

Getting Started with RF & Microwave Design with AWR Microwave Office

*Andriy Gordiyenko, RF-Engineering & Consulting
Aribonenstr 15, D-81669, München*

AWR MWO and VSS – is a next generation RF-design environment providing best simulation capabilities at all levels: system level, circuit level and layout level. You can simulate modulated signals, channels and functional blocks at system level; measure linear and non-linear characteristics of your device at circuit level; perform electromagnetic simulation at layout level. All levels can interoperate in the same design environment.

What is unique in MWO compared to other tools?

All the other RF design tools except AWR MWO use netlists for data transfer between the front-end, simulator, layout tool and so on. Netlist is generated for every simulation, and it is created again after any changes in the schematic. This process introduces a computational overhead, especially during optimization and tuning of the circuit. Another important issue is generation of RF-layout from ideal schematic circuit. All other programs process the netlist data of the schematic to create layout. This procedure is time consuming and can cause synchronization problems: if the layout is not generated after the last edits, then circuit in the schematic and layout will be not equal. In the other programs the tools for schematic, layout and simulation are logically disconnected, and information has to be transferred/extracted explicitly between the tools.

AWR MWO uses unique **Unified Data Model** approach, which eliminates the need to generate both the netlist and the layout. Every element in MWO has a few different presentations (or “views”) of the same object in database. The views are typically a schematic symbol, a geometric structure in layout and a mathematical model for the simulator. The point is that logically all of them are referring to the same object in MWO. For example, Microstrip line in schematic and layout has always same length, because length is the same parameter in MWO Object description of this element. MWO doesn't generate *netlist* in order to simulate circuit; instead, it uses the model information of circuit elements to create a mathematical model, and simulates it. And, if some elements are changed, simulator doesn't re-create the whole circuit and re-simulate it again; it re-simulates only the changed element and its interaction with the whole circuit. This makes MWO linear simulation and re-simulation extremely fast, and allows real time tuning and fast optimisation.

The next advantage of the unified data model is that it is not necessary to manually create layout from the schematic in MWO. Every schematic has layout automatically generated, because layout is just another representation of the schematic. This eliminates possibility to make mistakes and the need of creation of the layout after every change. And vice versa: there is no need for back-annotation, i.e. taking into account changes in layout back to schematic, because every change in layout will be simultaneously reflected in the schematic. There is only one object.

Andriy Gordiyenko,
RF-Engineering & Consulting

MWO's Unified Data Model reduces simulation time, and significantly helps in reducing mistakes at layout creation.

For illustration of the usage of MWO, let us consider a simple circuit, say a microstrip stub. You can see the design environment window at Fig. 1. To add a new circuit schematic, press the right mouse button over “Circuit Schematics”, choose “New Schematic”, and give an appropriate name for your schematic. A new schematic window is created. The layout view of the schematic can be accessed from menu item View>New Layout View. Alternatively, you may press the layout button on the toolbar. A 3D-view for this layout can be created from View>New 3D View. To setup the working space neatly, add a Graph by right-clicking the Graph item on the project tree, and choose Window>Tile Vertical to fit all four windows on the screen. To add elements, select Elements-tab in the low-left, expand Microstrip/Lines. You will see a list of microstrip items, along with a brief description (if the description is not visible, right-click next to the element list and choose “Show Details”). Drag-and-drop MLIN element to the schematic. Now, you can see new element in schematic and at the same time new element in layout editor and 3D-view. Try to change some parameters of the line, press “Enter” and you will see these changes in layout. Double Click on this line in layout editor and change its length, release the mouse bottom, and parameters in schematic will also be changed. At this point, one can see that this line has different representations of the same object.

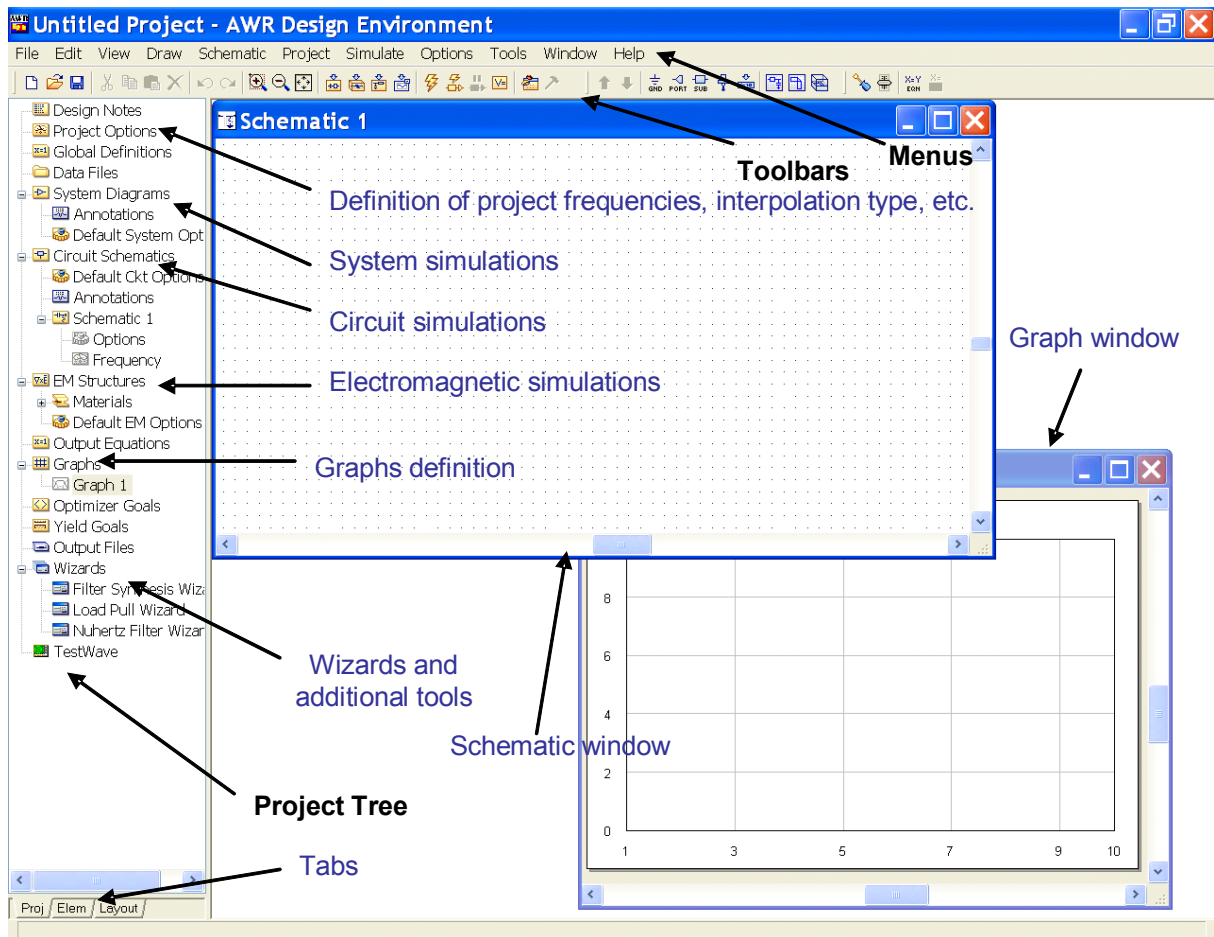


Figure 1a. AWR Design Environment window

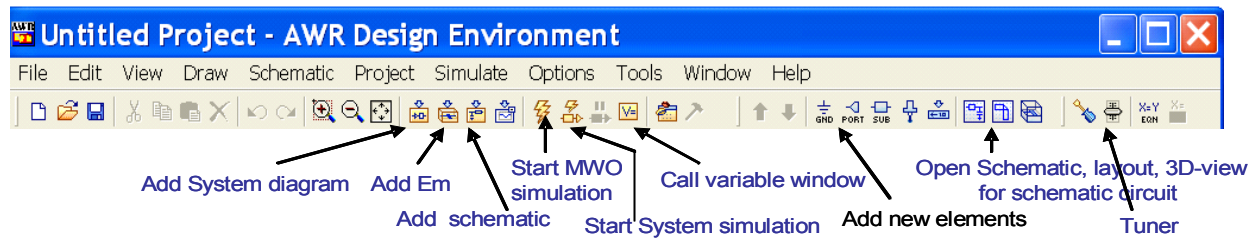


Figure 1b. AWR MWO, VSS Toolbars

Add two more lines in your schematic (or copy/paste). You can rotate schematic elements by right-click. You may note that the lines are initially placed on top of each other in the layout editor. Add junction between microstrip lines (Microstrip\Junctions\MTEES) and create stub, shown at Fig.2. It makes sense to use intelligent junction (MTEES) instead of common one, as it adjusts itself automatically to the connecting elements. Next, add MOPEN\$ from \Microstrip\Other to model open effect of Microstrip stub.

Go now to Layout Editor, select all elements (ctrl-a) and press “Snap Together” button. Now all elements are located in layout editor correctly. You can fit the view by pressing Home-button on your keyboard. Try to change something in layout or schematic and you will see the interaction live. Again, MWO doesn’t create any intermediate data, but just presents different features of the same objects in different windows.

Next, add ports to this structure (Port-button in the toolbar) and Substrate from Substrates\MSUB. Every structure needs ports to connect sources, in order to simulate something meaningful. Typically, one may want to simulate S-parameters of the structure. The measurements are associated to particular graphs (the “graph” can also be a table, for instance). To add measurement, right-click the graph on the project tree, and select “Add Measurement”. Under field “Data Source Name” you can select which schematic the measurement is related to. This way it is really straightforward to characterize different hierarchy levels of your project, as you can select any possible submodel here.

Set the measurement to S11, check dB box, and click “Apply”. You can define S21 measurement right away without having to re-open the measurement dialog. Set up correct frequency range under Project Options in project tree (set 0.1 to 40GHz step 0.1, for example) and simulate this structure (Simulate>Analyze or lightning-button). The response of this structure is shown in the Graph.

Here you can see one of the principles of MWO. It will not simulate data which you will not view or use somewhere in program. Start tuning slider from toolbar, and choose tunable parameters by pressing the screwdriver button, and clicking e.g. the length value of the stub in the schematic. Slide the tuner, and see a real-time tuning simulation. It works very fast and requires only a modest capability from your PC. Such high speed is possible because MWO doesn’t regenerate netlist for the whole circuit, but just re-simulates one element and re-connects it with the rest of the circuit.

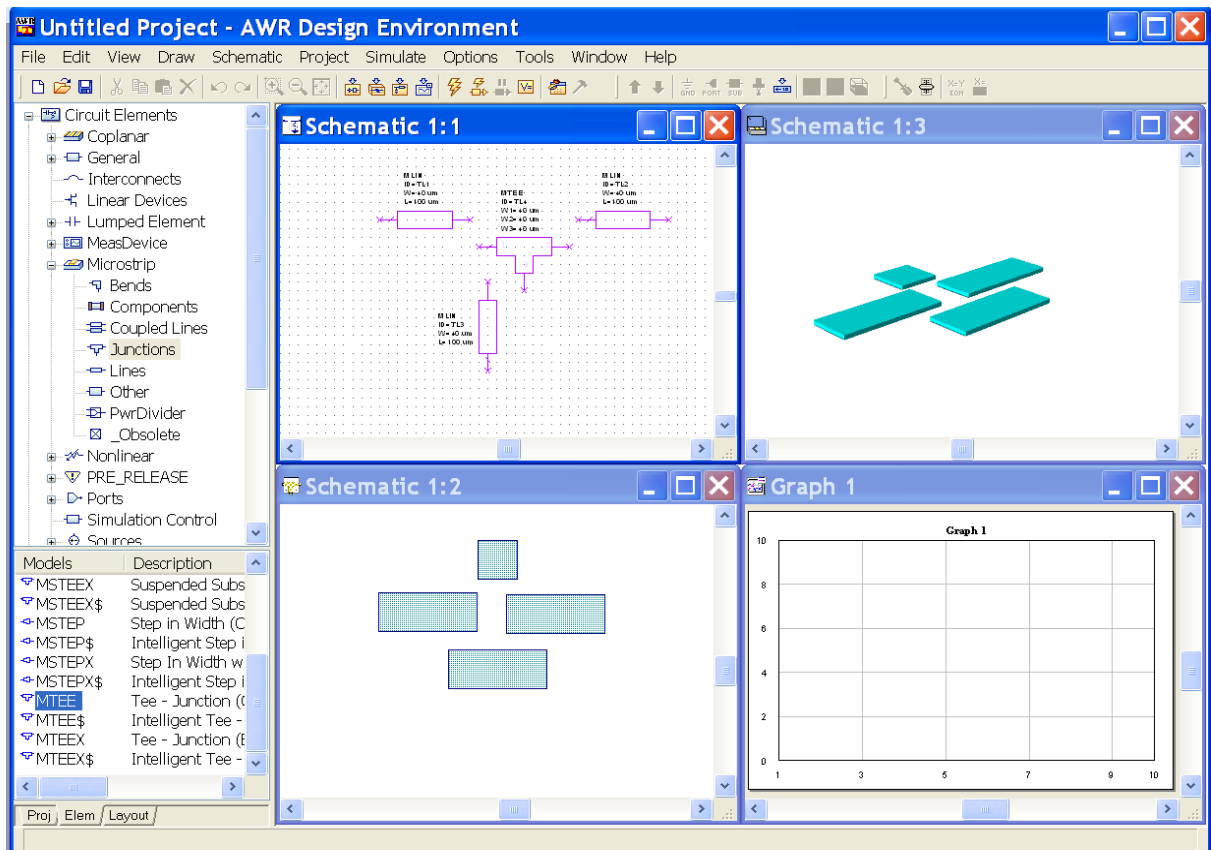
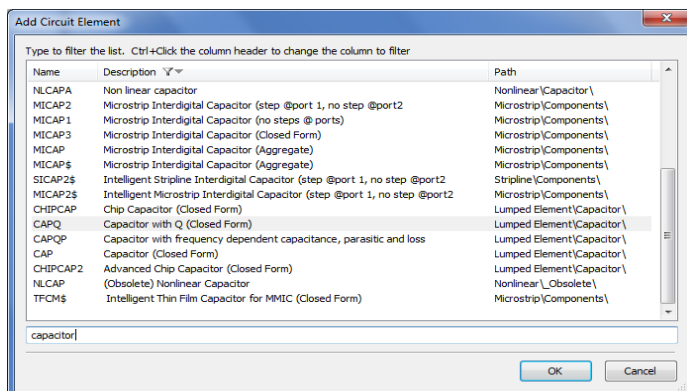


Figure 2. Inserting of MTEE element in MWO schematic

You can also see that all changes you do with tuner are reflected both in the schematic and in the layout, as well as in the 3D-view. Without this feature, you would have to generate your layout after a possibly laborious tuning, just to see that you can't fit your structure to your board! In conclusion, one needs minimum post-verification of the design, because everything necessarily keeps synchronized.

Another way to insert schematic element

From the version 8.5 is a key-combination Ctrl+L linked with add circuit element window. This window allows search of element to be insert with wildcard for name of description. For example, to insert capacitor one can type in "cap" into prompt line and will see all elements



with names starting with "cap", like "CAPQ", "CAPQ\$", etc with description of element function in "description" field and path to elements menu in path field. Alternatively, one can

search using tokens from description field. The field to be searched in can be chosen using Ctrl+Click.

So, using “capacitor” as search token one will see the same searched elements.

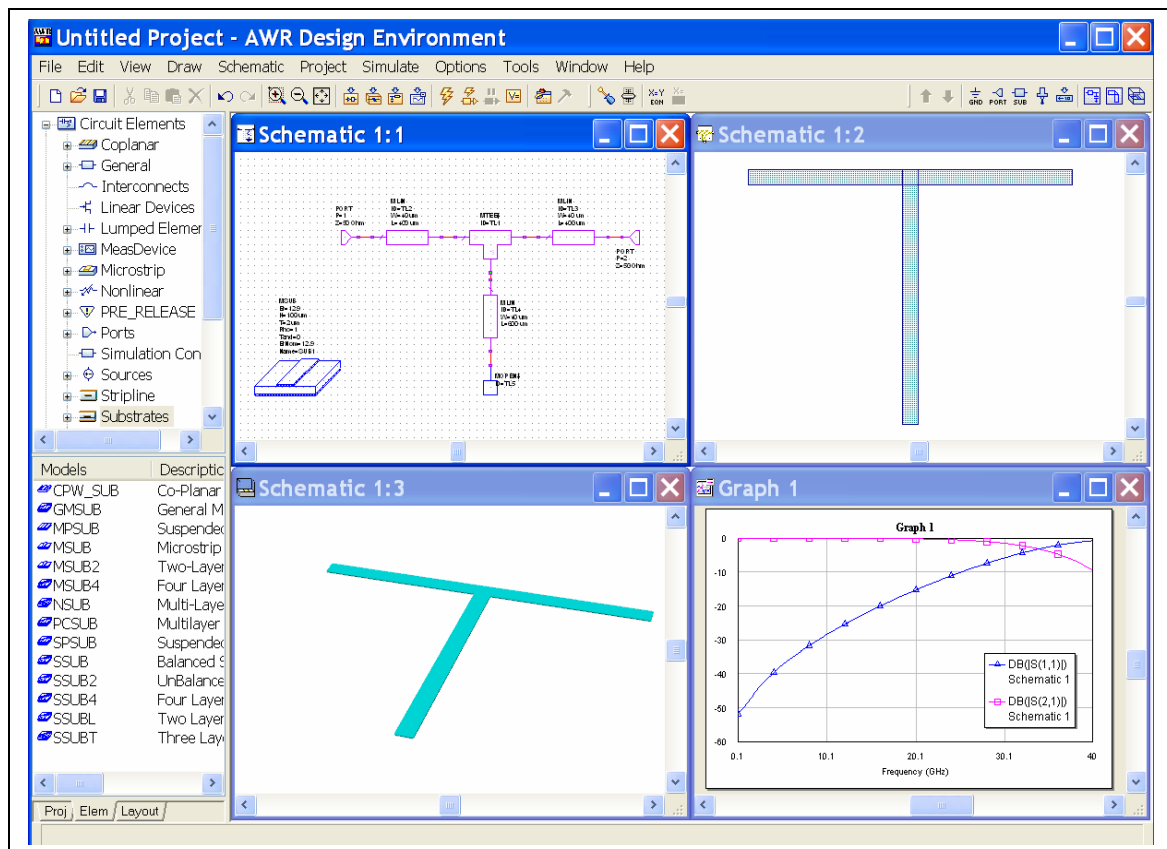


Figure 3. AWR DE window. Simulated schematic.

In this article, the use of Unified Data Model in MWO was discussed, the differences and advantages of MWO in comparison with other RF design tools were highlighted. A simple structure was created to illustrate this in practise, and much more complex designs can be built using the same principles. Necessarily, just a small portion of MWO capabilities are discussed, but nevertheless the basic principles of this innovative simulation software are demonstrated. Some of the key features not discussed here (but documented extensively by examples contained in the installation) are e.g. hierarchical schematic structure, use of on-fly EM-simulation, combined system and circuit simulation, and many more.

© Andriy Gordiyenko, RF-Engineering & Consulting
May, 2011. All rights reserved.
www.ag-rf-engineering.de

Andriy Gordiyenko,
RF-Engineering & Consulting