

## Microwave Office - 2006 Getting Started

Microwave Office (MWO) is available on the PCs in CAE2. There are only 20 licences and these need to be shared among all MWO users therefore you should only have one instance of MWO running on your PC at one time. This guide and the MWO project files available from the ENEL434 or ENEL432 WebCT site are a guide to getting started. You need to **save the MWO files to your local disk**. You may consult Dr Kim Eccleston but you should bring with you (on a memory stick or disk) the MWO project file.

The steps involved in running simulating a circuit are may be summarised as:

1. Schematic capture
2. Specifying analysis frequencies (and source power level in the case of large-signal analysis).
3. Specifying plots
4. Executing the simulation

### To start MWO:

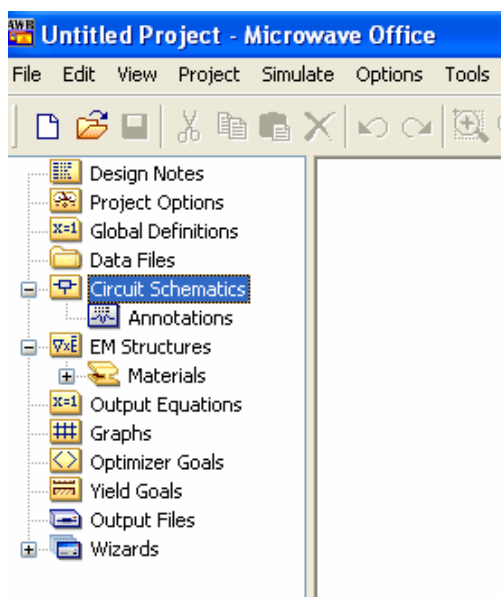
[Start] -> [All Programs] -> [AWR 2006] -> [AWR Design Environment]

- If you do not see "AWR 2006" then you may need to restart the PC to complete the installation process.
- You can **load in an existing project** file using the **MWO file menu**. Pressing F8 executes the simulation.
- What follows describes how you would begin a project from scratch, as well as add graphs and traces to an existing project.

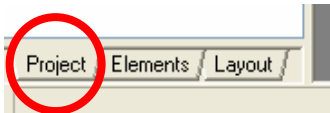
nb. Report MWO licence problems to Pieter Kikstra in the room next to CAE2.

### To add a schematic:

Right click on "Circuit Schematics" in the left column to reveal a menu.

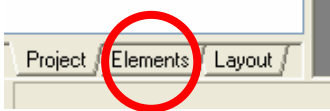


But make sure that the "Project" tab at the lower left corner is selected ...

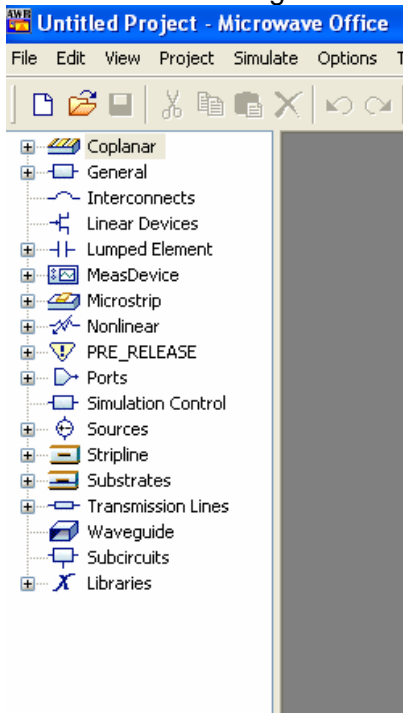


## To access the components palettes:

Click on the "Elements" tab at the lower left corner.



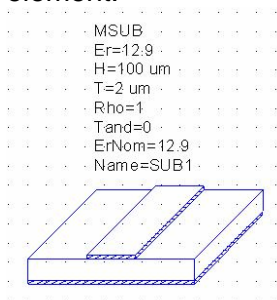
The elements are organised in palette groups:



The capturing of the schematic will be similar to PSPICE and similar circuit simulation software. To join terminals - simply hover the mouse pointer over the terminal and an icon depicting a spool of wire will appear. Drag and drop a length of wire running from one terminal to another.

## Substrate:

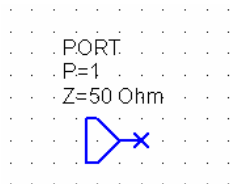
When using microstripline or related planar transmission lines, it is necessary to specify on the schematic the substrate parameters. This is done using a substrate element that can be found in the "Substrates" component palette. If you are using microstripline you should select the "MSUB" element.



## Ports:

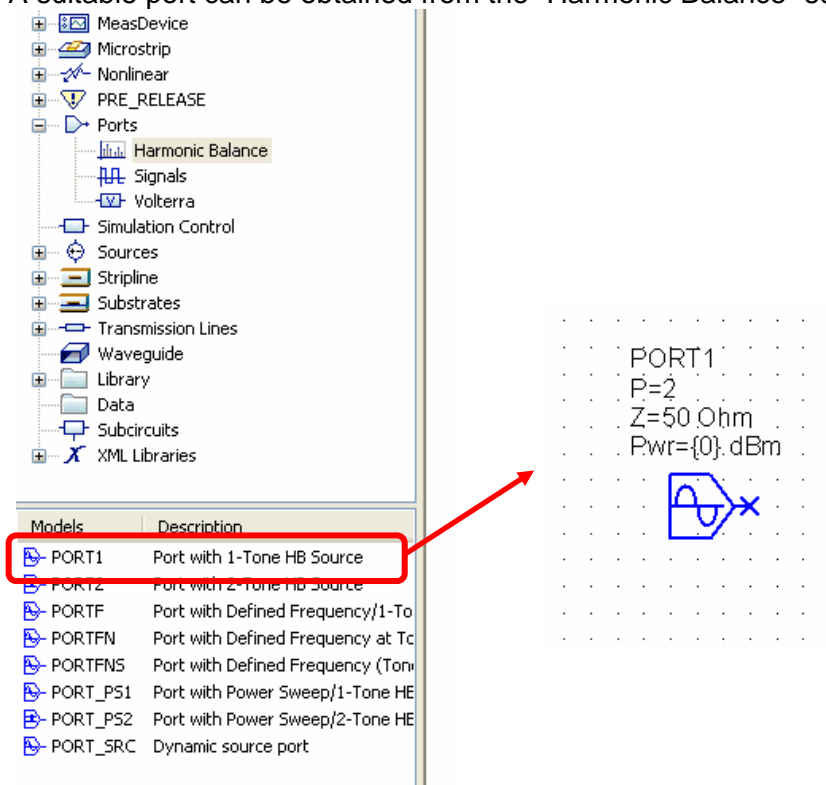
**You must use at least one "Port" element in a schematic.** This port specifies the port number and the port reference impedance used for reflection coefficient and S-parameter calculations. This port relates to say a coaxial plug or socket in a real circuit used to connect it to a signal generator (for example) with a coaxial cable.

For **small-signal simulations** all ports will be:



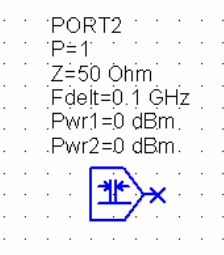
This port is available from the "Ports" palette.

For **large-simulations** one port (usually the input port) needs specify the power level to be applied. A suitable port can be obtained from the "Harmonic Balance" sub-palette of the "Ports" palette.



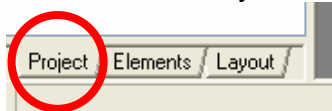
This type of port may be viewed as source with Thevenin impedance given by the parameter "Z" and available power given by the parameter "Pwr".

In some large-signal analyses it is necessary to apply two tones to the input of the circuit. This is done with the "Port with 2-Tone HB Source" available from the "Harmonic Balance" sub-palette. In this case you will need to specify the frequency separation between the two tones as well as their power levels.

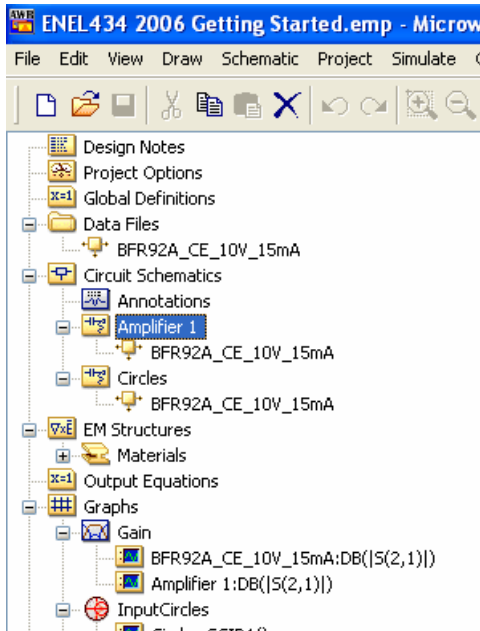


## Simulation Frequencies:

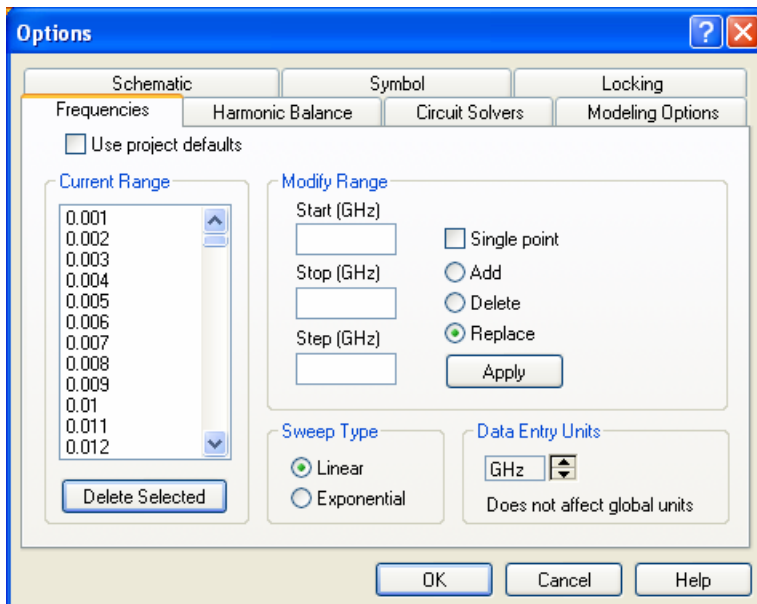
In the left column you will see tabs. Press the "Project" tab.



Under "Circuit Schematics", right-click on the schematic of interest.



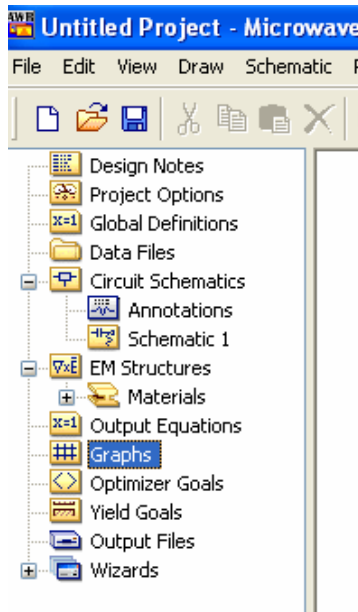
Select "Options" and then press "Frequencies". You will obtain a dialogue box which should be self-explanatory. You will be able to set the range of frequencies over which the circuit is analysed.



Advice: To achieve smooth curves in the plots, you should have at least 100 frequency points across the frequency range that you plot. The frequency range that you plot may be a subset of the analysis frequency range.

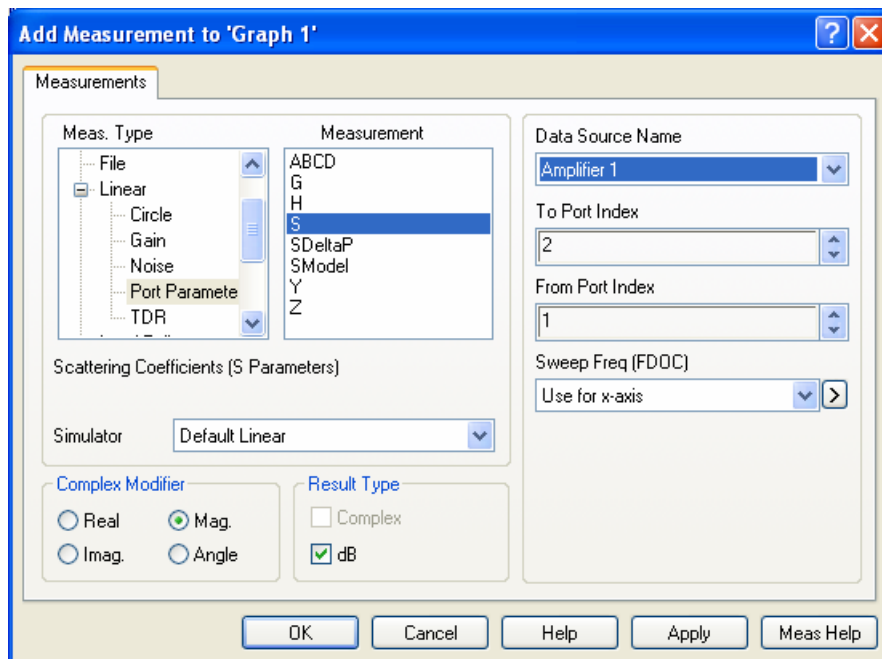
## To add a graph (for viewing simulations):

Right click on "Graphs" in the left column to reveal a menu. (Make sure that the "Project" tab at the lower left corner is selected.)



## To add a trace to the graph:

Right click on the respective graph icon in the left column. Select "Add Measurement". You will be greeted by a dialogue box with many choices. The following dialogue box demonstrates how you would plot the magnitude in dB of S21 of a 2-port circuit called "Amplifier 1":



## To execute simulations:

Press the F8 key.