WIPL-D Microwave Pro: What is New in v7.0?

New features/improvements introduced in v7.0 are:

1. **Import of WIPL-D CAD structures directly into schematic**
2. **Imported subcircuit can now be viewed and edited**
3. **Several schematics can be combined**
4. **Save As with Results option has been added to MW**
5. **GPU matrix inversion is now fully functional**
6. **Alternative way to specify a range of frequencies is implemented.**
7. **Frequency Dependent User-defined Symbols**
8. **Improved stability for Touchstone files with large number of ports**
9. **Maximum number of ports increased to 2047.**
10. **EM component and circuit can have different referent frequency**
11. **Working with symbols has been enhanced for increased user comfort**
12. **Minor improvements and bug fixes**

1. Import Of WIPL-D CAD Structures Directly into Schematic

In new version import of WIPL-D CAD projects to Microwave Pro schematic has been introduced. The import procedure will be demonstrated using an example of a filter presented in the next figure

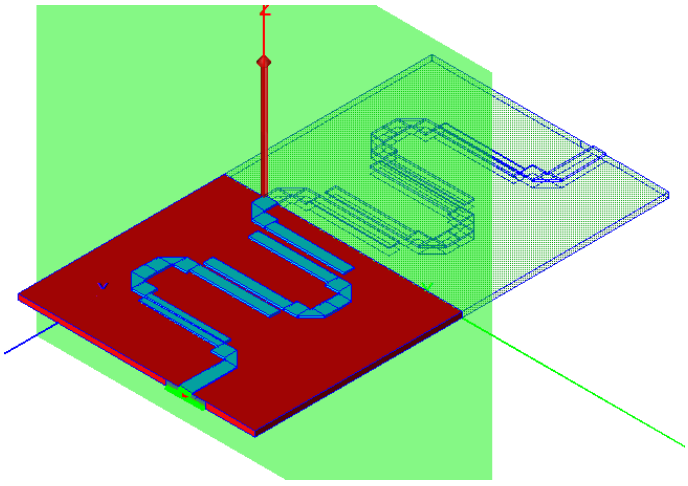

Fig. 1. CAD project containing hairpin bandpass filter 37-42 GHz.

Import is invoked by choosing Component/Import CAD command from the menu or by clicking on the icon, as shown in the next figure.

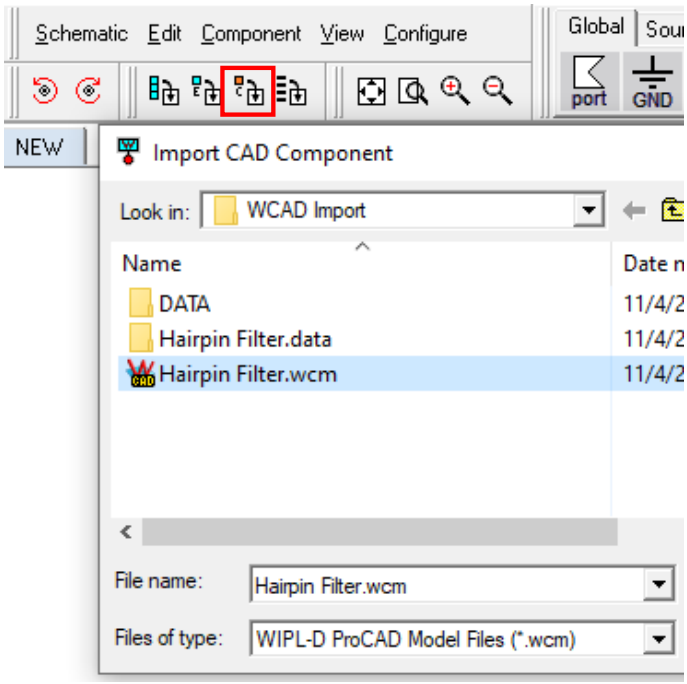

Fig. 2. How to import CAD project.

The user can then browse to the location of the CAD project to import. Once happy with the choice, the import is started by clicking on the project name from the list as shown in the previous figure.

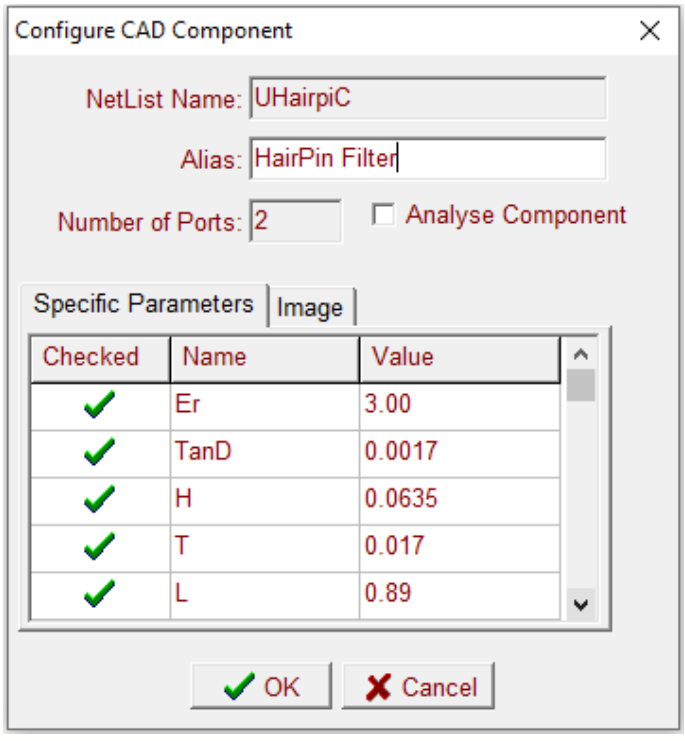

Fig. 3. Configure CAD Component dialog box.

The Configure CAD Component dialog box opens where user can input the schematic name for the imported component (HairPin Filter) and choose which symbols to make accessible for subsequent changes. By default, all the independent symbols are set as accessible which is marked with a green mark in the first column. To make a symbol inaccessible, click on the green mark.

If the CAD project has been solved, then the user can uncheck the Analyze Component field and the imported circuit will not be analyzed from Microwave. If this is the case, the imported component will be colored gray. If the box is checked, 3D EM analysis will be performed for the imported component by invoking CAD analysis directly from Microwave, and the color of the component will be cyan.

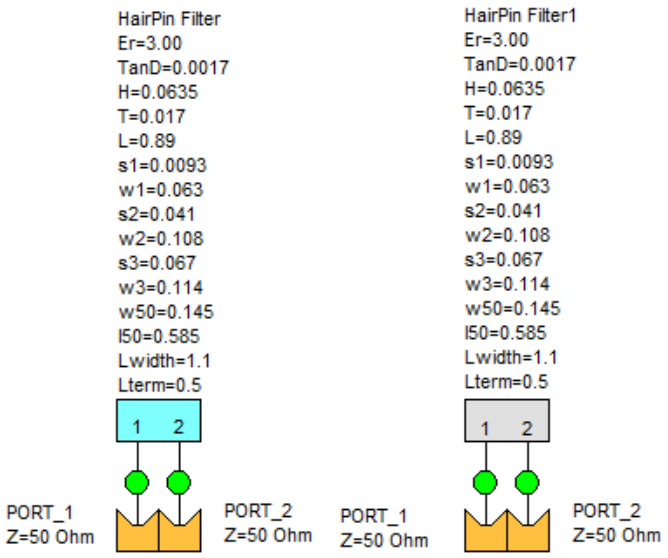

Fig. 4. Import of CAD project with (left) and without (right) Analyze component checked.

The frequencies of the analysis in Microwave project should correspond to the frequencies originally set up for the CAD project. After the simulations are started, the calculation will produce the results as in the original CAD project.

The response of the imported filter is presented in the next figure.

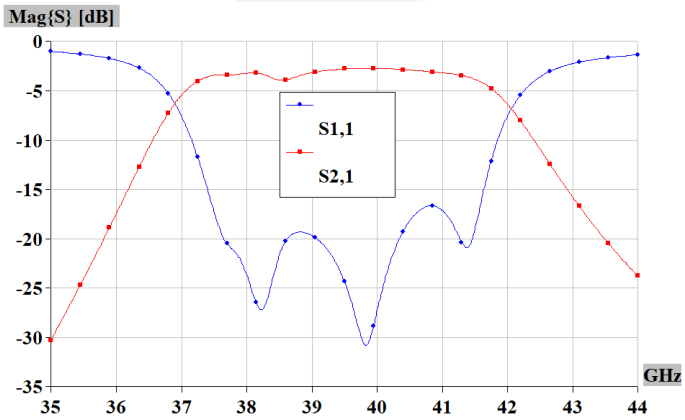

Fig. 5. Response of the hairpin filter form Fig. 1 obtained from simulations of Microwave schematics.

2. Imported Subcircuit Can Now Be Viewed and Edited

In previous versions of Microwave program, the option to import of a subcircuit in the schematic was available. However, once imported, it was not possible to edit or even view schematic of the imported subcircuit.

New version makes it possible to take the full control over the subcircuit schematic including both, viewing and editing. Building the top-level schematic of I/Q phasing network from imported subcircuits, branch line coupler and Wilkinson divider, will be used as example to demonstrate the newly introduced functionality.

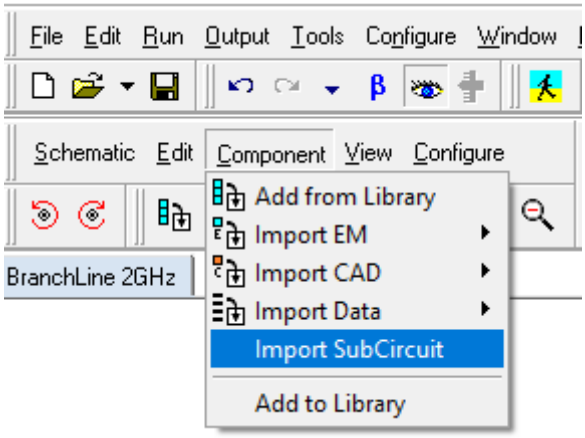

Fig. 6. How invoke subcircuit import.

Branch line subcircuit is imported first. Using Component/Import SubCircuit command, and selecting BranchLine 2GHz circuit, then pressing OK, the circuit is ready for import.

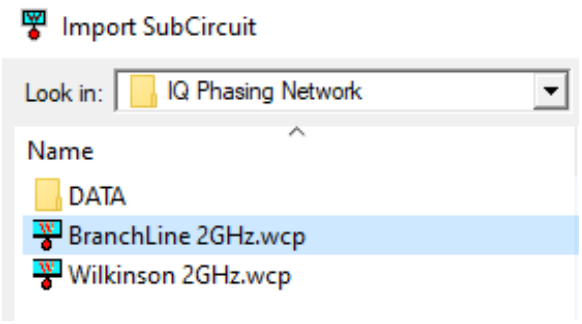

Fig. 7. How to import subcircuit – branch line.

A dialog window opens where user can give a schematic name to the imported circuit. We choose to name it BrancLine, as illustrated in the next figure.

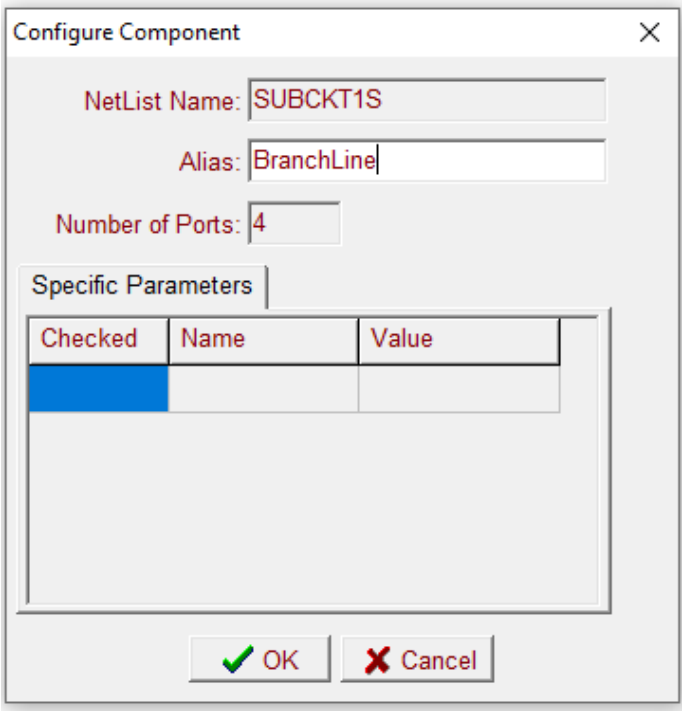

Fig. 8. Labeling the imported subcircuit.

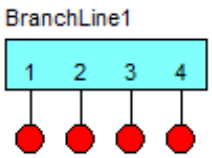

Fig. 9. imported branch line hybrid subcircuit.

After pressing OK the imported subcircuit appears in the schematic. Repeating the same procedure once more, a Wilkinson divider circuit can be imported into schematics.

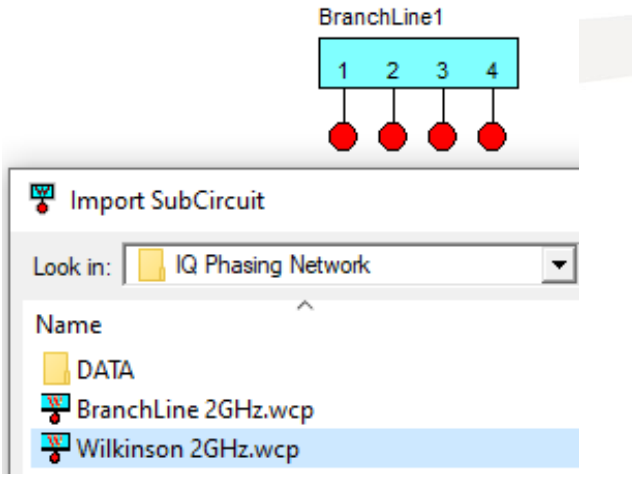

Fig. 10. How to import subcircuit – Wilkinson divider.

Fig. 11. Branch line and Wilkinson divider subcircuit imported into same schematic.

Following the same import subcircuit procedure or using Copy/Paste of the schematic elements, two more Wilkinson dividers and a 50Ω resistor can be added to the schematics and the connections established as they should be arranged for the network to perform like I/Q phasing network. The finalized schematic could look like the one presented in the next figure. The top-level schematic can be viewed by making the corresponding (leftmost) tab active.

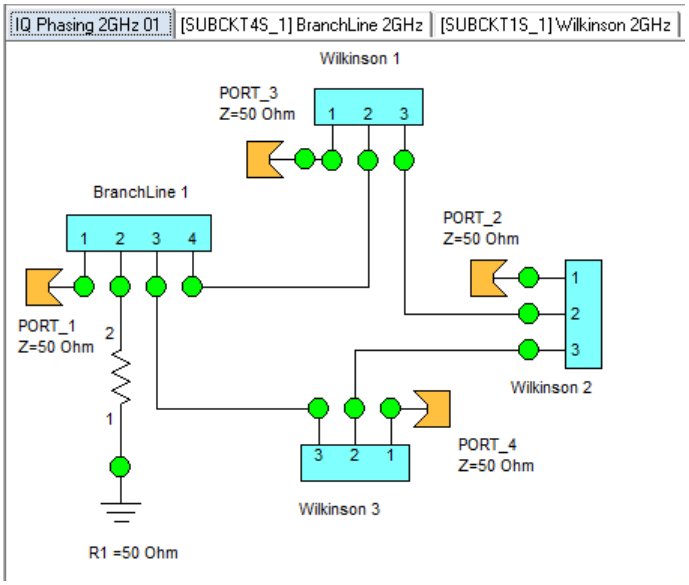

Fig. 12. Complete schematic of the I/Q phasing network.

Individual subcircuit can also be viewed and edited by pressing the corresponding tabs, as illustrated in the next figure.

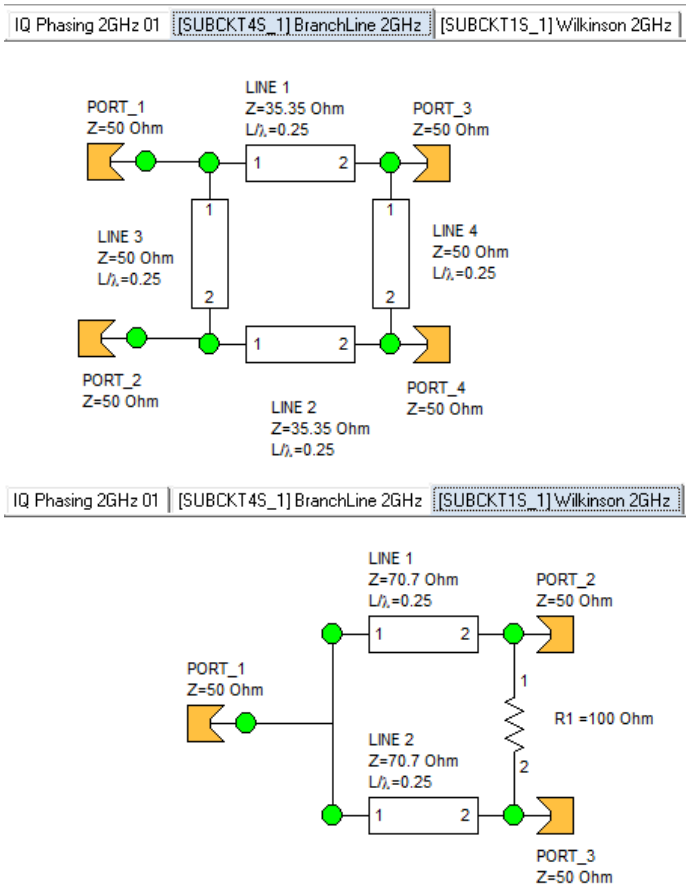

Fig. 13. Each circuit can be viewed by pressing the corresponding tab.

The circuit is simulated from the top-level schematic, and the results are presented in the next figure. It is obvious that the phase constellation is as it should be for the I/Q phasing network (port 4 – I, port 3 – Q).

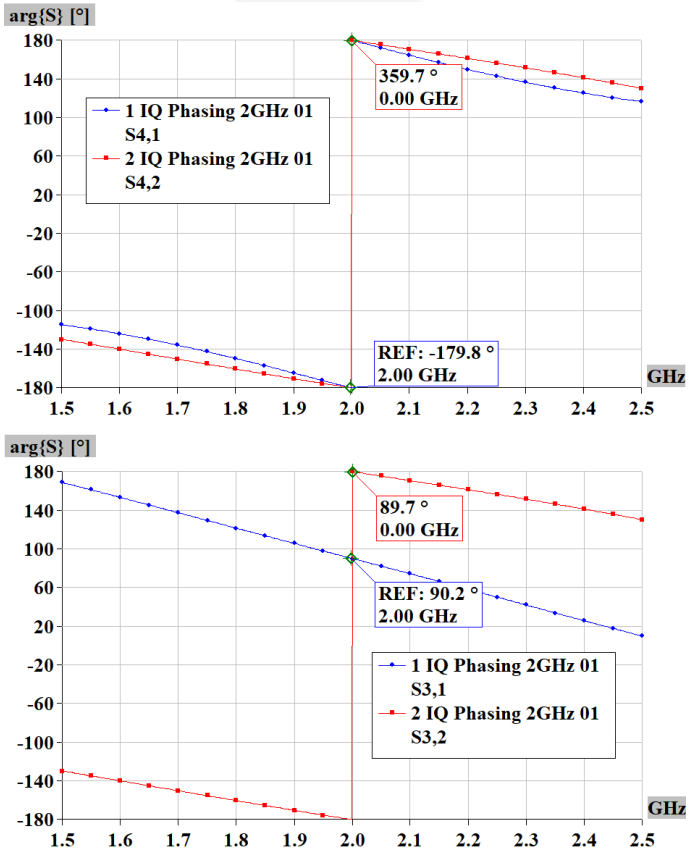

Fig. 14. I and Q phase constellation (port 4 – I, port 3 – Q).

3. Several Schematics Can Be Combined

Sometimes it is convenient to group several circuits schematics together and carry on a design by examining simultaneously response of each of the group members. Recently introduced feature Group can be used to combine multiple projects into a single entity. It will be explained next using a group of four projects which contain the same double stub tuner attached to different resistive loads. It is assumed that designer's intention is to check the quality of the matching as the resistive part of the load changes, while the capacitive part remains the same regardless of the change in the resistive part. The values of the resistive part are not equidistant and therefore cannot be covered using Sweeper.

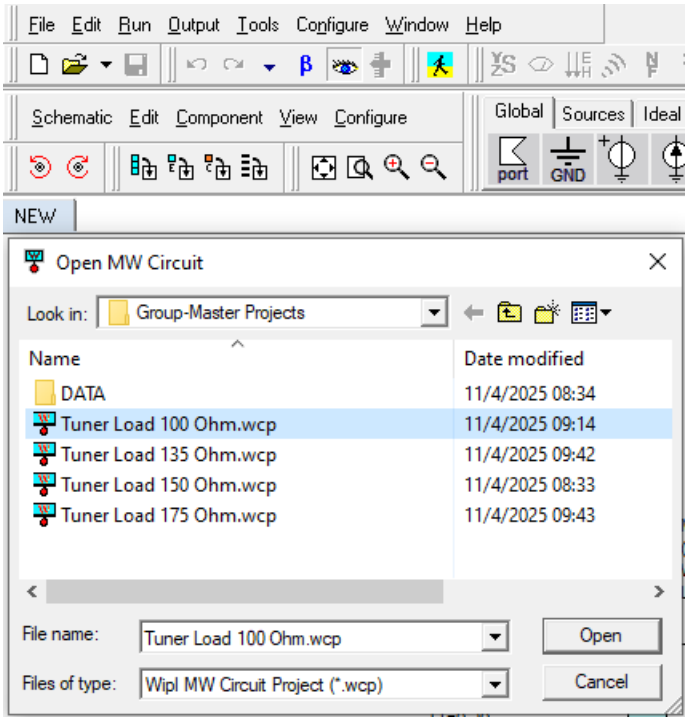

Fig. 15. Four individual tuner circuits which can be opened in turn.

Following standard procedures to open a project, all four of the projects can be opened using File/Open command. A double tuner circuit looks like the one presented in the next figure where a resistive part of the load has a value of 100 Ω.

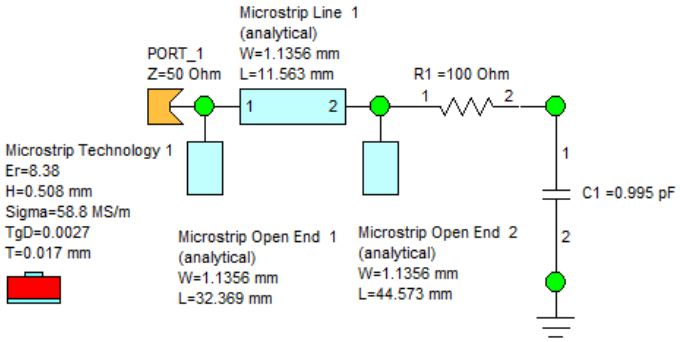

Fig. 16. Example of a double tuner circuit and 100 Ω load.

The look of the schematic window after all of the four tuner circuits are imported is presented in the next figure.

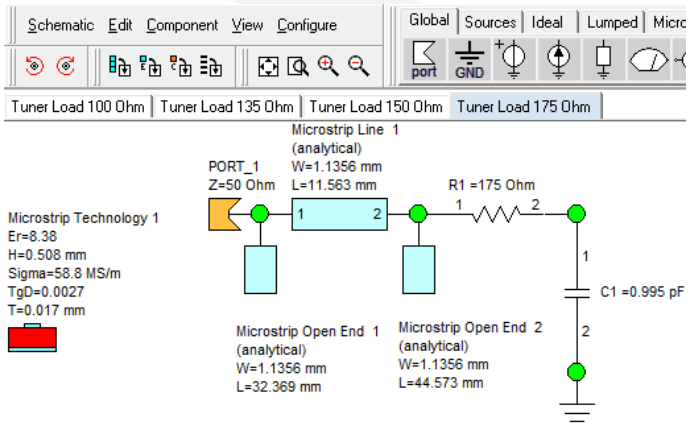

Fig. 17. Schematic windows with four of the tuner circuits opened.

All the projects can be saved as one group by executing File/Group/Save or File/Group/Save as commands, as presented in the next picture.

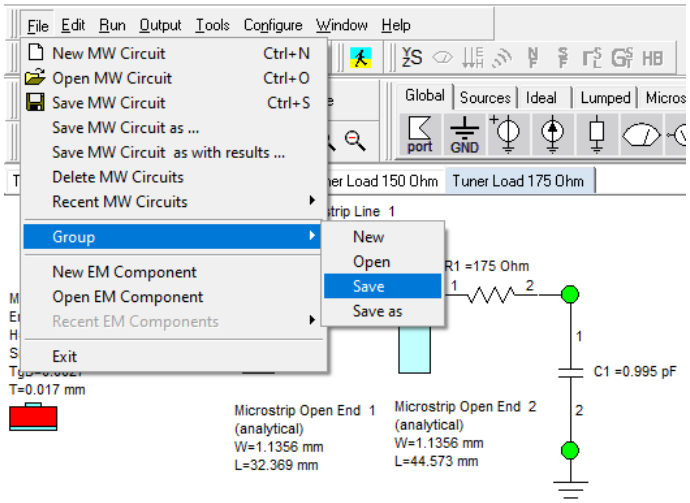

Fig. 18. Saving a group of projects.

The group of four tuners used in this example has been saved under the name Group_Tuners, as shown in the next picture.

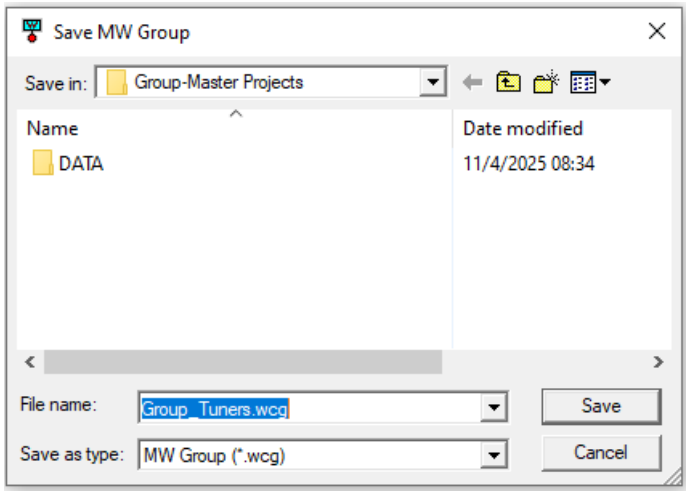

Fig. 19. Saving a group of four tuner projects under the name Group_Tuners.

To open again all the four projects belonging to the group Group_Tuners at the same time, use File/Group/Open command.

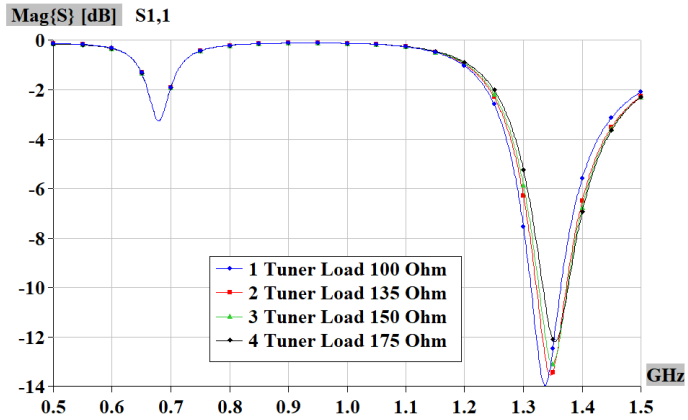

Fig. 20. Results of the simulations for a group of four tuners.

The results for the S_{11} parameter of all four of the tuners can be graphed in a usual way. The comparison presented in the previous picture suggests that around 1.35 GHz the quality of matching changes very mildly as the resistive part changes.

4. Save As with Results Option Has Been Added to MW

Starting from version 7.0, "Save as With Results" can be used when saving WIPL-D Microwave projects. To ensure consistency with other WIPL-D program modules, an option has been introduced that allows saving a solved project under a different name, while retaining immediate access to the results in the new project.

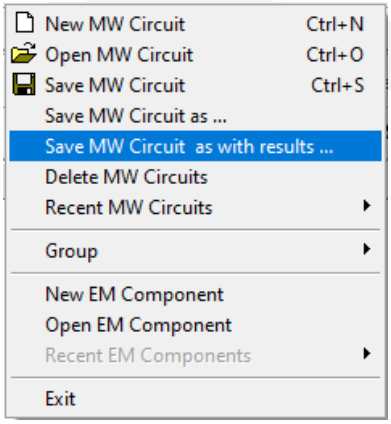

Fig. 21. Save as with results is now available in WIPL-D Microwave.

5. GPU Matrix Inversion Is Now Fully Functional

In previous versions, on some occasions, running the 3D EM analysis for EM schematic components, although specified to engage GPU matrix inversion would proceed on CPU i.e. would ignore the GPU settings. This bug has been resolved, and GPU is now utilized for matrix inversion when specified.

6. Alternative Way to Specify a Range of Frequencies Is Implemented

To conform with the other programs from WIPL-D suite, alternative way to define project frequencies has been introduced.

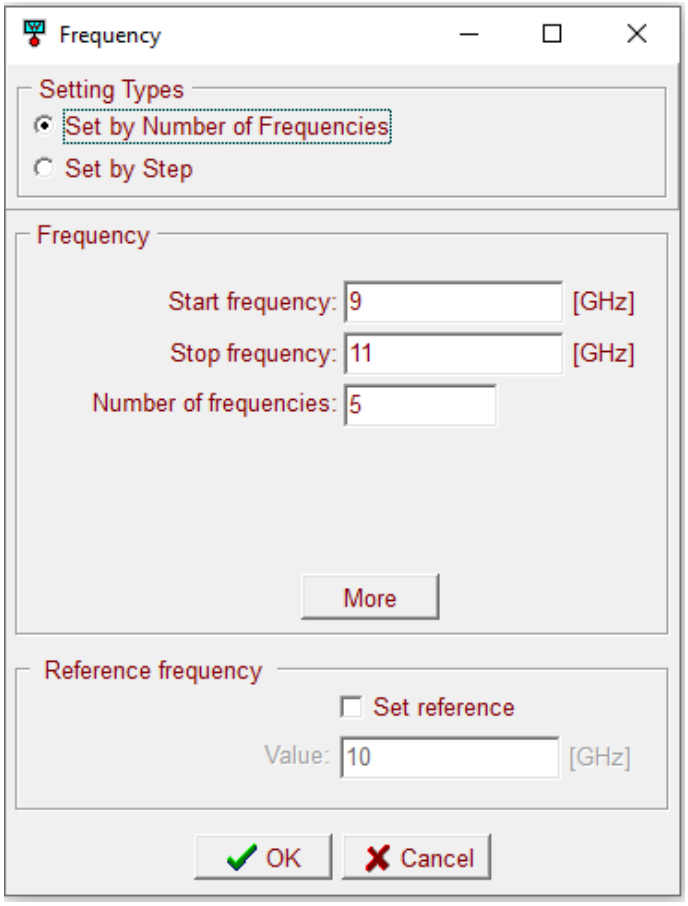

Fig. 22. Define project frequencies from start-stop frequencies and number of frequency information (increment is automatically calculated).

Besides the possibility to use start-stop frequencies and define a number of frequencies, as presented in the previous figure, it is possible to specify start-stop frequencies along with an increment, as illustrated in the next figure.

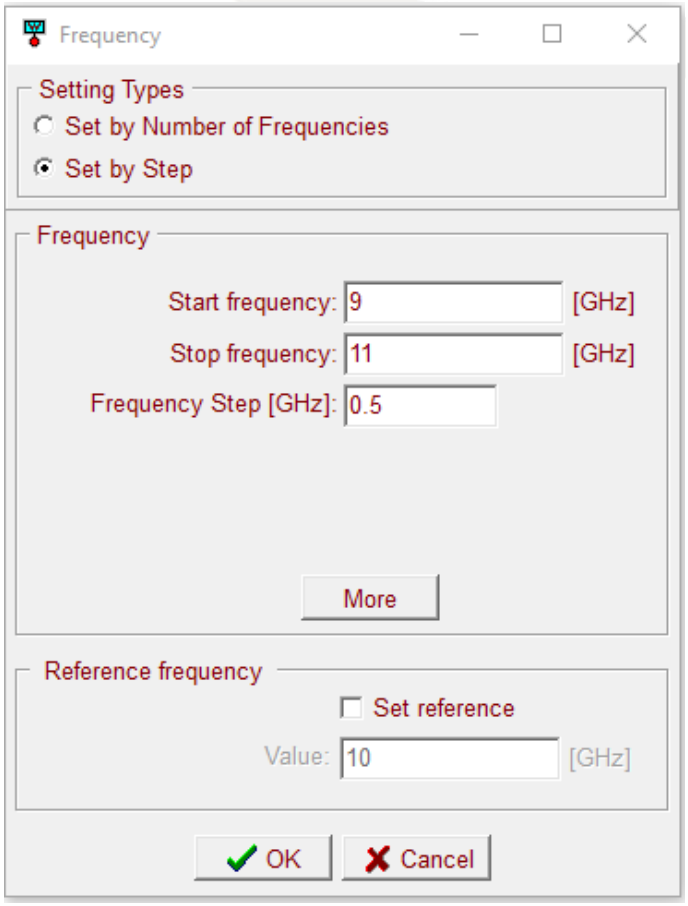

Fig. 23. Define project frequencies from start-stop frequencies and increment information (number of frequencies is automatically calculated).

The way of defining the frequencies is selected by clicking the desired radio button in the top section of the menu.

7. Frequency Dependent User-defined Symbols

Frequency dependent user defined symbols have been introduced in Microwave following the same concept as described in section 9.2 of WIPL-D Pro manual. Therefore, in this document we provide an example of how to use the freqtab() functionality along with Microwave program. It is demonstrated using the example of the double tuner where resistive part and capacitive part of the load has been made frequency dependent.

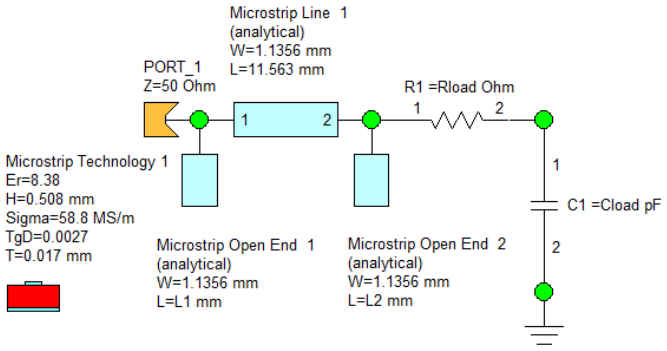

Fig. 24. Double tuner with symbolically defined resistive and capacitive part of the load.

Following the instructions from the manual, resistive and capacitive part of the load can be easily defined within a text file, which is presented in the next figure.

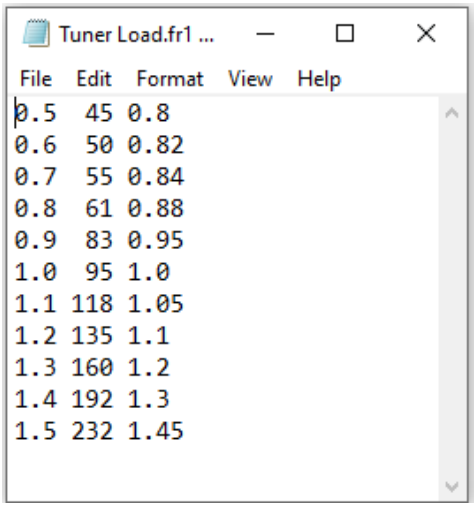

Fig. 25. Data from .fr1 file will be used to define frequency dependent load.

After desired data are entered in the file and it is present in the required location, the symbol list can be extended with the frequency dependent symbols, as shown in the next figure.

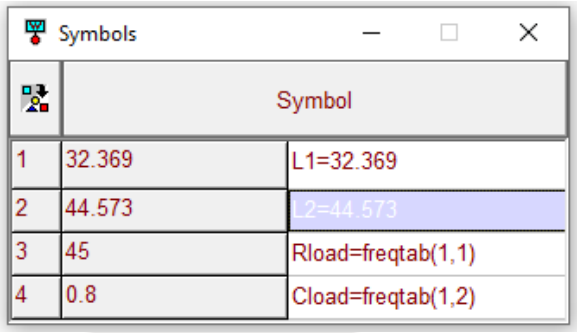

Fig. 26. Introducing frequency dependent resistive and capacitive part of the load.

The results of the simulation of the file with frequency dependent load can be compared with the original results where the load was 100Ω. The comparison is presented in the next figure.

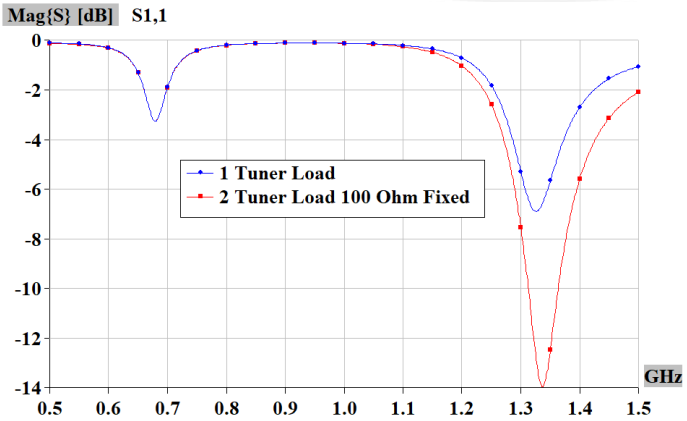

Fig. 27. The load with frequency dependent resistive and capacitive parts is not properly matched.

8. Improved Stability for Touchstone Files with Large Number of Ports

In previous versions, handling Touchstone files with a large number of ports has not been set correctly and in some situations the indices have been messed up so that wrong values were assigned to some S parameters when reading from the file. In version 7.0 these kinds of errors have been eliminated and the process of assigning S parameter values has been corrected.

9. Maximum Number of Ports Increased To 2047

Some users have found the previous setting of maximum number of ports prohibitively small, so we have extended the maximum number of ports to 2047.

10. EM Component and Circuit Can Have Different Referent Frequency

From this version on, a different approach has been chosen for setting the reference frequency for the project. Previously, this referent frequency, primarily intended to determine at which frequency electrical lengths of distributed schematic elements are defined, has also been taken as the referent frequency for the meshing and current expansion calculation of imported EM components. This choice was not the optimum, as the imported EM component should usually have a reference frequency set to ensure accuracy of EM simulation. It is because meshing and current expansion calculation of the EM component are performed at the reference frequency. Therefore, it should be specified within the 3D EM component model, i.e. set independently to the

frequency which relates to electrical lengths in the schematic.

For that reason, from version 7.0 on, the reference frequency value set inside the model of an imported EM component is not changed after the import to the schematic. It has been made independent on the reference frequency related to electrical lengths of distributed elements of the circuit schematic.

11. Working With Symbols Has Been Enhanced for Increased User Comfort

Several enhancements and fixes related to Symbol list have been introduced in version 7.0. Number of characters and expressions when defining a symbol has been increased. Occasional unexpected conversion from symbols defined through expressions to their numerical values, failure to execute Replace command in the symbol list, etc. have all been corrected

12. Minor Improvements and Bug Fixes

Several bugs related to minor program deficiencies and incremental improvements related to schematic components, symbol definitions, etc. have been included in the new version.