

MMIC Amplifiers

Several elements and commands are implemented in the Amplifier Design Wizard in order to speed up the design of MMIC amplifiers. These include

- Series plate capacitors (SPLC command)
- Overlay capacitors (GPLC command)
- Square rectangular spiral inductors (SPIN command)

- Two-port de-embedding command (ITPT command)

The **via** associated with an **overlay capacitor** can be centered or offset. When an offset via is used, one or two vias can be used. Examples of overlay capacitors with offset vias are provided in Figures 1 and 2 (red elements in Figure 2). When a centered via is used, a slot in the top plate, exposing the via, is allowed. The width of the **air-bridges** used should be the same as that used in modeling the capacitor. Ideally, the air-bridge (or other) lines used should be long enough to clear the via area. The **via** specifications in the circuit file are overridden by any via specifications made in the **substrate customization file**. Fixed size rectangular or round vias are allowed. When round vias are used, the via size can be stepped as specified in the circuit file.

Note that the **air-bridge** side (top plate connection) of a **series plate capacitor** (refer to Figure 1) is marked on the schematic. Also note that no allowance has been made for discontinuity effects and, therefore, the bottom plate should be connected to a line of the same width, while the air-bridge width should be the same as that used in modeling the capacitor.

Square rectangular **spiral inductors** providing in-line, as well as orthogonal connections are allowed. The first and last segments of a spiral can also be full length or have the launching sides centered, depending on the number of segments used. The number of segments allowed are 5 through 24. The artwork rules are summarized in the table shown below. Note that the air-bridge side of a spiral inductor is also indicated on the schematic. When customization information is not available for a spiral inductor used, the information provided in [1] or [2] is used to calculate the spiral inductance.

| | In-line | Orthogonal |
|---|--------------------------|--------------------------|
| Centered first and last segments | 7, 11, 15, 19, 23 | 8, 12, 16, 20, 24 |
| Full first and last segments | 5, 9, 13, 17, 21 | 6, 10, 14, 18, 22 |

As with the capacitors, no allowance is made for discontinuity effects in the connections to a spiral inductor and the width of the lines connected it should be the same as the line spiraled.

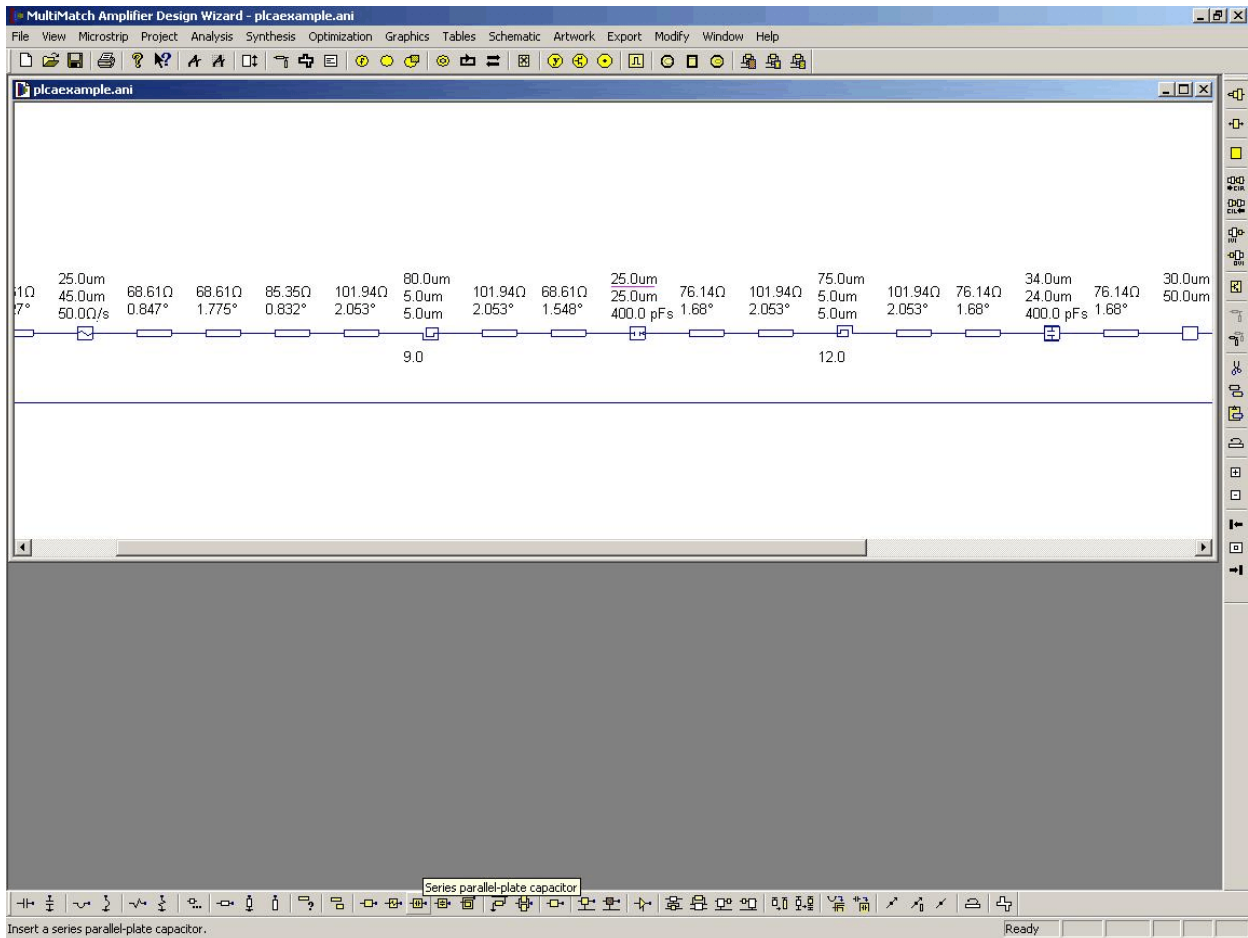


Figure 1 An example of the schematic created for a MMIC matching network in the Multimatch Amplifier Design Wizard.

The **orientation** of a MMIC component can be changed in the Artwork View by using the Flip Command provided on the Artwork Toolbar. If necessary, it can also be changed in the Text View by appending a “^-” or a “^+” string to the command.

Note that less **parameters** are required for a plate capacitor or a spiral inductor when the substrate customization option was used. The parasitic elements of the capacitors, and the spiral inductance and its parasitics, are then calculated from the models specified by the user. The parasitics can, however, still be specified on the schematic. If this is done, the specifications made will be used instead of the calculations made by using the model specified.

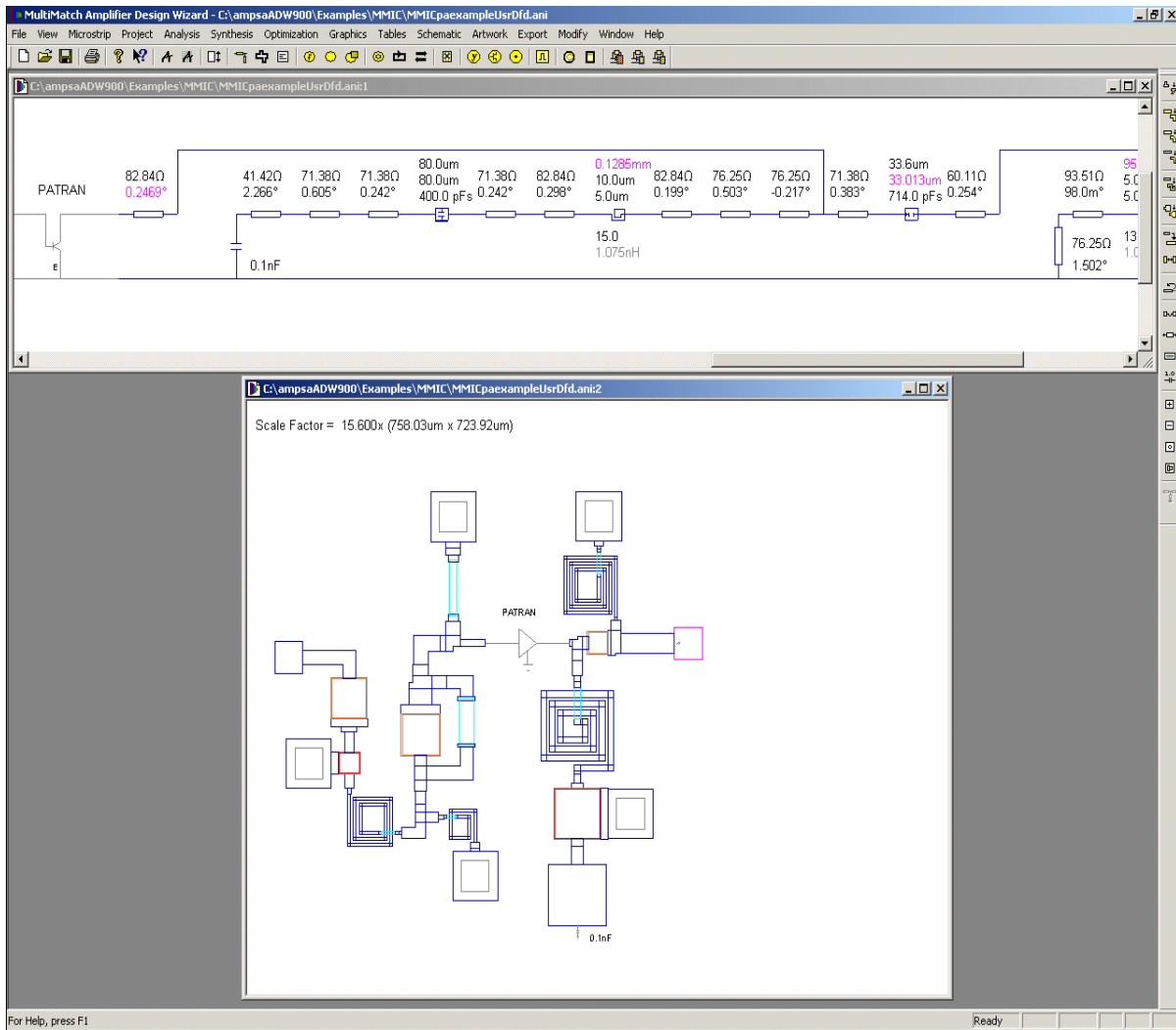


Figure 2 The artwork created in the Multimatch Amplifier Design Wizard for a MMIC power amplifier (user-defined customization).

The option to use the equations in [1] or [2] to calculate the inductance of spirals, for which no equations were specified in the customization file, is provided in the Preferences Dialog Box (Modify | Preferences Command; refer to Figure 3). Note that the zero value listed for the spiral inductance on the schematic only indicates that no value was explicitly specified by the user. The resistance per unit length to be used in this case can also be specified in the dialog box.

The Multimatch artwork created for the MMIC can be exported in DXF or HPGL format. An AWR Microwave Office™ script can also be created for the artwork. Note that several options can be set for the export to Microwave Office™ by using the Artwork | **Artwork Options/Parameters** command provided in Multimatch. The options provided are shown in Figure 3. The iCell (intelligent cells), X-models (accurate simulation by using EM models for the junctions) and MMIC TFR options are recommended for MMICs.

Note that **corrections** are usually required to the **thin-film resistors** after export to Microwave Office™. The Microwave Office™ TFRM\$ command expects some pads to be specified in the command, while the pads are external to the resistor in Multimatch. In order for the artwork to be created correctly, the minimum length of these pads should be 6µm. The lines attached to the thin-film resistor should be shortened accordingly. Note that the pad lengths in the Microwave Office™ command are set to 1µm by Multimatch. In general, this should be changed to 6µm and the lengths of the lines connected directly to the resistor should be reduced by 6µm. Also note that the **lines connected to the thin-film resistor** are slightly wider than the resistor. In this example, the lines are 4µm wider than the resistor.

Options to export specific components as **sub-circuits** to Microwave Office™ are provided (refer to Figure 3). Greater control over the artwork that will be created is possible when these options are used. The schematic and artwork for the relevant component(s) must, however, be set up in Microwave Office™ first before the script is run (a template file can be created for importing Multimatch projects). This option is recommended for overlay capacitors (GPLC command) and vias, and because of the issues mentioned in the previous paragraph, also for thin-film resistors. No corrections are required in Microwave Office™ to the Multimatch circuit imported when this is done.

The naming convention used when the sub-circuits are exported is demonstrated below. Note that n is the number associated with the custom substrate referenced. Also note that the alternative artwork option provided for the overlay capacitors is for the case where the via orientation is switched (rectangular via attached with its wide side to the capacitor).

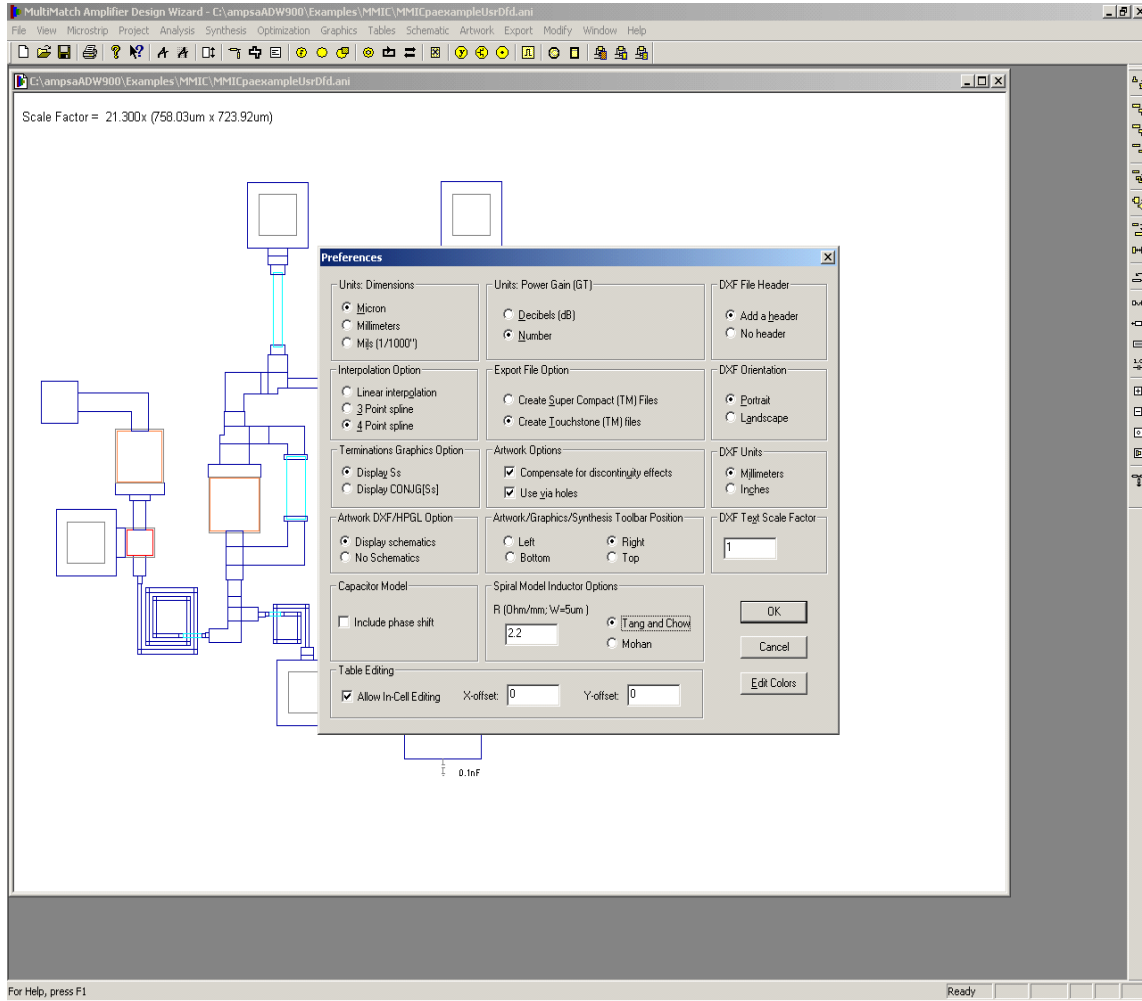


Figure 3 The options and parameters provided for the Multimatch artwork and exporting it to Microwave Office™.

| | Schematic | Artwork |
|---|------------------------------|--|
| Via (VIA) | adwVIAn | adwVIAnartw |
| Thin-film resistor (TFR) | adwTFRn | adwTFRnartw |
| Series parallel-plate capacitor (SPLC) | adwTFCMn | adwTFCMnartw |
| Overlay capacitor (GPLC) | adwGPLCn | adwGPLCnartw adwGPLCWWnartw |
| Square spiral inductor (SPIN) | adwSPINn | adwSPINnartw |

The thin-film resistor metal **overlap** (typically 3um) can also be set in the Multimatch Artwork Options Dialog. Also note that the artwork **resolution** can also be set. This resolution is used on all the dimensions when the artwork is exported, and also on the line widths in the Multimatch artwork created. The line lengths in the Multimatch artwork are not clipped because they may be adjusted to compensate for the discontinuity effects.

Options to specify **fringing capacitance** for plate capacitors and the **contact resistance** for resistors are also provided. The **bottom plate offset** to be used in parallel-plate capacitors can also be modified here (default value: 2um on each side). Note that the fringing capacitance option should not be used when user-specified models for the capacitors are implemented. Any such effects should have been included in the model. Also note that any specifications made for the contact resistance or the fringing capacitance will change the **simulation** results.

Depending on the foundry process used, a set of rules should be created and followed when the Multimatch artwork is created. These include:

- The dimensions of the lines used in air-bridge positions should be constrained to follow the rules set for air-bridges. It is a good idea to set the width slightly narrower (say, 2um) than the capacitor width for small capacitors, and equal to the maximum width (say, 20um) for larger capacitors. Multiple air-bridge connections on each side can also be used for larger capacitors. Only a single connection is currently allowed in the Multimatch artwork.
- The width of the line connected to the bottom plate of a series parallel-plate capacitor should be set to the capacitor width plus the difference in the bottom plate and top plate widths. The default offset used in Multimatch for the bottom plate relative to the top plate is 2um on each side, which implies that the connecting line should be 4um wider than the capacitor width (top plate width). The bottom plate offset used (2um) can be modified in the substrate customization file and by using the Artwork | Artwork Options Command.
- Thin-film resistors should be sandwiched between transmission lines which are slightly wider than the film (say, 4um) and which have some minimum length (say, 6um to 10um). The metal overlap (say, 3um) should be set as required in the Artwork Options Dialog. Note that the length set in Multimatch for a thin-film resistor excludes the overlap.
- Spiral inductors should be sandwiched between transmission lines with the same width as the spiral transmission line. The length of these lines must at least be equal to the minimum allowed (say, 5um). The maximum current expected should be taken into account when the line width of the spiral is decided.

- A minimum width (say, 25um) may be allowed in Microwave Office™ for the lines forming T-junctions or cross junctions. It is a good idea to use the same rules in Multimatch.

When the script has been created, it can be executed in Microwave Office™ to create the equivalent Microwave Office™ schematic and the associated artwork. The relevant Microwave Office views are shown in Figure 4. A second example is provided in Figures 5 and 6.

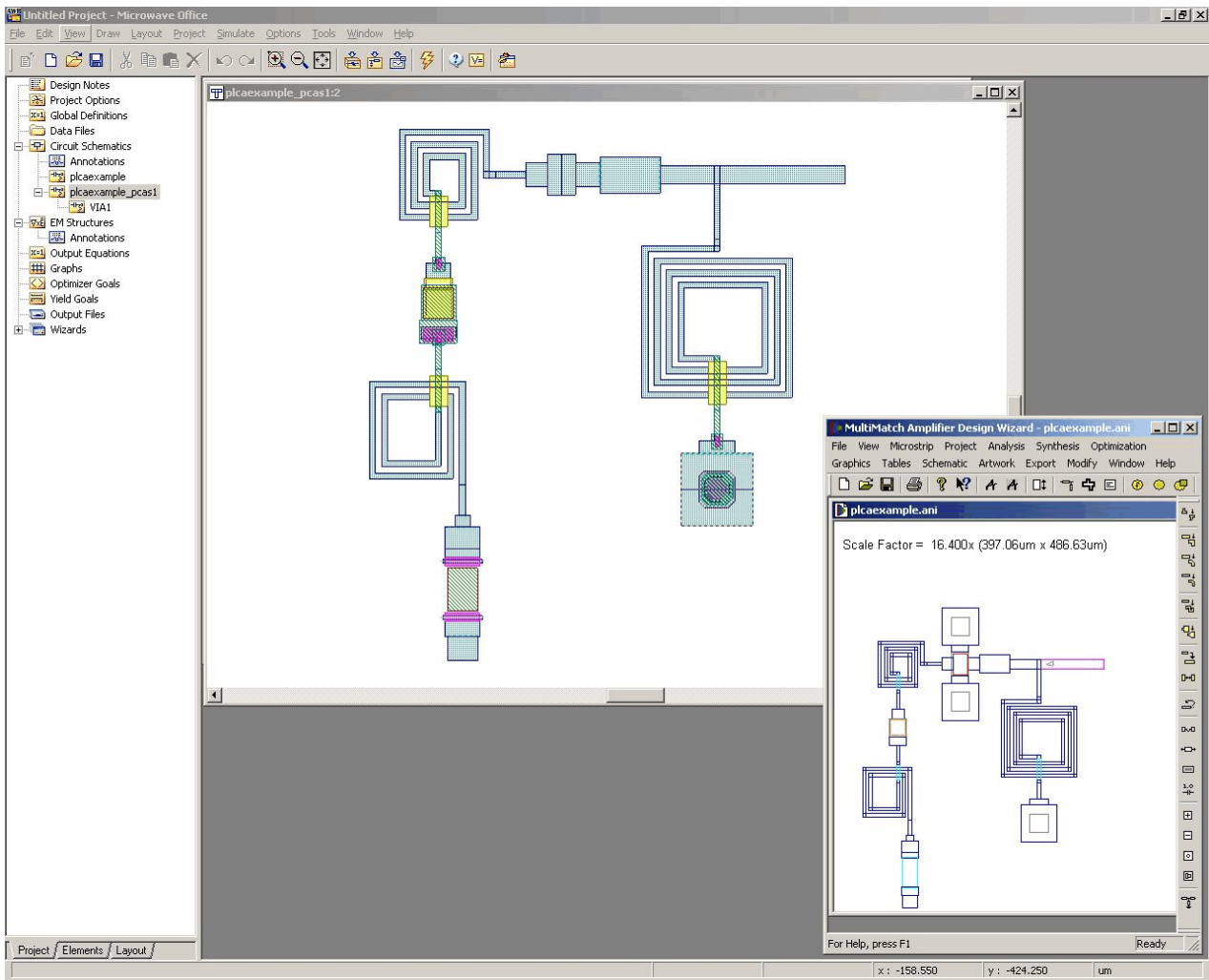


Figure 4 The Microwave Office™ layout created for the Multimatch circuit shown.

Note that the **overlay capacitor** vias are not shown in the Microwave Office™ layout. This is because there is no equivalent standard element in Microwave Office™ and a lumped capacitor (with parasitics) to ground, with the associated series plates (transmission lines), was exported by Multimatch. It is, however, possible to correct this by using the Multimatch option to **export the capacitor as a sub-circuit** and then import this script into a Microwave Office™ project file in which the relevant sub-circuit(s) is pre-defined.

Two specialized commands are provided on the Schematic Editing Toolbar to assist in **converting** any **lumped capacitors and inductors** in a mixed lumped/distributed matching network to equivalent MMIC components. The inductance command is illustrated in Figures 7 and 8. Note that the **negative length lines** introduced by the command must be absorbed into the surrounding circuit. The “pads” used during synthesis should be big enough to allow this.

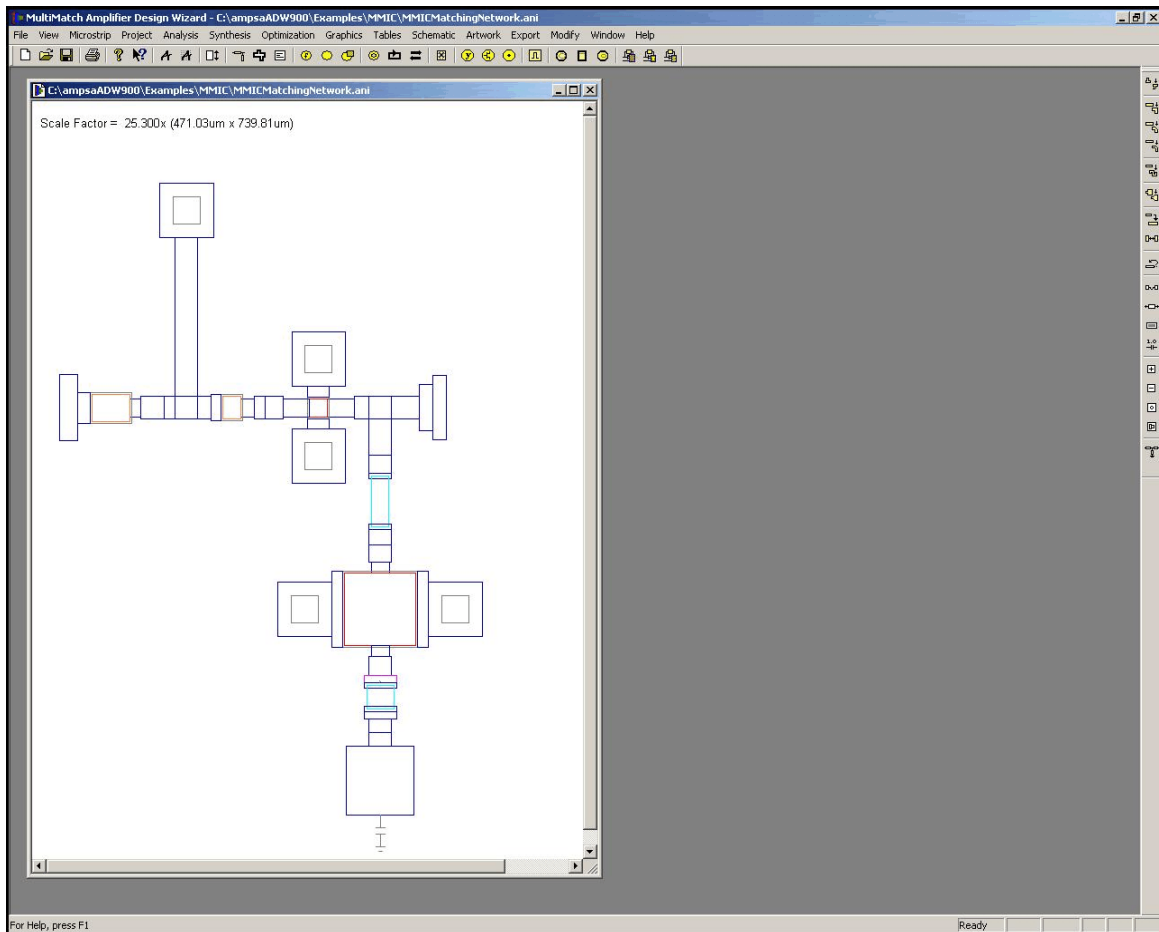


Figure 5 An example of a matching network setup for use in a MMIC.

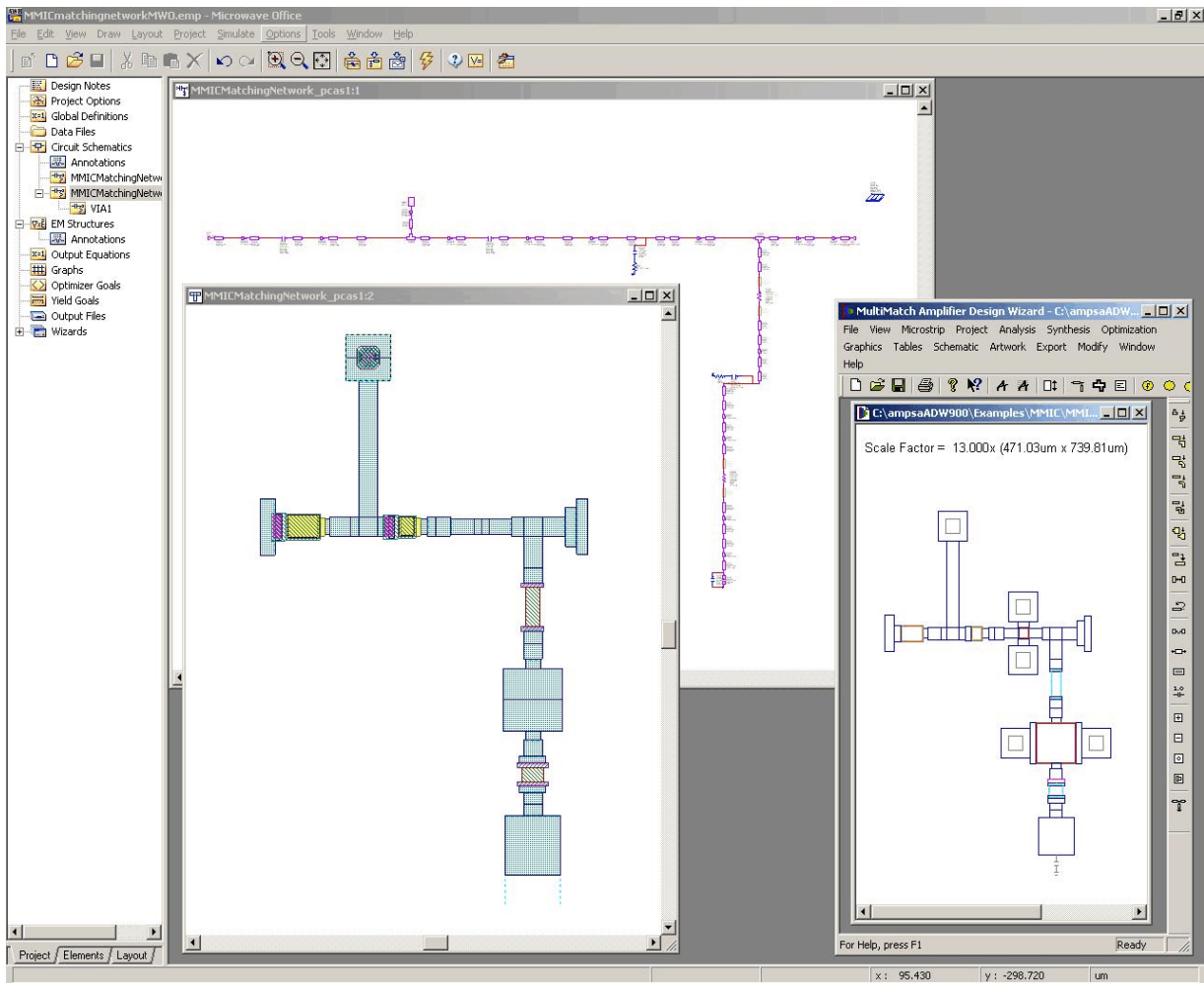


Figure 6 The matching network in Figure 5 after running the script created by Multimatch and making corrections for the thin-film resistors.

The options provided when a lumped inductor is transformed to a spiral inductor are also shown in Figure 7. Note that when customization data for the spiral inductor required is not available (or if the option to ignore such data is used), the inductance will be calculated by using [1] or [2] (former recommended). When a spiral inductor with a specific number of segments is required, the minimum and maximum number of segments allowed should be set to the required value.

The command provided for converting a series or shunt connected capacitor to a parallel-plate capacitor is shown in Figure 9. Negative length lines are also introduced by this command and these lines should again be absorbed into the surrounding network. Allowance should be made for adequate “pads” during synthesis in order to allow for absorption of the lines.

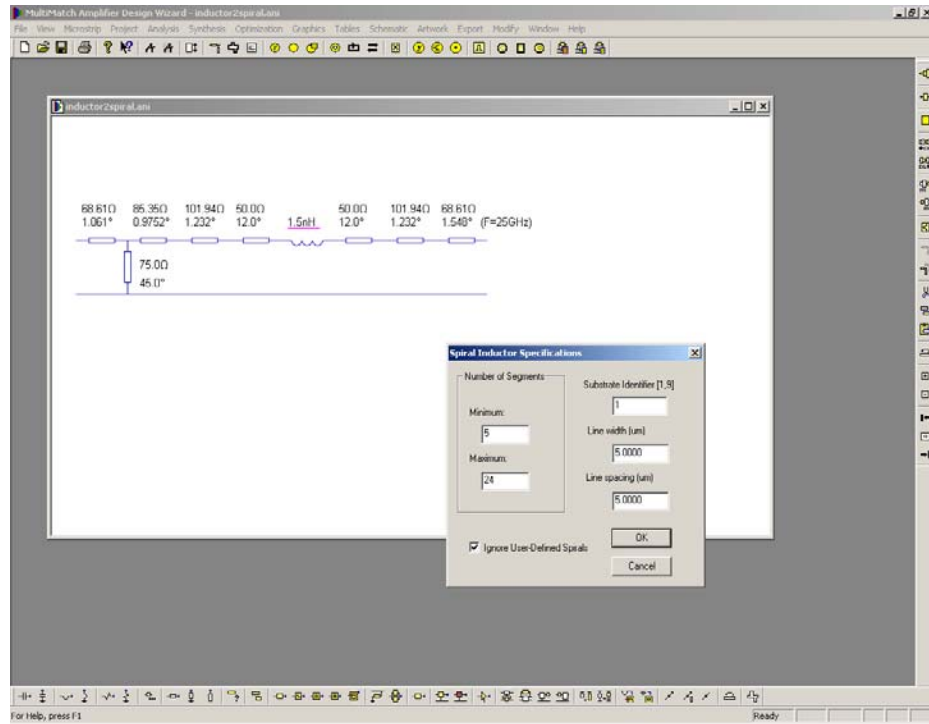


Figure 7 The options provided when a lumped inductor in a mixed lumped/distributed network is transformed to an equivalent spiral inductor.

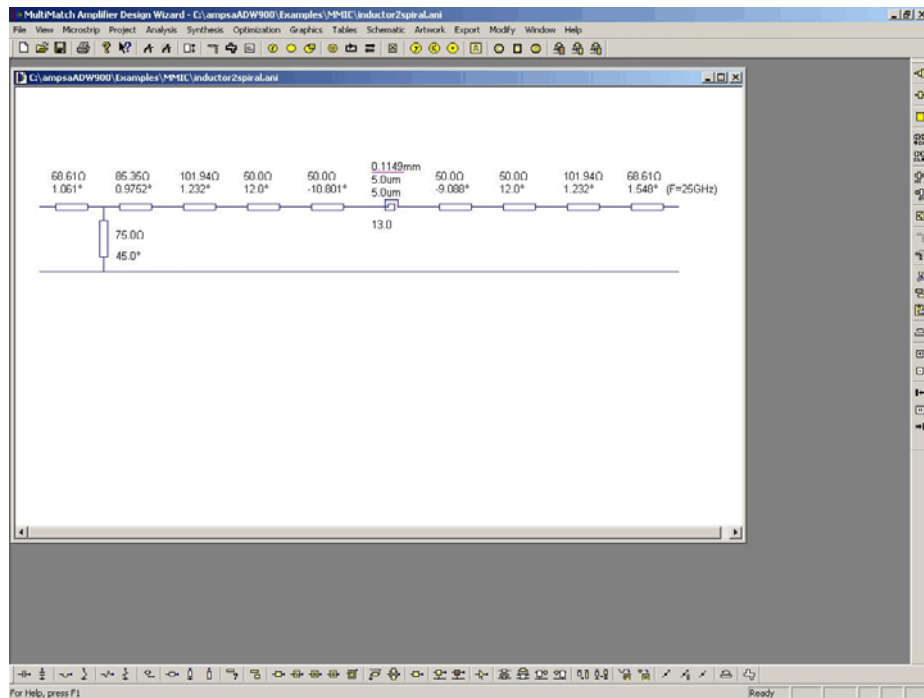


Figure 8 The lumped inductor shown in Figure 7 was transformed to a spiral inductor by using the 7th command from the right on the Schematic Editing Toolbar. The negative length lines introduced, must be absorbed into the surrounding positive length lines.

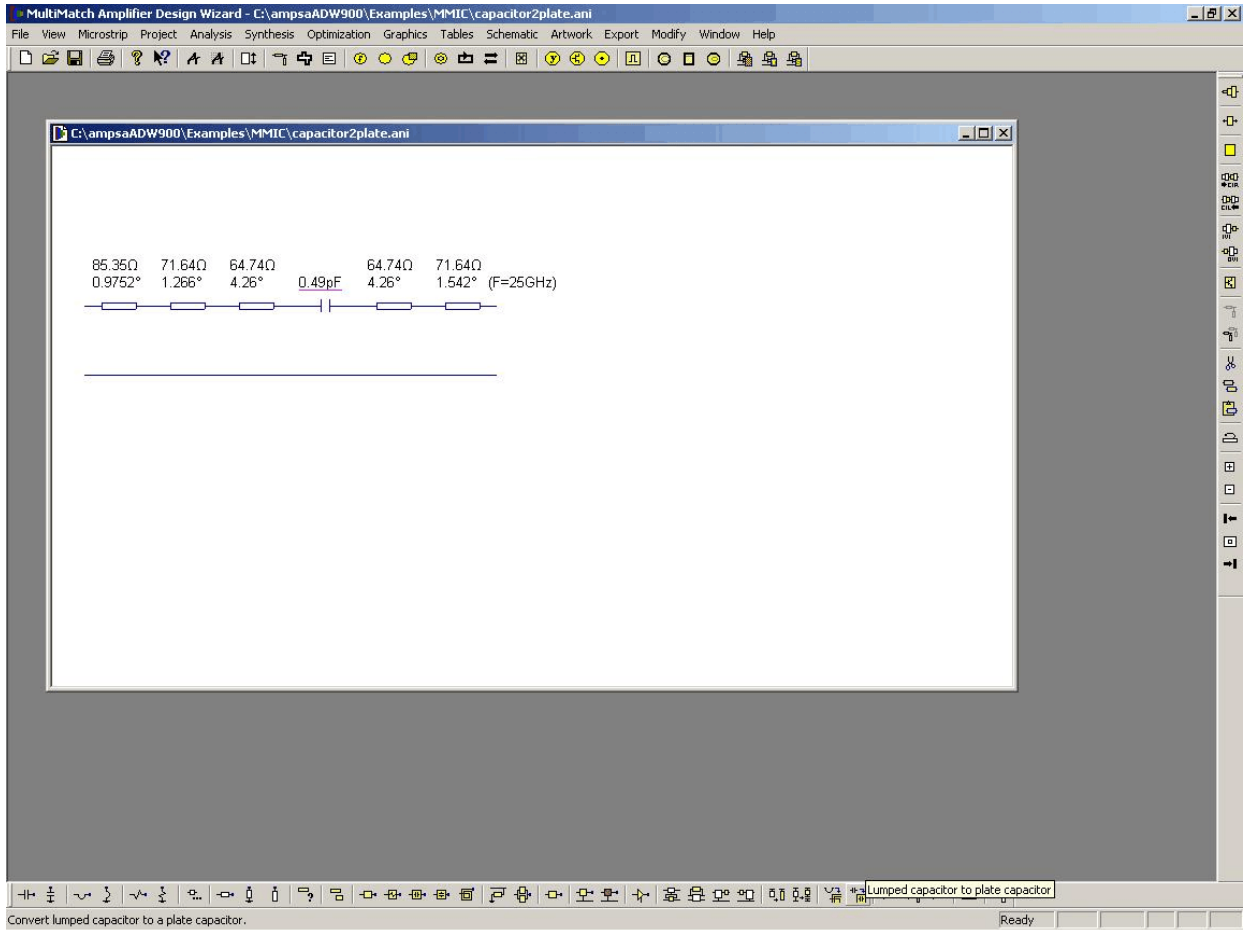


Figure 9 The command for converting a lumped capacitor (series or shunt) in a mixed lumped/distributed network to a plate capacitor is shown above.

The two-port **de-embedding command** provided on the Schematic Editing Toolbar (Small box with the red cross; refer to Figure 9) can be used to de-embed the **S-parameters and the noise parameters** associated with a two-port from the circuit analyzed. This is useful when a model for the intrinsic transistor is extracted.



Note

It is convenient to set the **project units** to micron when MMICs are designed. This can be done by first using the Modify | Preferences Command and then using the Analysis | Options Command.

References.

1. Wan. C. Tang and Y. Leonard Chow, “Simple CAD Formula for Inductance Calculation of Square Spiral Inductors with Grounded Substrate by Duality and Synthetic Asymptote”, Microwave and Optical Technology Letters, Vol. 34, No. 2, July 20, 2002.
2. S.S. Mohan, M. del Mar Hershenson, St P. Boyd and T.H. Lee, “Simple Accurate Expressions for Planar Spiral Inductances”, IEEE Journal of Solid-State Circuits, vol. 34, no. 10, Oct. 1999.