CAD Techniques for RF & Microwave Circuits

This section reviews the different CAD packages that are available and describes how these are used in conjunction with various models in the circuit and subsystem design process. It is shown that there are a wide range of different component models that can be used, the choice of which depends on the level of component characterisation that has been carried out and on the specific CAD package that is available. At one end of the scale it is possible to design a simple circuit using a basic shareware-type simulator with simple equivalent circuit models. At the other end of the scale, it is possible to use a fully integrated CAD workstation which has SMART™ libraries for chip layout automation and direct integration of electromagnetic simulation for non-standard structures.

The key players in the major microwave CAD business are listed in Table I. There is a wide range of software on offer, and the function of each of the different types of microwave CAD package is now described.

1. Schematic Capture

This is a means of entering designs directly into a circuit diagram using symbols. This is now a fundamental part of all major CAD packages, and the old "net-list" is often not accessible. The advantages of schematic capture are that it is easy to visualise your circuit and prevent mistakes. Designs should use hierarchy extensively in order to keep them tidy and easy to follow. Figs. 1 and 2 show the ADS™ schematic of a parallel-coupled microstrip filter: In the high-level schematic (Fig. 1) two identical filter halves (sub circuits) are connected together with the input and output ports. No signal generators are required since this is an S-parameter simulation. The detail within the sub-circuit is shown in Fig. 2; each microstrip CAD element has its own symbol and these are wired together on the circuit page. An MSub data block defines the substrate parameters.

2. Linear Frequency-Domain Analysis

This type of simulator generally uses Y-parameter matrix techniques to solve for the steady-state frequency response of linear circuits. Since individual components are treated essentially as frequency-dependant admittances within a nodal matrix, solution for the overall circuit response is a case of using matrix reduction techniques to end up with an overall set of Y-parameters. These can then be converted to S-parameters, etc., for display. The simulator does not need to calculate the currents or voltages in the circuit, and the analysis is very fast as a result. In particular, working in the frequency-domain means that only the steady-state response of distributed elements (transmission-lines) needs to be considered. This is very important, because the multiple reflections encountered in a microwave circuit would take a lot of time to solve in the time-domain. Furthermore, frequency-dependence of transmission-line parameters (e.g. dispersion, resulting in a change of Z_o with frequency) can easily be handled. However, non-linear elements cannot be modelled since the currents and voltages in the circuit are not calculated. The shareware programs RFSim99, SCALC, and PUFF all use this method. The linear simulator "engines" within ADS, Serenade, and Microwave Office will also all be based on this method, although it is not the only analysis method available in these cases.

** all trademarks acknowledged **

TABLE I

A Selection of Commercial Simulators

COMPANY	PRODUCT (all trademarks acknowledged)	TYPE
Agilent	ADS Momentum	Full linear, non-linear and system suite, various bundles to choose. 3D Planar
Ansoft	Serenade Harmonica Symphony Ensemble HFSS	Integrated environment Harmonic balance System level 3-D Planar Full 3D
Applied Wave Research	Microwave Office VSS EMSight	Linear/nonlinear System level 3D planar electromagnetic
Sonnet Software	em xgeom emvu patgen/patvu	3-D Planar Layout entry Current display Patch antenna patterns
Cadence	Analog Design Environment AMS SpectreRF	RFIC design suite Mixed signal design Non-linear transistor-level simulation
Mentor Graphics	RF Architect / RF Layout ADMS EldoRF	Board-level design linking to ADS^{TM} Mixed signal simulation Transistor level RFIC design
Silvaco	Athena, Atlas SmartSpice, Celebrity UTMOST	Physical-level device simulation Spice-based VLSI design & layout Device model parameter extraction
AC Microwave (Jansen)	LINMIC+/N MLSIM	Linear & Non-linear Planar electromagnetic
IMST	Coplan Topas Empire	CPW model add on Model extraction FDTD 3D electromagnetic
Flomerics	MicroStripes	3D Arbitrary (TLM)
Zeland Sofware	IE3D Fidelity	3D planar FDTD 3D Arbitrary
Computer System Technologies	Microwave Studio	Full 3D
Optotek	MMICAD Lasimo	Linear simulator (opt. time domain) Model extraction
Eagleware	Genesys Harbec Spectrasys Empower Match/Filter, & others	Linear Harmonic Balance System Planar electromagnetic Synthesis
Vector Fields	Concerto	FDTD 3D Arbitrary
COMSOL	FEMLAB	3D electromagnetic

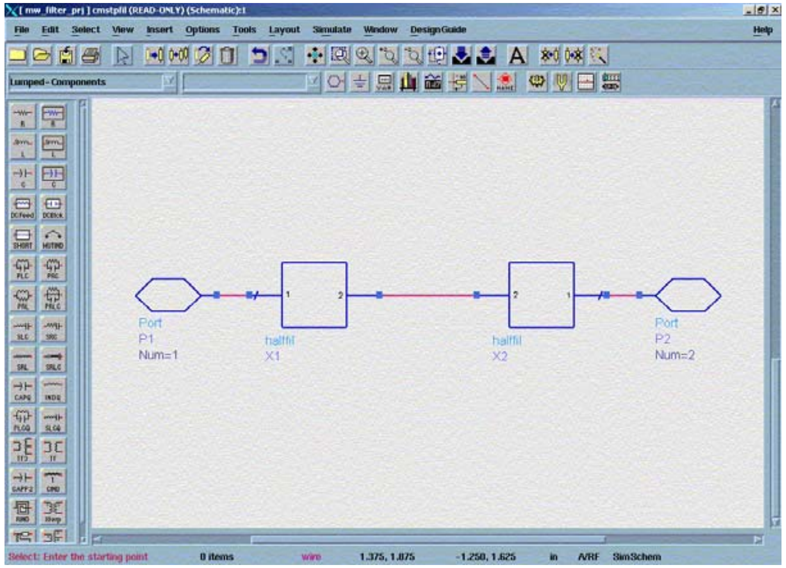
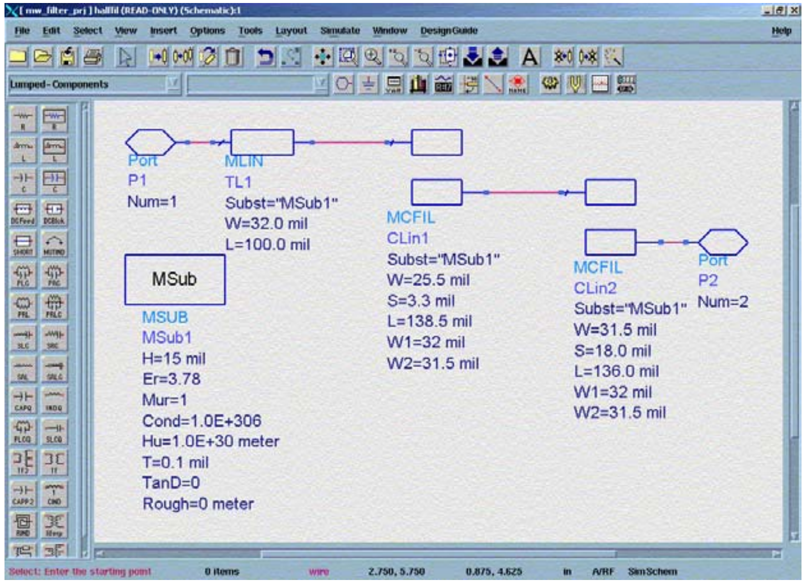


Fig. 1 Two “halfil” subckts connected together



Port
P1
Num=1

MLIN
TL1
Subst="MSub1"
W=32.0 mil
L=100.0 mil

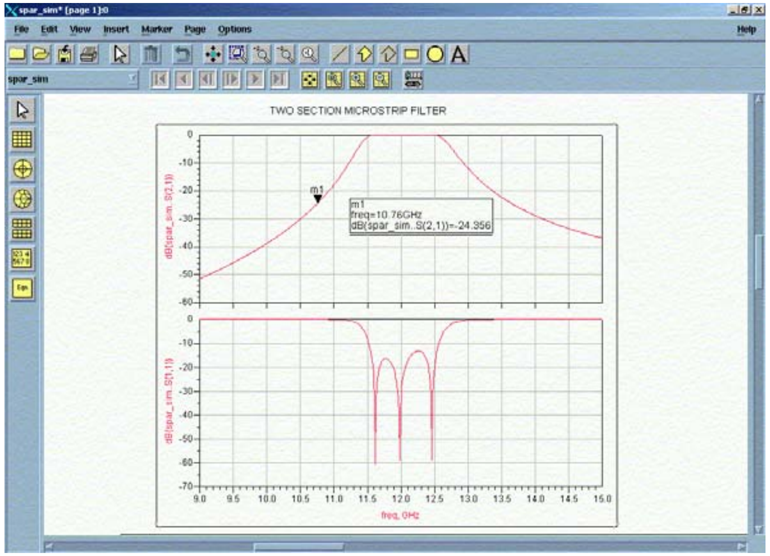
MSUB
MSub1
H=15 mil
Er=3.78
Mur=1
Cond=1.0E+306
Hu=1.0E+30 meter
T=0.1 mil
TanD=0
Rough=0 meter

MCFIL
CLin1
Subst="MSub1"
W=25.5 mil
S=3.3 mil
L=138.5 mil
W1=32 mil
W2=31.5 mil

MCFIL
CLin2
Subst="MSub1"
W=31.5 mil
S=18.0 mil
L=136.0 mil
W1=32 mil
W2=31.5 mil

Port
P2
Num=2

Fig. 2 The detail within the “halfil” sub ckt



m1
freq=10.76GHz
dB(spar_sim..S(2,1))=-24.356

Fig. 3 S-parameter simulated results

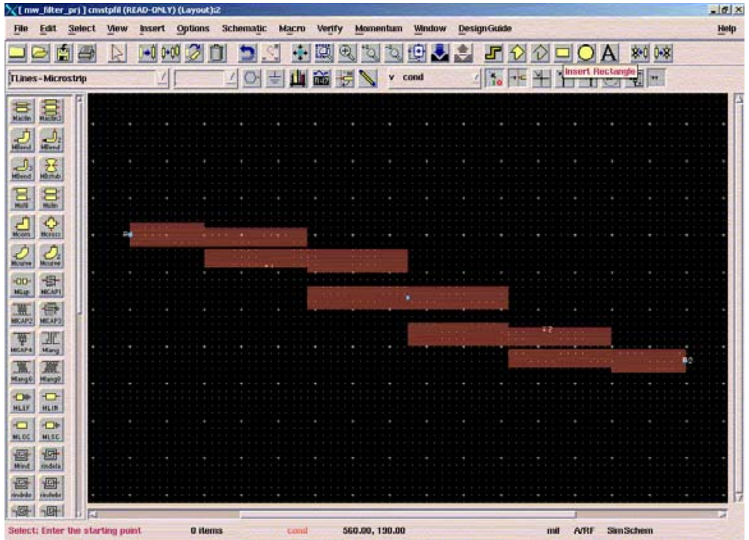


Fig. 4 Automatically-generated layout

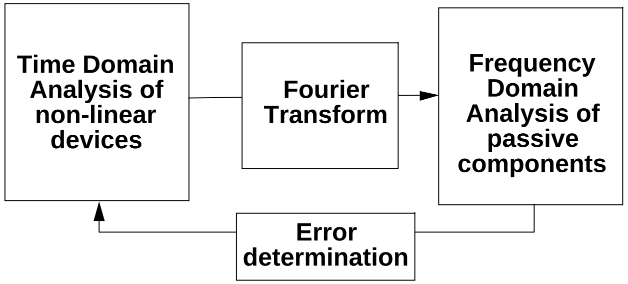
3. Non-Linear Analysis

In order to calculate the true response of non-linear components such as transistors it is necessary to work in the time-domain. This is because the equivalent circuit elements vary with signal amplitude, and are thus functions of time. Direct time-domain methods are generally based on SPICE, with more convenient display options added.

The handling of transmission-line elements is difficult in the time-domain because of the long time taken to reach a steady state and the need to consider dispersion. Some of the newer time-domain simulators use a convolutional technique which does makes it possible to handle dispersion.

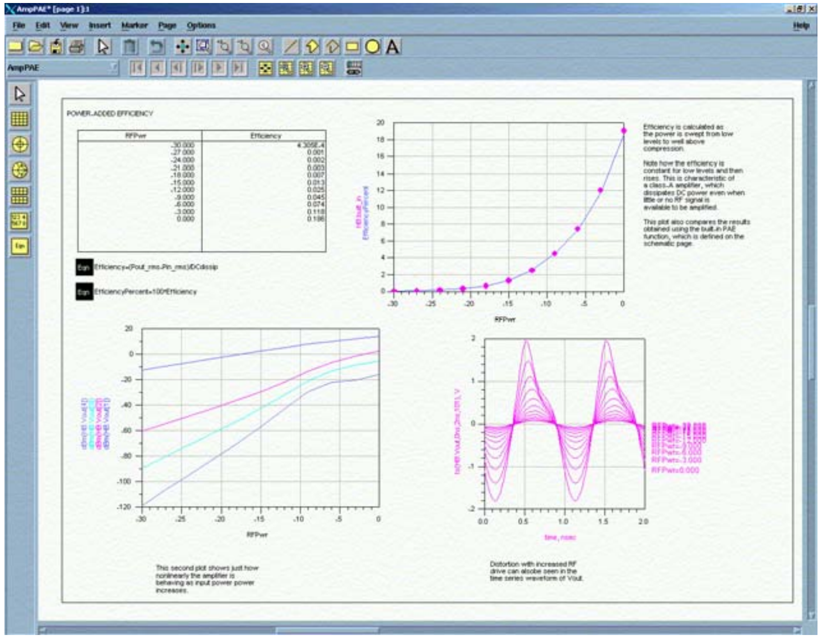
For steady-state large-signal analysis (such as gain compression of a power amplifier) the harmonic balance technique is faster than time-domain methods and is now an industry-standard tool. This uses a combination of time- and frequency-domain analysis. The voltages & currents in the non-linear components are calculated in the time domain. The linear elements are treated in the frequency domain. Fourier transformation is then used to compare the currents and voltages of the non-linear components with the terminating impedances presented by the linear components at all the harmonic frequencies used in the analysis. Any error is reduced by successive iterations, as shown in Fig. 5. The signals must be periodic in order to use the Fourier transform. Due to the enormous number of calculations required, and the need for iterations to ensure convergence, this type of analysis requires a fast computer with a lot of memory. These requirements go up dramatically as the number of harmonics increases. The harmonic balance analysis is available in ADS, Microwave Office and Harmonica™. Fig. 6 shows simulations of the output waveforms, intercept diagram, and efficiency vs. output power for a power amplifier. Load-pull contours can often also be simulated.

For weakly non-linear circuits, the Volterra series analysis is fast and sufficiently accurate; this is available in Microwave Office.



The voltages & currents in the non-linear subcircuit are transformed out of the time domain and compared with the terminating impedances presented by the linear subcircuit at all the harmonic frequencies used in the analysis. Any error is reduced by sucessive iterations.

Fig. 5 Simplification of the harmonic balance technique



POWER-ADDED EFFICIENCY

RFPwr	Efficiency
-30.000	4.305E-4
-27.000	0.001
-24.000	0.002
-21.000	0.003
-18.000	0.007
-15.000	0.013
-12.000	0.025
-9.000	0.045
-6.000	0.074
-3.000	0.118
0.000	0.186

Eqn Efficiency=(Pout_rms-Pin_rms)/DCdissip

Eqn EfficiencyPercent=100*Efficiency

Efficiency is calculated as the power is swept from low levels to well above compression.

Note how the efficiency is constant for low levels and then rises. This is characteristic of a class-A amplifier, which dissipates DC power even when little or no RF signal is available to be amplified.

This plot also compares the results obtained using the built-in PAE function, which is defined on the schematic page.

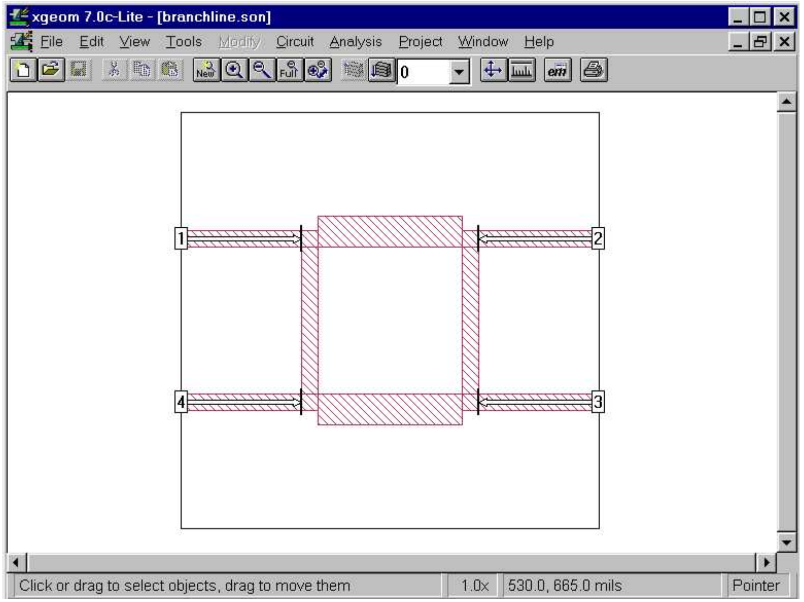
This second plot shows just how nonlinearly the amplifier is behaving as input power power increases.

Distortion with increased RF drive can alsobe seen in the time series waveform of Vout.

Fig. 6 Typical harmonic balance PA output

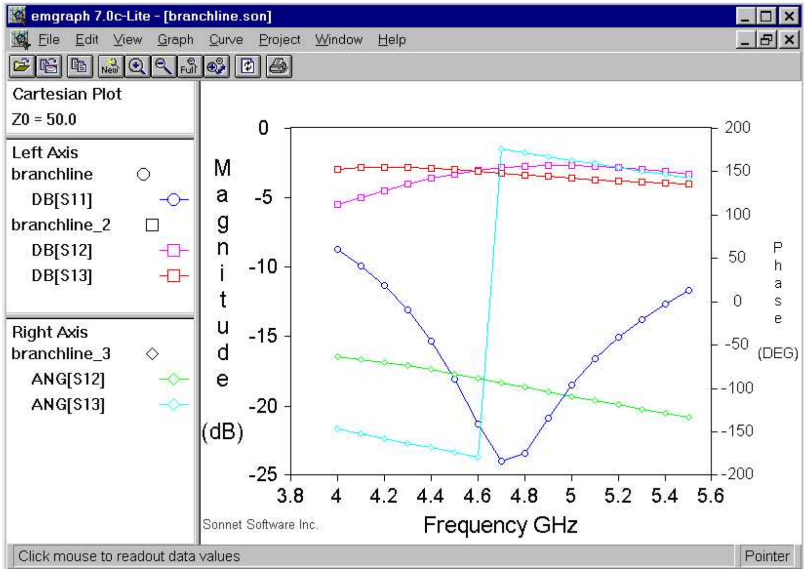
4. Planar Electromagnetic Analysis

These programmes are intended to solve for the S-parameters of arbitrarily shaped microstrip or CPW structures. Usually, the circuit conductors are divided into subsections and the method of moments is used for the electromagnetic analysis. Examples of this type of simulator are *em*™ (Sonnet), Ensemble™, and Momentum™. Often, many metal and dielectric layers can be handled, but these are assumed to be planar, and so the term 3D planar has been coined (there are also carefully-defined cases of 2.5D, 2.6D…2.9D simulation!). These simulators cannot normally analyse true 3-D structures, such as microstrip-to-stripline transitions where the dielectrics are not planar. 3D planar simulators are used extensively in MMIC design to analyse non-standard microstrip structures such as coupled bends and meandered lines. Fig. 7. shows the layout of a branch-line coupler on *xgeom* ™ by Sonnet Software. The simulator *em*™ can handle any number of ports and metal or dielectric layers.



Click or drag to select objects, drag to move them

Fig. 7 Screenshot of Sonnet Lite 7



Click mouse to readout data values

Fig. 8 EM simulation of the branch-line coupler

5. 3D Electromagnetic Simulation

The High Frequency Structure Simulator (HFSS™) has dominated this market for many years and is now a product unique to Ansoft. This uses finite element techniques to analyse completely arbitrary metal and dielectric structures and can thus be used for analysing components such as waveguide-to-microstrip transitions and MMIC packages. However, this type of analysis is extremely demanding of CPU time and memory: At least 300Mb of RAM is needed. Since most MIC and MMIC components are at present basically planar in nature, 3-D simulation is not used extensively. Other notable 3D simulators include CST Microwave Studio and Microstripes. Microwave Studio uses the Finite Difference Time Domain (FDTD) technique. Microstripes uses the transmission-line matrix (TLM) method which is a fast technique requiring less memory than finite element methods.

6. System Block Diagram Simulation

These programmes treat individual circuits as black boxes with some kind of describing function (not just S-parameters). This enables complex subsystems, such as complete transmitters and receivers, to be designed at a high level. An example of this is shown in Fig. 9 which shows part of a CDMA transmitter. Fig. 10 shows the output spectrum. Since systems are becoming increasingly complex, and as digital signal processing becomes closely integrated with microwave hardware, system level and mixed-signal simulation is becoming more important and is receiving a great deal of attention. Note that where pseudorandom baseband signals and high frequency carriers are simultaneously present, there may be ***huge*** numbers of sample points to be calculated; the "modulation domain" analysis in ADS addresses this problem.

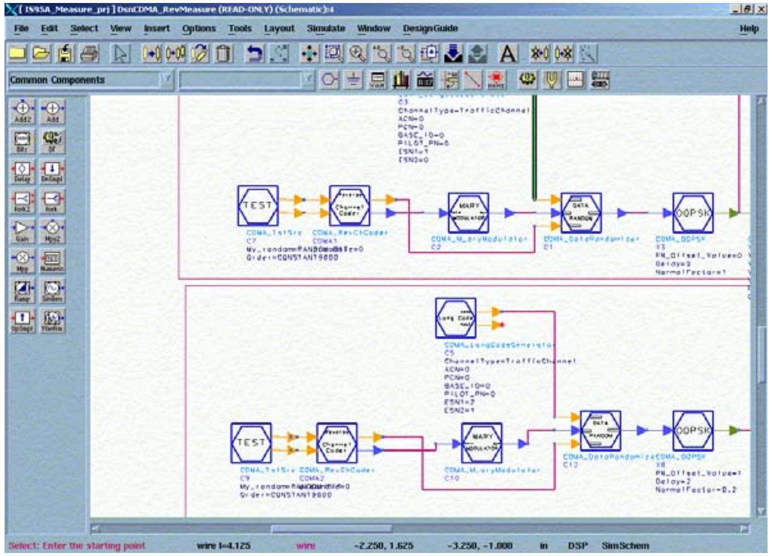


Fig. 9 Part of a CDMA transmitter in ADS^{TM}

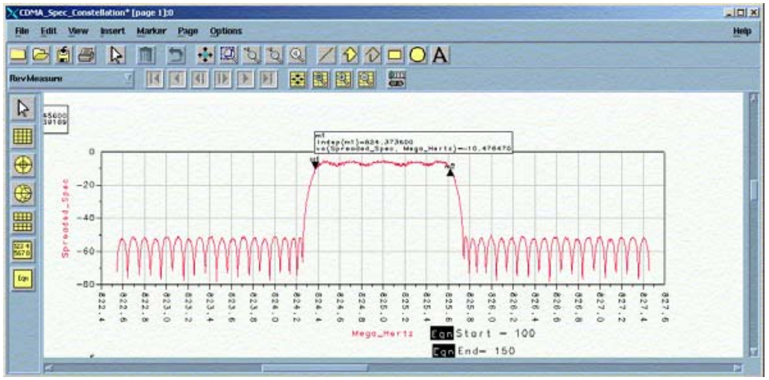
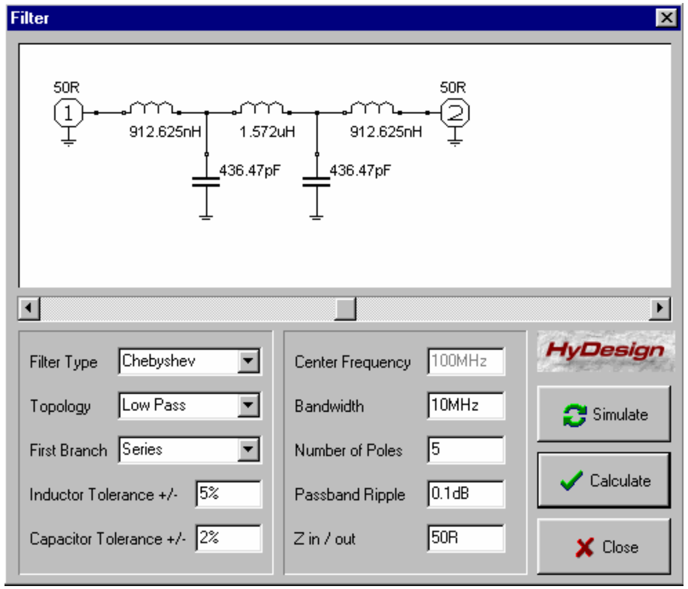


Fig. 10 Simulated output spectrum

7. Filter / Matching network synthesis

These programmes carry out the design of filters and matching networks using established synthesis techniques. They should be used with caution, however: Too tight a specification (ripple, bandwidth, etc.) leads to over complex networks with extreme component values.



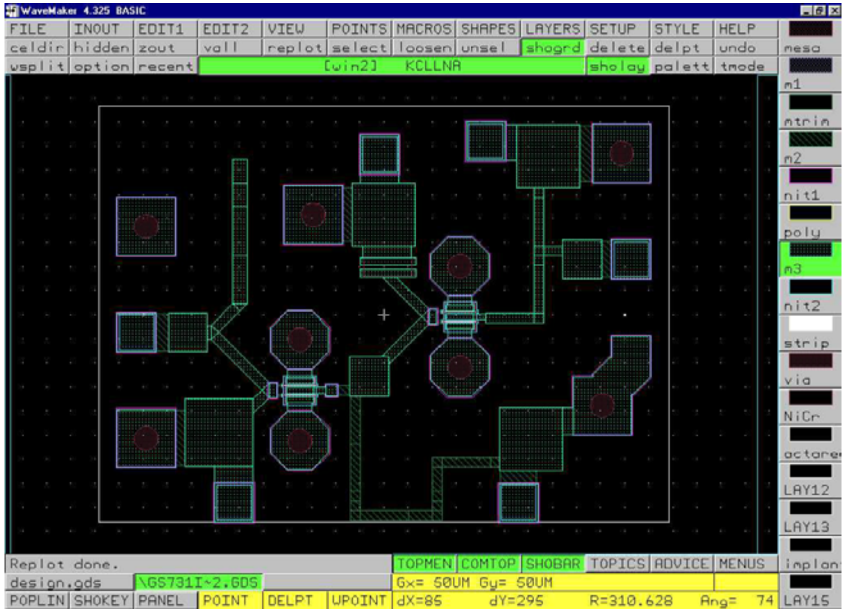
Filter Type: Chebyshev
Topology: Low Pass
First Branch: Series
Inductor Tolerance +/-: 5%
Capacitor Tolerance +/-: 2%
Center Frequency: 100MHz
Bandwidth: 10MHz
Number of Poles: 5
Passband Ripple: 0.1dB
Z in / out: 50R

Fig. 11 Lumped element filter synthesis with RFSim99

8. Layout and Design Rule Checking

RFIC and MMIC layouts have to be in the CALMA™ GDSII industry standard format. Layout packages are usually available within the simulation package; if not, WAVEMAKER™ Layout offers a tremendous range of powerful layout commands yet is economical and can run on anything from a modest PC upwards. This type of dedicated programme is often superior to the layout facilities in microwave CAD packages. Fig. 12 shows a screen shot of an MMIC amplifier layout on WAVEMAKER™.

However, to create a complete mask set reticle requires a more powerful tool such as CADENCE or MENTOR GRAPHICS. For serious integrated transceiver designs, CAD packages are now offering their simulator "engines" for use with these industry standard interfaces.



Replot done.
design.gds \GS731I~2.GDS
Gx= 50UM Gy= 50UM
dX=85 dY=295 R=310.628 Ang= 74

Fig. 12 MMIC amplifer layout on Wavemaker

9. Types of Component Models

9.1 S-parameters

Often, component manufacturers and IC foundries supply a large number of S-parameter files to cover the most commonly used components. For example, data for 1 turn, 1.5 turn.....up to 6 turn spiral inductors may be provided. If the measured data has been de-embedded accurately, and the reference planes defined properly, then this data should be very reliable. This is discussed later. However, it is often the case that, for example, a non-standard number of turns is required. This data-file approach has clear limitations for design and optimisation.

9.2 Standard CAD elements

The standard elements from the CAD package can be used (e.g. the microstrip lines and discontinuities in ADS™ and others). However, these are not necessarily that accurate: Examples of where these may not apply include very high frequencies (say > 20 GHz) where dispersion effects are not accurately predicted, and circuits where lines and discontinuities are so close together that coupling occurs.

9.3 Custom equivalent circuit models

In this approach a range of components is measured, and an equivalent circuit model is fitted to the measured S-parameters. Different model parameters can be looked up for different components, OR, preferably, curve-fitting equations can be generated which give the values for the equivalent circuit as a function of either the layout dimensions or other parameter (such as the number of turns of an inductor). This approach has the advantage that the model is not limited to discrete values of standard elements. Optimisation can therefore be used. However, care must be exercised to ensure that the models are not used outside their range of validity and that they are accurate at high frequency. The most common problems are encountered with lumped spiral inductor models, since these are often inaccurate past the self-resonant frequency of the inductor.

9.4 Special CAD elements

Many CAD packages now include special library elements which have been developed for various situations. An example is the 3-port capacitor in LINMIC+™, shown in Fig. 13. These elements are often based on rigorous theoretical treatment of the component, resulting in an accurate model which requires many input parameters. The important feature is that distributed effects in the component are taken into account (e.g. at 20 GHz this is important even for a very small capacitor). Another example of special CAD elements is the library of multilayer PCB transmission-line elements offered in ADS.

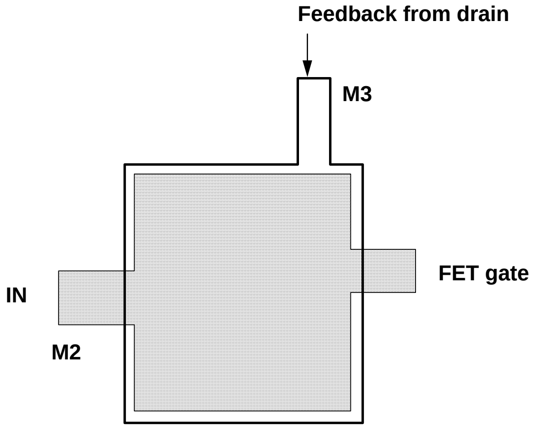


Fig. 13 Three-port capacitor example

9.5 S-parameters from EM simulation

Electromagnetic simulation of arbitrary planar structures is particularly useful for microstrip circuitry when standard CAD elements are invalid due the close proximity of bends, Tee-junctions, etc. So, after the bulk of the design has been carried out using CAD elements, the initial layout is carried out. Once the required geometry of the microstrip is known, the layout (of a meandered line, for example) can be imported directly into the em analysis package.

The S-parameters are calculated and the new circuit response checked. This process is now transparent with many of the latest CAD packages.

9.6 Linear Active Device Models

For linear simulations these can be S-parameters, equivalent circuits for a fixed device and bias point, or scaleable/bias-dependent equivalent circuits. A scaleable model is one which can model different gate-widths or areas. A bias-dependant model is one in which the small-signal equivalent circuit elements vary with a bias parameter. For example, the drain current of a FET might be a parameter: This allows the designer to see how the bias affects the gain of an amplifier, but it is not a large-signal model.

9.7 Large-Signal Active Device Models

Large-signal device models (transistors and diodes) are ones in which the DC and RF parameters are all calculated dynamically for the instantaneous voltages applied to the device. As a result, when using a large-signal model the DC bias supplies must be added to the simulation as voltage or current sources. Many different large-signal models exist for FETs, such as the Curtice Cubic, Materka, Parker-Skellern, Triquint's Own, HP Root, etc. The accuracy of large-signal models still leaves a lot to be desired, especially for highly non-linear circuits such as mixers. One of the reasons for this is that some devices (especially FETs) suffer trapping effects which have a significant effect at DC and low frequencies, but do not respond at RF due to the long time constant. The upshot of this is that the DC I-V curves are sometimes nothing like the RF dynamic behaviour. Furthermore, heating effects in devices can seriously affect the IV curves. Pulsed I-V measurement systems can overcome this problem.

10. Simulator customisation for RFIC & MMIC design: SMART Libraries

RFIC and MMIC technology is firmly established and is widely acknowledged as being a key factor in the widespread commercial exploitation of the microwave and millimetre-wave frequency range. Fierce competition requires that designs work first-time to reduce time-to-market. Since successful MMIC design relies very much on the accuracy of the available models for active and passive components, it is vital that manufacturers can offer extensive model libraries and design information. The extent to which a process has been characterised and carefully developed into a fixed production process is often more important to a customer than the ultimate performance achievable. The characterisation and process definition task, from the first RFOW testing of a new device up to the official release of a model library and design manual, may take several years. The model library is then the fruit of a great deal of labour, and whilst it represents only a small part of the manufacturer’s modelling and design information, the model library is the basis of all the customer’s design work.

The various types of microwave CAD package have been briefly described already. In a fully integrated CAD package there is seamless integration of the schematic and layout. In order to achieve this level of design integration for ICs, the simulator must be customised to the foundry. This means that the foundry's models and layout library elements form a core part of the simulator's element library. In this way the layout and schematic can be linked together: A change made in the layout window will automatically result in a change of the parameters in the schematic. For example, if you edit the length of a microstrip line in the layout editor then the new length information is passed to the schematic, and the simulation can be re-run directly. This transfer of information directly between the layout and the schematic/simulation is called back annotation. The ability to edit the layout (i.e. the artwork) of a particular library element and then re-simulate directly has led to the phrase **S**imulateable **M**icrowave **ART**works, which was coined by EEsof some years ago. These SMART™ library elements have the component model and layout integrated, so the design becomes partially automated. This is illustrated in Fig. 14. It should be noted, however, that there is a limit to the complexity of a layout which can be created in this way. Also, despite all the apparent sophistication of such a CAD suite, it is as important as ever that the designer is aware of the nature and limitations of the models that he/she is using.

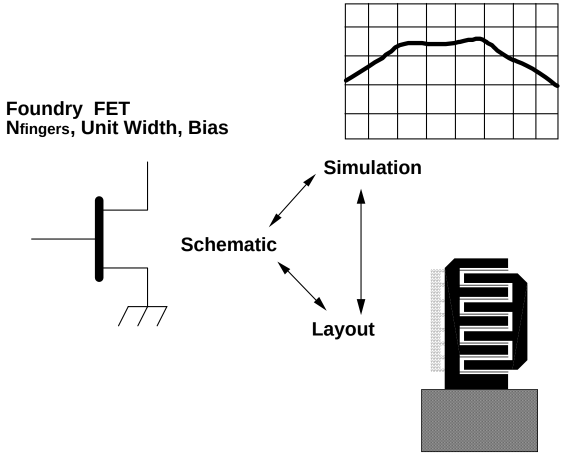


Fig. 14 Smart Library concept

10.1 Test Mask Measurements

The creation of a new MMIC process starts with the layout of a test, or characterisation, mask-set. This will include a large number of active and passive components and a number of test structures for process monitoring, as shown in Fig. 15. The layout of the active devices will rely on experience, engineers' intuition and perhaps physics-based device simulations; the mask set will certainly undergo a few iterations during the development of the process. Wafer testing and process characterisation starts with DC and low frequency tests on the PCM (process control monitor) components. These tests yield vital process information such as carrier concentration, doping profile, sheet resistivity, contact resistance, metal conductivity, and dielectric permittivity. From these results, key enhancements and changes can be made to various stages of the process to centre the process. RF-on-wafer testing is conducted to investigate the performance of the active and passive components. The active devices are always most important; the passive components vary little from one process type to another and are relatively insensitive to minor changes in the process. In contrast, the active devices are extremely sensitive to process variations and it is a major task to develop a device/process with the optimum combination of performance and manufacturability.

Accurate MMIC measurements can easily be made using a microwave probe station. Already, the Cascade Microtech / Agilent or Karl-Suss / GGB / Anritsu wafer probing systems can perform repeatable measurements up to 110 GHz with a single frequency sweep. Fig. 16 shows an arrangement for measuring a MIM capacitor for modelling purposes: here, the capacitor is laid-out with microstrip feed lines to represent a typical scenario where it might be used in a circuit. Probe stations can use a TRL/LRL (through-reflect-line/line-reflect-line) or LRM (line-reflect-match) calibration procedure, with the reference standards located either on-wafer or on a precision impedance standard substrate (ISS). Since the probe is coplanar-waveguide (CPW), but the device-under-test (DUT) is in a microstrip environment there are a number of options for calibrating and de-embedding. CPW calibrations are believed by many to give the best accuracy and traceability, and have been studied by the National Physical Laboratory amongst others. But, for accurate capacitor measurements it is essential to exclude the CPW-microstrip transition; hence it may be better to use on-wafer through-reflect-line microstrip standards. In this way the probe-to-microstrip transitions are calibrated out to give accurate measurements for the capacitor in a microstrip environment. Care must be taken, however, because the microstrip characteristic impedance is affected by dispersion and is also dependant on the wafer thickness.

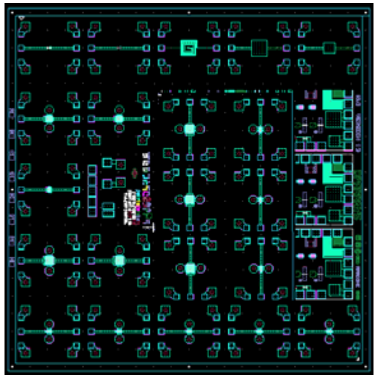


Fig. 15 Part of a typical test mask

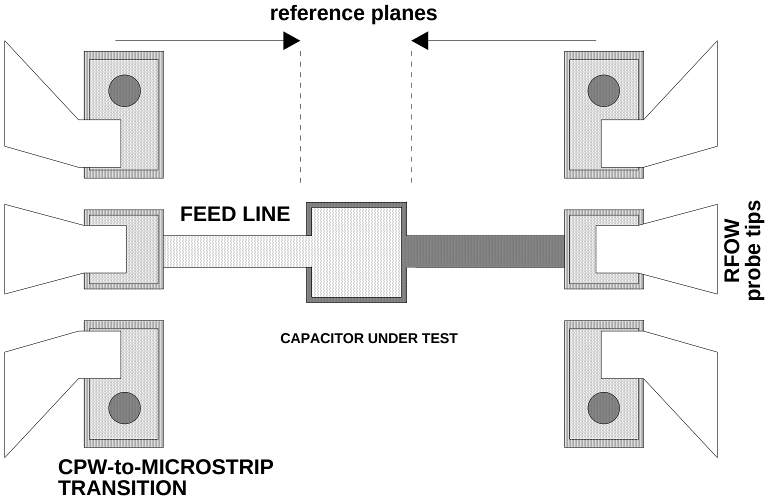


Fig. 16 RFOW measurement of a MIM capacitor for model parameter extraction

10.2 Passives’ Equivalent Circuit Models

Measured data is not ideal since it cannot be easily used for optimisation, and designers will need values between those that have been measured. Equivalent circuit models provide a way of producing a variable component value; the parasitic elements are derived from the measurements, and curve-fitting used to interpolate between the measured data.

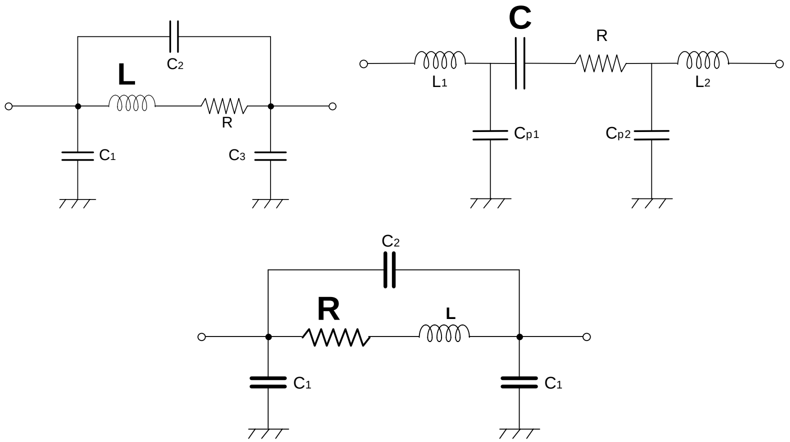


Fig. 17. Lumped-element equivalent circuit models (a) Inductor (b) Capacitor (c) Resistor

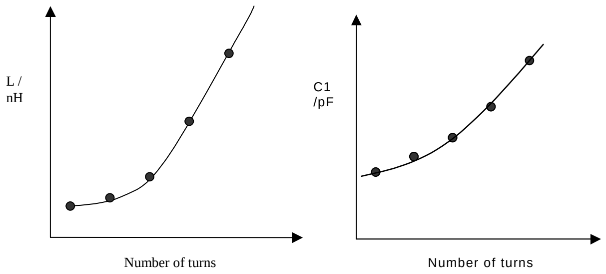


Fig. 18 Typical curve-fitting results

10.3 Active device models

From measurements of a wide range of device **geometries**, a scaleable bias-dependant model is reasonably easy to generate. The geometry refers to the number of gate/emitter fingers, the length and width of each, the spacing, etc, as shown in Fig. 19, for example. The basic device within, the **intrinsic device**, is not different. Hence, the parameters of the small-signal model (Fig. 20) which are within the dotted box, can easily scale with the area (gate width). For a FET, the intrinsic parameters scale closely with size:- g_m is proportion to the gate width; the capacitances Cgs, Cds, Cdg are proportional, and the resistance Rds is inversely proportional. The other model parameters, the extrinsic parameters, do not scale so simply and are extracted from device measurements on a range of geometries. By curve fitting, the values are then made simple polynomial functions of the geometric parameters (area, no. fingers etc.).

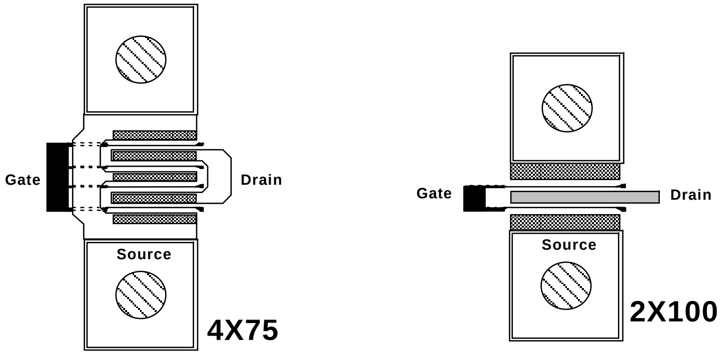


Fig. 19 4x75 and 2x100 HEMTs

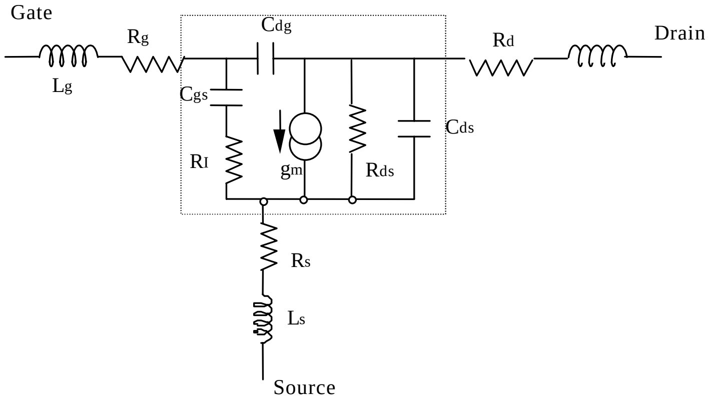


Fig. 20 Typical small-signal equivalent circuit

For large-signal models, every parameter in Fig. 20 will vary with input/output signal level. A model typically has >30 parameters to find. Assumptions in the model (say, tanh dependance of Ids on Vgs) will make it nearly impossible to get a good fit at all bias points. Fortunately, parameter extraction software exists which can help automate the process (IC-CAP, LASIMO, TOPAS, UTMOST and others).

This device modelling exercise is important because a major advantage of RFICs/MMICs compared with hybrid circuits is the ready availability of active devices and the freedom the designer has over the device geometry.

10.4 Programming the model library

There are four key stages in creating the custom library element:

(1) Measure a range of components
(2) Fit the equivalent circuit to each measurement and
    derive equations for the element values
(3) Programme the model, link it into the simulator
    and create a layout representation
(4) Create a user interface with customised menus, palettes and icons

Steps (1) and (2) have been described already. The procedures of steps (3) and (4) are entirely dependant on the CAD package under consideration. For example, it may be necessary to programme the model in the “C” language, compile the model, and then link it into the simulator. Files describing the icons, library palette, help files, etc, must also be created in the correct format. The amount of work involved is extremely significant. Thus, if a design house wishes to use a certain Foundry (for reasons of cost, performance, etc), then the design house probably has little choice but to adopt the CAD tools supported by that foundry. A design house using different Foundries may then have a large bill for CAD! Hence, “pay per use” and other flexible licensing payment schemes have been introduced.

10.5 Model Limitations

Foundry models are the most reliable means of simulating a circuit since they are based on measurements of components made with the relevant MMIC process. However, the models cannot be expected to cater for every structure that is encountered in a circuit, and model uncertainties become increasingly important for designs above 20 GHz. There are some inevitable potential limitations which the designer must be aware of:-

1) Differences from the test mask layout: for example, a capacitor might be used with different feed line widths, or with multiple feeds.

2) Distributed effects not incorporated into the model: e.g. most spiral inductor models are lumped models and are not accurate at frequencies above self-resonance.

3) Measurement uncertainty: as well as the vexed questions surrounding calibration techniques, a small probe placement inconsistency during measurement can result in significant phase discrepancy at >30GHz.

4) Grounding problems: even mutual inductance between neighbouring via-hole grounds has been shown to be potentially significant.

5) Coupling between components: all components are considered isolated from one another unless electromagnetic simulations are used.

6) Active device model limitations: large-signal models should not be expected to be accurate under all bias and signal conditions. Models can only represent a “typical” device, and sensitivity to device parameters must be checked.

11 Summary

This set of notes has described the various CAD techniques used in microwave circuits, without looking at the theory behind them. It has been shown how RFIC/MMIC library elements are derived from an extensive chracterisation programme requiring a coherent programme of measurements and equivalent circuit parameter extraction. The result is a set of models which are easy to use and represent the "typical" components whilst allowing statistical analysis to be performed for yield optimisation. Nevertheless, the models are based on measurements on a set of known standard structures, and great care must be exercised if using non-standard layout features, such as multi-port capacitors. Electromagnetic simulation packages provide a useful tool for analysing such non-standard elements and for investigating coupling effects.