Getting Started with EMIT - Tutorial 6

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2025 R1
January 2025

Conventions Used in this Guide

Please take a moment to review how instructions and other useful information are presented in this documentation.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, "**copy file1**" means you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). For example, "click **HFSS** > **Excitations** > **Assign** > **Wave Port**."
  - Labeled keys on the computer keyboard. For example, "Press **Enter**" means to press the key labeled **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, "**copy** *filename*" means you must type the word **copy**, then type a space, and then type the name of the file.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, "Press Shift+F1" means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Schematic** > **Line**"

  This instruction means that you should click the **Line** command on the **Schematic** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation> Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, http://www.ansys.com/Support. You can also contact your Ansys account manager in order to obtain this information.

All Ansys software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu

To access help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** - opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** - opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** - opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Introduction

The focus of this tutorial is to learn about the use of the EMIT Elements Library and explore how EMIT's simulation engine runs for larger scenarios.

Key Concepts

- Introduction to libraries in EMIT
- Creating libraries and adding components
- Using library components in EMIT projects
- Tunable filters in EMIT
- Understanding how EMIT runs large projects

Project Configuration

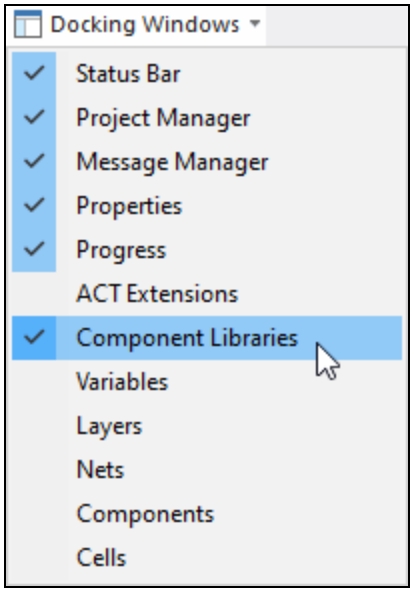
We will build on the project begun in Tutorial 4 and use the EMIT Elements Library to complete the scenario. You can load a completed version of the Tutorial 4 project archive from [Installation Folder]\Win64\Examples\EMIT\Tutorials\Tutorial 4 or work with your own version of the Tutorial 4 project.

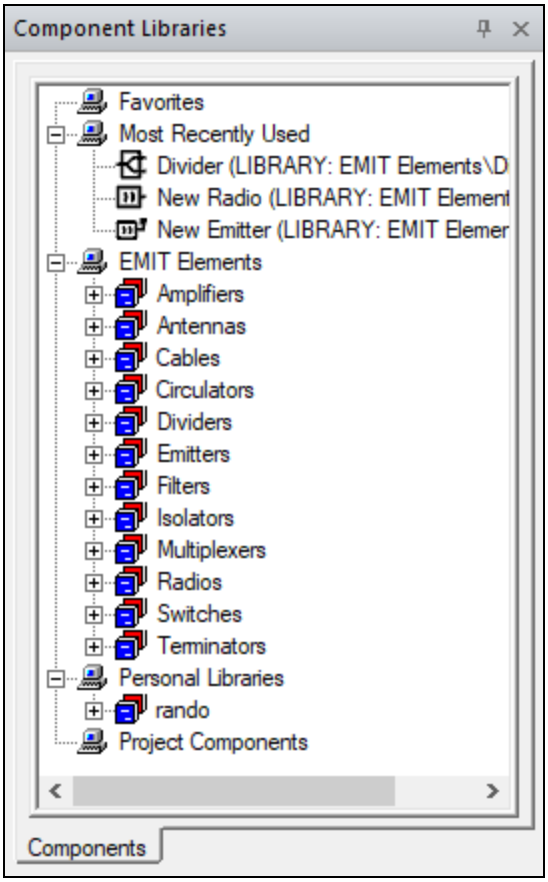
2 - EMIT Libraries

The EMIT Elements Library provides a way to create and store any components and RF systems that can be used in an EMIT project. The models stored in the EMIT Elements Library are available for use in any EMIT project and can be archived for storage and for portability to share libraries between users.

The EMIT Elements Library is controlled from the **Component Libraries** window. Access it one of two ways:

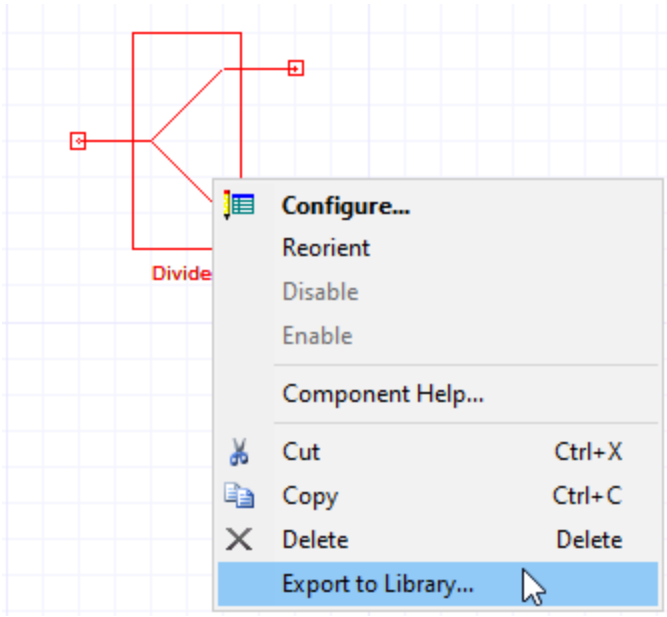
- From the **View** menu, select **Component Libraries**, or
- From the **View** ribbon, click **Docking Windows** > **Component Libraries**.





A new installation of EMIT includes multiple Ansys libraries containing components that can be used in EMIT projects. These are permanently locked libraries that cannot be changed by the user. However, components can be exported into your Personal Libraries, where they are fully editable.

After making changes to a component, right-click it and select **Export to Library** to save it to a personal library.



This makes the component easily accessible across multiple designs and/or projects.

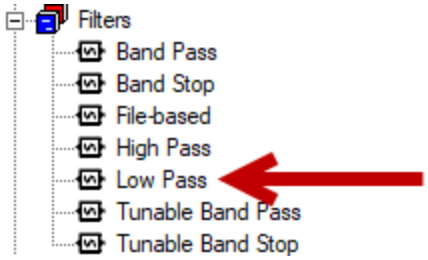
Once stored in a personal library, right-click a component for options to rename or delete it.

3 - Adding Components to Libraries

Components added to a library will be available for use in any project containing an EMIT design. Components can be easily moved from a project to a library.

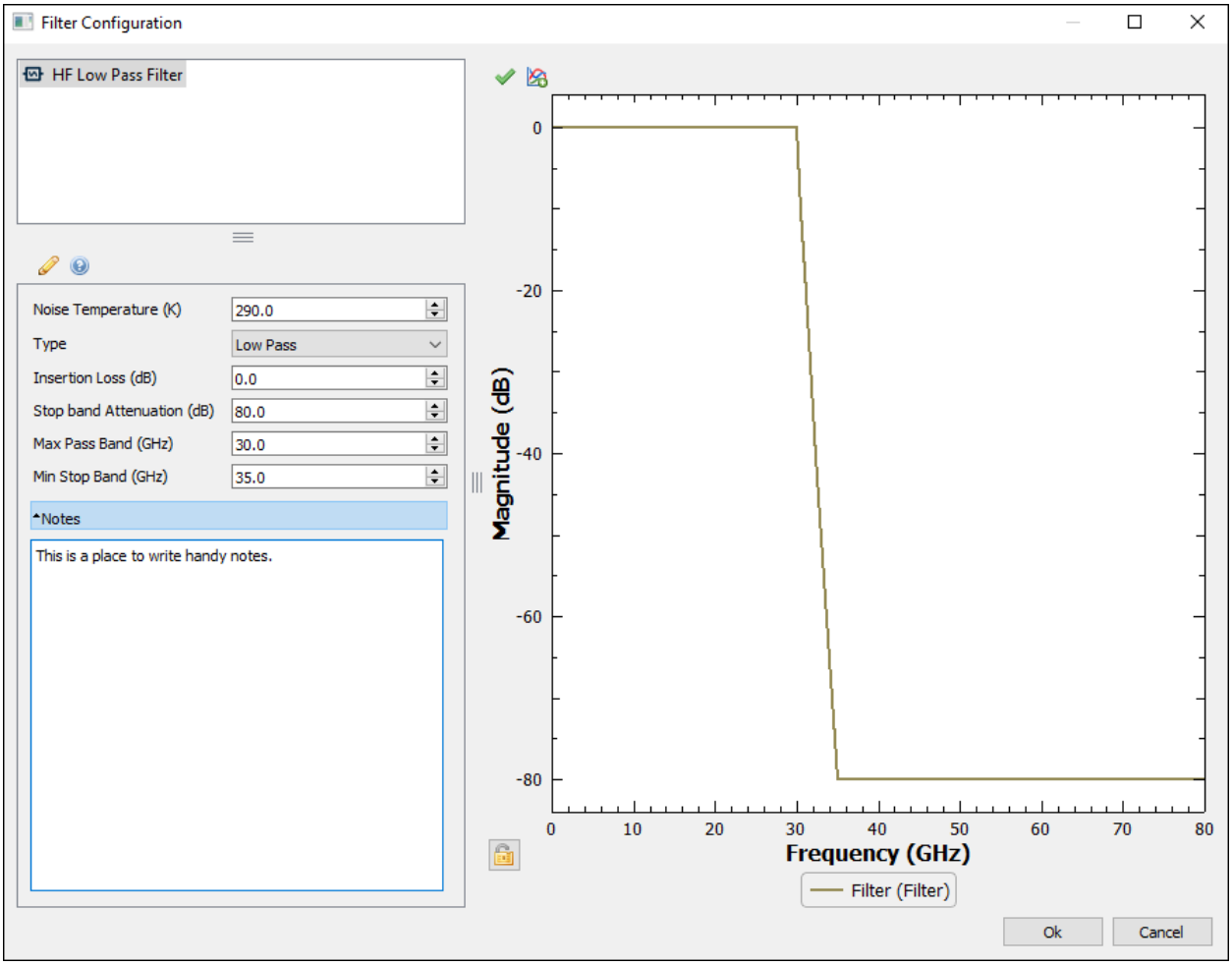
Suppose we wish to create a filter that will be available in a library.

To begin, insert a low pass filter into the project from the **Components** window:



Using the **Properties** window, rename the filter to *HF Low Pass Filter*, then double-click to edit the filter.

The **Filter Configuration** window appears:



Input the parameters:

- Insertion Loss (dB): 0.0
- Stop Band Attenuation (dB): 80.0
- Max Pass Band (GHz): 30.0
- Min Stop Band (GHz): 35.0

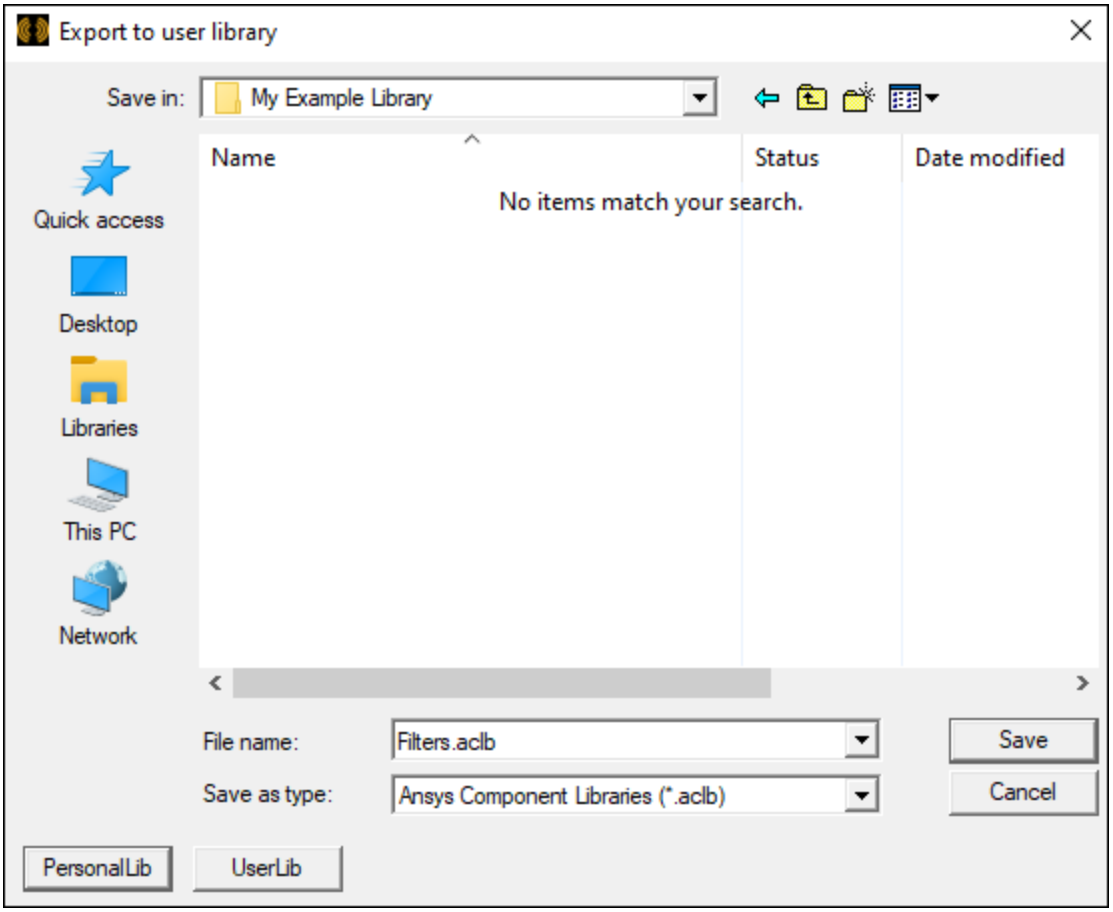
> **Tip:**
>
> The **Notes** field is a useful way to document your library components for future reference.

To add the filter to a library, right-click on the filter in the schematic and select **Export to Library**. This opens a file explorer in the PersonalLib directory. The location of this directory can be

changed in Electronics Desktop from the menu **Tools** > **Options** > **General Options** > **General** > **Directories**.

Create a folder named “My Example Library” and navigate into that folder.

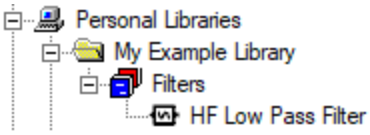
Next, save the filter in a new library named Filters.aclb:



An alert appears, asking if you'd like to create a new user library. Click **Yes**.

The **Component Libraries** window updates. Under **Personal Libraries**, there is now a folder named **My Example Library** containing a library called **Filters**.

This library contains the HF Low Pass Filter:



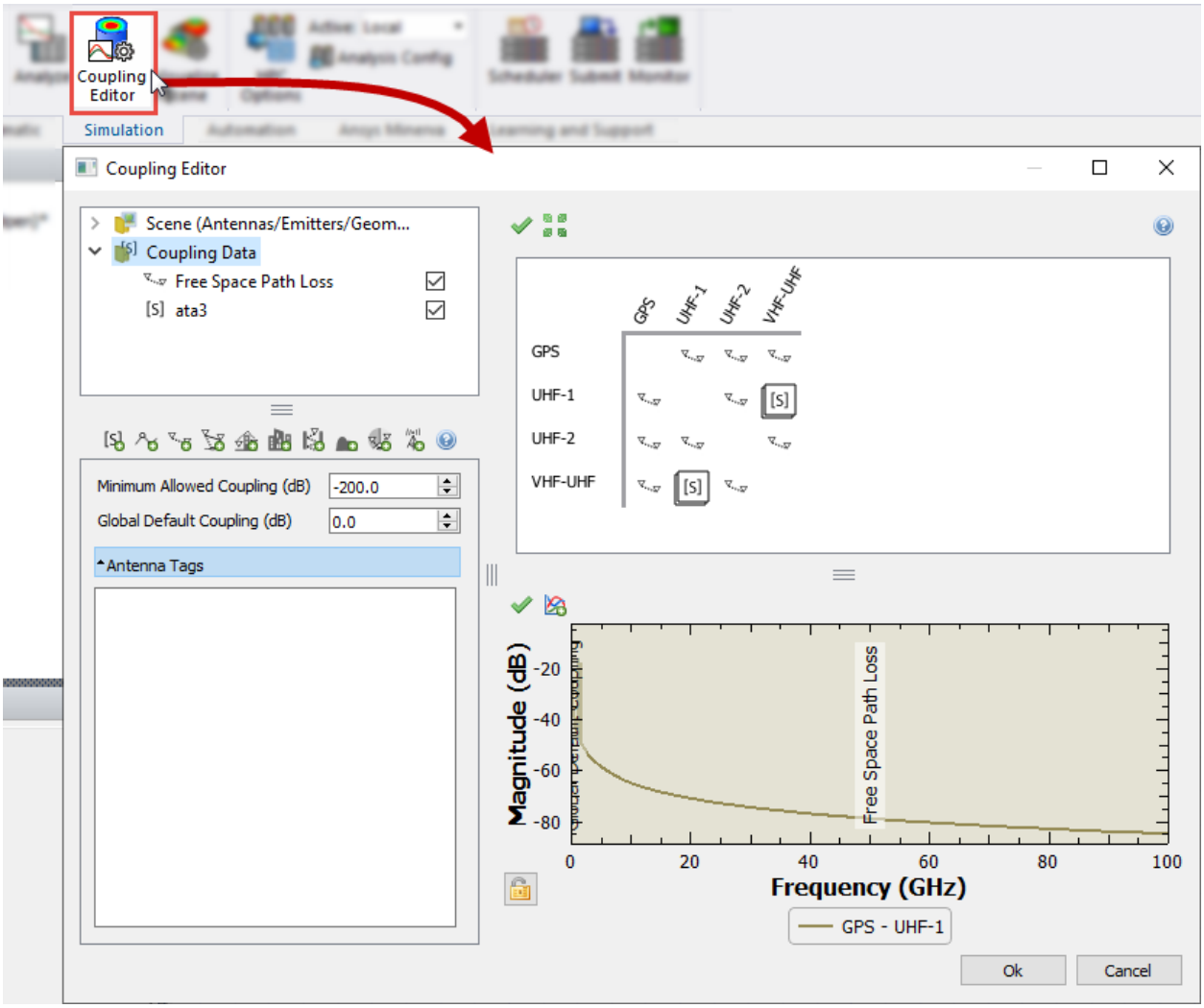
Any changes made to libraries are automatically saved by AEDT and will be available each time you open an EMIT design.

4 - Using Library Components in an EMIT Project

We will use the Ansys Navigation Systems Library to add RF systems to this scenario. The goal is to demonstrate the use of the library in building EMIT scenarios. The result is not intended to represent a real-world or realistic scenario.

First, load the project that you began building in Tutorial 4. From the **File** menu, save the project as Tutorial 6.

From the **Simulation** ribbon, launch the **Coupling Editor** and enable all coupling models:

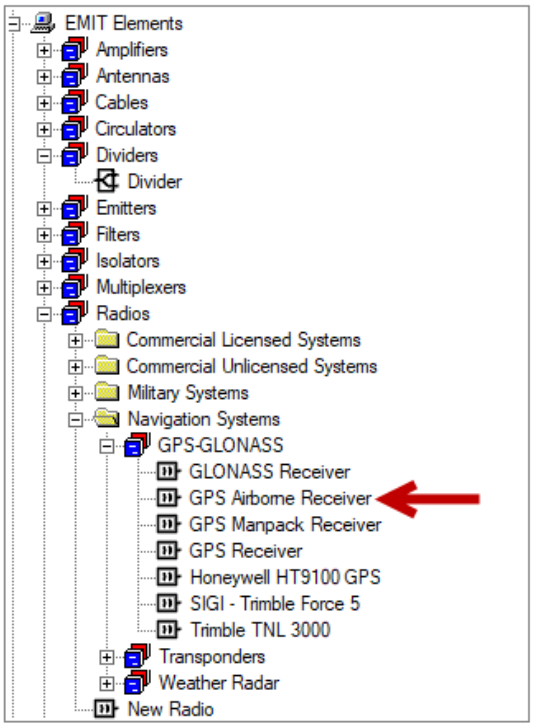


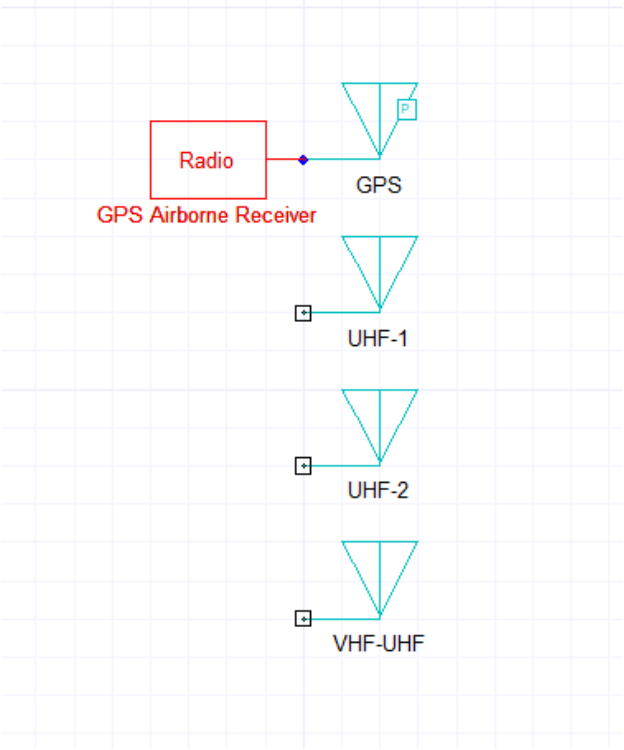
When you are finished, click **OK** to close the **Coupling Editor**.

Next, we will add a GPS Rx to the project using a GPS Rx provided in the Navigation Systems Library. We must first create the desired RF system in the project, and then add the radio from the library to an existing RF system.

Most radios in the Ansys Libraries are not intended to represent any specific brand or model of radio. Instead, they provide a model that is broadly representative of radios used for a particular function. The parameters are based on typical values averaged over a range of actual hardware specifications. After adding a radio to an RF System, the user still needs to set up the RF System's configuration, including the hardware connections and the radio's sampling.

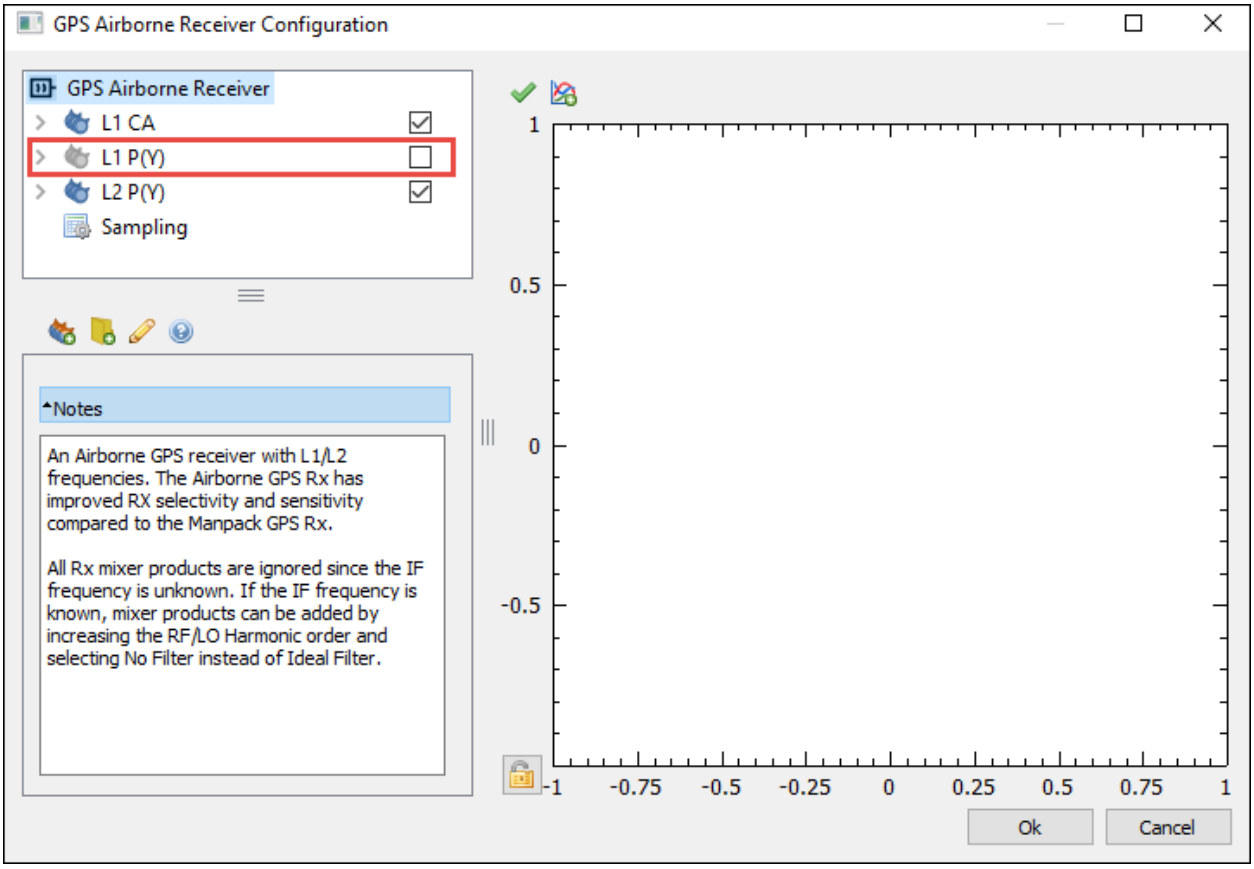
The GPS Airborne Receiver can be found in the **Component Libraries** window under **EMIT Elements** > **Radios** > **Navigation Systems** > **GPS-GLONASS**. Add a **GPS Airborne Receiver** to the schematic, and ensure the radio is connected to the previously placed GPS antenna:





Right-click the radio and select **Configure** to launch the **GPS Airborne Receiver Configuration** window.

The radio has 3 bands (L1 CA, L1 P(Y), and L2 P(Y)), each of which has 1 channel. We are only interested in the L1 CA and L2 P(Y) bands, so disable the L1 P(Y) band.



Suppose that the spec sheet for the particular GPS Rx that we will be using provides sensitivities of -140 and -150 dBm @ -32 dB SNR for L1 CA and L2 P(Y), respectively. In this case, we wish to adjust the radio model to more reliably represent the performance of the specific hardware that we will be using. Let's also assume for the analysis that we require reception at sensitivity. That is, the lowest signal that we wish to receive is at the sensitivity level of the Rx.

We can adjust the generic library radio model to more accurately reflect the specifications of the particular radio that we will actually use on the platform. The Processing Gain for our model is specified at the antenna terminal, so we want to apply it to both the broadband noise and the narrowband noise. Thus, we will make sure Apply PG to Narrowband Only is False for both bands.

Expand the nodes under the GPS Airborne Receiver radio node and select the **Rx Spectral Profile**.

For the L1 CA band, apply the following settings:

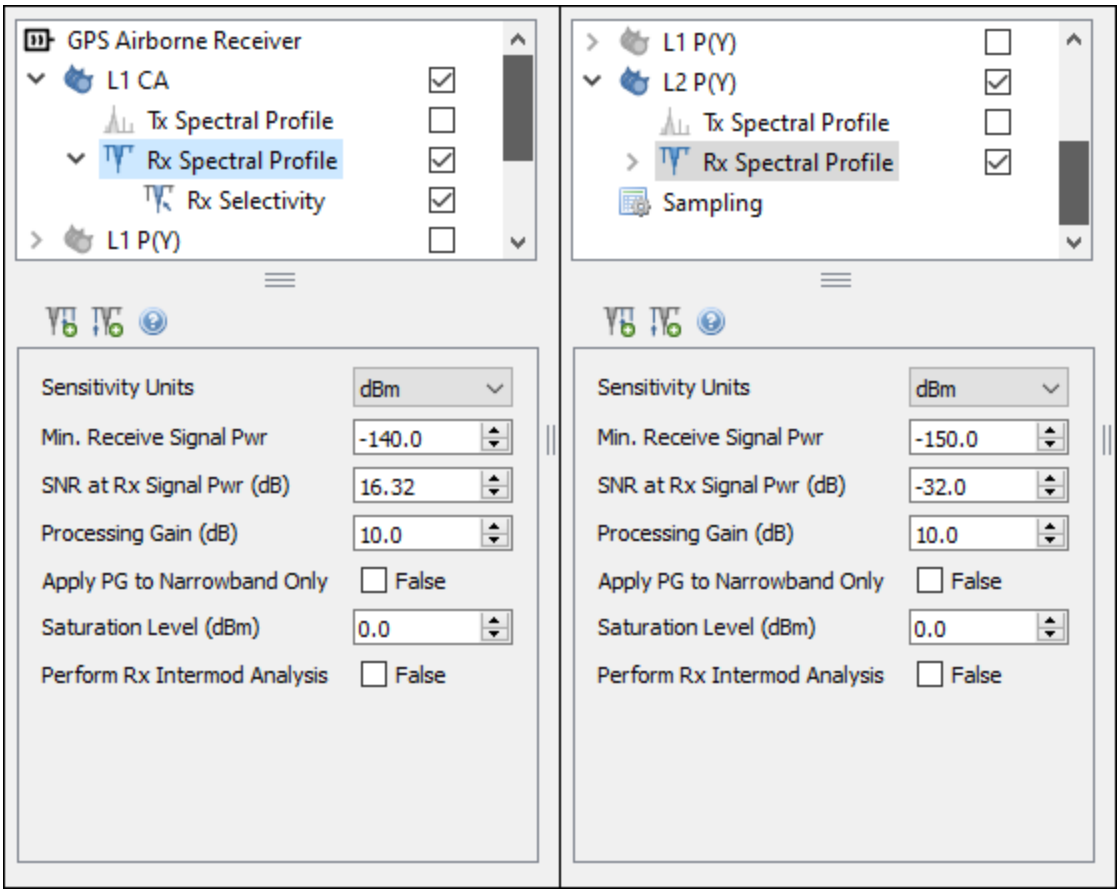
- Min. Receive Signal Pwr (dBm): –140 dBm
- SNR at Rx Signal Power (dB): 16.32
- Processing Gain (dB): 10.0

- Apply PG to Narrowband Only: False
- Saturation Level (dBm): 0.0
- Perform Rx Intermod Analysis: False

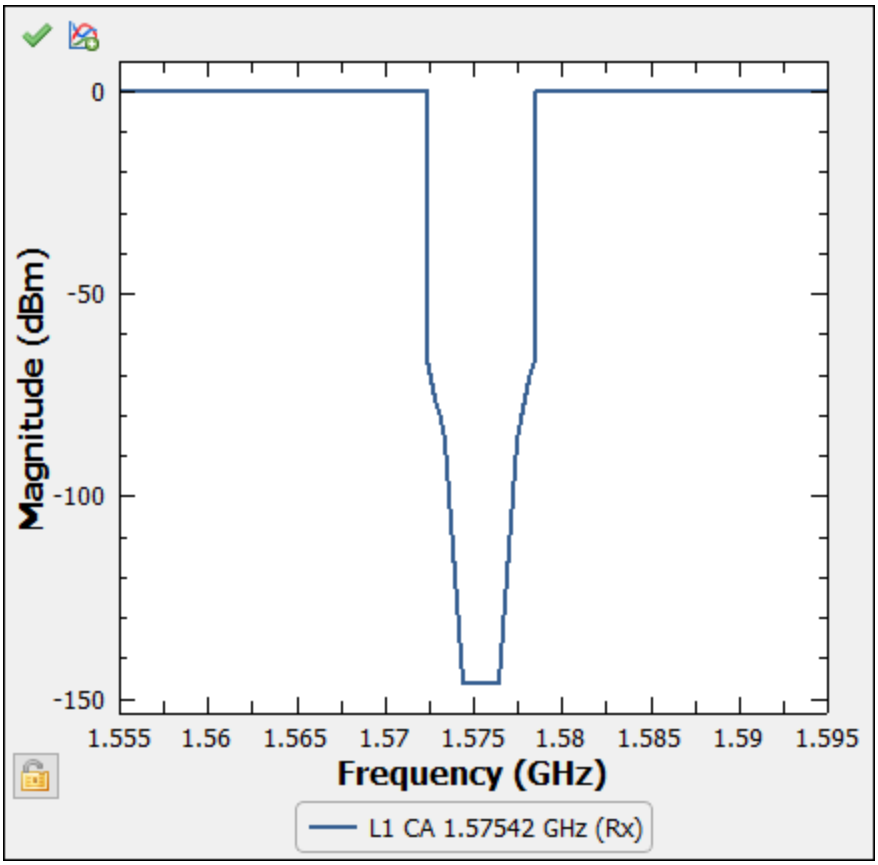
For the L2 P(Y) band, apply the following settings:

- Min. Receive Signal Pwr (dBm): –150 dBm
- SNR at Rx Signal Power (dB): –32.0
- Processing Gain (dB): 10.0
- Apply PG to Narrowband Only: False
- Saturation Level (dBm): 0.0
- Perform Rx Intermod Analysis: False

The spectral profiles for the L1 CA and L2 P(Y) bands are shown on the left and right in the figure below, respectively.



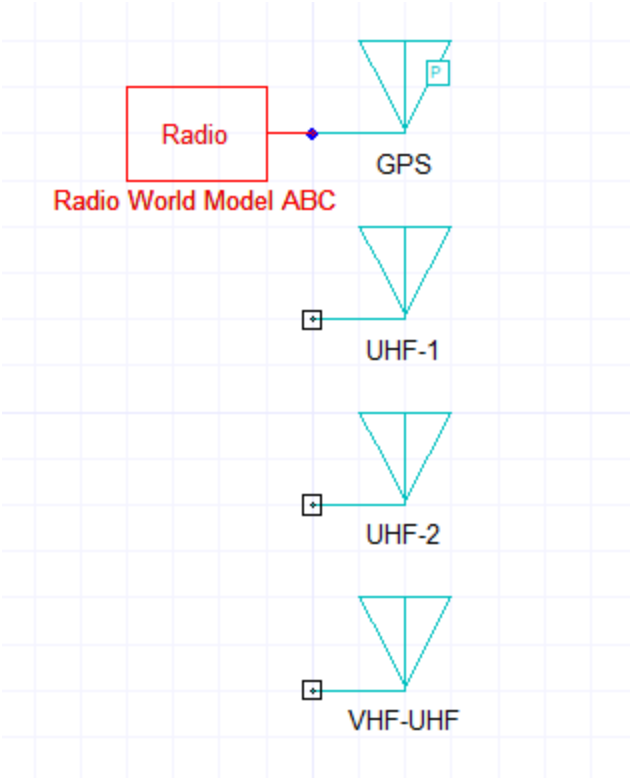
Plot the L1 CA band to see the Rx susceptibility spectral profile for the Rx on the L1 frequency. You may need to adjust the plot axis limits in the Plot configuration panel to get the exact plot shown below.



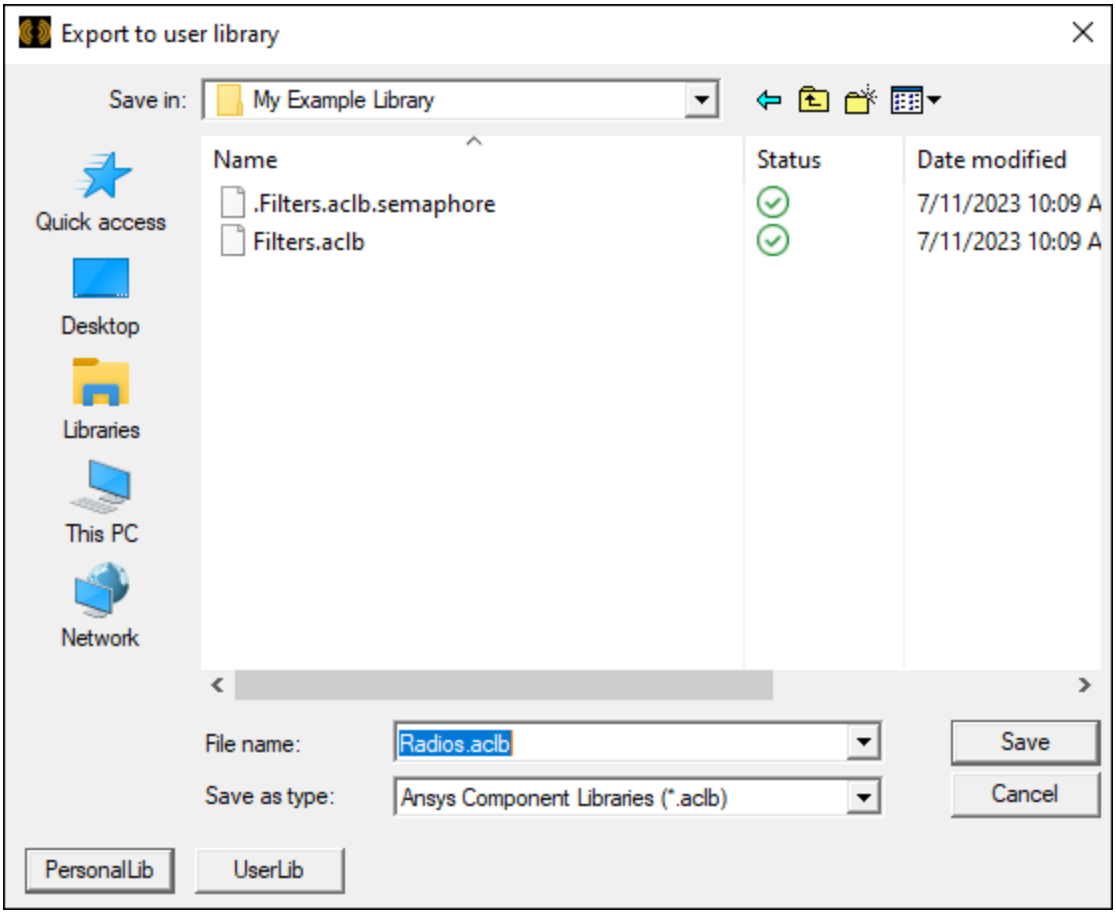
Notice that the Rx response only contains the in-channel response (that is, no spurious responses are included). This is because for a generic radio model, the IF of the Rx and the location of other spurs is not known. If you have this information available for a specific radio to be used, you will want to add this to the generic radio model from the library.

Once you've modified the generic model in your project to represent the actual hardware more closely, you may want to preserve this model in the library for future use in other projects.

Using the Properties window, rename the generic radio to something more descriptive. For this example, name it **Radio World Model ABC**.

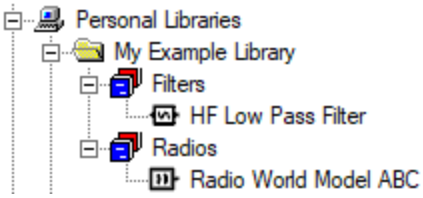


Save the modified radio to the My Example Library folder you created earlier. First, right-click on the component and select **Export to Library**. Navigate into the **My Example Library** folder and save the new library as Radios.aclb.

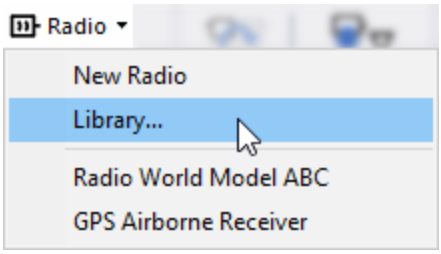


Upon clicking **Save**, an alert appears asking if you'd like to create a new library. Click **Yes**.

The **Component Libraries** window updates to show two libraries in the **My Example Library** folder:



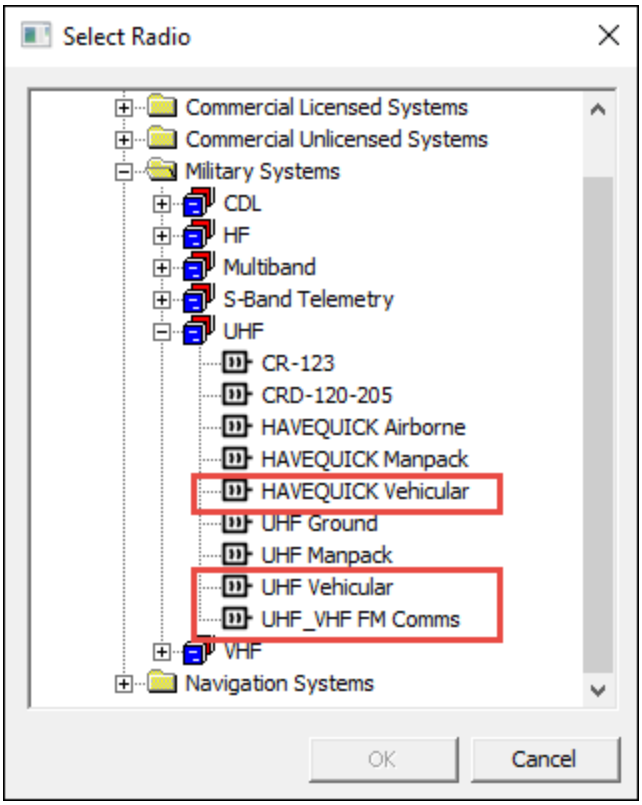
Next, navigate to the **Schematic** ribbon and add three new radios to the project using **Radio > Library**.



The **Select Radio** window appears, containing the EMIT Elements library.

All three radios to add are under **Military Systems** > **UHF**.

Add the radios named **HAVEQUICK Vehicular**, **UHF Vehicular**, and **UHF_VHF FM Comms**.

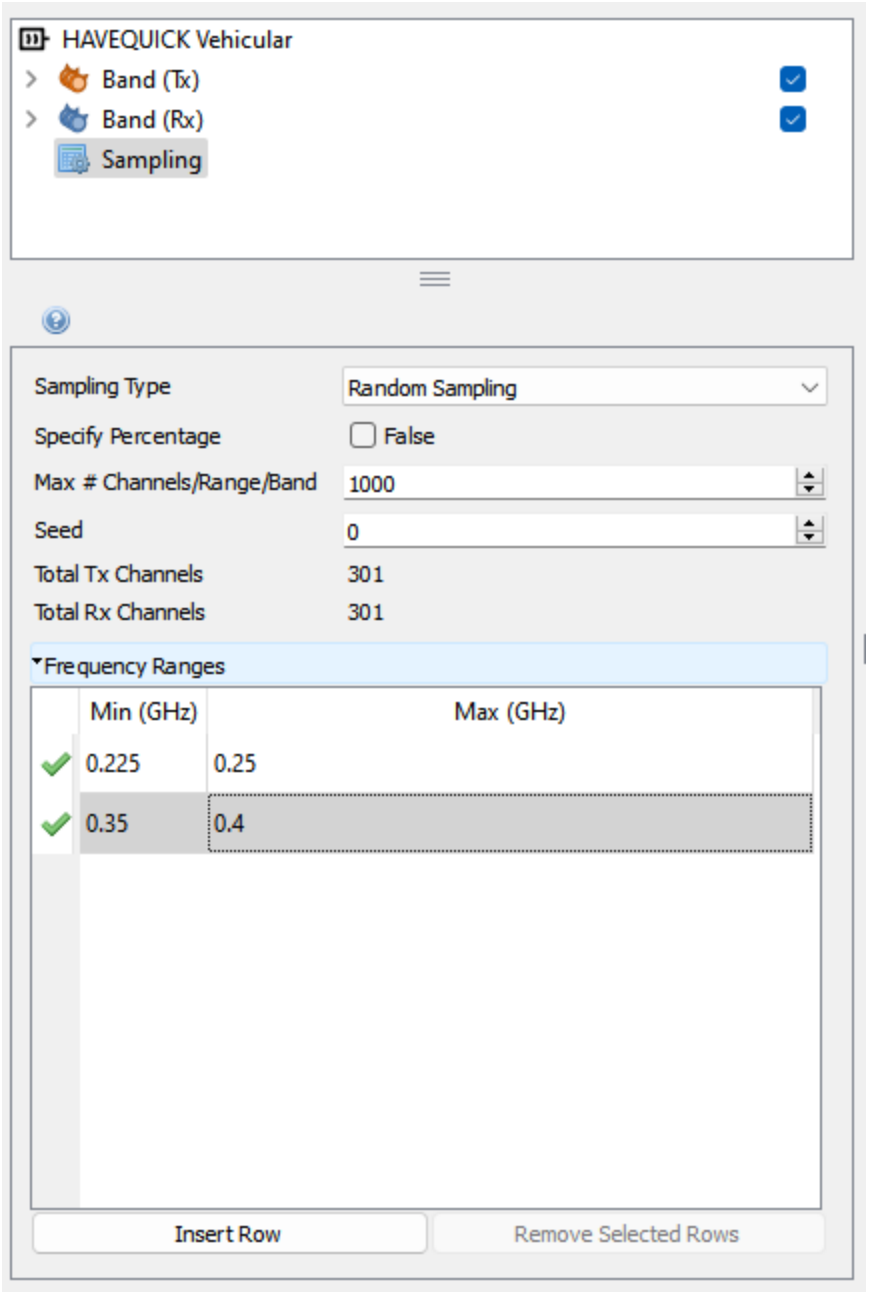


Next, we will modify the radios' samplings and the configurations for these RF systems to use our existing antennas.

Open the configuration window for the HAVEQUICK radio. We want to operate the HAVEQUICK system in its random frequency hopping mode, so set the **HAVEQUICK Vehicular** radio's Sampling parameters to the following:

- Sampling Type: Random Sampling
- Specify Percentage: False
- Max # Channels/Range/Band: 1000
- Seed: 0
- Frequency Ranges: Insert two rows.
  - Row 1: 225 Min (MHz) and 250 Max (MHz)
  - Row 2: 350 Min (MHz) and 400 Max (MHz)

> **Note:**
>
> The unit of measure displayed in the configuration window is determined by settings in **Tools** > **Options** > **General Options** > **EMIT**. You may need to convert the numbers above.



Connect this radio to the UHF-1 antenna.

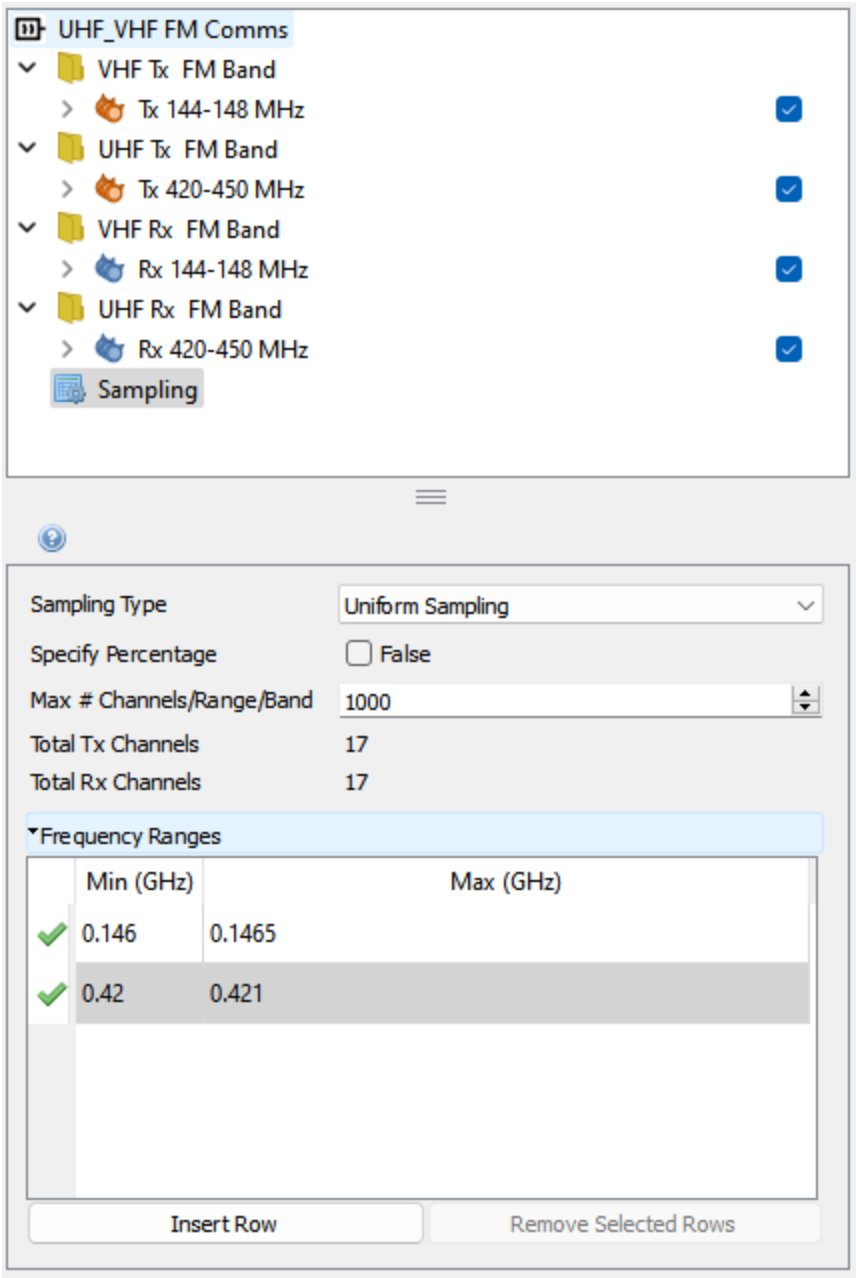
Open the configuration window for the **UHF_VHF FM Comms** radio.

Radios in the Ansys libraries with multiple bands have all the bands disabled by default. This is a result of the radio models being generic and often containing more capabilities than is required for a simulation. In this tutorial, we want all the bands enabled for the UHF_VHF FM Comms RF System, so enable them all.

Set the **Sampling** parameters to the following:

- Sampling Type: Uniform Sampling
- Specify Percentage: False
- Max # Channels/Range/Band: 1000
- Frequency Ranges: Insert two rows.
  - Row 1: 146 Min (MHz) and 146.5 Max (MHz)
  - Row 2: 420 Min (MHz) and 421 Max (MHz)

> **Note:**
>
> The unit of measure displayed in the configuration window is determined by settings in **Tools** > **Options** > **General Options** > **EMIT**. You may need to convert the numbers above.



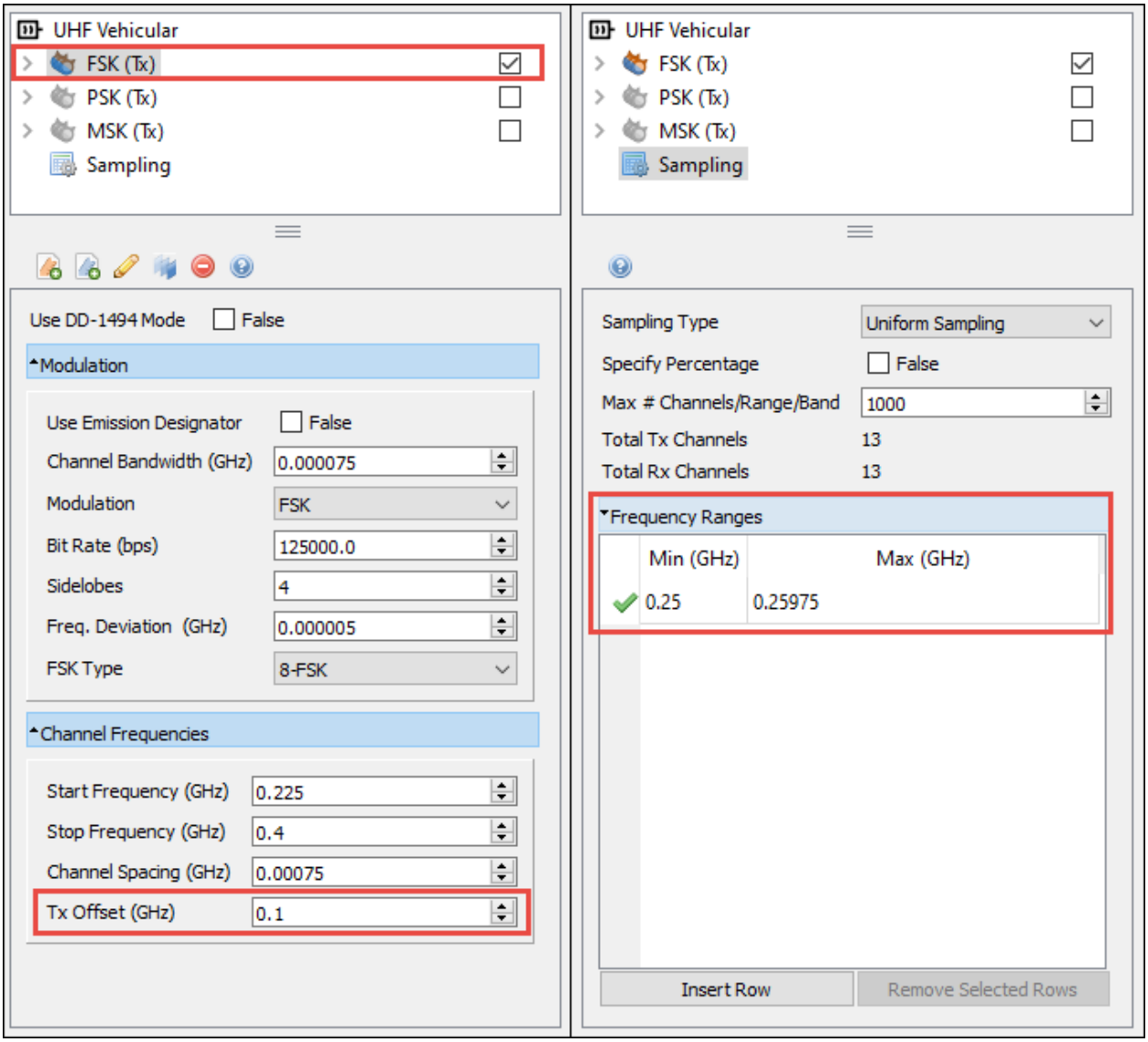
Connect this radio to the VHF-UHF antenna.

The **UHF Vehicular** radio will be operated in full-duplex mode using FSK to receive on channels from 250 to 259.75 MHz and to transmit at a +100 MHz offset from the selected Rx channel. Enable the **FSK (Tx)** bands under the UHF Vehicular radio node. Select the **FSK band** and set the **Tx Offset** to **100 MHz**. Leave the **Sampling** parameters as their defaults.

Insert one row into the Frequency Ranges, with a 250 Min (MHz) and a 259.75 Max (MHz).

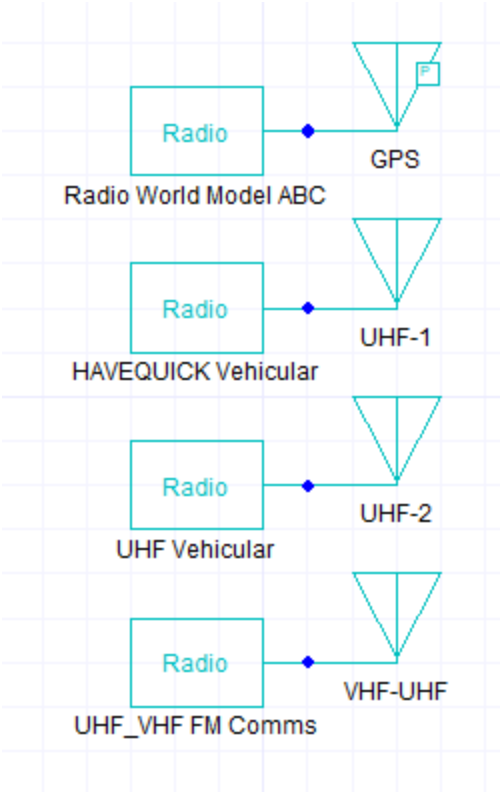
Note:

The unit of measure displayed in the configuration window is determined by settings in **Tools** > **Options** > **General Options** > **EMIT**. You may need to convert the numbers above.



Connect this radio to the UHF-2 antenna.

The schematic should now look like the following:



If other specifications on actual radios' performance were available, we would also want to refine these generic radio models. Remember that the scenario we are creating is not intended to represent a real operational scenario.

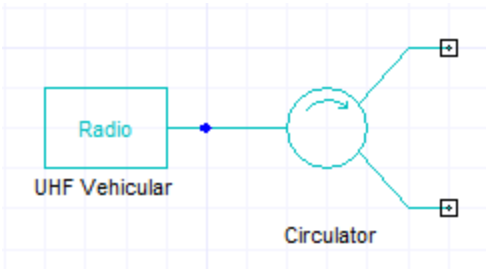
5 - Tunable Filters

Next, we will add tunable (aka. tracking) bandpass filters to the **UHF Vehicular** radio. Tracking filters automatically change their passband characteristics with changes to the Tx and/or Rx frequency of the radio. EMIT supports bandpass as well as bandstop tracking filters.

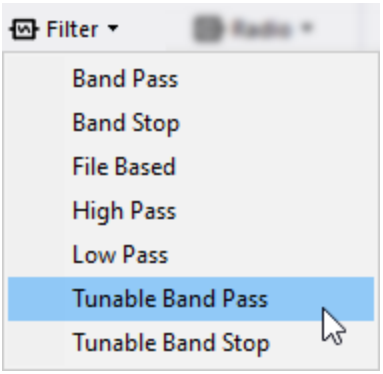
In the case of the UHF Vehicular system in our scenario, the full duplex operation with 100 MHz separation between Tx and Rx channels requires separate filters for the Tx and Rx paths of the radio and EMIT supports this.

The easiest way to add components to an RF system is using the Reorient and Reposition actions in the Schematic Toolbar. Select the **UHF-2** antenna and click **Reposition**.

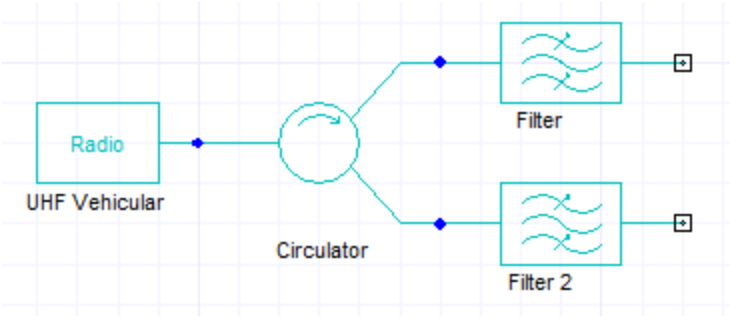
This will automatically disconnect the antenna from the radio. Next, add a Circulator to separate the Tx and Rx paths. To add a Circulator to the diagram, click the **Circulator** button on the Schematic ribbon. With the circulator still selected, connect it to the radio's output port. We will use the default parameters for the circulator in this tutorial. The connection should look like the following:



Next, we will add tunable filters. Add a filter by clicking the **Filter** button on the **Schematic** ribbon and selecting **Tunable Band Pass** from the drop-down menu.

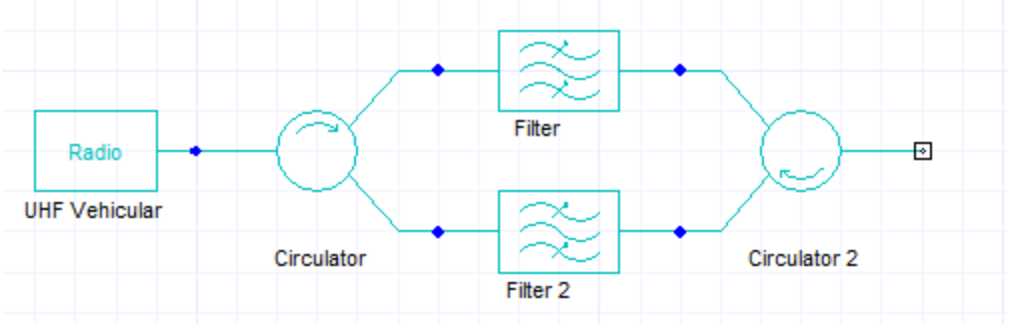


Repeat this to add a second tunable band pass filter. Click **Reposition** to connect it to the circulator's other port.



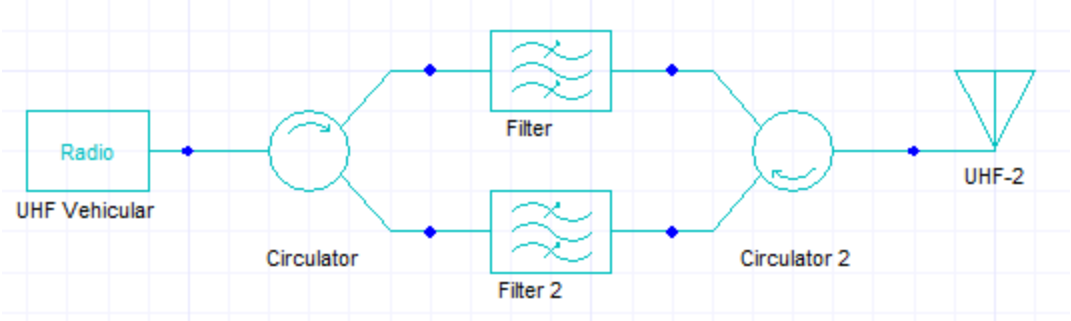
Finally, add a second circulator. By default, circulators are added to the schematic with the single port on the left and the two port side to the right. We can change this quickly by selecting the circulator and clicking **Reorient**.

Click **Reposition** until the circulator is connected as shown below.



As before, we will use the default parameters for the circulator in this tutorial.

Finally, select the **UHF-2** antenna and click **Reposition** until it is connected to the second circulator:



The components have been added to the schematic, but their properties are undefined. Double-click the top filter to open its **Filter Configuration** window.

Set the properties as follows:

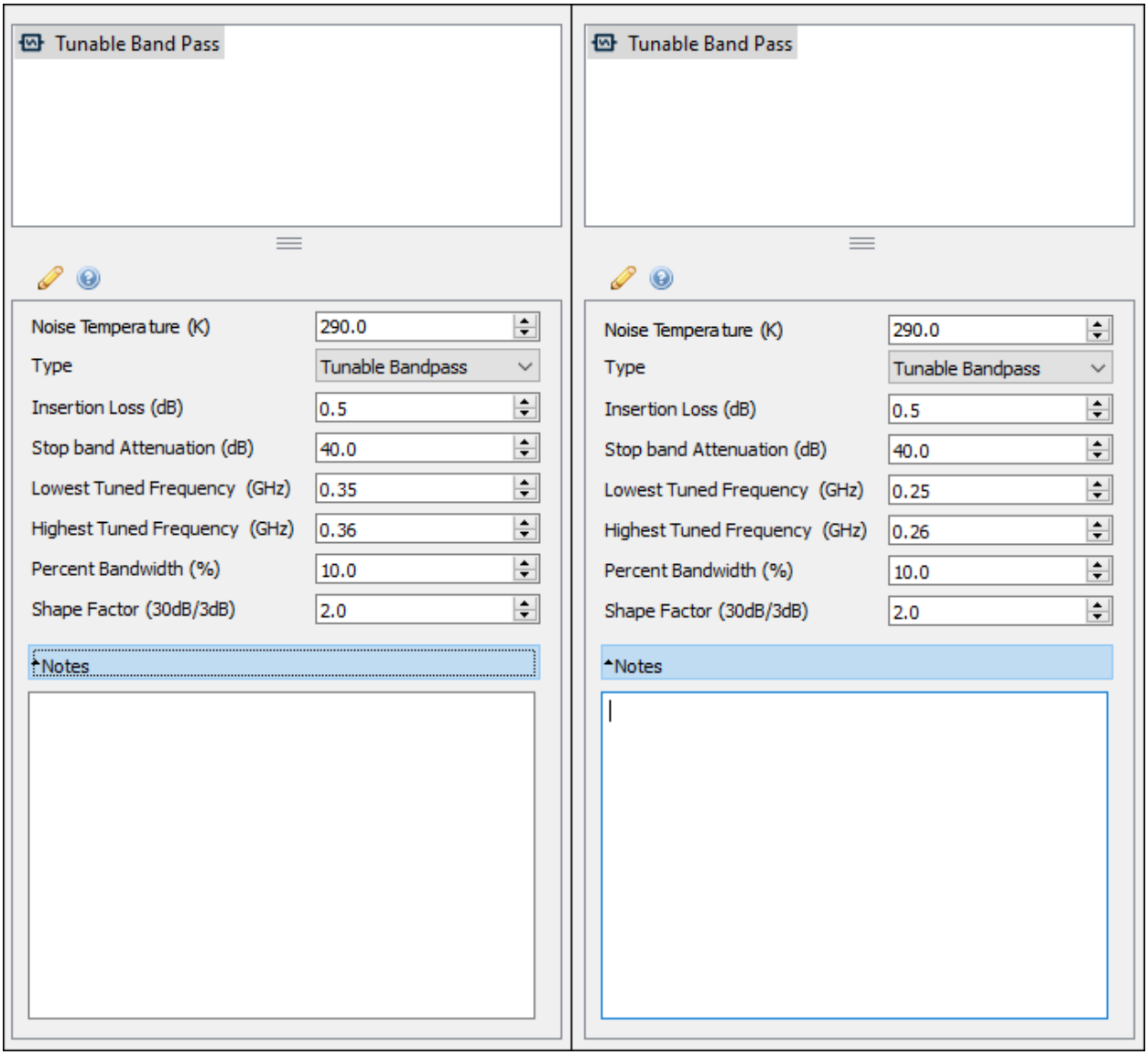
- Insertion Loss (dB): 0.5
- Stop Band Attenuation (dB): 40.0
- Lowest Tuned Frequency (GHz): 0.35
- Highest Tuned Frequency (GHz): 0.36
- Percent Bandwidth (%): 10.0
- Shape Factor (30dB/3dB): 2.0

Double-click on the second filter in the schematic to open its **Filter Configuration** window.

Set the properties as follows:

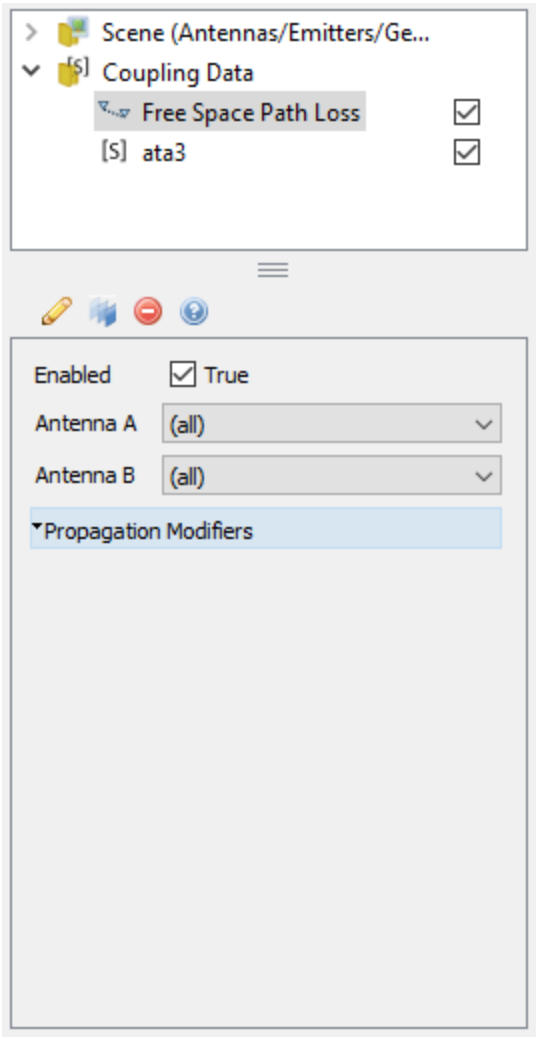
- Insertion Loss (dB): 0.5
- Stop Band Attenuation (dB): 40.0
- Lowest Tuned Frequency (GHz): 0.25
- Highest Tuned Frequency (GHz): 0.26
- Percent Bandwidth (%): 10.0
- Shape Factor (30dB/3dB): 2.0

The properties for both filters are shown below.

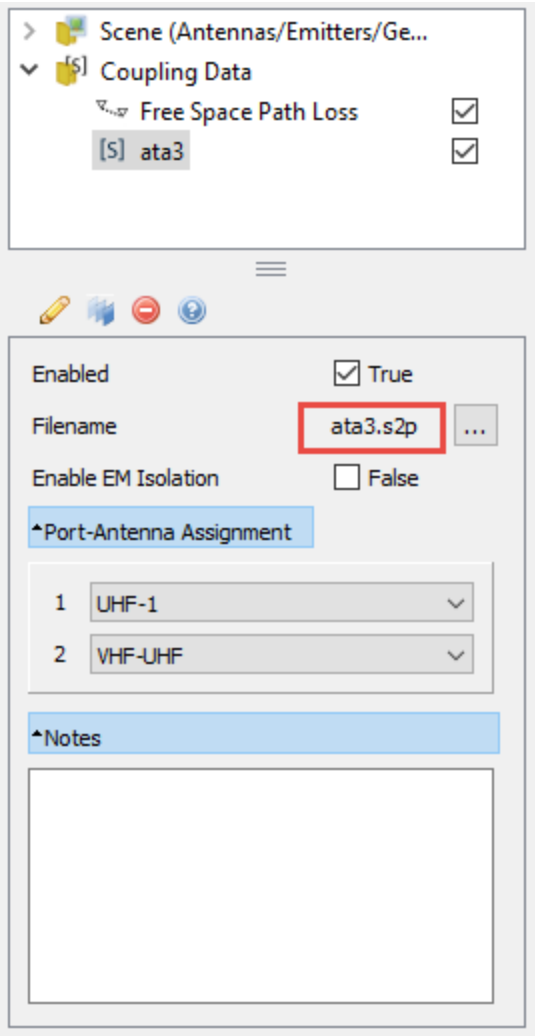


6 - Running the Simulation

Before running the simulation, we must ensure that the coupling is set. From the **Simulation** ribbon, open the **Coupling Editor**. Under **Free Space Path Loss**, ensure **Antenna A** and **Antenna B** are set to **(all)**.



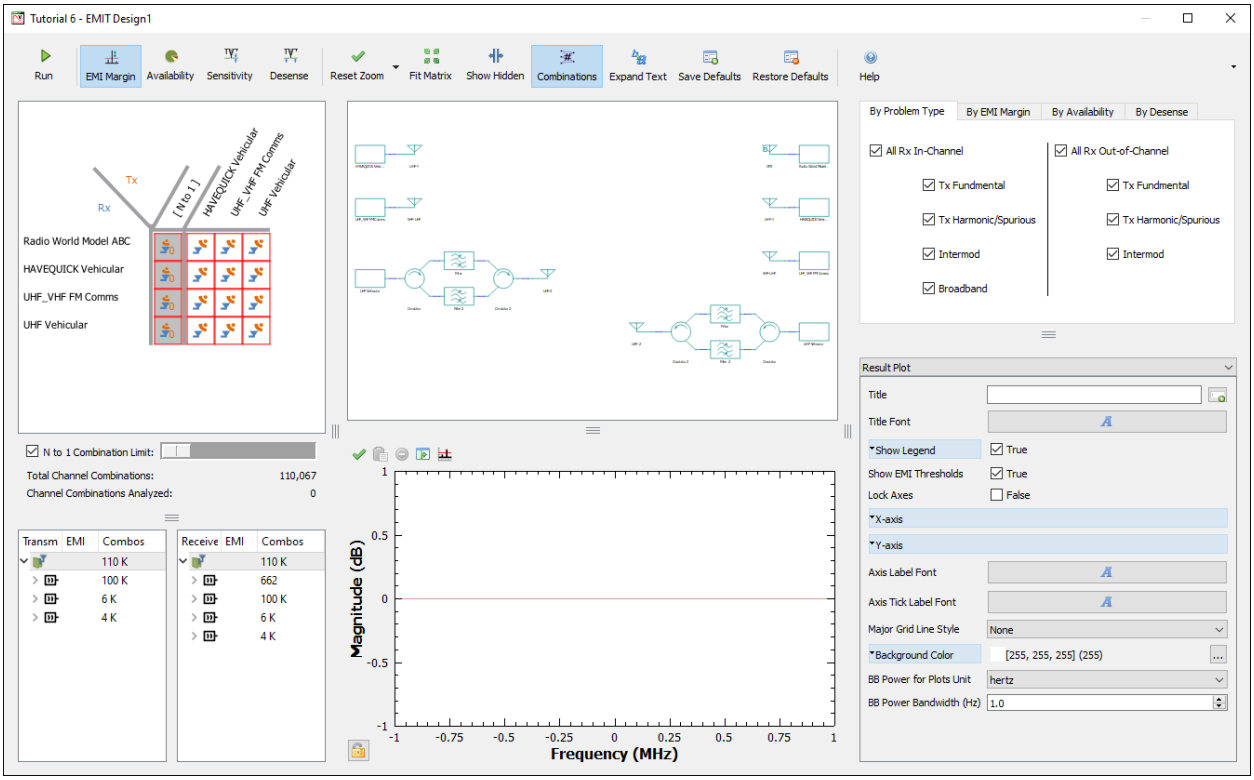
Next, select ata3 settings and ensure that **ata3.s2p** is selected for **Filename**. This file can be found in [Installation Directory]Win64\Examples\EMIT\Tutorials\Tutorial 4.



When you are finished, click **OK** to close the Coupling Editor.

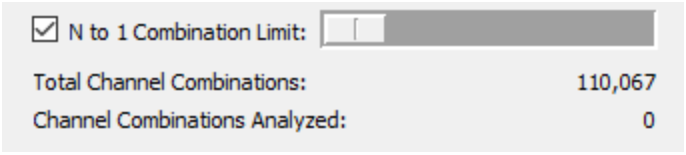
Next, we run the simulation. From the **Simulation** ribbon, click **Analyze**.

The analysis setup launches:

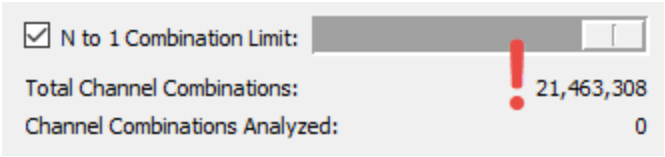


While it is not the intent of this example to examine the results of this scenario in any detail, it will be used to discuss several aspects associated with running larger simulations in EMIT. This is not a particularly large simulation, but it is large enough to demonstrate several features of EMIT designed to deal with large simulations.

At the bottom of the Scenario Matrix, the total number of channel combinations in this scenario is *110,067*:

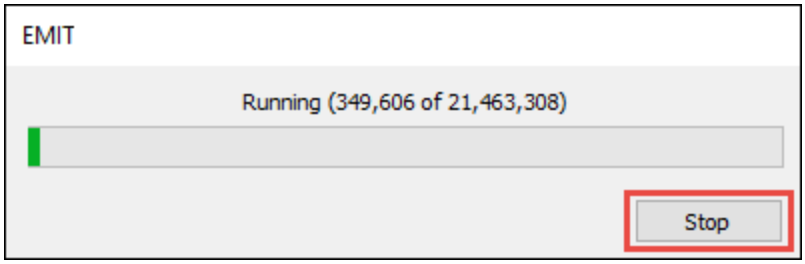


This is the number of Tx/Rx channel pairs that EMIT needs to analyze to complete all of the 1-on-1 simulations. If we slide the N-to-1 Combination slider all the way to the right or deselect the check box, the number balloons to *21,463,308* since multiple channel combinations and permutations must now be accounted for.



A simulation with this many channel combinations could take a long time to run in EMIT, but EMIT has several features that make a potentially time-consuming simulation more palatable:

- EMIT's computational engine can take advantage of computers with multi-core processors to run parts of the simulation in parallel. The user can specify the number of cores (also called threads) to use for the simulation. By default, EMIT uses all available cores, but this is not always desired as it will leave few system resources for other tasks. To control the number of cores used for the simulation in AEDT, go to **Tools** > **Options** > **General Options** > **Desktop Performance**. The maximum number for the base license is 4. Additional cores can be specified if you have an Ansys HPC. This can be controlled from **Tools** > **Options** > **HPC and Analysis Options**.
- EMIT simulations can benefit from the use of GPUs (requires an Ansys HPC license). However, the benefits are problem-dependent. In projects that use tunable filters, as this one does, there is little benefit to utilizing a GPU.
- EMIT simulations can be paused at any time by the **Stop** button in the simulation progress window. When a user clicks **Stop**, EMIT saves the current results and pauses the run. The next time the simulation is run, EMIT will resume the simulation.

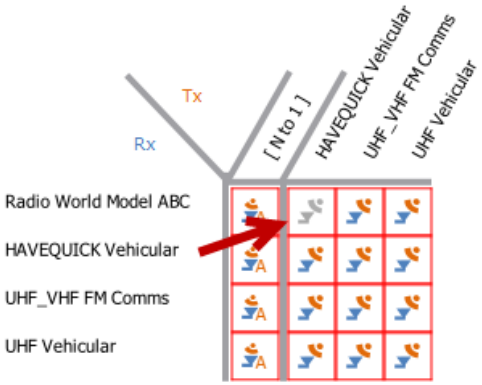


Running the entire simulation at this point will take a long time to run with Multiple Tx's simulations active. When performing N-on-1 simulations (that is, those associated with the [N to 1] column of the Scenario Matrix), all potential Tx/Rx channels must be analyzed. Even for systems with a modest number of operational channels this can result in a very large number of interactions for EMIT to compute.

You can activate/deactivate Tx/Rx pairs for simulation by pressing **Ctrl** and clicking the corresponding entry in the Scenario Matrix. Deactivated entries in the Scenario Matrix appear as gray icons. Entire rows/columns in the Scenario Matrix can be enabled or disabled (as well as run or hidden) via the **Row** or **Column** menu accessed by right-clicking on the RF System name in the Scenario Matrix.

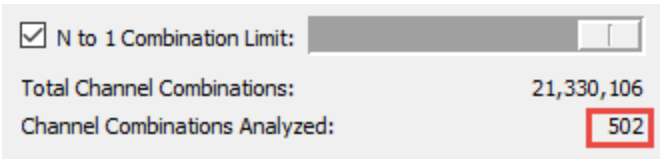
Suppose we wish to simulate possible interference in the Radio World Model system when only the UHF_VHF FM Comms and UHF Vehicular systems are transmitting simultaneously. This is accomplished as follows:

1. Deactivate the HAVEQUICK Vehicular versus Radio World Model ABC square by pressing **Ctrl** and left-clicking on its Scenario Matrix square. Deselect **Enabled**. The square grays out:

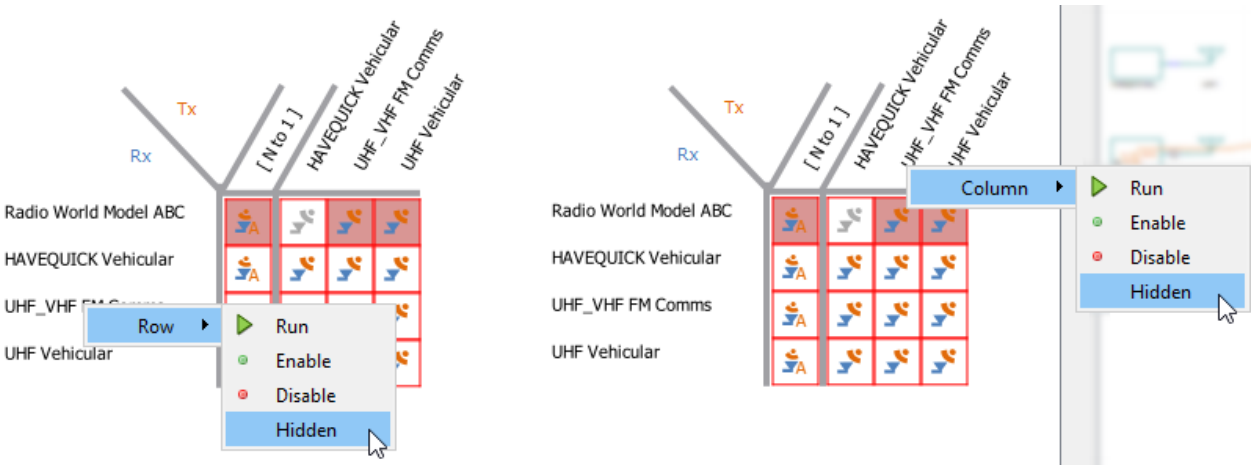


2. Right-click on the Radio World Model ABC vs [N to 1] square and select **Run**.

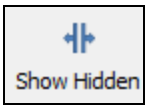
The analysis completes very quickly, as there are only 502 channel combinations.



Since we are only analyzing the Radio World Model, we may also wish to hide the other entries in the Scenario Matrix. Entire rows and columns can be hidden by right-clicking an RF System name and selecting **Hidden** from the **Row** or **Column** menu.



To restore the display of hidden rows and/or columns, click the **Show Hidden** button on the Scenario Matrix toolbar:



Feel free to run various interactions in this scenario and explore the results. You'll find that there are many interference problems, and it might be interesting to explore the various phenomena causing the interference. Just keep in mind that this example isn't representative of a typical or even realistic cosite scenario.

When you are finished, save your project.

This concludes EMIT Tutorial 6.