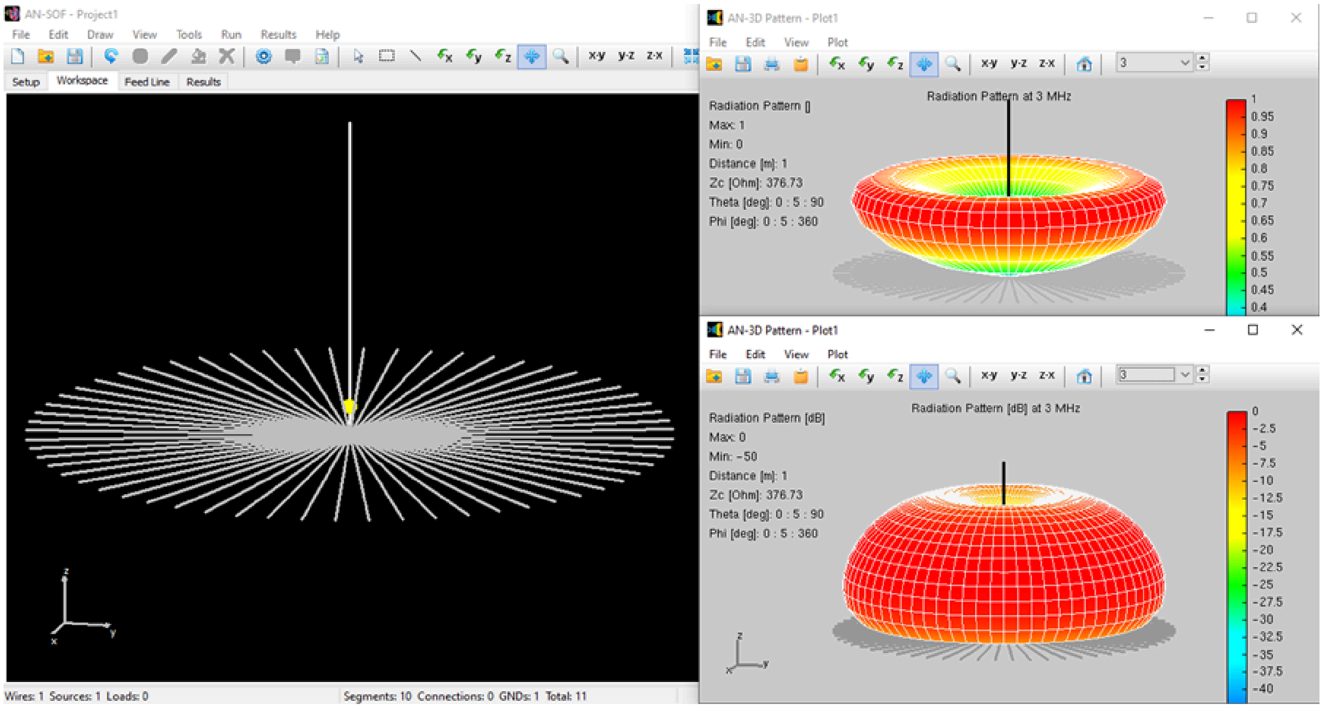


Fig. 4: Quarter-wave monopole antenna model (left) with its 3D radiation pattern: linear scale (top right) and decibel scale (bottom right), simulated using AN-SOF.

Download Model

Setup | Workspace | Feed Line | Results

No.	Freq.	Rin	Xin	VSWR	Dir.	Gain	Eff.
---	MHz	Ohm	Ohm	---	dBi	dBi	%
1	3	52.2151	30.0568	1.78892	5.17579	-2.39475	17.4963

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

Helix Antenna in Axial Mode

The helix is a good example where we need curved segments to describe the geometry of the antenna. When the length of the helix is of the order of or greater than the wavelength, it can work in the so-called "axial mode". To do this, we need to add a ground plane as a reflector.

Step 1 | Setup

Go to the Setup tabsheet and set an operating frequency of 100 MHz in the Frequency panel. Then, go to Environment panel > Ground Plane box, select **Perfect**, and set the ground plane position at Z = 0 (xy-plane), Fig. 1. Make sure the **Discrete Sources** option is selected in the **Excitation** panel.

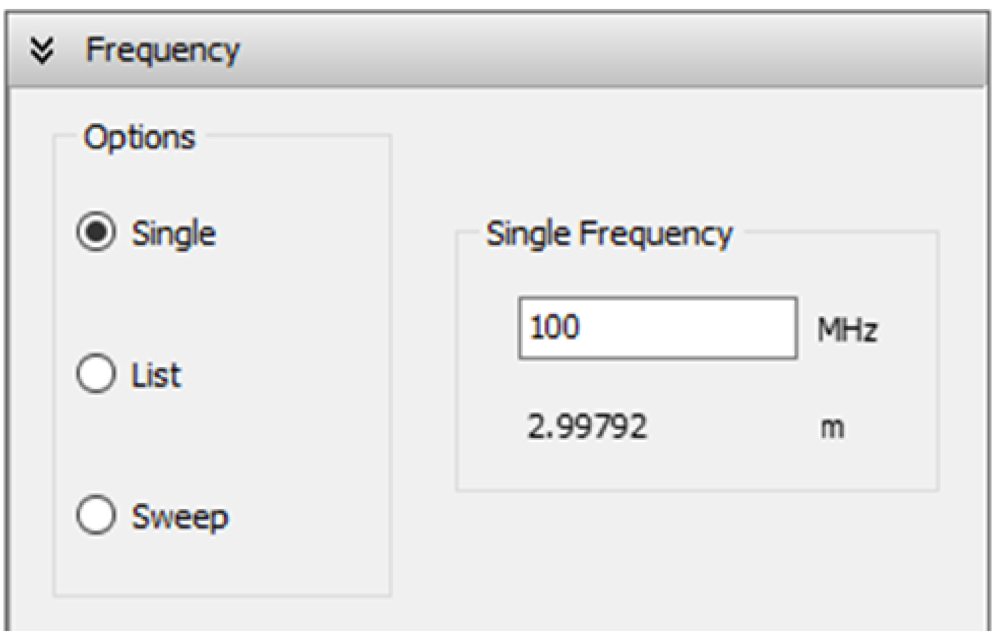


Fig. 1(a): Setting the operating frequency for the helix antenna.

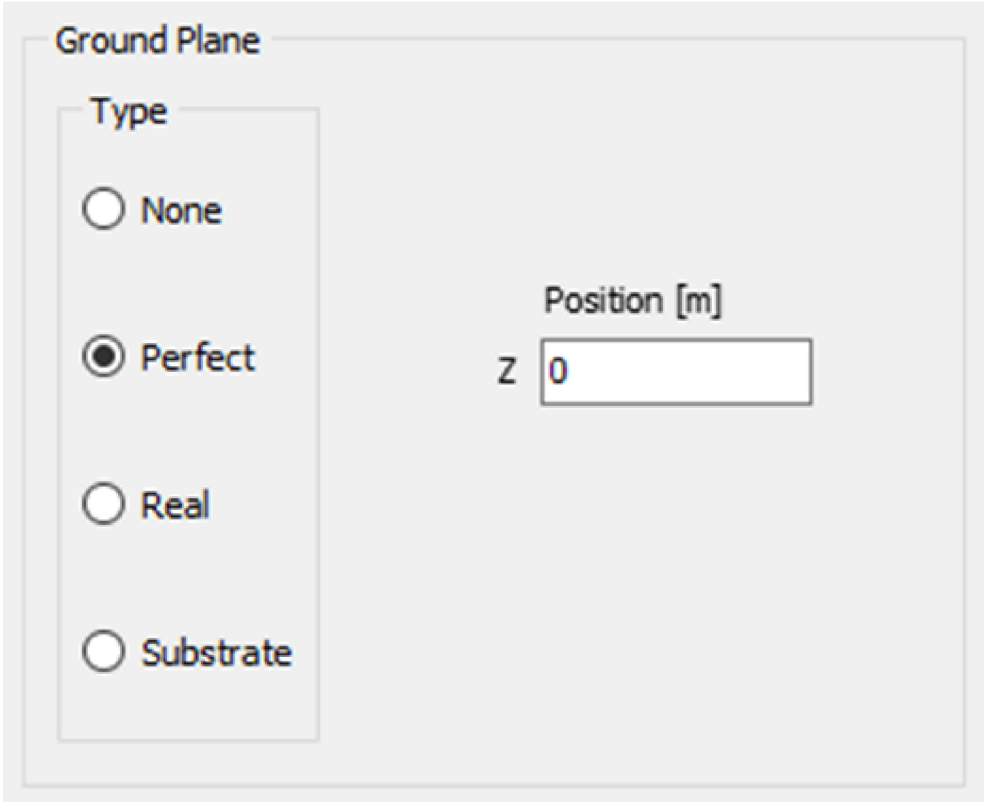


Fig. 1(b): Setting the ground plane for the helix antenna.

Step 2 | Draw

Go to the Workspace tab, right click on the screen, and select **Helix** from the displayed **pop-up menu >**. The **Draw** dialog box for the Helix will be shown, Fig. 2. The helix will start above the ground plane, at the point (0,0,0.3) m, and run along the Z axis. We will then add a vertical wire that will connect the helix to the ground plane and where we will place the source.

The recommended helix dimensions for the axial mode can be obtained from any antenna book. Here we will set the helix radius, pitch (spacing between turns), and number of turns shown in Fig. 2. In the **Attributes** tab, we will leave the recommended number of segments of 103. The wire cross-section will be circular with 3 mm in radius

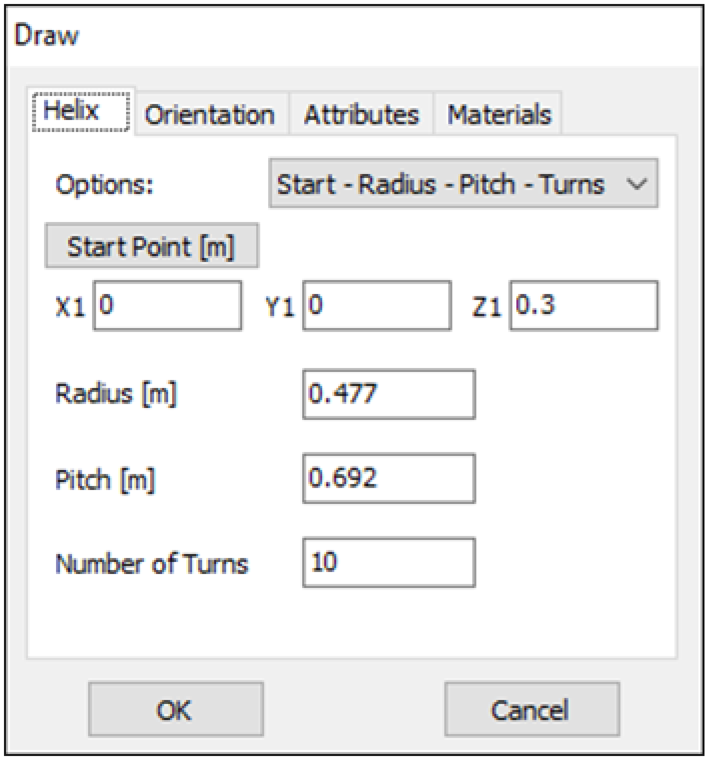


Fig. 2(a): Specifying the helix dimensions.

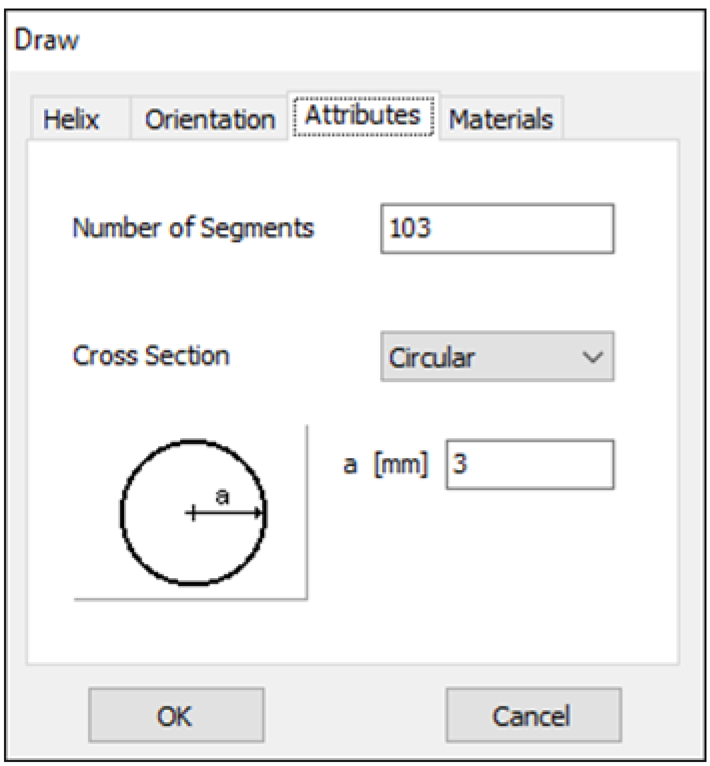


Fig. 2(b): Specifying the helix segmentation and cross-section.

Download Model

After drawing the helix, right click on the helix and choose the **Start Point to GND** command from the pop-up menu. The **Draw** dialog box for a **Line** will be displayed, where the coordinates of the ends of the wire are already set to connect the helix to the ground plane, Fig. 3. Set up 2 segments and a radius of 3 mm for this vertical wire. Finally, right click on the vertical wire, choose the **Source/Load** command, and connect a voltage source to the segment that is closest to the ground plane. Refer to **Adding Sources >** to add the source.

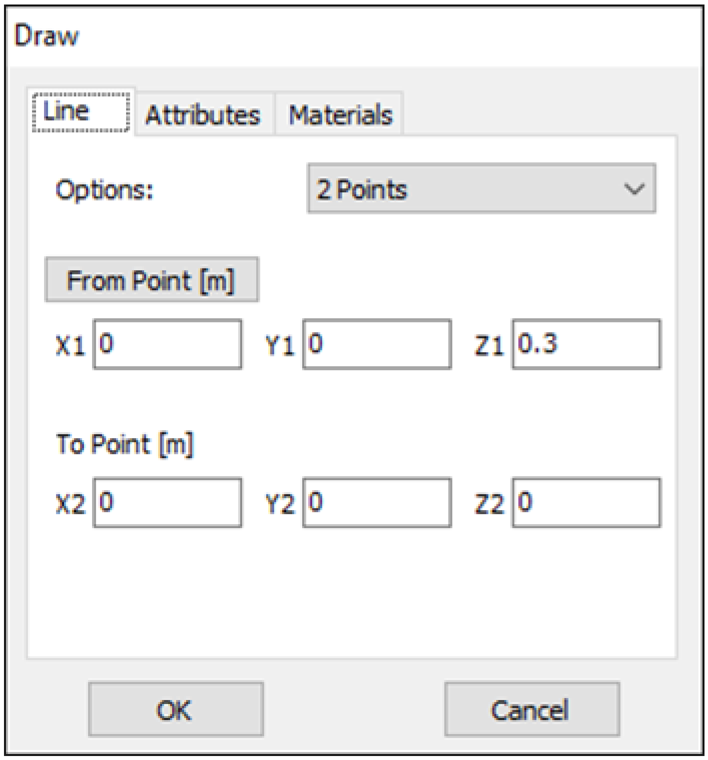


Fig. 3(a): Specifying the vertical wire that connects the helix to the ground plane.

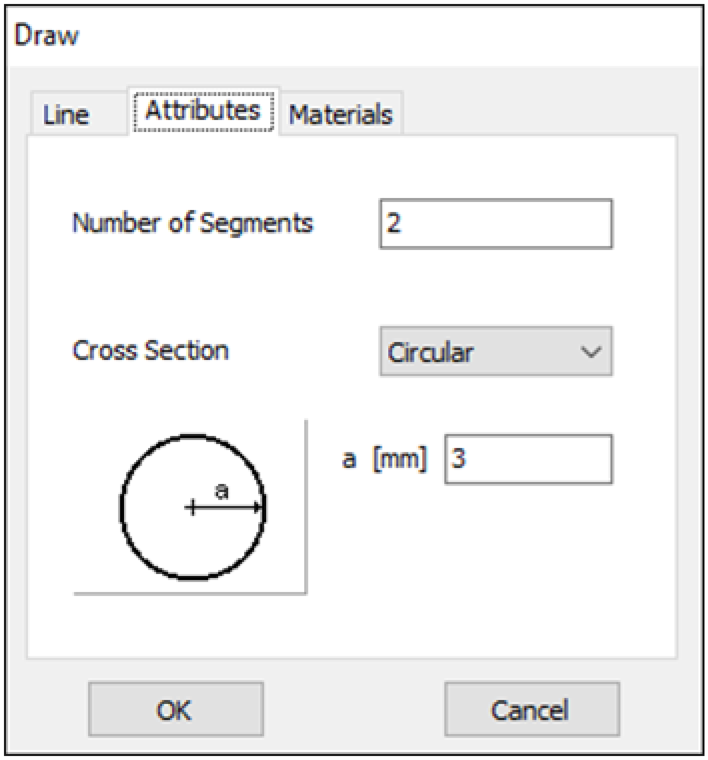


Fig. 3(b): Specifying the segments and cross-section for the vertical wire.

Step 3 | Run

Click on the **Run Currents and Far-Field (F11)** button on the toolbar. After the calculations are complete, click on the **Far-Field 3D Plot** button on the toolbar to display the radiation pattern, Fig. 4(a). The main lobe is on the axis of the helix, hence the name “axial mode”.

Because the helix is right-handed, the radiated field is circularly polarized, and the right-handed component predominates. Go to the AN-3D Pattern **Plot** menu and choose **E-right** or **E-left** to see the difference between both components, Figs. 4(b) and 4(c). To make the comparison between the color scales meaningful, go to **Edit > Preferences** in AN-3D Pattern and set the maximum of E-left to the same value as the maximum of E-right.

To draw a left-handed helix, specify a negative number of turns. Repeat the calculations and compare the E-right and E-left components.

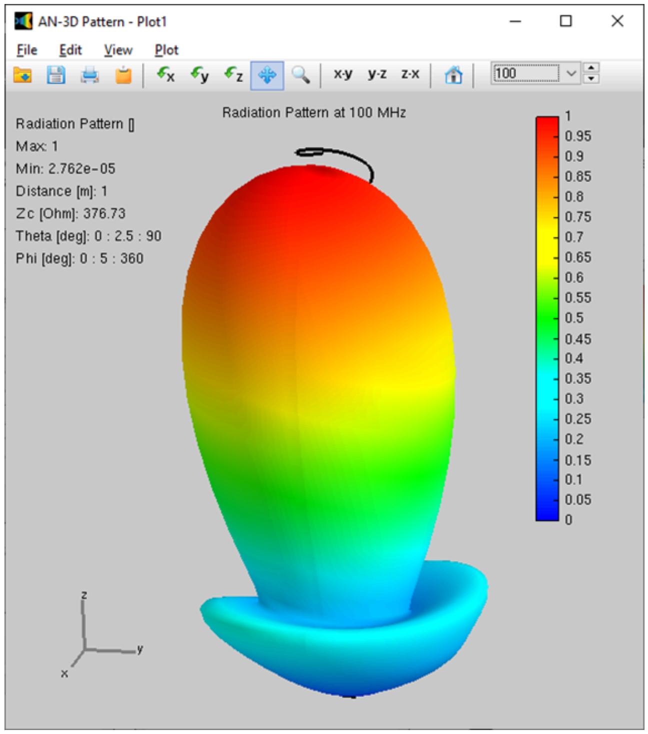


Fig. 4(a): Normalized radiation pattern of the helix.

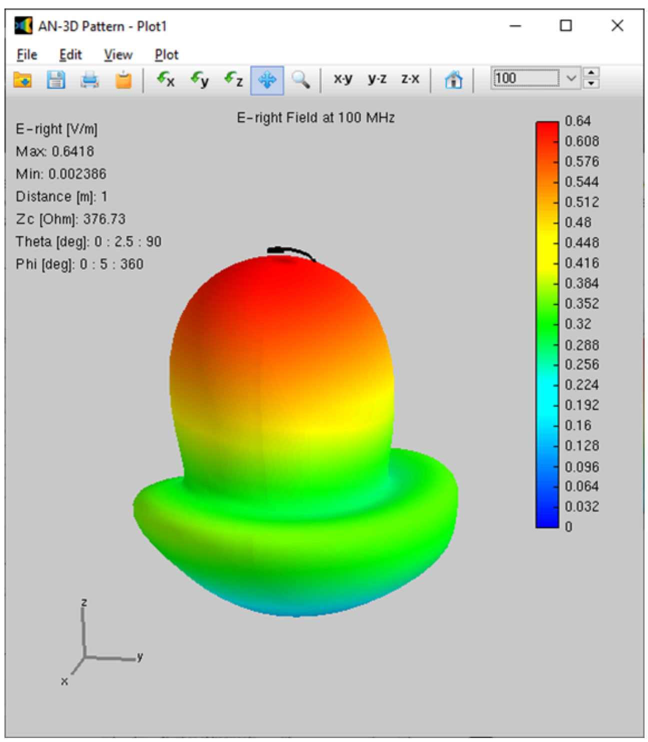


Fig. 4(b): Right-handed circularly polarized component of the far-field.

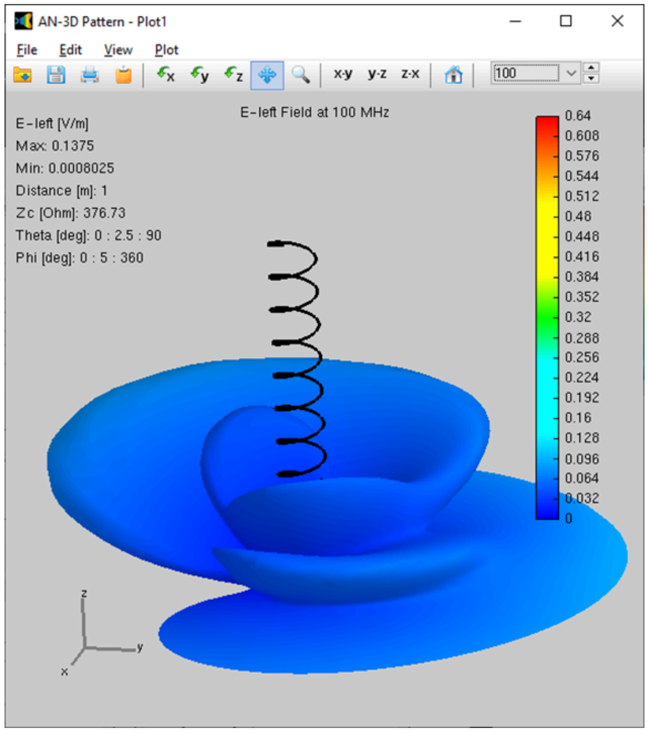


Fig. 4(c): Left-handed circularly polarized component of the far-field.

Yagi-Uda Array

After learning how to simulate a **Cylindrical Antenna >**, we are ready to build a dipole array. A 3-element Yagi-Uda antenna, consisting of a reflector, a driven element, and a director, is shown in Fig. 1, where the coordinates of the wire ends are indicated in meters.

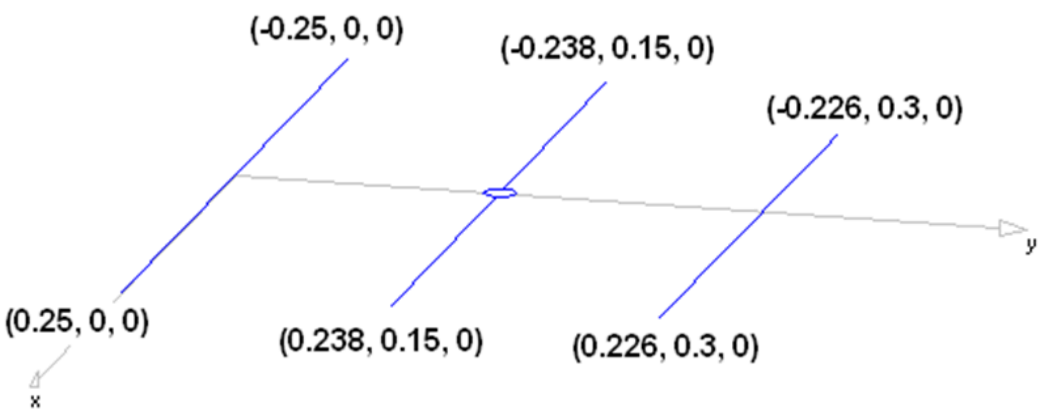


Fig. 1: Geometry definition for the Yagi-Uda array. The coordinates are in meters.

Download Model

Step 1 | Setup

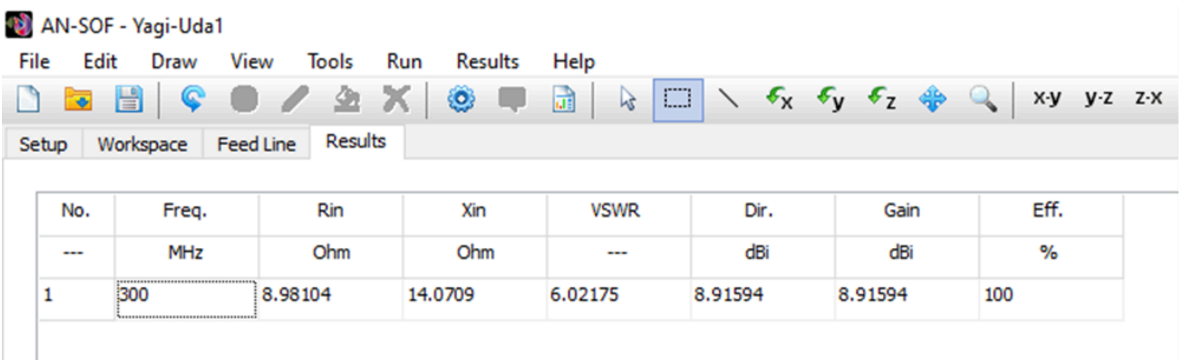
Go to the Setup tabsheet and set an operating frequency of 300 MHz in the Frequency panel. **None** must be selected in Environment panel > Ground Plane box and **Discrete Sources** in the Excitation panel.

Step 2 | Draw

Follow the procedure described in **Cylindrical Antenna >** to draw one wire at a time. Set the coordinates of the ends of the wires indicated in Fig. 1. Set 15 segments for each wire and a radius of 5 mm. Then, right click on the driven element, select the **Source/Load** command, and connect a voltage source at the middle segment. Refer to **Adding Sources >**.

Step 3 | Run

Click on the **Run Currents and Far-Field (F11)** button on the toolbar. Fig. 2 shows the table in the Results tabsheet, where a peak gain of 8.9 dBi is obtained. This can also be seen in the gain pattern of the Yagi-Uda array shown in Fig. 3. Click on the **Far-Field 3D Plot** button on the toolbar to plot the 3D radiation pattern.



No.	Freq.	Rin	Xin	VSWR	Dir.	Gain	Eff.
---	MHz	Ohm	Ohm	---	dBi	dBi	%
1	300	8.98104	14.0709	6.02175	8.91594	8.91594	100

Fig. 2: Results tabsheet, where a peak gain of 8.9 dBi is obtained for the Yagi-Uda array.

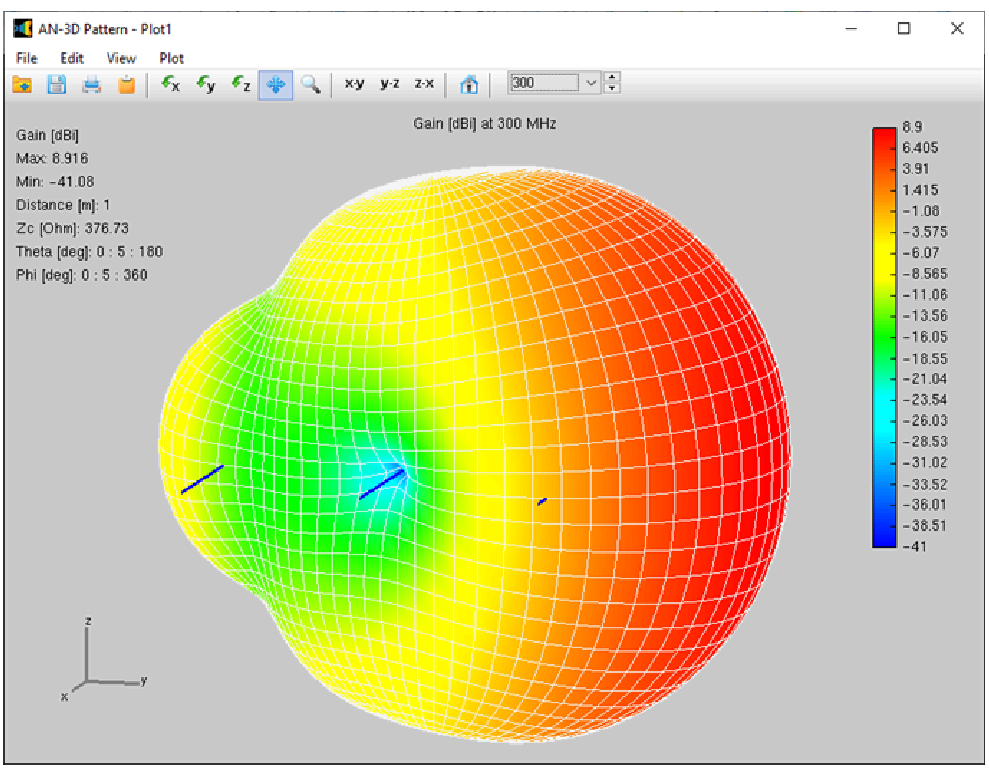


Fig. 3: Gain pattern [dBi] for the Yagi-Uda array of Fig. 1 at 300 MHz.

A Transmission Line

Two-wire transmission lines can be modeled explicitly in AN-SOF. In this example, the line will have a single wire but there will be a ground plane below it, so we have the mirror image of the wire as the return of the line.

Step 1 | Setup

Go to the Setup tab and select **Single** in the **Frequency panel >**. Set a frequency of 100 MHz. Then, go to the **Environment panel >** and set a perfect ground plane at Z = 0, Fig. 1.

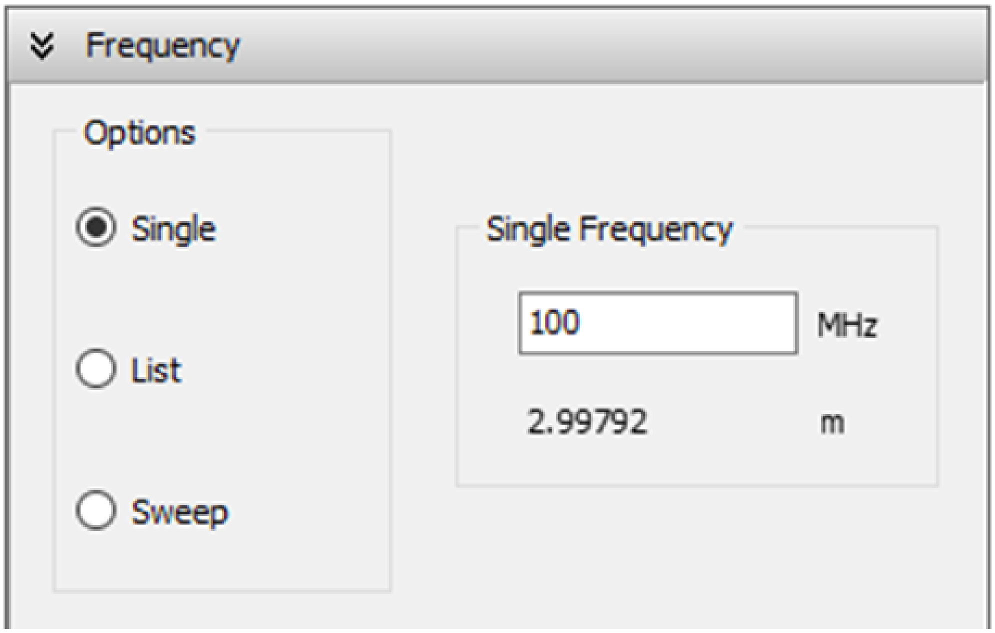


Fig. 1(a): Setting up the frequency for the transmission line.

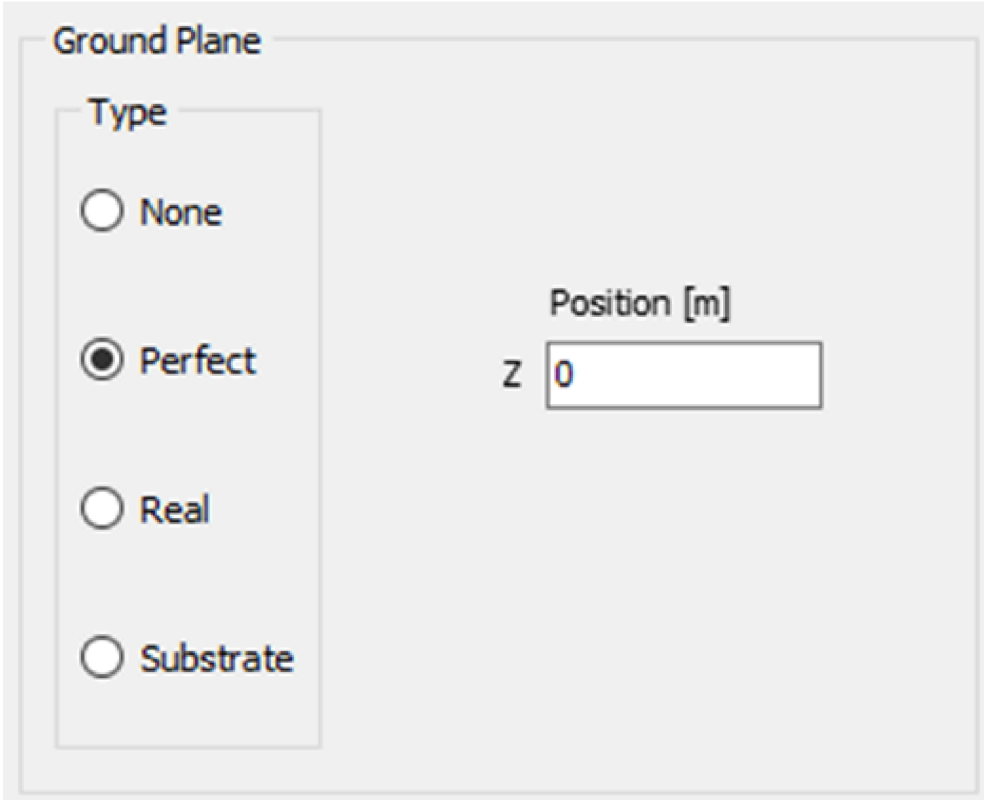


Fig. 1(b): Setting up the ground plane for the transmission line.

Step 2 | Draw

Go to the Workspace tab, right click on the screen, and select **Line** from the **pop-up menu >**. Draw a horizontal line with the coordinates indicated in Fig. 2. Next, connect the ends of the line to the ground plane by drawing two vertical wires. You can right click on the line and select the commands **Start point to GND** and **End point to GND** to connect the ends to ground.

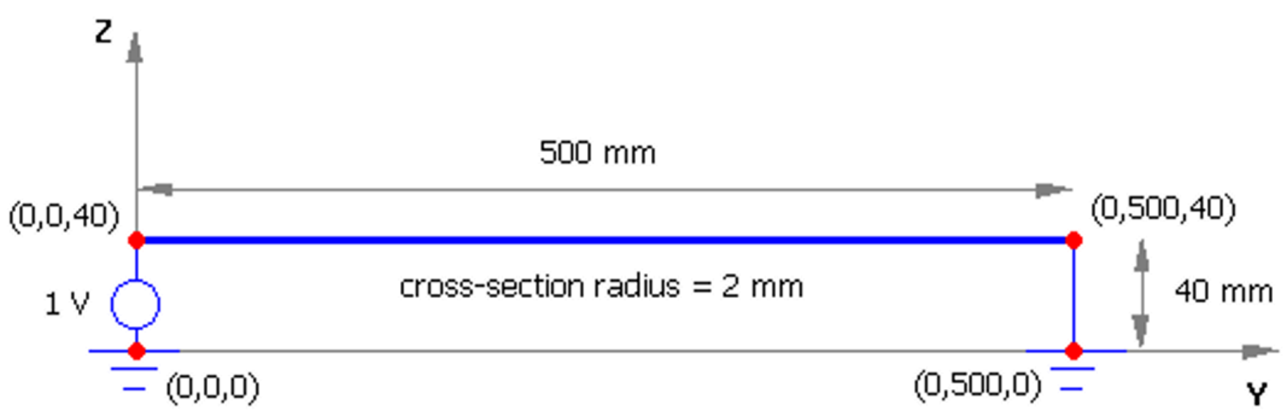


Fig. 2: Transmission line dimensions.

Set 40 segments for the horizontal wire and 1 segment for each of the vertical wires. Note that dimensions in Fig. 1 are in millimeters. To change the unit of length, go to **Tools menu > Preferences > Units tab >**.

Right click on the vertical wire at (0,0,0), select **Source/Load** from the displayed pop-up menu and put a 1 Volt voltage source on it. Refer to **Adding Sources >** to add the voltage source.

Step 3 | Run

Go to the Run menu and click on **Run Currents**. Since we are only interested in the current distribution and the input impedance, it is not necessary to calculate the radiated field (you can do it to check that it is practically negligible). Click on the **Zin (List Input Impedances)** button on the toolbar to display a table where the input impedance is shown as a function of frequency, Fig. 3. Refer to **Listing Input Impedances >**.

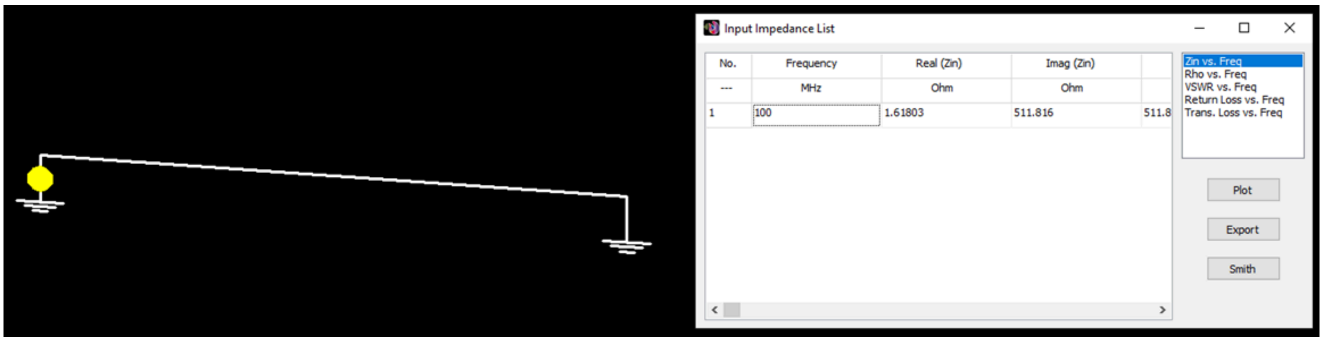


Fig. 3: Transmission line in the workspace and table of input impedances.

The impedance obtained is practically reactive, j512 Ohm. The small real part is the radiation resistance, since the line radiates a small amount of power, which is negligible but not zero.

This is a short-circuited line. Now right click on the vertical wire at (0,500,0) mm and select **Delete** from the pop-up menu to remove it. You will get an open-circuited line in this way. Rerun the calculations with the **Run Currents** command in the Run menu. The input impedance will now be -j105 Ohm.

Summarizing, we have,

- Z_{in} (short-circuited line) = j512 Ohm.
- Z_{in} (open-circuited line) = -j105 Ohm.

According to transmission line theory, the characteristic impedance can be calculated as the geometric mean of the short-circuit and open-circuit line input impedances, hence

$$Z_c = \sqrt{512 \times 105} = 232\ Ohm$$

On the other hand, the expression for the characteristic impedance of a line above a ground plane is given by:

$$Z_c = 138 \ \log\left(\frac{2h}{a}\right) = 138 \ \log\left(\frac{2 \times 40}{2}\right) = 221 \ Ohm$$

where *a* is the wire cross-section radius and *h* is the line height above the ground plane. As we can see, the agreement between the characteristic impedance obtained from AN-SOF and that from theory is quite good. The difference is since the theory neglects the radiation of the line, and the logarithmic formula is an approximation that is valid when *h* >> *a*.

An RLC Circuit

The ability of AN-SOF to simulate at extremely low frequencies can be demonstrated with a model of an RLC circuit that will resonate at only 800 Hz, so the wavelength is 375 km!

Step 1 | Setup

Go to **Tools > Preferences >** in the main menu and select Hz, mm, mH and uF as the units for frequency, length, inductance, and capacitance, respectively. Then, go to the Setup tab and select **Sweep** in the **Frequency panel >**. Choose **Lin** for a linear sweep and set the **Start**, **Step**, and **Stop** frequencies. The frequency sweep will start at 600 Hz and end at 1,000 Hz, incrementing by 10 Hz for each calculation, Fig. 1. In the **Environment panel >**, set a perfect ground plane at Z = 0.

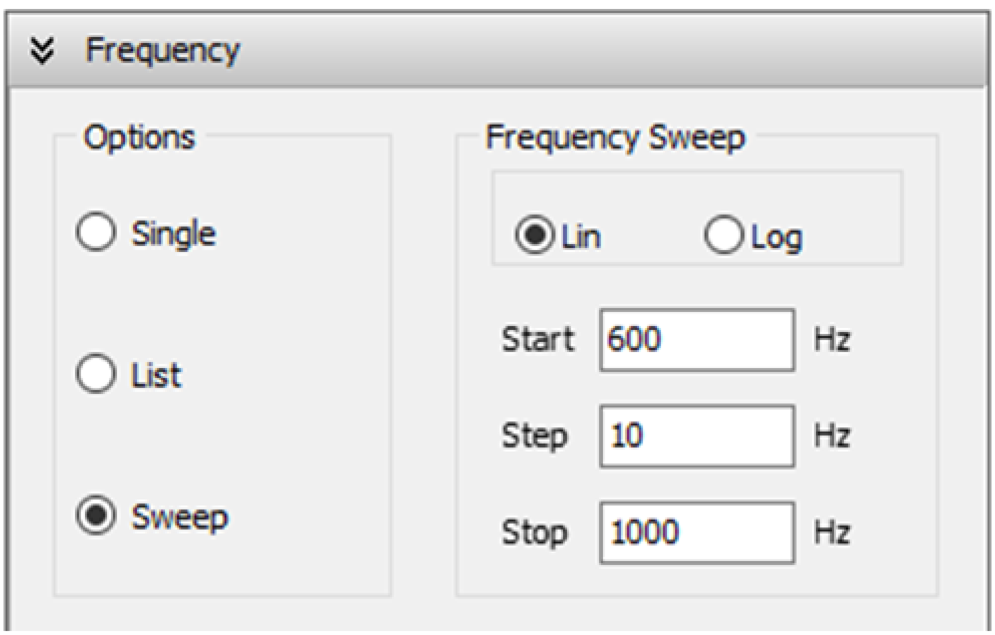


Fig. 1(a): Setting up frequencies for the RLC circuit.

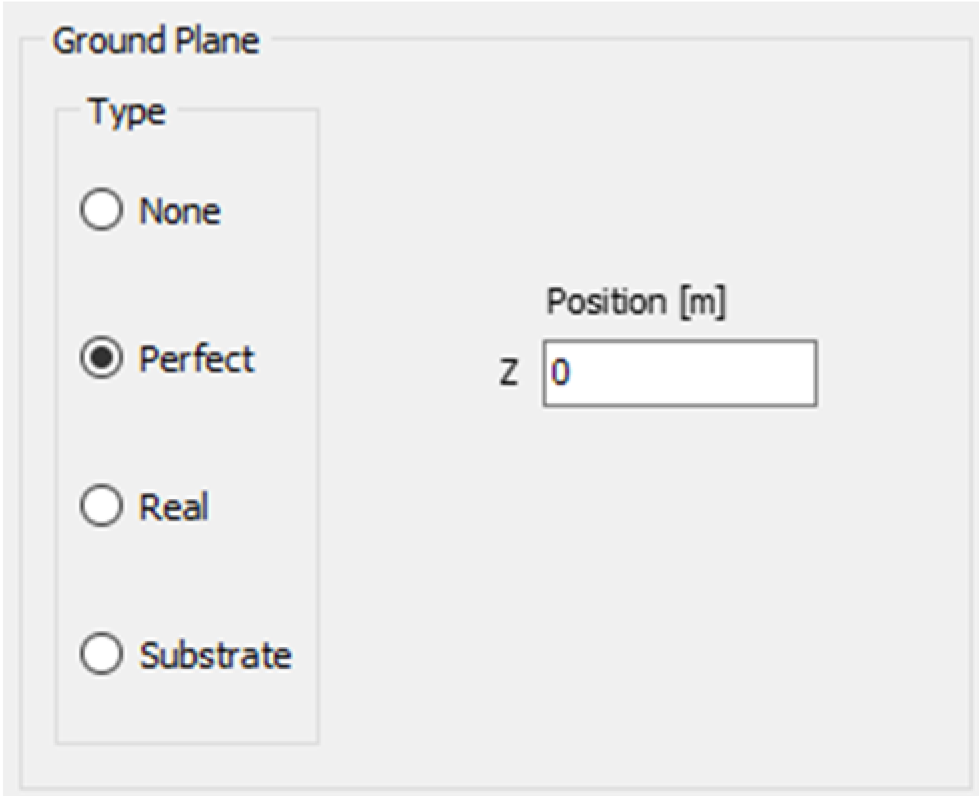


Fig. 1(b): Setting up the ground plane for the RLC circuit.

Step 2 | Draw

Go to the Workspace tab, right click on the screen, and select **Line** from the **pop-up menu >**. Draw the three wires with the coordinates indicated in Fig. 2 using the Line dialog box. The left vertical wire has 1 segment, the horizontal wire has 1 segment, and the right vertical wire has 2 segments. The wire radius is 0.5 mm.

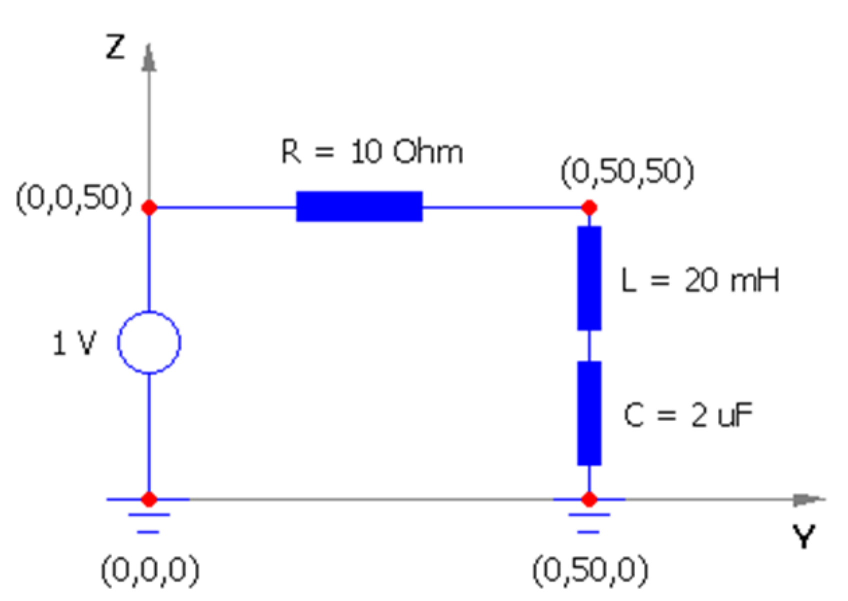


Fig. 2: RLC circuit dimensions. The coordinates are in millimeters.

Right click on the left vertical wire, select the **Source/Load** command from the pop-up menu and put a 1 Volt voltage source. Then, right click on the horizontal wire, select **Source/Load** from the pop-up menu and connect a load impedance with R = 10 Ohm. Finally, right click on the right vertical wire, select **Source/Load** from then pop-up menu and put an inductance L = 20 mH on the first segment and a capacitance C = 2 uF on the second segment. Refer to **Adding Sources >** and **Adding Loads >** for adding sources and load impedances.

Step 3 | Run

Go to the Run menu and click on the **Run Currents** command. Since we are only interested in the input impedance, it is not necessary to calculate the radiated field (you can do it to check that it is practically negligible).

Right click on any of the three wires composing the circuit, select the **List Currents** command and click on the **Current on Segment** button of the displayed **toolbar >**. A table will be shown, where the current is tabulated vs. frequency. Next, press the **Plot** button to the right of the table to plot the current versus frequency, Fig. 3.

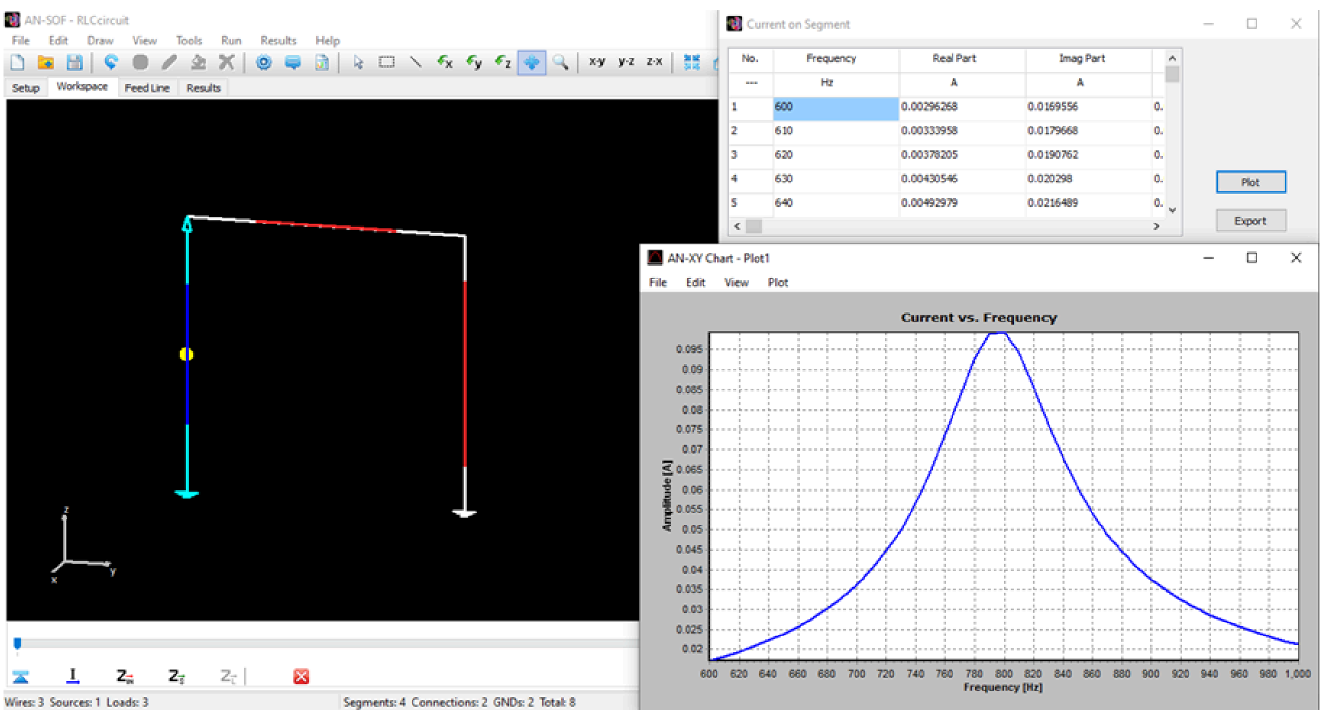


Fig. 3: Current amplitude vs. frequency in the RLC circuit.

Since this is a series RLC circuit, the current flowing must be the same in all three wires (check this). As can be seen, resonance occurs at a frequency near to 800 Hz. Repeat the calculation for frequencies around 800 Hz, with a step of 1 Hz, and verify that the resonant frequency is **796 Hz**. On the other hand, according to circuit theory, the resonance frequency is given by

$$f_r = \frac{1}{2\pi\sqrt{LC}} = \frac{1}{2\pi\sqrt{20 \times 10^{-3} \times 2 \times 10^{-6}}} = 796\ Hz$$

The agreement between AN-SOF and theory is remarkable!

Background Theory

The AN-SOF Calculation Engine

The AN-SOF engine is written in the C++ programming language using double-precision arithmetic and has been developed to improve the accuracy in the modeling of wire antennas and metallic structures in general.

The computer code is based on an Electric Field Integral Equation (EFIE) expressed in the frequency domain. The current distribution on wire structures is computed by solving the EFIE using a Method of Moments (MoM) formulation with curved basis and testing functions, called the **Conformal Method of Moments (CMoM) >**. In this method, curved wires are modeled by means of conformal segments, which exactly follow the contour of the structure, instead of the traditional approximation based on straight wire segments. The linear approximation to the geometry can be a very inefficient method in terms of unknowns or computer memory. By using curved segments, the number of unknown currents, simulation time and memory space can be greatly reduced, allowing for the solution of bigger problems.

Old MoM codes suffer from several drawbacks due to the linear approximation to geometry and the use of the so-called **thin-wire Kernel**, such as: divergent input impedance, poor convergence for curved antennas (helices, loops, spirals) and bent wires, and singularities that appear when two parallel wires are close to each other or close to a lossy ground plane. With the **CMoM and an exact Kernel formulation** we have removed these limitations and obtained the following advantages:

- Decreased number of calculations and increased accuracy of results.
- Decreased simulation time and computer memory usage, allowing us to model larger and more complex designs.
- Ability to simulate from extremely low frequencies (circuits at 60 Hz) to very high ones (microwave antennas).

Electric Field Integral Equation

The current distribution on metallic surfaces with ideal conductivity can be found by solving an Electric Field Integral Equation (EFIE) expressed in the frequency domain:

$$\mathbf{n} \times \mathbf{E}_i(\mathbf{r}) = \mathbf{n} \times \frac{j}{\omega\varepsilon} \iint_S \left[k^2 \mathbf{J}(\mathbf{r}')G(\mathbf{r},\mathbf{r}') + div'(\mathbf{J}(\mathbf{r}'))grad\,(G(\mathbf{r},\mathbf{r}')) \right] dS'$$

Eq. (1)

where:

$\mathbf{E}_i$: Incident Electric Field on the surface *S*.

n: unit vector at point **r** on the surface *S*.

k: wave number.

J: unknown electric current density flowing on the surface.

G: Green's function, which in free space is given by:

$$G(\mathbf{r},\mathbf{r}') = \frac{e^{-jk|\mathbf{r}-\mathbf{r}'|}}{4\pi|\mathbf{r}-\mathbf{r}'|}$$

Eq. (2)

The EFIE is an expression of a boundary condition on the surface, namely zero tangential electric field. When we are dealing with a wire structure, the EFIE reduces to:

$$\mathbf{T} \cdot \mathbf{E}_i = \mathbf{T} \cdot \frac{j}{\omega\varepsilon} \int_{\Gamma} \left[k^2 I(s')K(s,s')\mathbf{T}' + \frac{dI(s')}{ds'} grad\,(K(s,s')) \right] ds'$$

Eq. (3)

where **T** is the tangential unit vector describing the wire contour along its axis, *I*(*s*) is the unknown electric current on the wire, and *K*(*s,s*') is the integral equation Kernel defined as:

$$K(s,s') = \frac{1}{4\pi^2} \int_0^{2\pi}\int_0^{2\pi} G(\mathbf{r},\mathbf{r}')d\phi' d\phi$$

Eq. (4)

The EFIE is averaged about the wire circumference, resulting in the EFIE (3) with the Kernel (4). The current distribution *I*(*s*) is then the average value of the current density **J** in the axial direction; the current in the transversal direction is neglected. This is a good assumption provided that the wire radius is small with respect to the wavelength.

The wire axis is defined by its parametric equations that can be written in the compact form:

$$\mathbf{r}(s) = x(s)\mathbf{i} + y(s)\mathbf{j} + z(s)\mathbf{k}$$

Eq. (5)

which points from the origin to any point on the wire, Fig. 1. The parameter *s* varies over a real interval. The tangent unit vector can be obtained from the first derivative of (5):

$$\frac{d\mathbf{r}(s)}{ds} = \frac{dx}{ds}\mathbf{i} + \frac{dy}{ds}\mathbf{j} + \frac{dz}{ds}\mathbf{k} \qquad \mathbf{T} = \frac{d\mathbf{r}}{ds}\left|\frac{d\mathbf{r}}{ds}\right|^{-1}$$

Eq. (6)

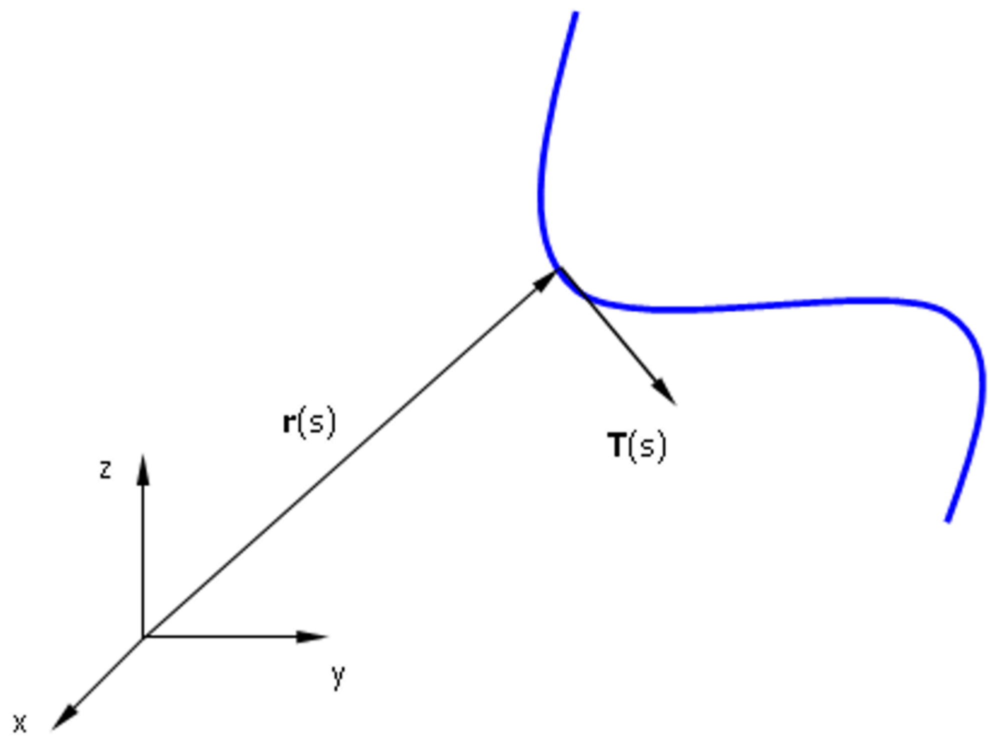


Fig. 1: Parametric description of a curved wire. The tangent unit vector is obtained from the first derivative of the position vector **r**(*s*).

This parametric description is the key for the accurate modeling of wire structures. A straight wire approximation to the geometry produces a loss of geometrical information that can never be completely restored. However, this information is not lost if a parametric representation is used to describe the wire locus. It is also possible to improve on the straight wire approximation by using quadratic segments to model the geometry.

Thus, the definition of a wire must include its parametric description and its first derivative if an exact representation of the geometry is required, as shown in Fig. 1.

The Kernel (4) can be approximated by the following generalized thin-wire approximation:

$$K(s,s') \cong \frac{e^{-jkR}}{4\pi R} \qquad\qquad R = \sqrt{\left|\mathbf{r}(s) - \mathbf{r}(s')\right|^2 + a^2}$$

Eq. (7)

where *a* is the wire radius.

When the observation point **r**(*s*) and the source point **r**(*s*') are both in the same straight wire, the distance *R* reduces to the usual thin-wire approximation:

$$R = \sqrt{(s - s')^2 + a^2}$$

Eq. (8)

Thus, the EFIE and its Kernel are also valid for straight wires.

It is well known that the thin-wire approximation produces numerical oscillations in the computed current distribution near wire ends and near the position of discrete sources when wire segments are relatively thick. To avoid this undesired behavior and obtain the maximum accuracy, **the exact Kernel in (4) is used in AN-SOF by default** instead of the thin-wire approximation in (7). A closed-form expression for the exact Kernel has been found so its use practically does not compromise the speed of the simulation. However, an extended thin-wire Kernel has been calculated that also avoids the current distribution inaccuracies for a thin-wire ratio (wire diameter/segment length) < 3, which is far better than the thin-wire ratio < 1 that must be used when the standard thin-wire approximation is used.

In the **Settings panel >** of the **Setup** tabsheet check the **Exact Kernel** option to use the exact Kernel in (4). Uncheck this option to use the extended thin-wire Kernel.

The existence of a PEC ground plane is modeled in AN-SOF by means of image currents. This method can be easily implemented by adding an image term to the Green's function, resulting in an additional term for the Kernel.
The Exact Kernel

The kernel is the core of the integral equation that is solved in AN-SOF by means of the **Method of Moments (MoM)** to obtain the current distribution on metallic wires. Since the kernel cannot be calculated analytically in closed form, several approximations exist, among which the best known and widely used is the so-called **thin-wire approximation**.

The integral equation kernel is given by

$$K(s, s') = \frac{1}{4\pi^2} \int_0^{2\pi} \int_0^{2\pi} G(r, r')\, d\phi' d\phi \quad (1)$$

where s and s' are coordinates along a wire axis that represent an observation point and a source point, respectively. G is the free space Green's function, and ϕ and ϕ' are angles that determine points on the circumference of the wire cross-section. Thus, $r = (s, \phi)$ represents an observation point on the wire surface, and $r' = (s', \phi')$ represents a source point. With that said, we can see that the kernel is obtained by averaging the Green's function along the contour of the wire's cross-section.

In the **thin-wire approximation**, the kernel is approximated as follows,

$$K(s, s') \approx \frac{e^{-jkR}}{4\pi R}, R = \sqrt{|r(s) - r(s')|^2 + a^2} \quad (2)$$

where a is the wire radius, and R is the distance between the source and observation points. When both points are on the same wire, the current will be represented by a filament on the wire axis, as the expression for R shows. In particular, when both points coincide ($r = r'$), which happens when calculating the electromagnetic interaction of a wire with itself, the thin-wire kernel will vary as $1/a$, but **the exact kernel will have an integrable singularity**.

When a **wire segment is thin**, that is, its length is at least twice its diameter, the thin-wire approximation works well. However, as we increase the number of segments and therefore **the segments get shorter**, we will reach a threshold where the segments are **too thick** for the thin-wire approximation to work. This situation is particularly restrictive when we are investigating the convergence of some parameter or even when we want to fill the source gap at the antenna terminals using a short wire.

In AN-SOF, we have implemented the **exact kernel** instead of the thin-wire approximation. The solution involves separating the kernel into two terms: one containing an **analytically integrable singularity**, and the other containing a **remainder** that can be numerically integrated with minimal computational effort since it does not have any singularity. We have also provided the option of using an "extended kernel," similar to the one used in NEC (Numerical Electromagnetics Code), where a series expansion of the kernel in terms of the wire radius is utilized, and the a^2 and a^4 terms are retained.

> **Note**
>
> In summary, it should be noted that **AN-SOF implements both the Exact Kernel and the Extended Kernel**, while NEC uses an extended kernel and the thin-wire approximation.

Fig. 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 in this case.

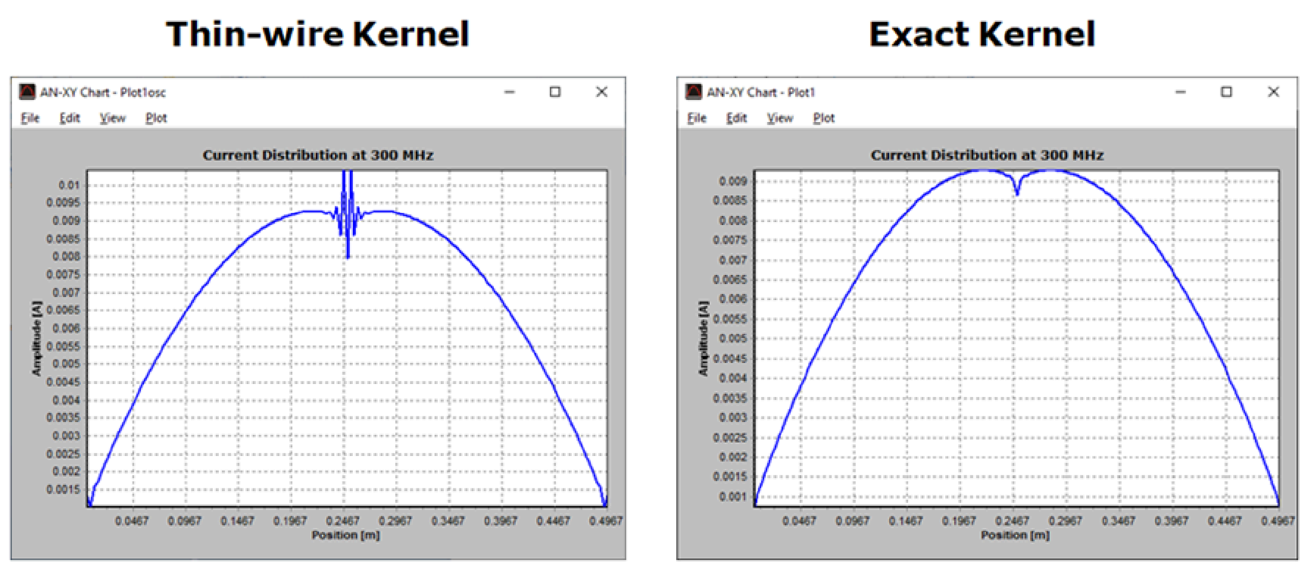


Fig. 1: Current distribution along a center-fed half-wave dipole divided into segments having a diameter-to-segment length ratio of 3.

The thin-wire kernel exhibits the well-known **oscillatory effect** on the current distribution **near the position of discrete sources** and at **wire ends** for a segment diameter-to-length ratio greater than 1. As we can see, these oscillations disappear when the exact kernel is used instead of the thin-wire approximation.

The effect of not using the exact kernel can also be observed in the **lack of convergence of the input impedance** when the number of segments increases, as shown in Fig. 2 for the AN-SOF vs. NEC-2 results.

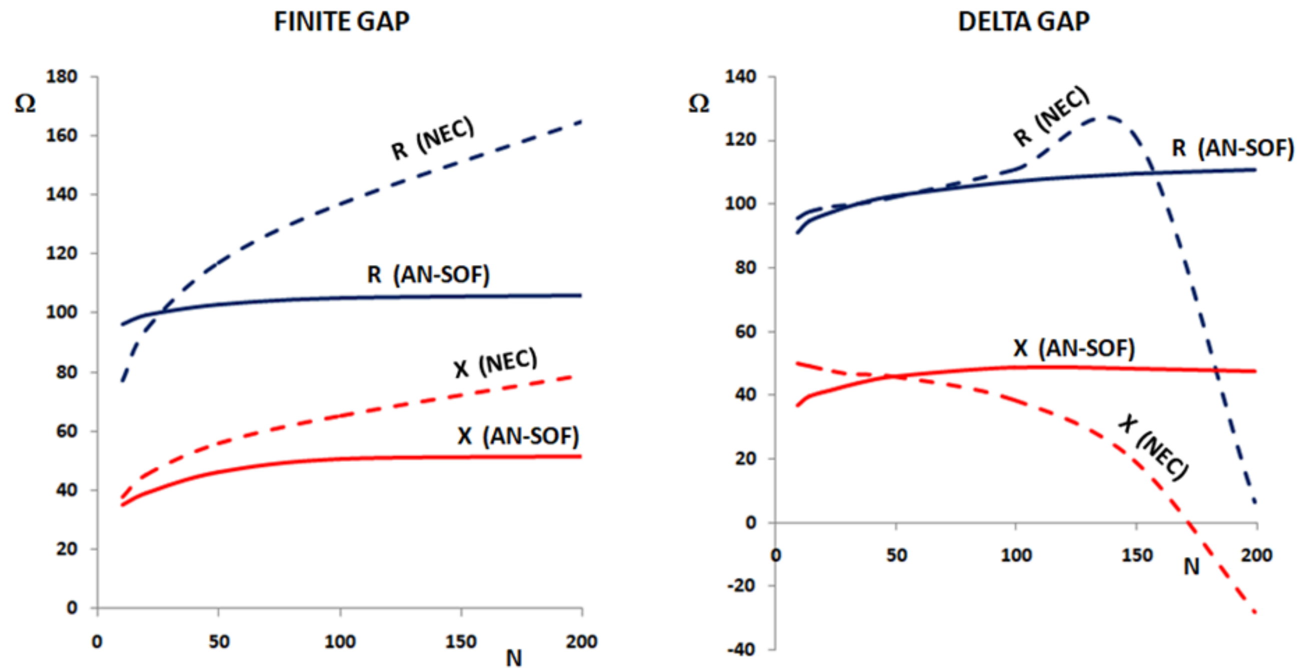


Fig. 2: Comparison between the AN-SOF and NEC-2 results for the input impedance of a center-fed half-wave dipole as a function of the number of segments used in each dipole arm. Radius = 0.005λ, gap = 0.025λ.

> **Note**
>
> In conclusion, we will achieve the **highest possible accuracy** in the calculation of the current distribution and antenna input impedance by using the **Exact Kernel**.

Conformal Method of Moments

The Method of Moments (MoM) is a technique used to convert the EFIE into a system of linear equations that then can be solved by standard methods. For simplicity, the integral (linear) operator in the **Electric Field Integral Equation >** (EFIE) will be denoted by *L*. Then, the EFIE takes the form:

$$L(I) = E_T$$

Eq. (1)

where E_T is the tangential component of the incident electric field. The current distribution is approximated by a sum of *N* basis functions with unknown amplitudes I_n, giving:

$$I = \sum_n I_n F_n$$

Eq. (2)

With this expansion and using the linearity of the operator *L*, we can write:

$$\sum_n I_n L(F_n) = E_T$$

Eq. (3)

To obtain a set of *N* equations, the functional equation (3) is weighted with a set of *N* independent testing functions w_m, giving:

$$\sum_n I_n \int w_m L(F_n) ds = \int w_m E_T ds$$

Eq. (4)

where the integrals are calculated over the domain of *L*. Now we have as many independent equations as unknowns, so (4) can be written as:

$$[Z] \cdot [I] = [U]$$

Eq. (5)

where

[*Z*]: impedance matrix with dimension *NxN* and the elements

$$Z_{mn} = \int w_m L(F_n) ds$$

[*I*]: current matrix with dimension *Nx1* and the elements I_n.

[*U*]: voltage matrix with dimension *Nx1* and the elements

$$U_m = \int w_m E_T ds$$

This fully occupied equation system must be solved for the unknown currents I_n. LU decomposition is used in AN-SOF. The MoM is applied by first dividing the wire structure into *N* segments, and then defining the basis and testing functions on the segments. Triangular basis and pulse testing functions are used in AN-SOF, Fig. 1.

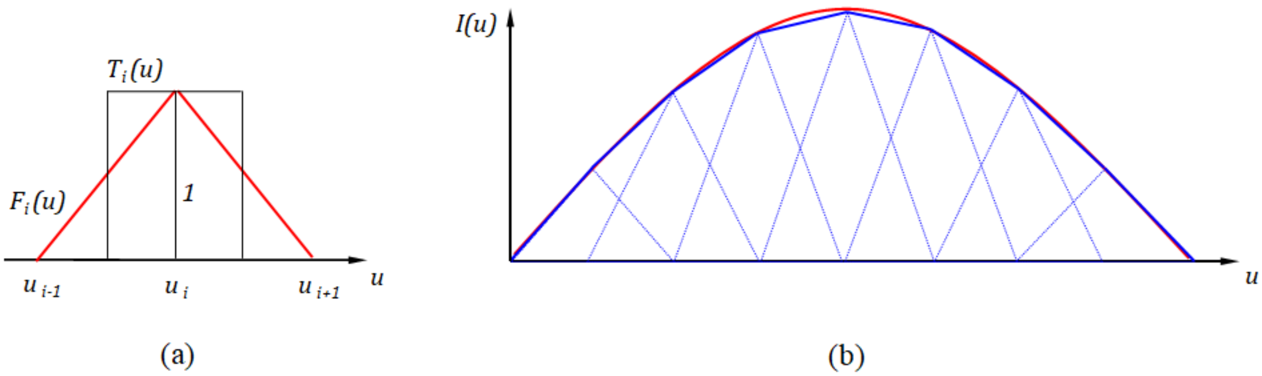


Fig. 1: (a) Triangular basis functions, $F_i(u)$, and pulse testing functions, $T_i(u)$. (b) Current distribution approximated by triangular functions.

When a curved wire is described parametrically by a vector function as in **Eq. (5) here >**, the basis and testing functions are curved in the sense that their support is a curved subset of the wire. Therefore, when curved basis and testing functions are used, the **Conformal Method of Moments (CMoM)** is obtained.

To fill the impedance matrix [*Z*], an adaptive Gauss-Legendre quadrature rule is applied to compute the involved integrals. After having solved the equation system, the currents I_n are known and other parameters of interest, such as input impedances, voltages, radiated power, directivity, and gain can be computed.

The MoM can also be used to calculate the electromagnetic response of metallic surfaces, which are modeled using wire grids. In AN-SOF, with the CMoM the accuracy of the calculation of wire grids is remarkably improved compared to the traditional MoM, as demonstrated in **this article >** . Another extension of the calculation includes wires that do not have a circular cross section. In AN-SOF an equivalent radius is calculated for these wires.

Excitation of the Structure

If a discrete voltage source is placed at the i-th segment, the corresponding element in the voltage matrix is simply equal to the voltage of the generator. Thus,

$$[U] = \begin{bmatrix} 0 \\ \cdots \\ U_i \\ \cdots \\ 0 \end{bmatrix}$$

Eq. (1)

When an incident plane wave is used as the excitation, each wire segment is excited by the incoming field, which has the form:

$$\mathbf{E}_i(\mathbf{r}) = \mathbf{E}_0 e^{-j\mathbf{k}\cdot\mathbf{r}}$$

Eq. (2)

where **k** is defined by the direction of propagation, so that |**k**| = *k* is the wave number, and **r** is the evaluation point, Fig. 1. The elements of the voltage matrix are then defined by:

$$U_m = \int_{S_m} \mathbf{E}_i(\mathbf{r}(s)) \cdot \mathbf{T}(s) ds$$

Eq. (3)

where the integration is performed over the m-th segment, and the vectors **r**(*s*) and **T**(*s*) are given by **Eqs. (5) and (6) here >**.

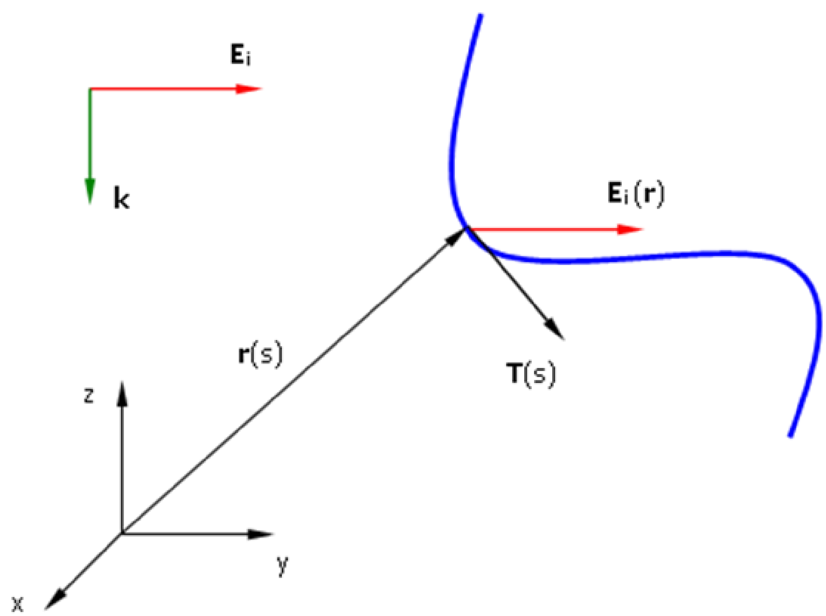


Fig. 1: Incident plane wave exciting a wire.

Curved vs. Straight Segments

Many examples show the advantages of using curved segments with respect to the stability and convergence properties of the solutions. Due to the improved convergence rate, accurate results can be obtained with reduced simulation time and memory space.

Fig. 1 shows the dimensions of a center-fed helical antenna in free space (normal mode). Figs. 2 and 3 show a comparison between AN-SOF, which uses curved segments, and a straight wire approximation to the helix of Fig. 1. The convergence properties of the input impedance and admittance versus the number of segments are investigated.

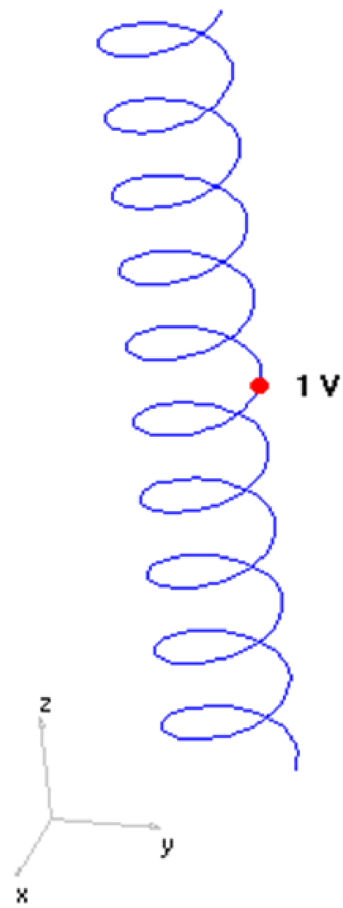


Fig. 1: Helix radius = 0.0273λ. Pitch = 0.0363λ. Number of turns = 10. Wire radius = 0.001λ.

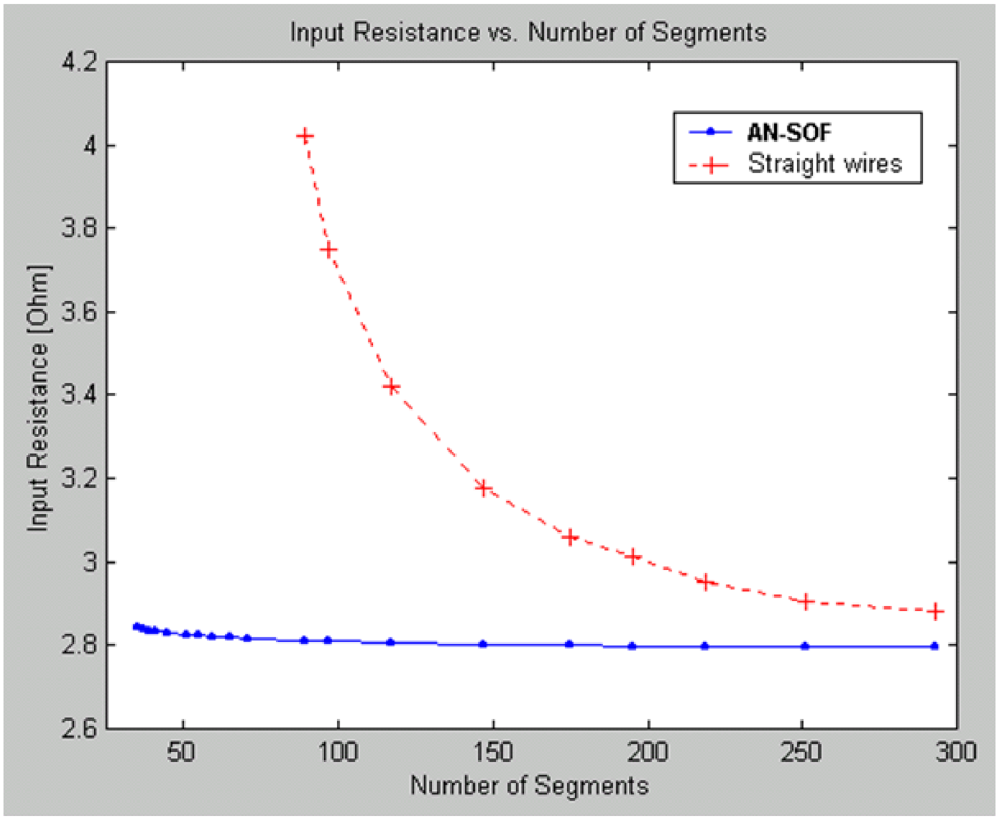


Fig. 2(a): Resistance convergence plot for the helix of Fig. 1.

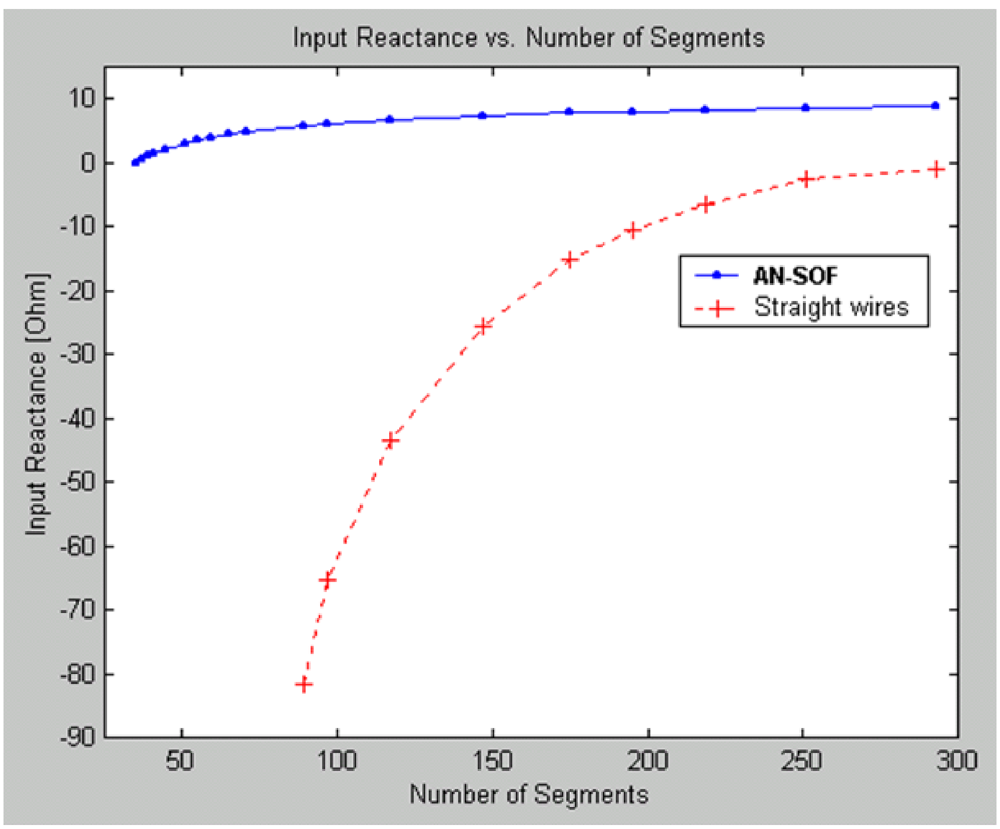


Fig. 2(b): Reactance convergence plot for the helix of Fig. 1.

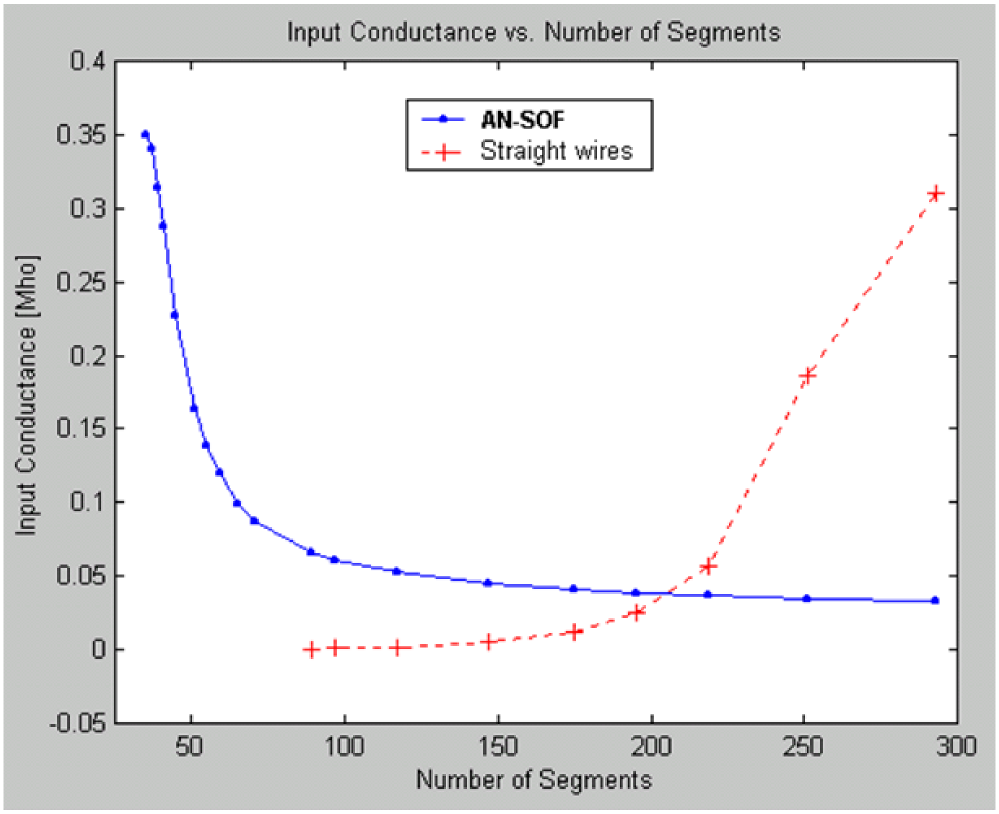


Fig. 3(a): Conductance convergence plot for the helix of Fig. 1.

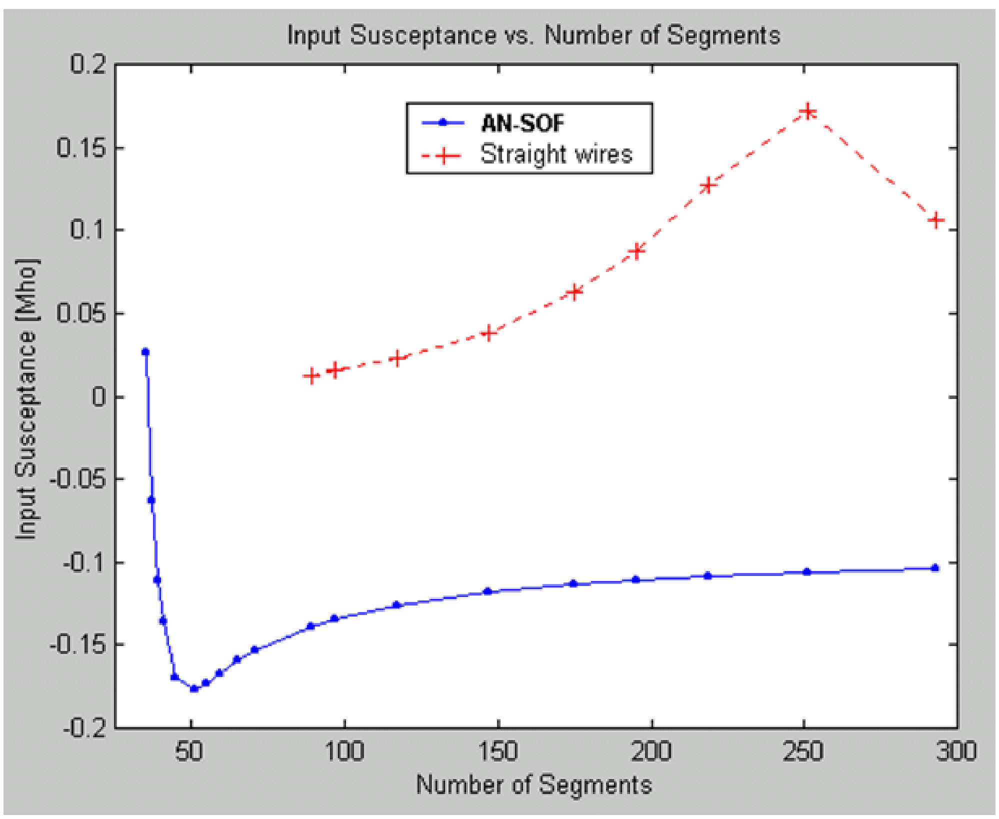


Fig. 3(b): Susceptance convergence plot for the helix of Fig. 1.

As can be seen from these results, by using curved segments significantly fewer unknowns are needed to predict the input impedance. However, the admittance convergence is questionable for the straight wire case, while it has a notorious convergent behavior for the curved case.

The improvement depends on the geometry and frequency, but generally, if N straight segments are needed to obtain a convergent value, then N/p curved segments are needed to obtain the same value, with p = 2…10. For a straight geometry the improvement factor is p = 1, as can be expected, because there are no curved segments in this case.

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?

No. There is only a limit of **50 unknowns** (segments + wire connections + ground connections) **to run the calculations**. All example files included in the installation directory are pre-computed and can be opened with the trial version of AN-SOF to display tables and graphs without limitations. Download examples that have less than 50 unknowns from **this link >**.

3. Will AN-SOF stop working when my plan ends?

No, AN-SOF will continue to function even after your plan expires. However, you will lose access to the following features:

- **Download updates:** You will no longer be able to download and install the latest versions of AN-SOF, including bug fixes and security updates.
- **Technical support:** You will no longer be eligible to receive technical support from our team.
- **Installer access:** Installers for discontinued versions of AN-SOF will only be available for download for **three years after their discontinuation**. After this period, you will need to purchase a new plan to access the installers.

- **Activation key:** If you lose the activation key for a discontinued version of AN-SOF, **we will not be able to provide you with a new one after three years of discontinuation.**

Therefore, while AN-SOF will continue to function, it is recommended to renew your plan to ensure access to all features and ongoing support.

4. What happens if I don’t renew my plan?

If you don’t renew your plan, you will experience the following:

- **Continued functionality:** AN-SOF will continue to function, but it will be limited to the version you have installed.
- **No updates:** You will not receive any updates, including bug fixes, security patches, or new features.
- **Limited support:** You will not have access to technical support from our team.
- **Version discontinuation:** Your currently installed version of AN-SOF will eventually be discontinued. **After three years of discontinuation, the installer and activation key for that version will no longer be available.**

To continue enjoying the full functionality and support of AN-SOF, it is important to renew your plan before it expires.

5. What is the difference between the GOLD and PLATINUM plans?

The **PLATINUM plan** includes live chat support in addition to ticket and email support, **priority support**, and **early access** to updates and exclusive content. PLATINUM customers also gain **Premier Membership**, granting access to the Early Access Portal and additional exclusive benefits.

In contrast, the **GOLD plan** offers ticket and email support only. However, customers who renew the GOLD plan annually without interruption also qualify for **Premier Membership**.

6. 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.

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?

There is no theoretical upper limit for the frequency since the structure size is **measured in wavelengths**. So, we talk about a size limit in wavelengths instead of a frequency limit. The simulation requires solving a matrix equation of increasing order as the structure size is increased in wavelengths, so modeling large structures will require more computer memory and time on a particular PC.

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 be simulated with AN-SOF?

Although AN-SOF was developed to simulate wire structures, we have been able to extend the calculation engine to also model PCBs and microstrip antennas that meet the following requirements:

- The structure must have a **single layer of dielectric substrate**. Multiple layers are not supported.
- When the substrate has a finite size, **the traces must be separated from the edges** by at least 5 times the width of a trace.
- The dielectric substrate is backed up by a **perfectly conducting ground plane** that cannot be removed. Vias that go through the substrate and connect traces to the ground plane can be added to feed an antenna or PCB.

17. What types of DXF files can be imported into AN-SOF?

Only DXF files containing LINE objects can be imported into AN-SOF. See the description of the LINE object and download examples from **this link >**.

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.
Disclaimer of Warranty

This Knowledge Base and associated software are provided “as is” without warranty of any kind, expressed or implied. Golden Engineering Ltd. disclaims all warranties, including, but not limited to, the implied warranties of merchantability, fitness for a particular purpose, and non-infringement.

Golden Engineering Ltd. 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 Ltd. 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.