

RF Power Amplifier Simulation with Microwave Office



Louis Brown
rfengr@everestkc.net



Simulation Methods

- A simulator is an equation solver. The models are the equations. Accurate results require accurate models and the skill to apply them.
- Large signal models of power transistors are not very accurate. Simulation should be used for verifying concepts and statistical analysis.
- Several types of simulators are used for PA design. Most all include layout for PCBs and some have RFIC process kits:

Linear – Simulation of passive networks and linear controlled sources. Prediction of power output using “Cripps Contours”.

Harmonic Balance – Nonlinear frequency domain analysis. Periodic sources decomposed into harmonics. KVL and KCL solved for user specified number of harmonics and order of products. Good for soft to medium nonlinearities. Fast runtimes for closely spaced tones.

Spice - Nonlinear time domain analysis. Good for strong nonlinearities. Long runtimes for closely spaced tones. Must handle distributed elements and s-parameters via equivalent circuits or convolution.

Envelope – Represents modulated waveforms with complex envelope. Nonlinear circuits may be behaviorally modeled, or envelope source sliced into time steps and circuit solved with HB. Amplifiers and mixers may be incorporated into high level systems such as digital communication links, AGC loops, and linearization schemes.

2.5D EM – Microstrip, stripline, RFICs, multilayer PCBs, thin printed antennas. Primary EM coupling in XY plane.

3D EM – Packages, wirebonds, waveguide, antennas. Not a substitute for 2.5D (ex. Lange couplers).

HB Simulation

- Distributed components and s-parameters pose some problems:

DC is the zero harmonic.

Closely spaced tones produce low frequency products.

S-parameter data should have a 0 Hz line or else the simulator will extrapolate a low frequency response or model it as a short, likewise for EM simulations.

High order products appear at high frequencies. Simulator may extrapolate data.

- Hard nonlinearities cause convergence problems. Tricks to achieve convergence:

Back off power

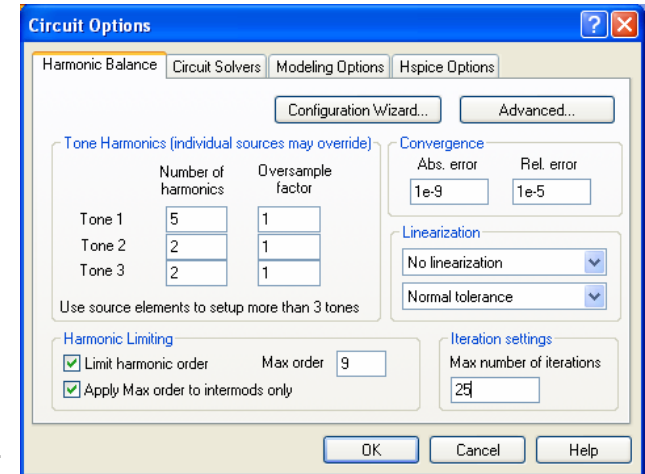
Adjust oversample factor.

Reduce error tolerance.

Turn off breakdown diodes in model or linearize them. Same for nonlinear capacitors.

! Coilcraft 0402CS-10N Chip Inductor

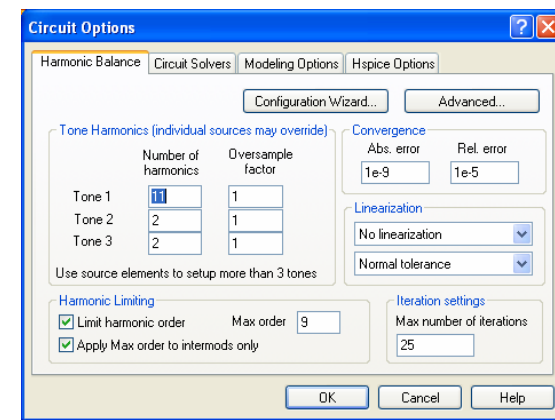
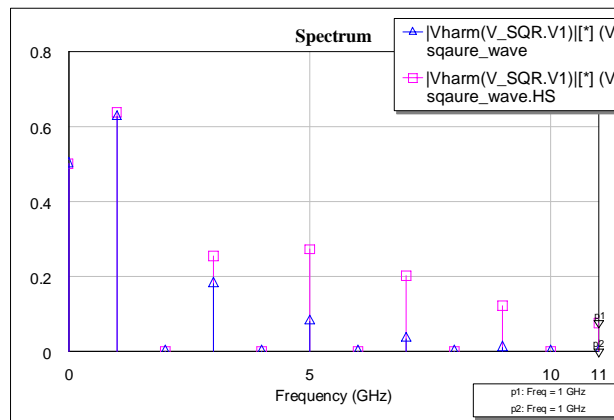
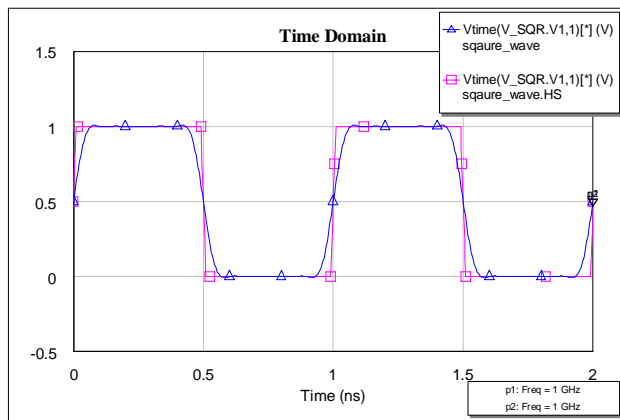
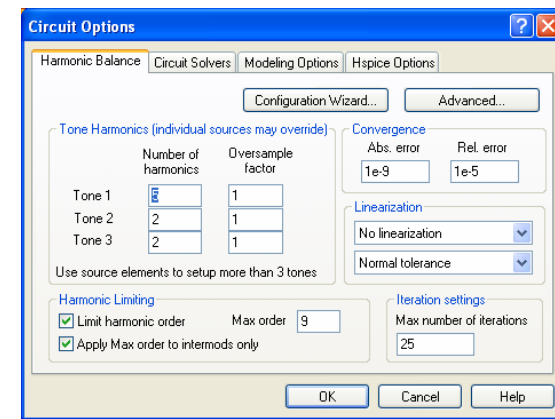
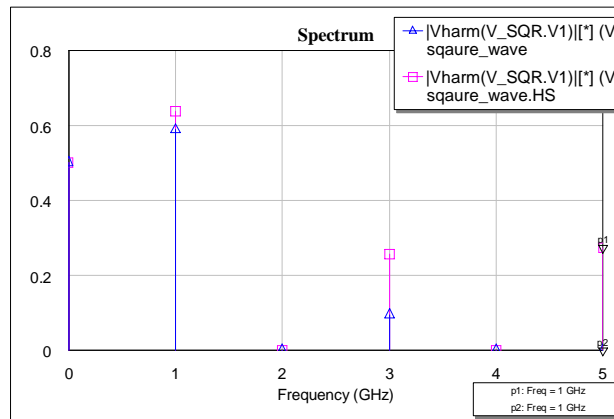
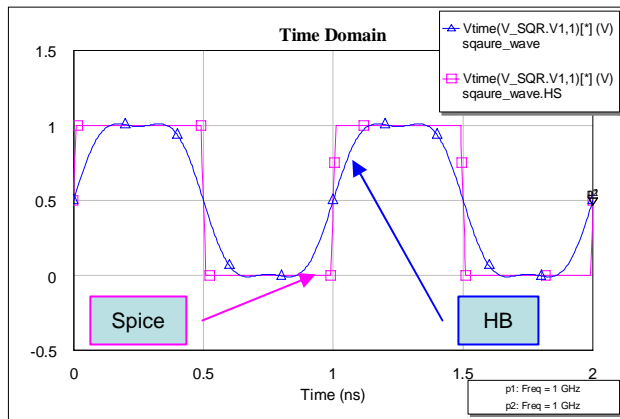
! 29 May 2002	# MHz S MA R 50	! Freq	MagS11	AngS11	MagS21	AngS21	MagS12	AngS12	MagS22	AngS22
0	1E-9	0	0.0007713331	54.474609	0.9999999999	0	0.9999999999	0	1E-9	0
1	0.0007713331	54.474609	0.999552003	-0.0359838749	0.999552003	-0.0359838749	0.999552003	-0.0359838749	0.000771333108	54.474609
21.7	0.0137349967	81.4213296	0.9980436	-0.779702275	0.9980436	-0.779702275	0.9980436	-0.779702275	0.0137349967	81.4213296
42.4	0.0266883634	82.9323629	0.997068075	-1.52215299	0.997068075	-1.52215299	0.997068075	-1.52215299	0.0266883634	82.9323629



- Good model should be “continuously derivable” meaning there should be a solution (no matter how small) for every harmonic.

HB Waveforms

- Sharp transitions difficult to model



Example Amplifier

Freescall MRF5S21150H reference design:

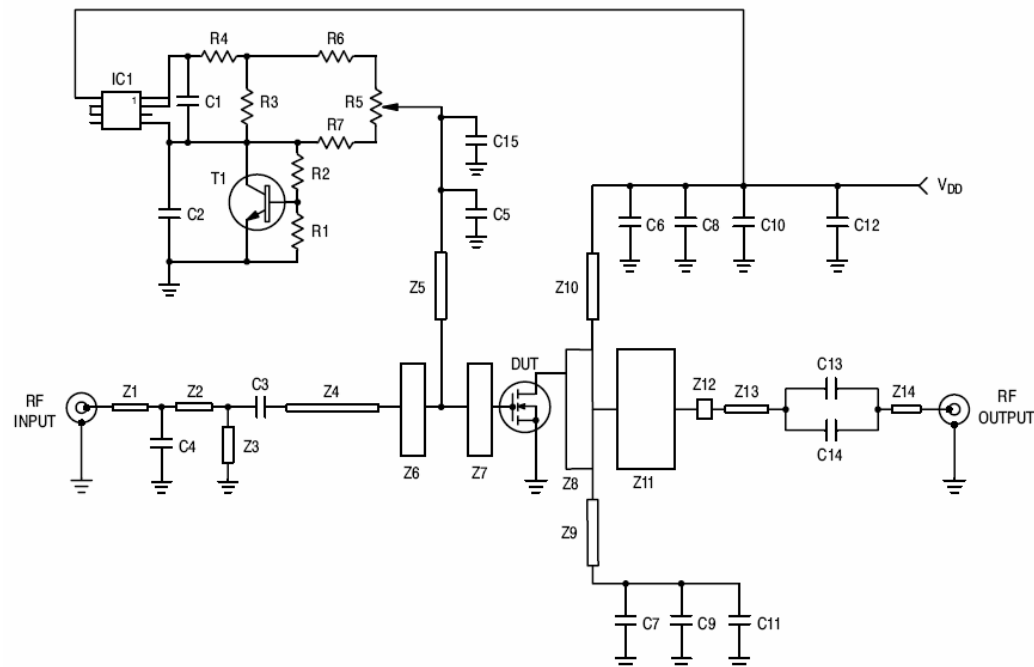
$P_{out}=33$ Watts at 2140 MHz, 2 carrier CDMA

Gain=12.5 dB

Eff=25 %

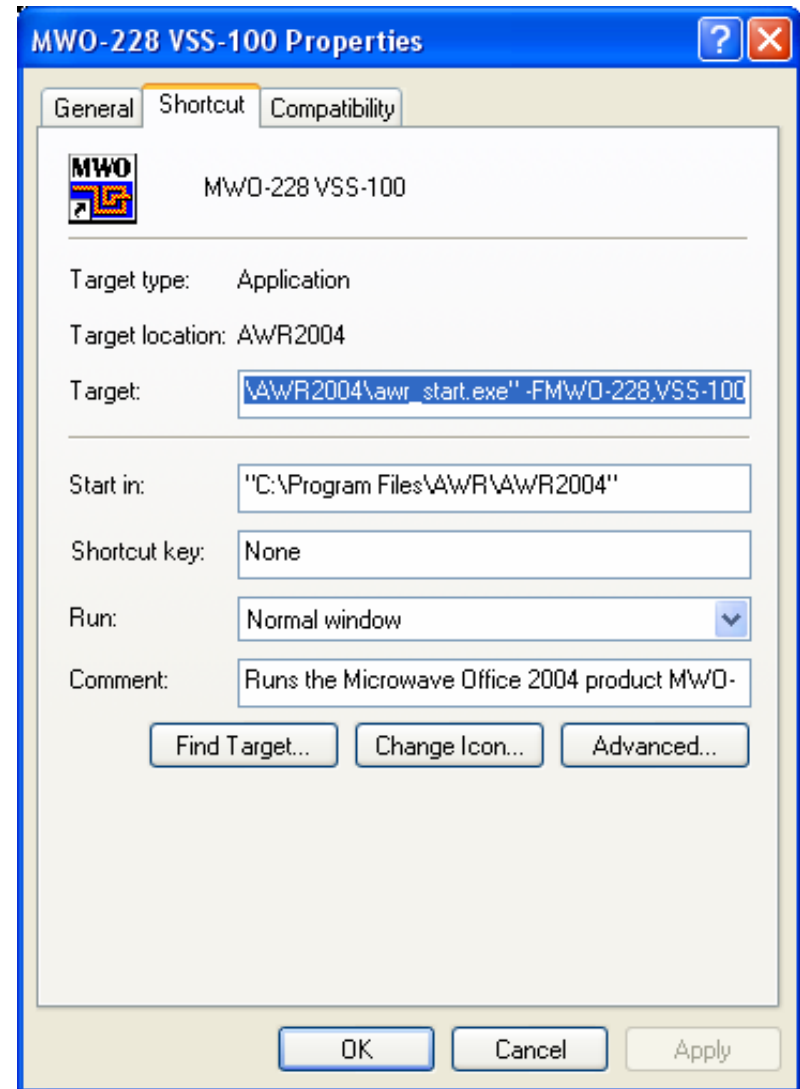
IM3=-37 dBc at 10 MHz offset

VCC=28 V, IDQ=1300 mA



Launching Microwave Office

- MWO-228, VSS-100 :
Linear
Harmonic Balance
EM
Layout
System
- May need to edit desktop icon and modify start parameters. May also use MWO-225



Project Setup

- Set Project Options, Global Units to:
 - MHz
 - mils
- Set Project Options, Frequencies to:
 - MHz units
 - 2140 MHz single point

The screenshot shows the 'Project Options' dialog box with the 'Global Units' tab selected. The 'Use Base Units' button is highlighted. The units are set as follows:

Category	Unit
Frequency	MHz
Angle	Deg
Temperature	DegC
Time	ns
Voltage	V
Resistance	Ohm
Conductance	S
Inductance	nH
Capacitance	pF
Current	mA
Power (Linear)	mW
Power (Log)	dBm
Length (Metric units)	mil

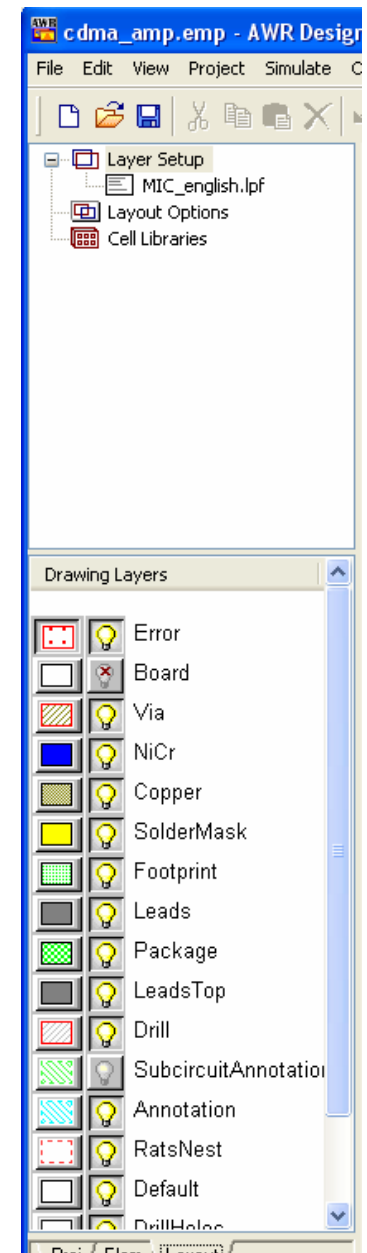
Buttons at the bottom: OK, Cancel, Help.

The screenshot shows the 'Project Options' dialog box with the 'Frequencies' tab selected. The 'Current Range' list contains '2140'. The 'Modify Range' section has 'Point (MHz)' set to '2140', 'Stop (MHz)' is empty, and 'Step (MHz)' is empty. The 'Single point' checkbox is checked. The 'Sweep Type' is set to 'Linear'. The 'Data Entry Units' are set to 'MHz'.

Buttons at the bottom: OK, Cancel, Help.

Layout Setup

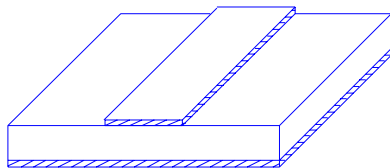
- Setup drawing layers by importing pre-defined Layout Process File “MIC_english.lpf”
- Project, Process Library, Import LPF
- Example process files located in:
 - C:\Program Files\AWR\AWR2004



Global Definitions

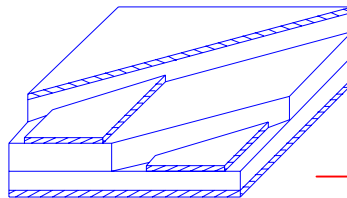
- Substrate information may be placed on Global Definitions page

MSUB
Er=3.48
H=20 mil
T=1.378 mil
Rho=0.7
Tand=0.004
ErNom=3.48
Name=RO/RO4350B

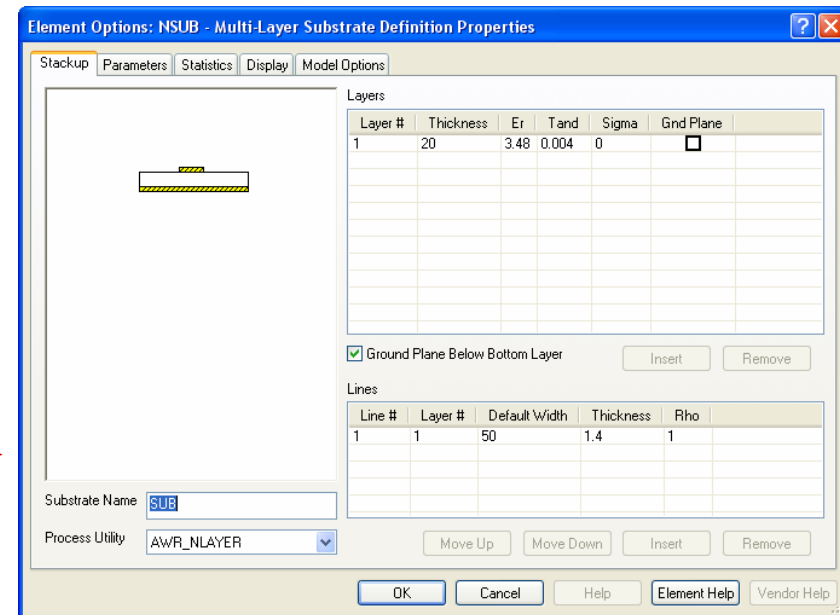


Microstrip substrate from vendor library

NSUB
HSub=20 mil
Er=3.48
Tand=0.004
ProcUtil="AWR_NLAYER"
Name=SUB

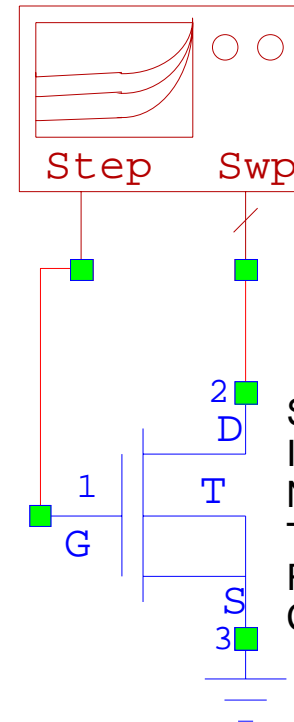


Multilayer substrate used for EM extraction



DCIV Schematic

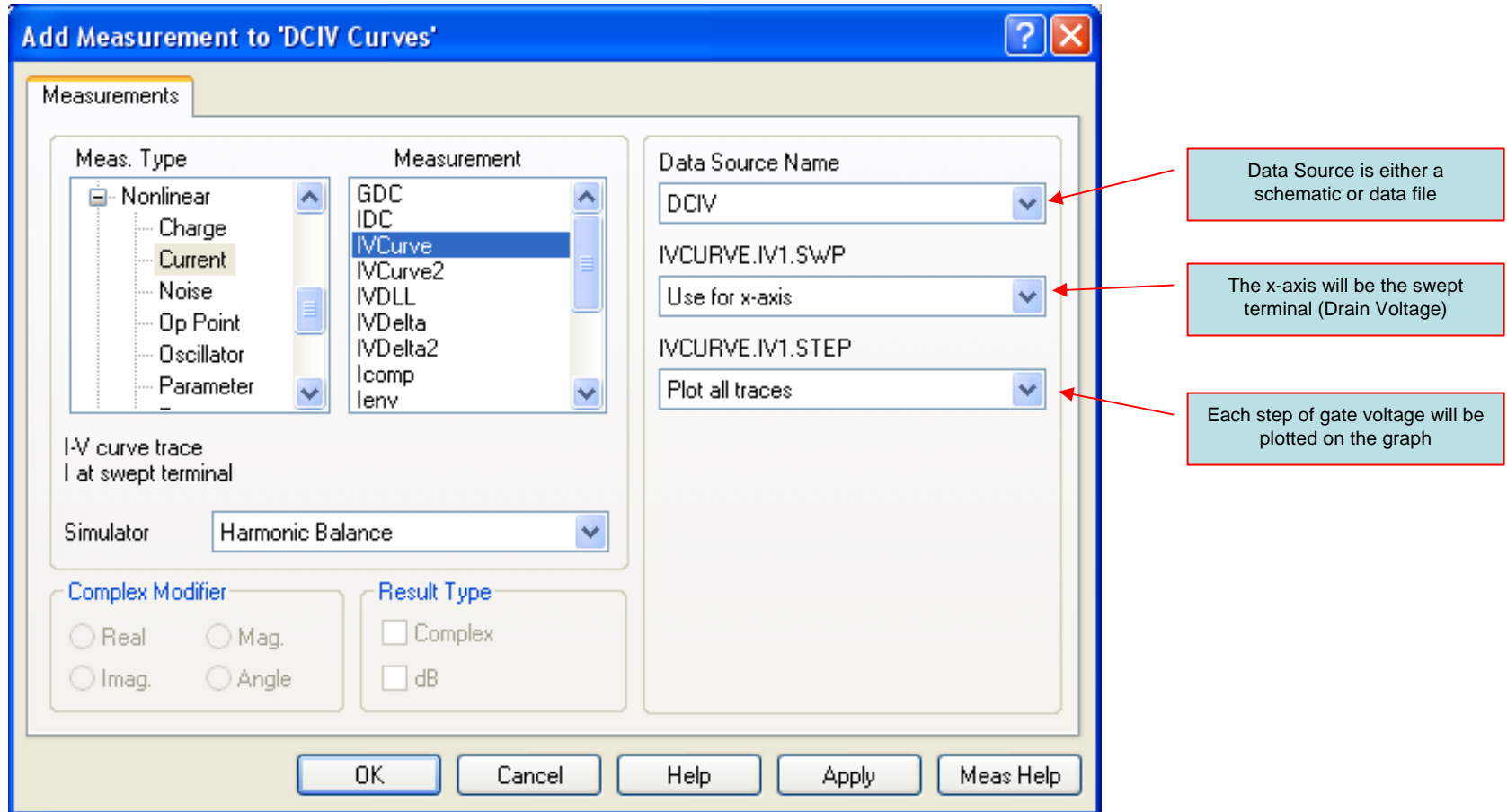
- Create a circuit schematic named “DCIV” consisting of the non-linear model and IVCURVE measurement device
- Use a curve tracer measurement to find V_{GS} that establishes $I_{DQ}=1300$ mA at $V_{DD}=28$ V



```
IVCURVE
ID=IV1
VSWEEP_start=0 V
VSWEEP_stop=60 V
VSWEEP_step=1 V
VSTEP_start=3.8 V
VSTEP_stop=4 V
VSTEP_step=0.05 V
```

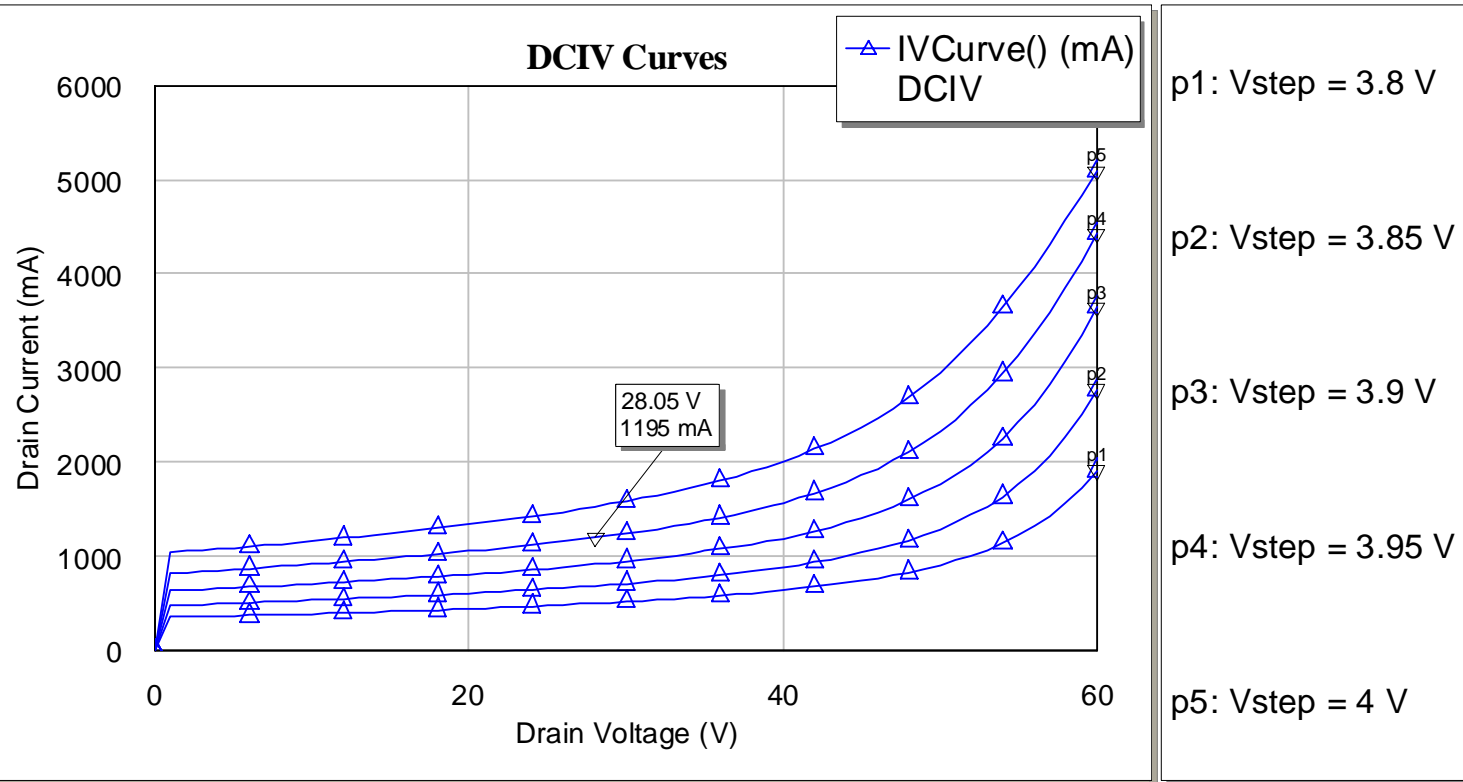
DCIV Measurement

- Create a rectangular graph named “DCIV Curves”
- Add an IVCurve measurement



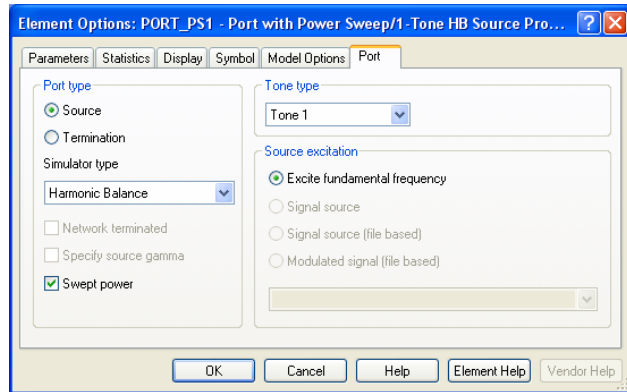
Finding the Bias Point

- Simulate then place a marker showing $V_{GS}=3.95\text{ V}$ yields (approximately) $I_{DQ}=1300\text{ mA}$ at $V_{DD}=28\text{ V}$



Model Verification Schematic

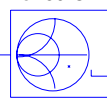
- Create a new schematic named “model_test” to verify performance using impedance tuners.



PORT_PS1
P=1
Z=50 Ohm
PStart=30 dBm
PStop=40 dBm
PStep=1 dB



LTUNER
ID=source_tuner
Mag=0.84
Ang=-154 Deg
Zo=50 Ohm



$Z_{source} = 4.70 - j11.03$

CAP
ID=C1
C=1000 pF



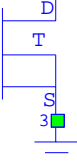
TLIN
ID=TL1
Z0=50 Ohm
EL=90 Deg
F0=2140 MHz



TLIN
ID=TL2
Z0=50 Ohm
EL=90 Deg
F0=2140 MHz



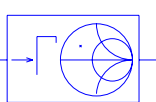
SUBCKT
ID=S1
NET="MRF5S21 150H"
TSNK=-1
RTH=-1
CTH=-1



CAP
ID=C2
C=1000 pF



LTUNER
ID=load_tuner
Mag=0.95
Ang=-174 Deg
Zo=50 Ohm



$Z_{load} = 1.16 - j2.46$

PORT
P=2
Z=50 Ohm



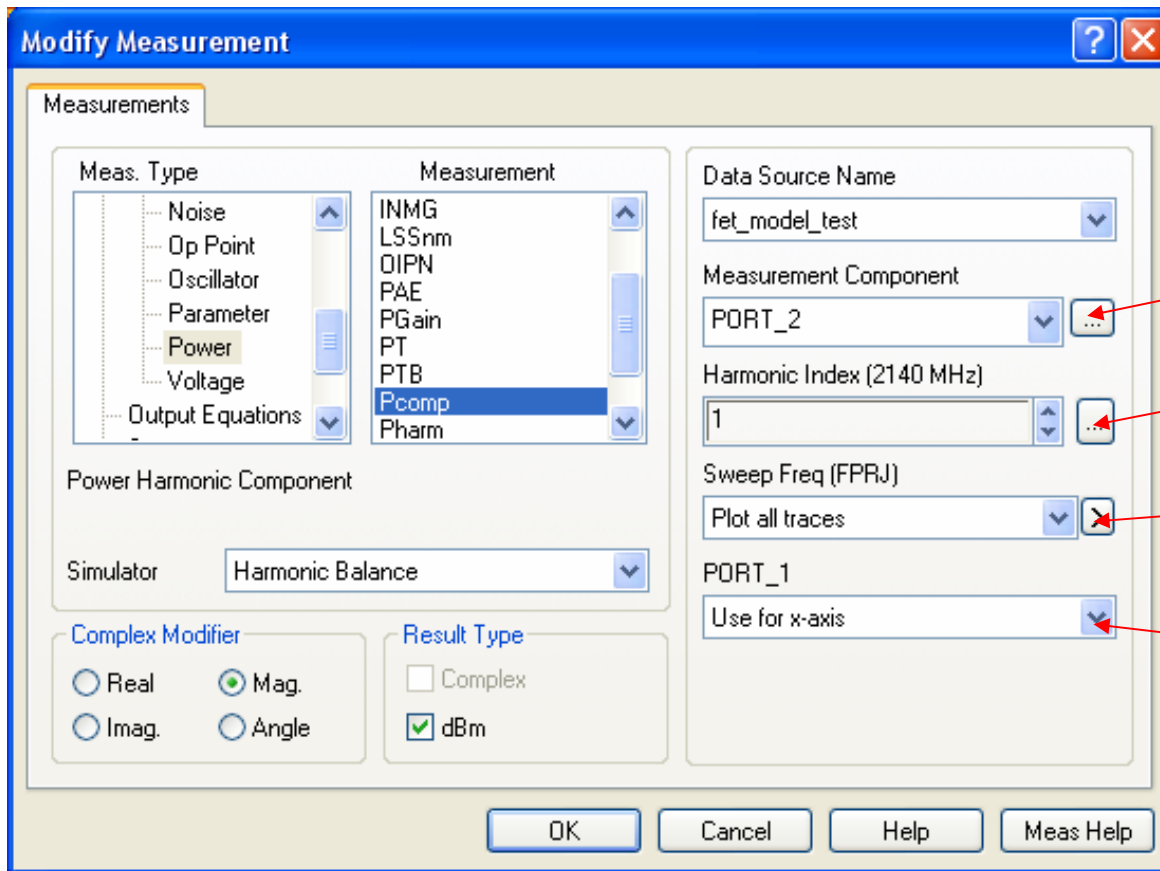
Right-click on standard port and modify to single tone source

Tuner gamma calculated from optimum impedance given in data sheet

1/4 wave bias used in reference design. 2nd harmonic will be shorted.

Model Verification Measurements

- Create a rectangular graph named “FET Model Output Power and Efficiency” and add an Pcomp measurement



Click this button to graphically select the measurement element

Click this button for “harmonic helper” pop-up window

Click this button to select “Project Frequencies” or “Document Frequencies”

Input power used for x-axis

Model Verification Measurements

- Add a DCRF measurement. The DC input power is calculated from all the DC sources on the schematic.

Modify Measurement

Measurements

Meas. Type: Charge, Current, Noise, Op Point, Oscillator, Parameter, **Power**

Measurement: AMtoAM, AMtoPM, **DCRF**, INMG, LSSnm, OIPN, PAE, PGain, PT

DC to RF Efficiency (%)

Simulator: Harmonic Balance

Complex Modifier: ☐ Real, ☐ Mag., ☐ Imag., ☐ Angle

Result Type: ☐ Complex, ☒ dB

Data Source Name: fet_model_test

Power Out Component: PORT_2

Sweep Freq (FPRJ): Plot all traces

PORT_1: Use for x-axis

OK Cancel Help Meas Help

Model Verification Results

- Comparison at $P_{in}=5$ W (37.0 dBm)
 - Model: $P_{out}=105$ W (50.2 dBm), $G=13.2$ dB, $Eff=55$ %
 - Measured: $P_{out}=102$ W, $G=13.1$ dB, $Eff=42$ %

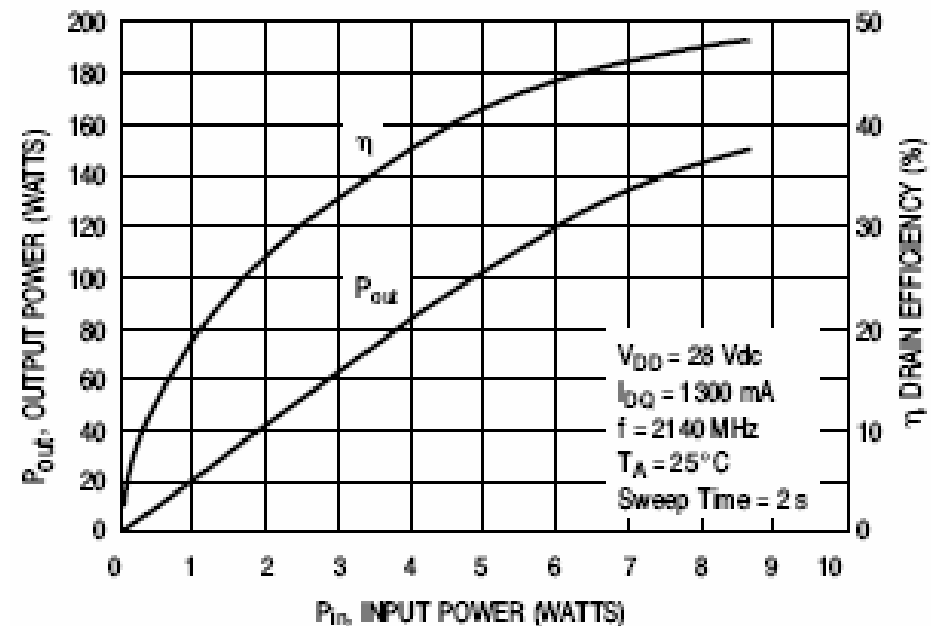
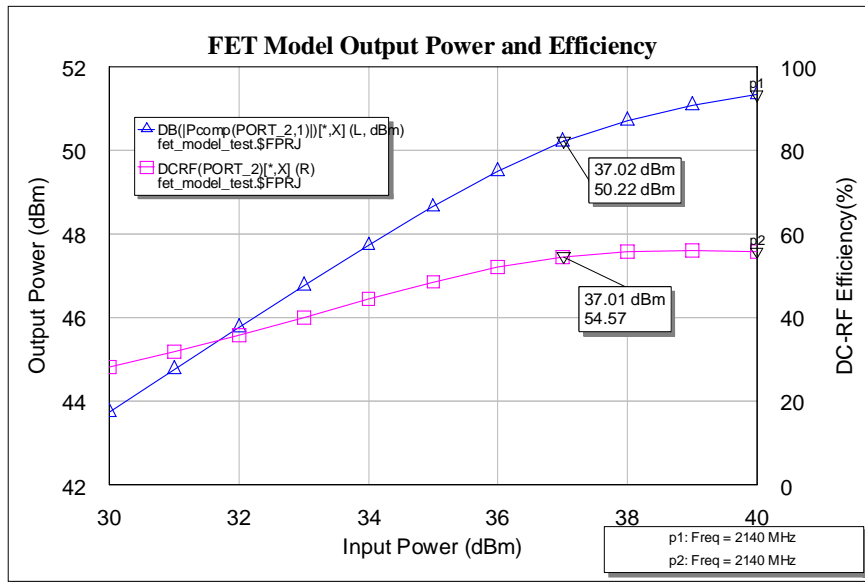
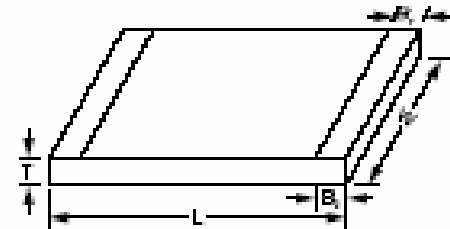
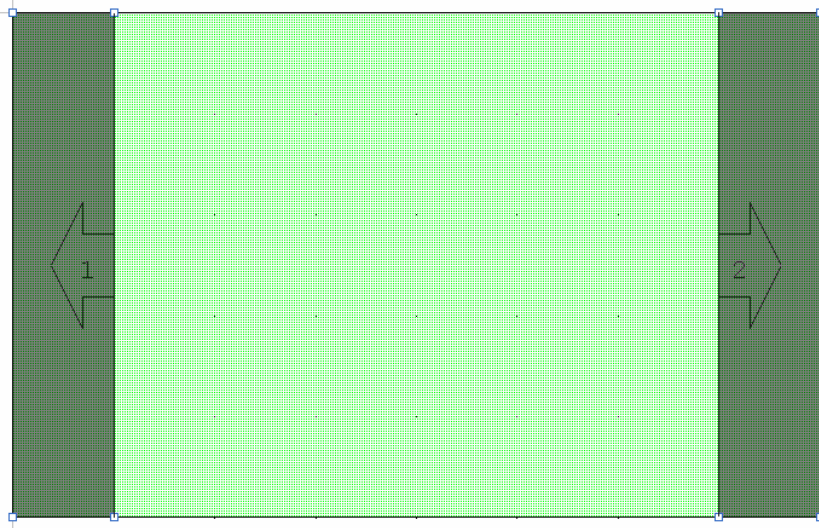


Figure 9. Output Power and Drain Efficiency versus Input Power

Layout Cell Creation

- Create a new GDS cell library named “RF_footprints”. Create a new cell named “0805_AVX”.
- Use a 10 mil snap grid to draw the footprint and leads on their respective layers.
- Add cell ports (reference planes) at the inside of the pads. Hold down CTRL to snap to geometry.



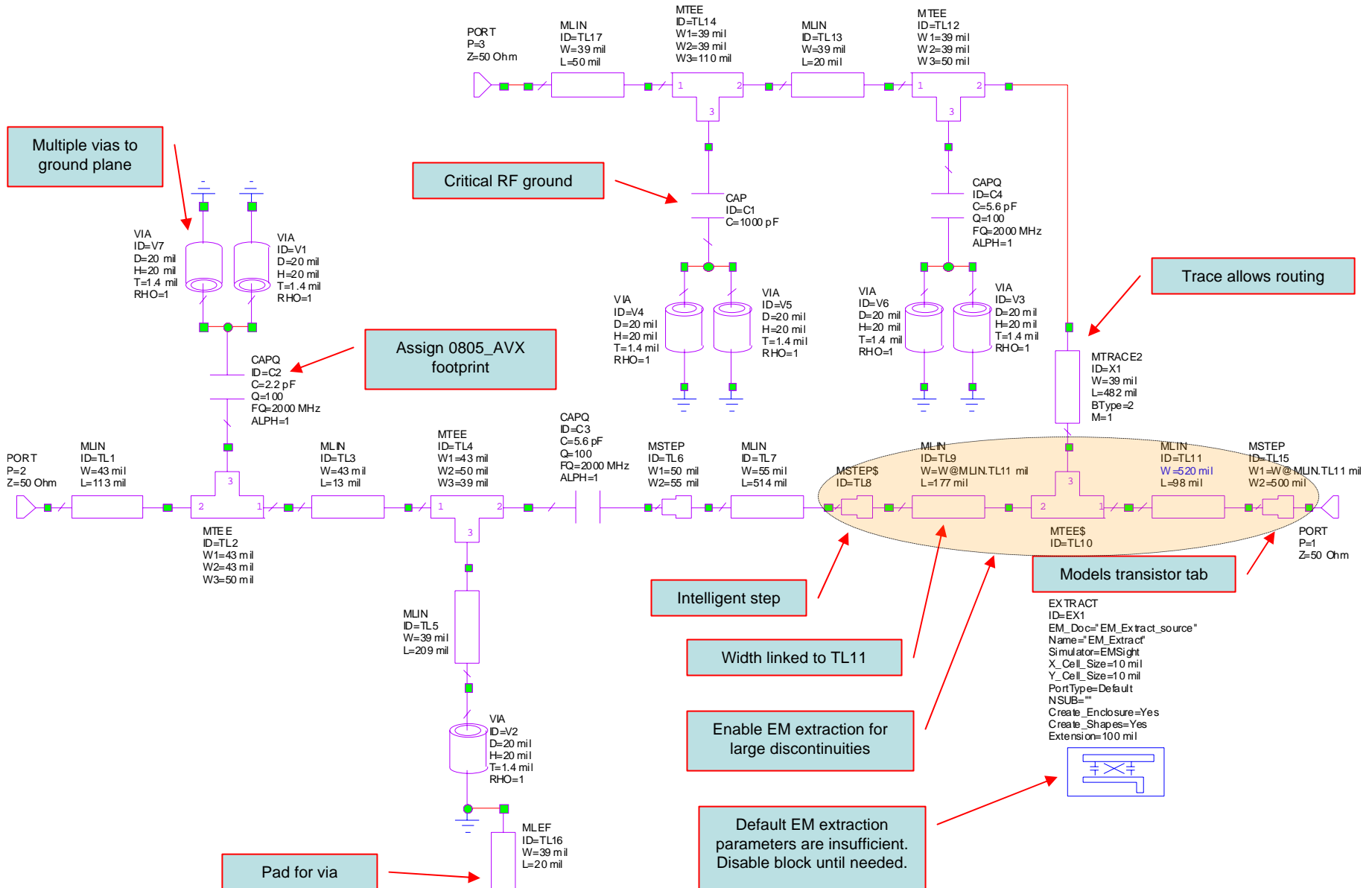
ACCU-F® *(Signal Type Capacitors)

	0603	0805
L	1.60±0.1 (0.063±0.004)	2.01±0.1 (0.079±0.004)
W	0.81±0.1 (0.032±0.004)	1.27±0.1 (0.050±0.004)
T	0.63±0.1 (0.025±0.004)	0.63±0.1 (0.025±0.004)
B	0.30±0.1 (0.012±0.004)	0.30±0.1 (0.012±0.004)

*Not recommended for new designs.
Accu-P's are recommended.

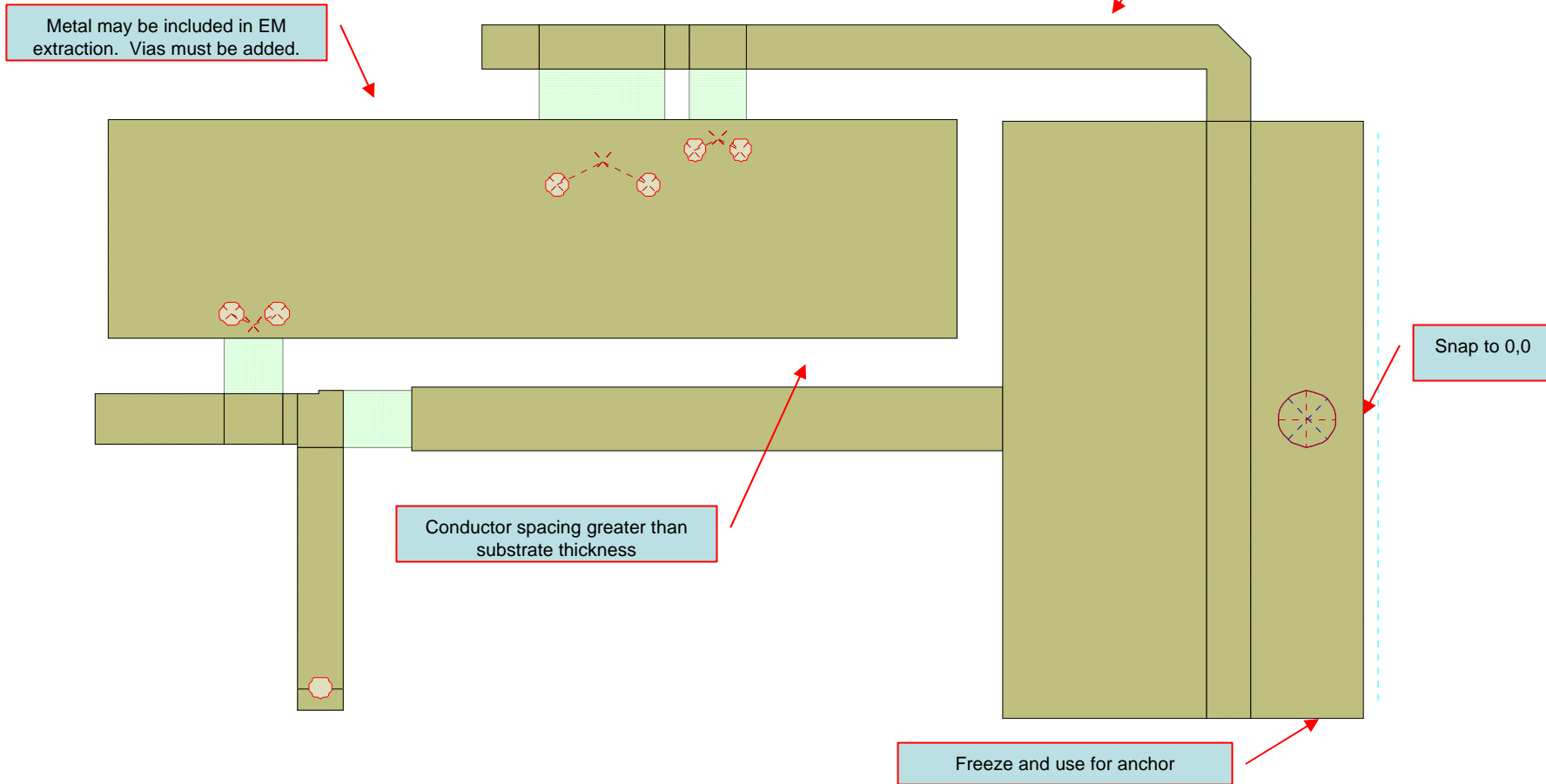
DIMENSIONS:
millimeters (inches)

Source Match Schematic



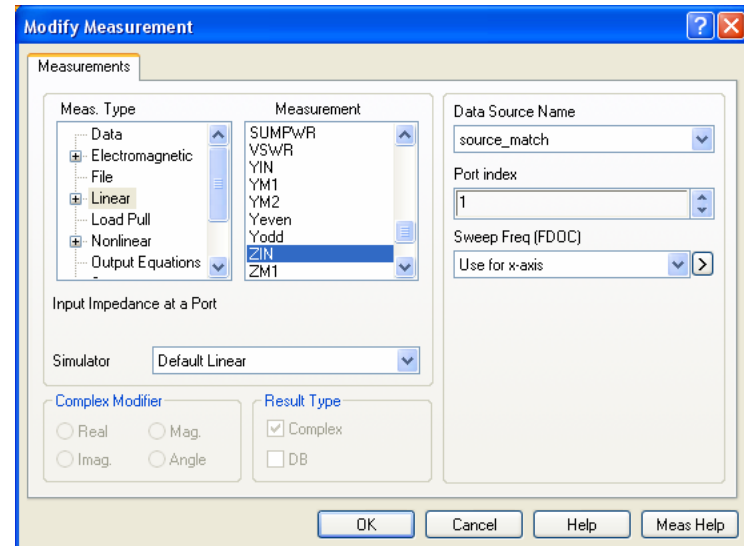
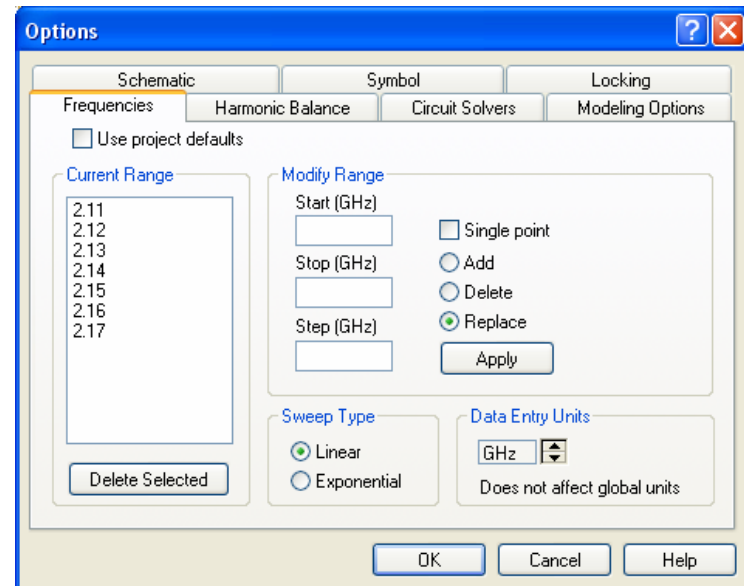
Source Match Layout

- “Select All” then “Snap All Together”
- Change the grid (ex. 10 mils)



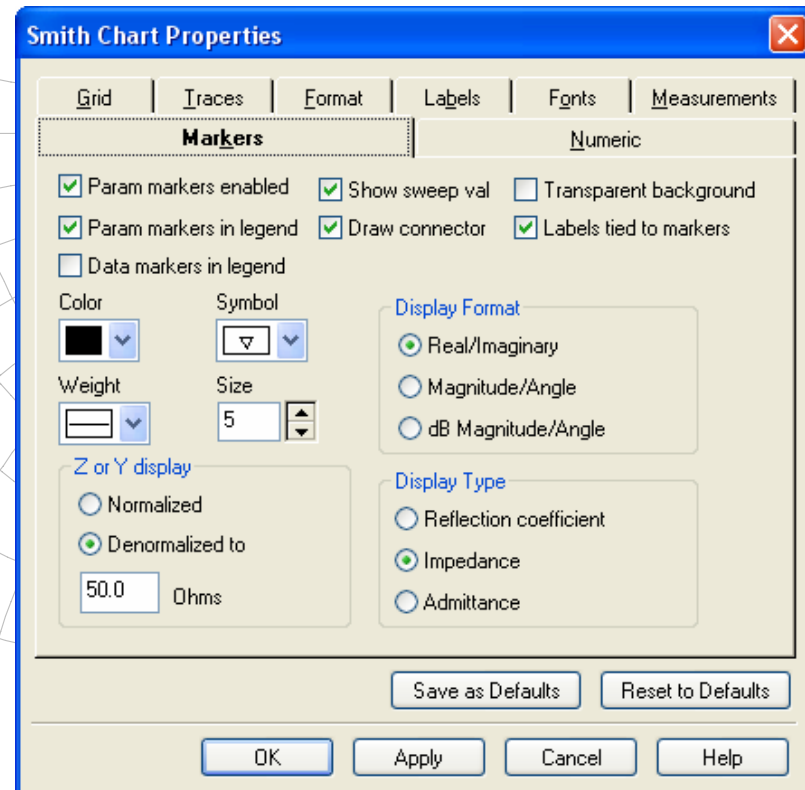
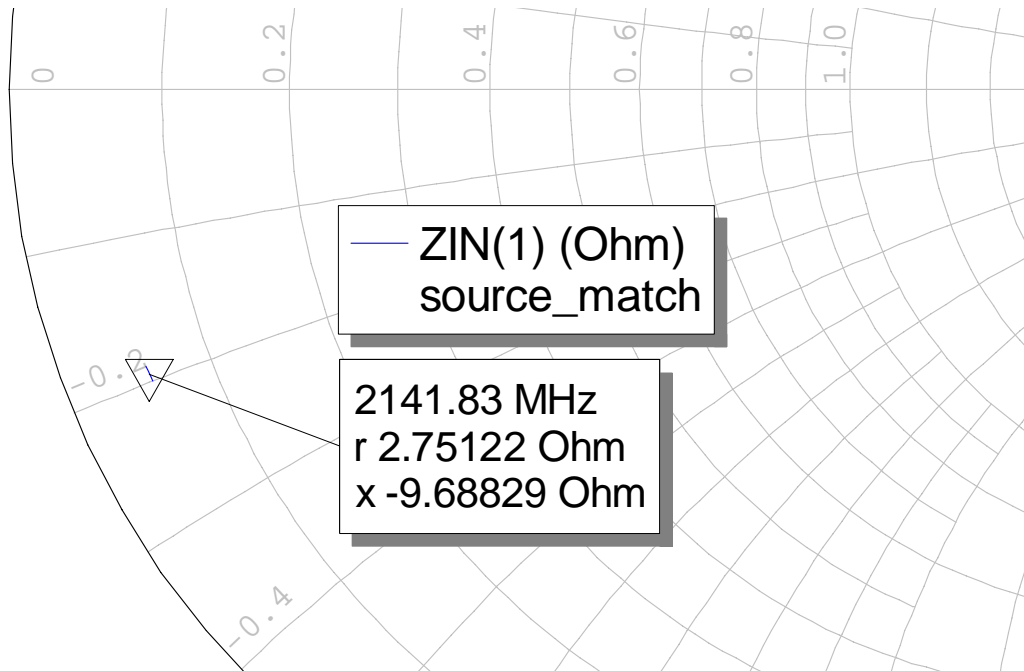
Source Match Measurements

- Modify the frequency points of “source_match” by right clicking the options in the tree view.
- Create a new Smith Chart graph named “Matching Network Impedances”. Add a Zin measurement for Port 1.



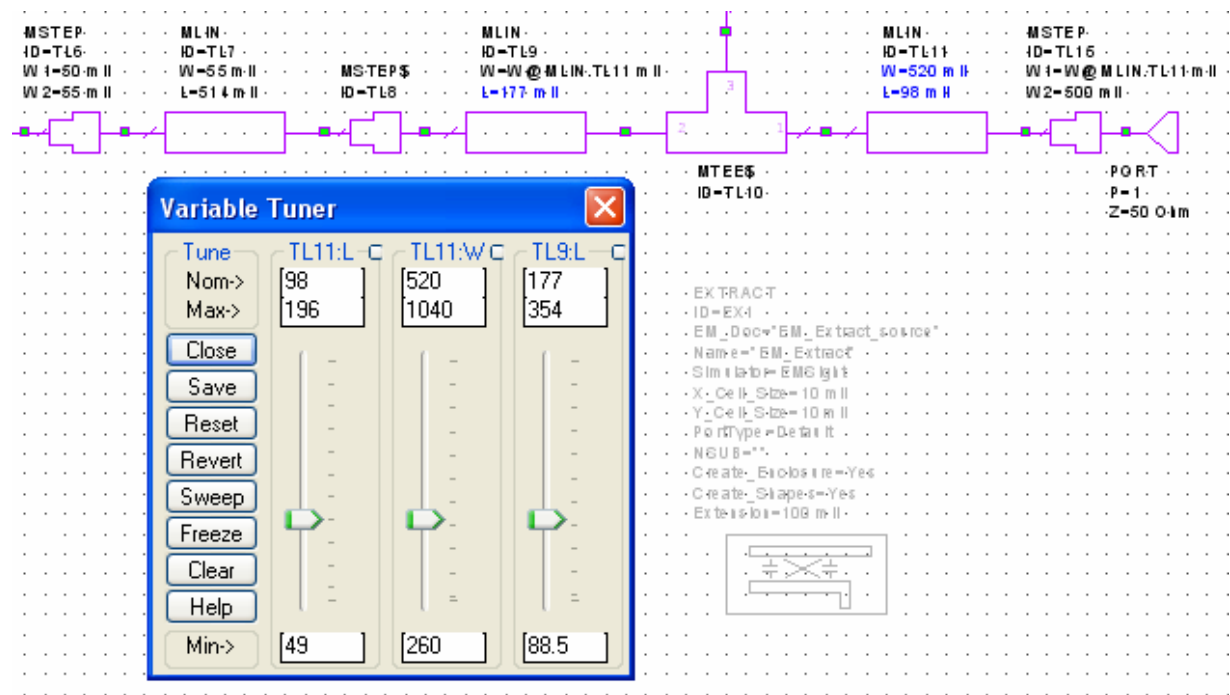
Source Match Results

- Simulate then place a marker at 2140 MHz.
- Modify the marker to display impedance

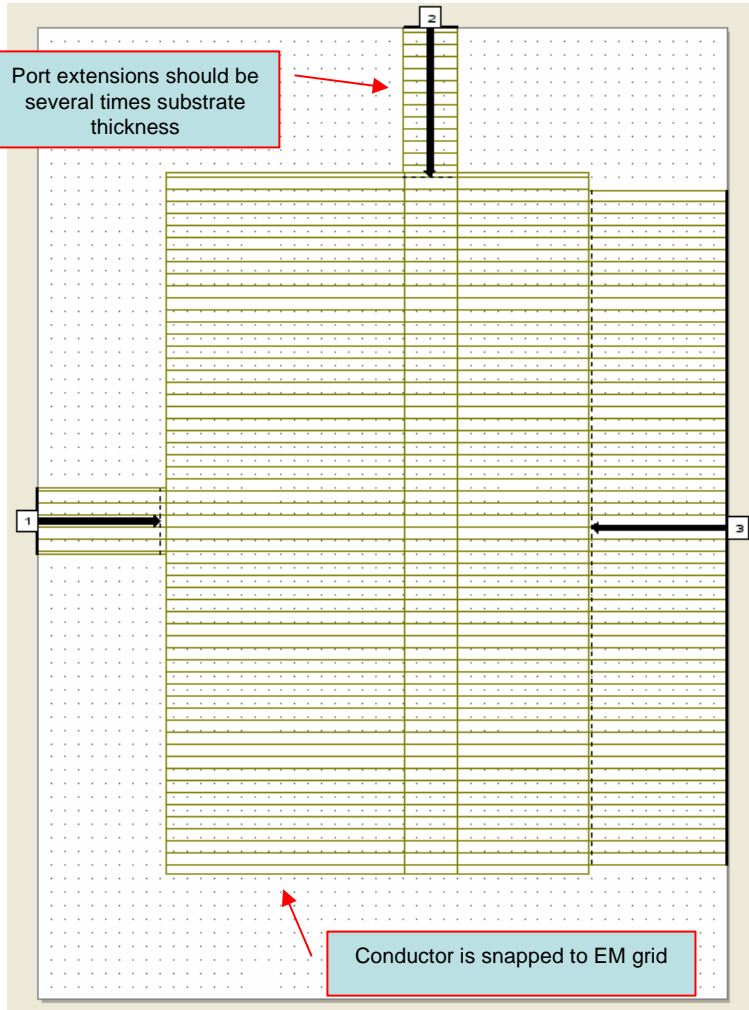


Source Match Tuning

- Is the source impedance close the $4.70 - j*11.03$ ohms recommended in the data sheet ?
- Tune the circuit to improve the result.
- Enable the EM Extraction element and observe the result.

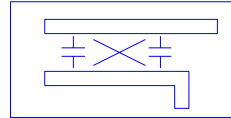


EM Extraction



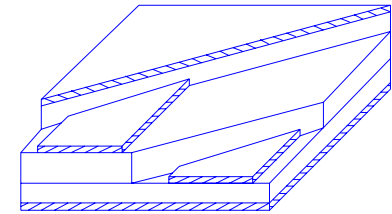
```

EXTRACT
ID=EX1
EM_Doc="EM_Extract_source"
Name="EM_Extract"
Simulator=EMSignit
X_Cell_Size=10 mil
Y_Cell_Size=10 mil
PortType=Default
NSUB=""
Create_Enclosure=Yes
Create_Shapes=Yes
Extension=100 mil
    
```



```

NSUB
HSub=20 mil
Er=3.48
Tand=0.004
ProcUtil="AWR_NLAYER"
Name=SUB
    
```

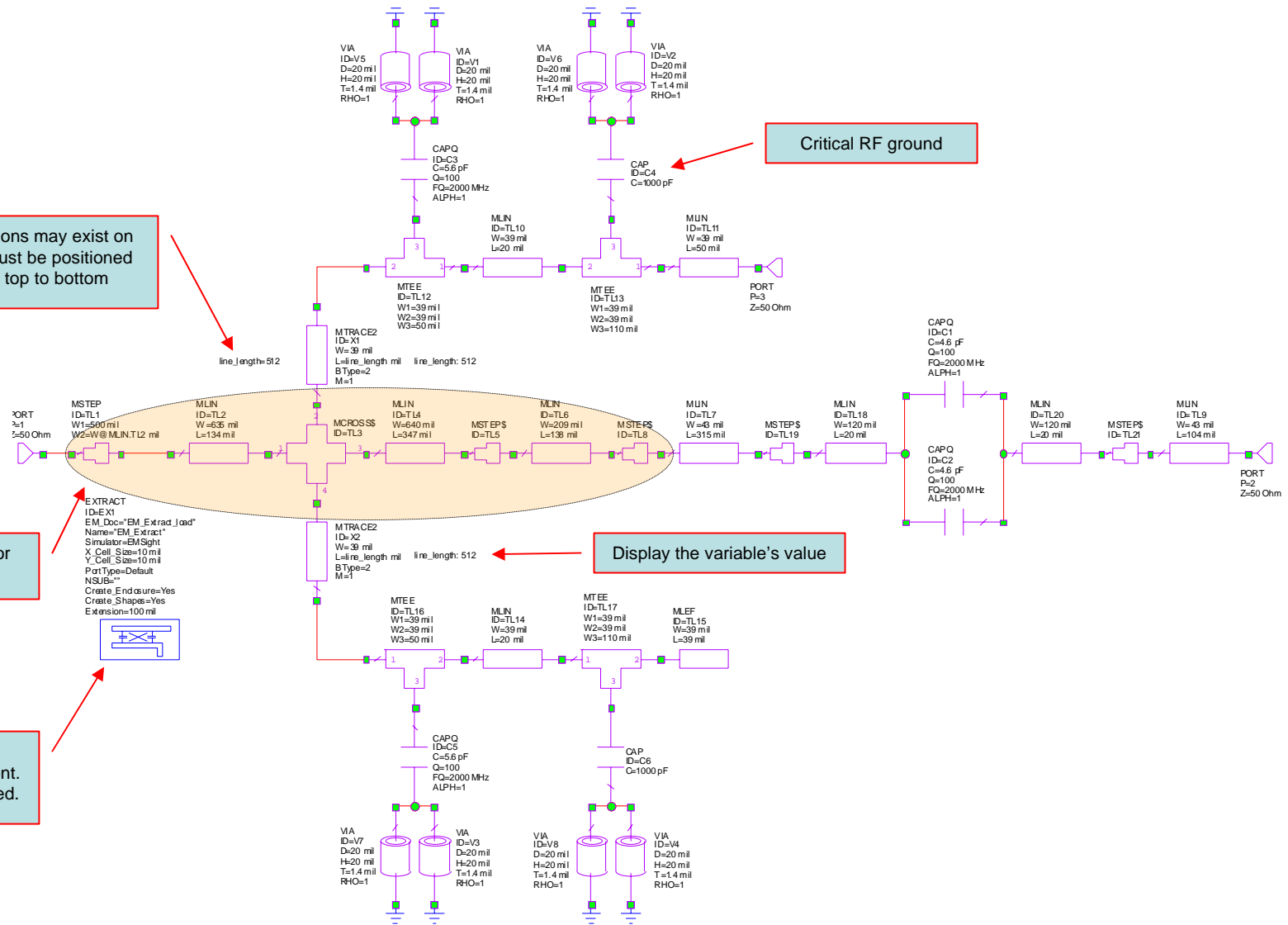


Substrate Information

Enclosure			Dielectric Layers				Boundaries	
Dielectric Layer Parameters								
Layer	Hatch	Via Hatch	Thickness mil	er	Loss Tangent	Bulk Cond. (S/M)	View Scale	
1			80	1	0	0	1	
2			20	3.48	0.004	0	1	

☐ ☐ ☐ 80 1 0 0 1

Load Match Schematic



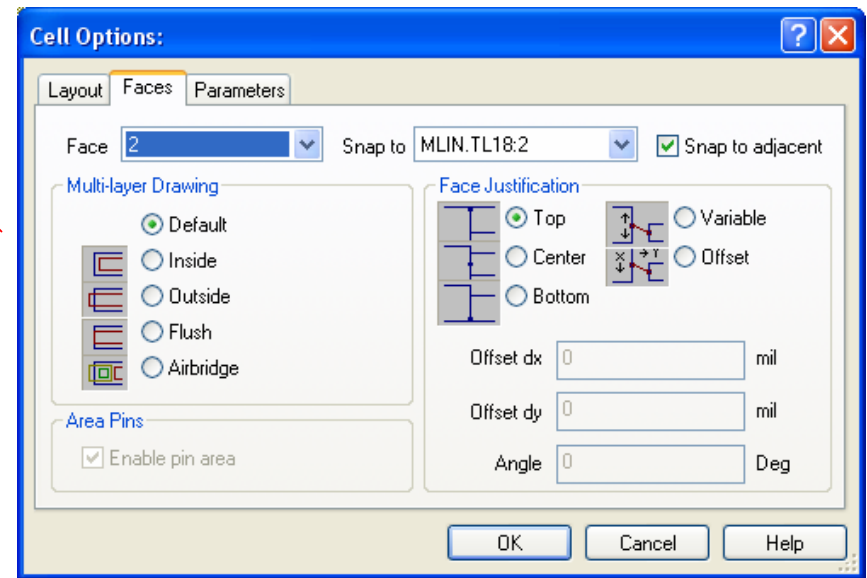
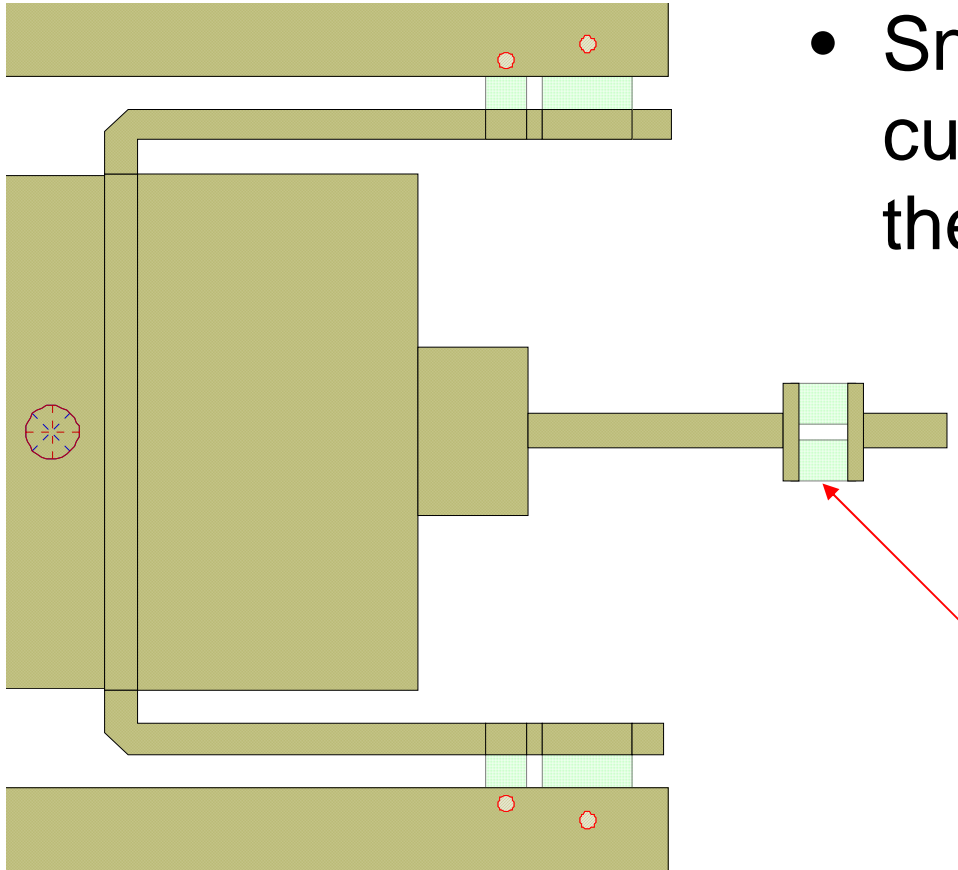
Variables and equations may exist on the schematic but must be positioned in sequence from top to bottom

Enable EM extraction for large discontinuities

Default EM extraction parameters are insufficient. Disable block until needed.

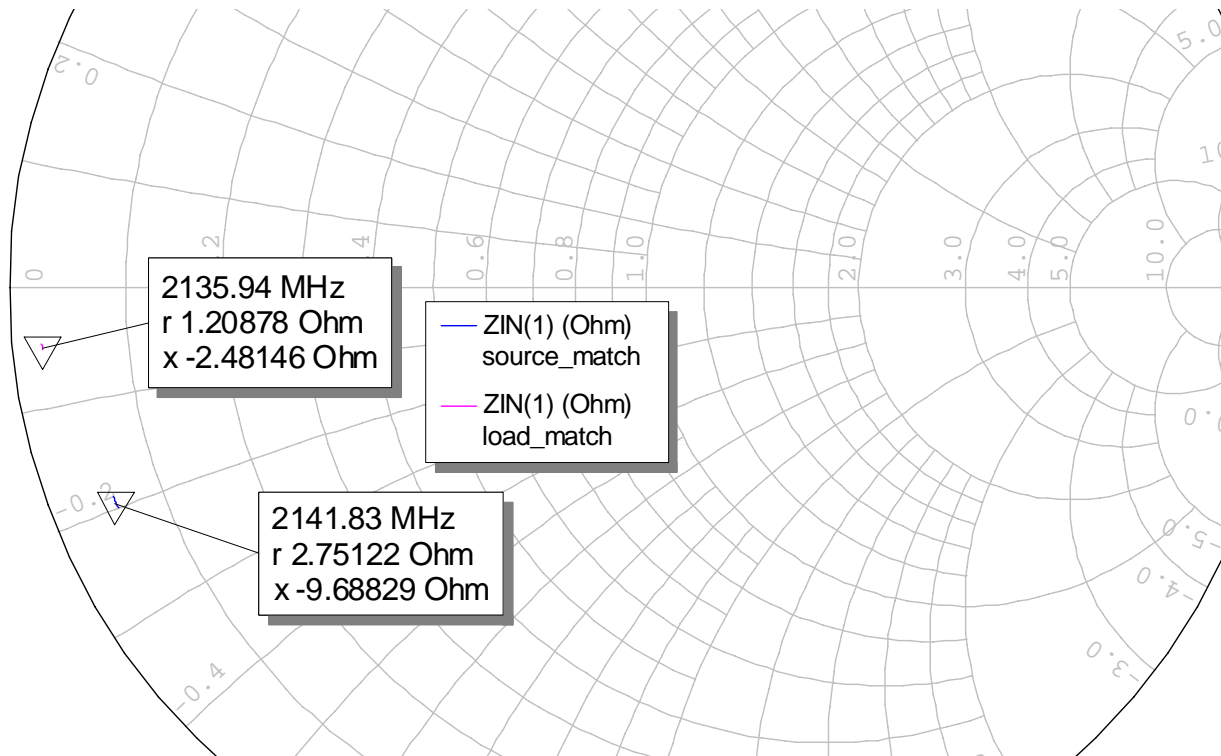
Load Match Layout

- Snapping may be customized by changing the shape properties.

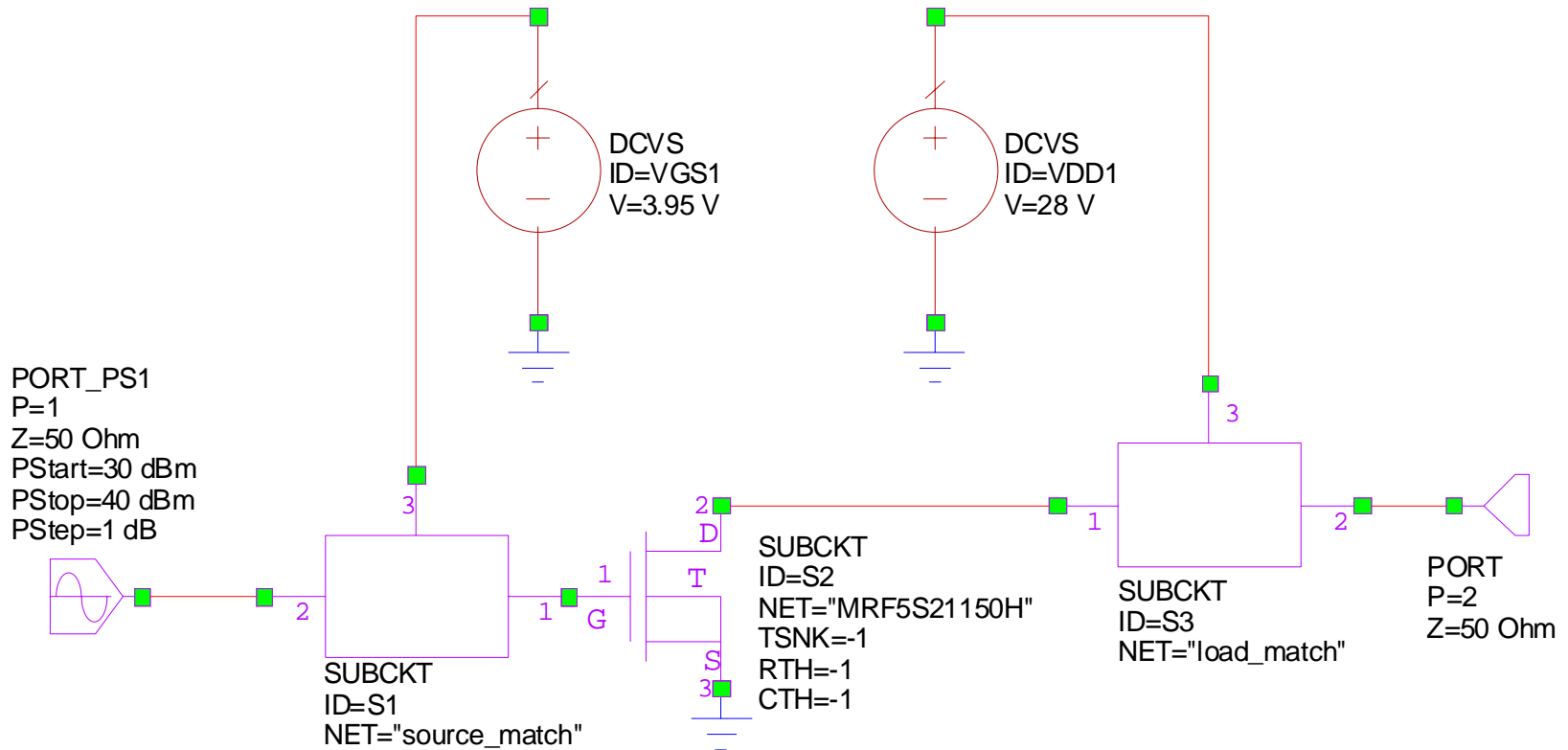


Load Match Results

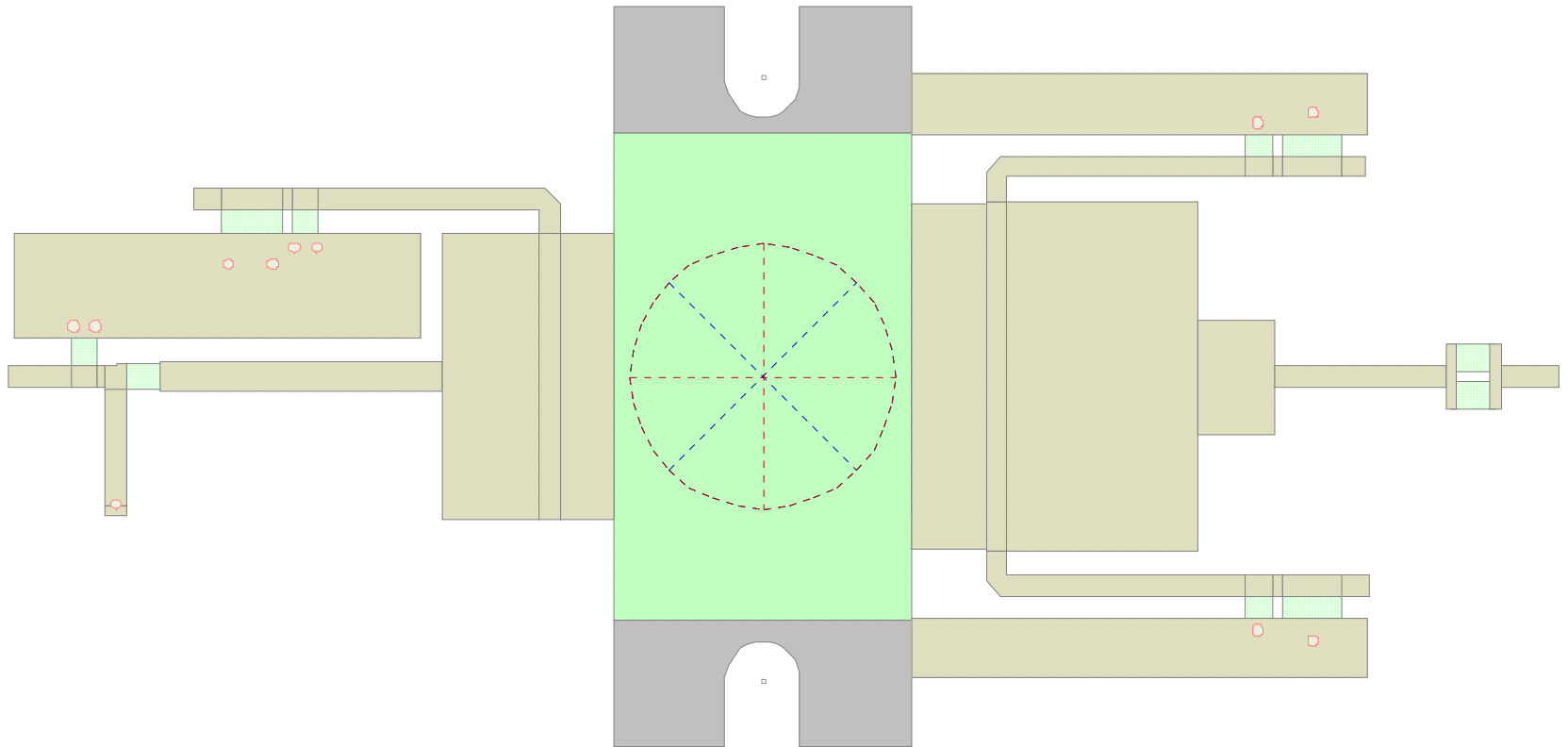
- Is the load impedance close the $1.16 - j2.46$ ohms recommended in the data sheet ?
- Tune the circuit to improve the result.
- Enable EM extraction and observe the results.



Amplifier Schematic



Amplifier Layout



Amplifier Pout and Eff

- Comparison at $P_{in}=5\text{ W}$ (37.0 dBm)
 - Model: $P_{out}=93\text{ W}$ (49.7 dBm), $G=12.7\text{ dB}$, $Eff=48\%$
 - Measured: $P_{out}=102\text{ W}$, $G=13.1\text{ dB}$, $Eff=42\%$

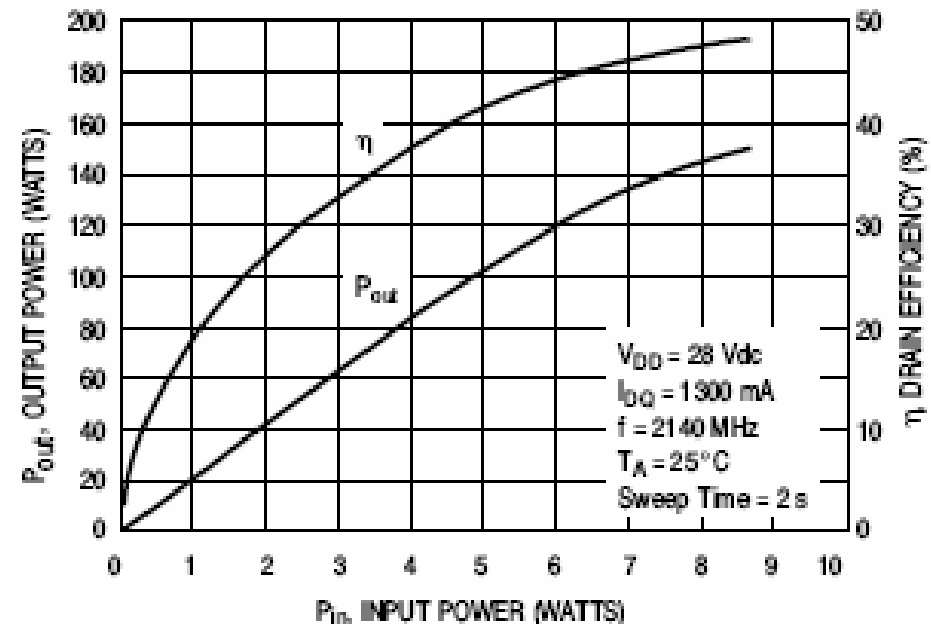
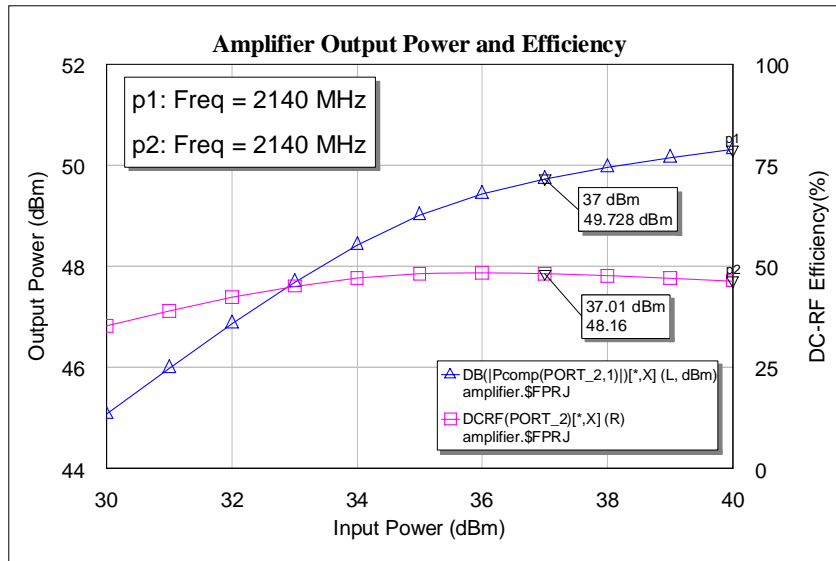
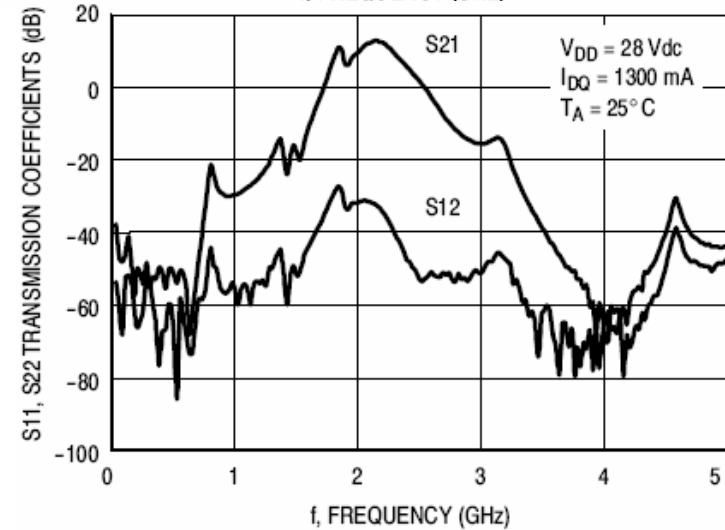
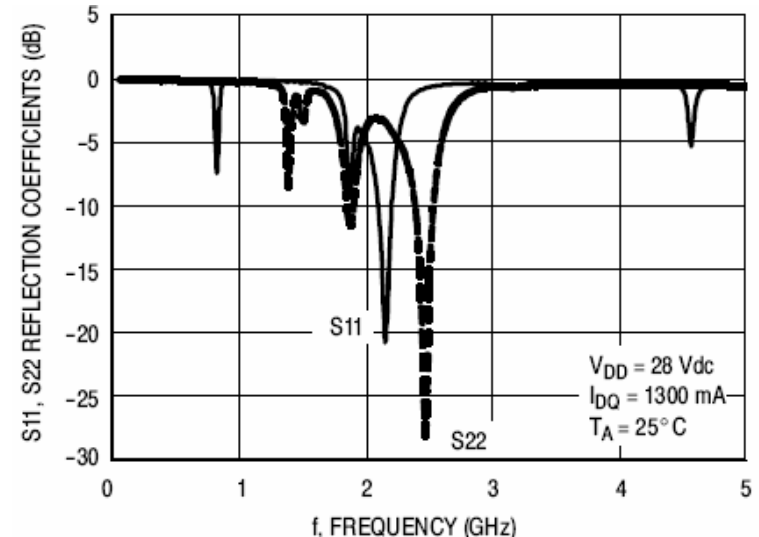
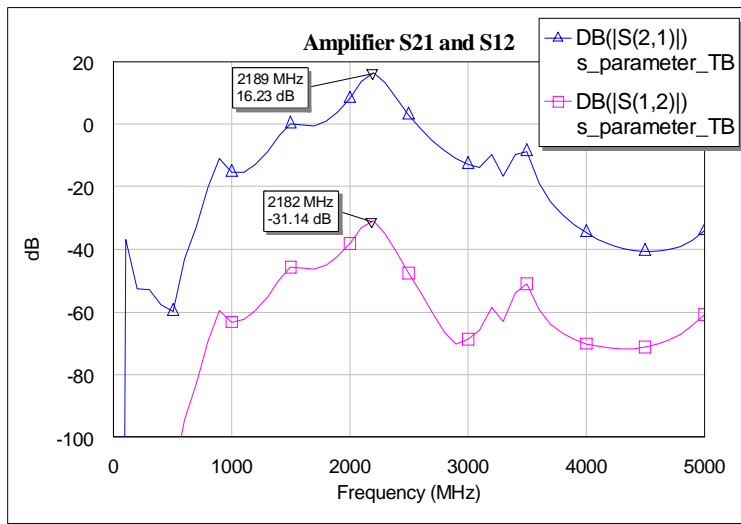
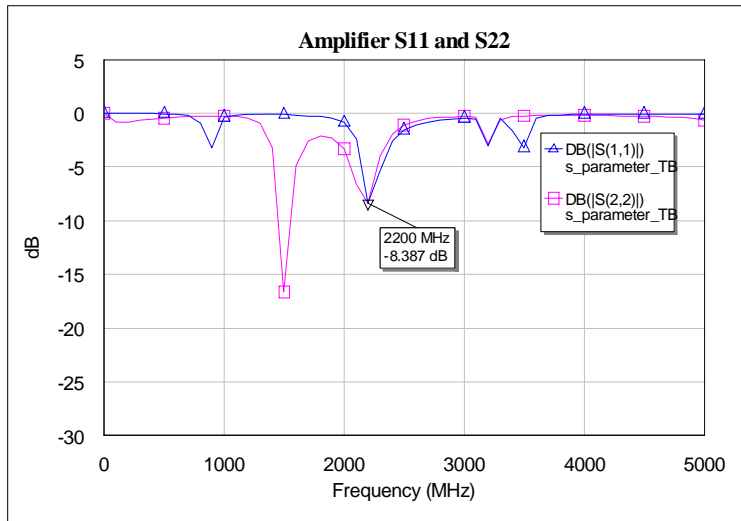
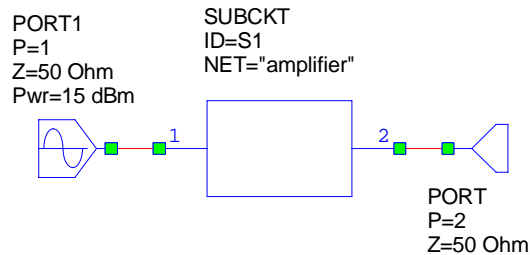


Figure 9. Output Power and Drain Efficiency versus Input Power

Amplifier S-Parameters



Amplifier Power Gain and Input Return Loss



Modify the document options
(instead of project options)

Increased to obtain
convergence

Options

Schematic Symbol Locking Hspice Options
Frequencies Harmonic Balance Circuit Solvers Modeling Options

☐ Use project defaults Configuration Wizard... Advanced...

Tone Harmonics (individual sources may override)

	Number of harmonics	Oversample factor
Tone 1	5	3
Tone 2	2	1
Tone 3	2	1

Use source elements to setup more than 3 tones

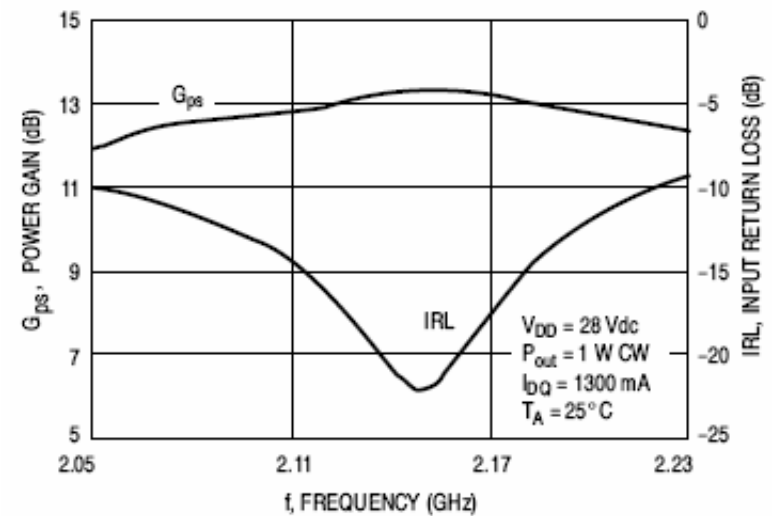
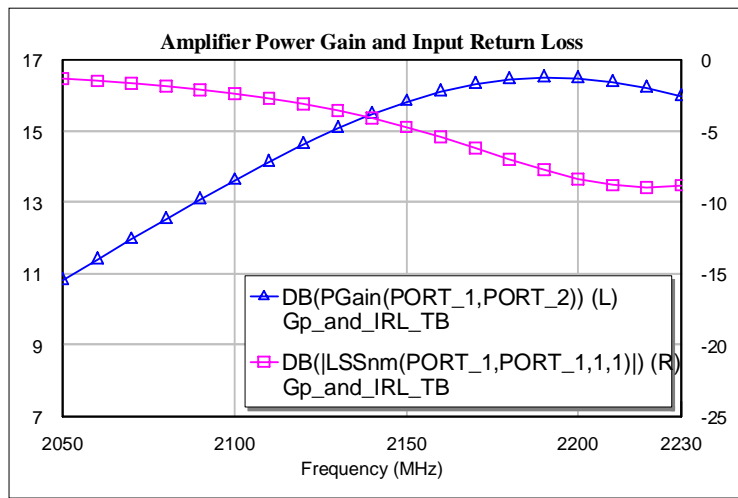
Convergence
Abs. error: 1e-9
Rel. error: 1e-5

Linearization
No linearization
Normal tolerance

Iteration settings
Max number of iterations: 25

Harmonic Limiting
☒ Limit harmonic order Max order: 9
☒ Apply Max order to intermods only

OK Cancel Help



Output Equations

- Use output equations to assign measurements to variables.
- Output equations are always in base units
- Data may then be manipulated using a regular equation.

$P_{fund} = \text{two_tone_power_sweep}:|Pcomp(PORT_2,0_1)|[* ,X]$

Output Equation (green text)

$P_{PEP} = P_{fund} * 4$ The Peak Envelope Power for a balanced 2-tone signal is approximately 4 times the single tone
http://en.wikipedia.org/wiki/Peak_envelope_power

Regular Equation

$P_{3rd_order} = \text{two_tone_power_sweep}:|Pcomp(PORT_2,-1_2)|[* ,X]$

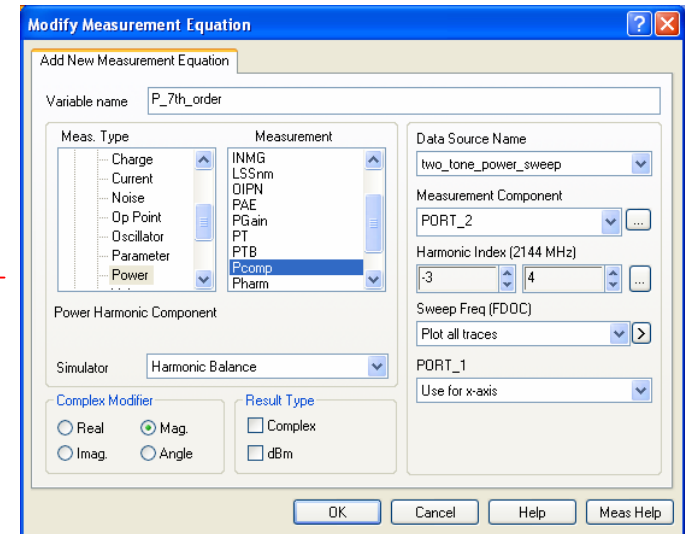
$P_{5th_order} = \text{two_tone_power_sweep}:|Pcomp(PORT_2,-2_3)|[* ,X]$

$P_{7th_order} = \text{two_tone_power_sweep}:|Pcomp(PORT_2,-3_4)|[* ,X]$

$IMD3 = \text{plot_vs}(10 * \log_{10}(P_{3rd_order}/P_{PEP}), P_{PEP})$

$IMD5 = \text{plot_vs}(10 * \log_{10}(P_{5th_order}/P_{PEP}), P_{PEP})$

$IMD7 = \text{plot_vs}(10 * \log_{10}(P_{7th_order}/P_{PEP}), P_{PEP})$

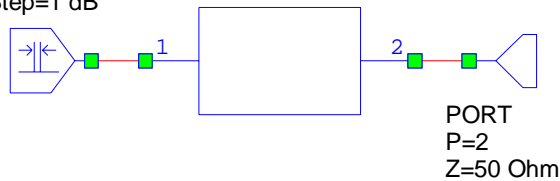


Amplifier IMD

PORT_PS2
P=1
Z=50 Ohm
Fdelt=1 MHz
PStart=10 dBm
PStop=36 dBm
PStep=1 dB

Default HB settings for document
must be changed

SUBCKT
ID=S1
NET="amplifier"



Options

Schematic Symbol Locking Hspice Options
Frequencies Harmonic Balance Circuit Solvers Modeling Options

☐ Use project defaults Configuration Wizard... Advanced...

Tone Harmonics (individual sources may override)

	Number of harmonics	Oversample factor
Tone 1	5	3
Tone 2	5	3
Tone 3	2	1

Use source elements to setup more than 3 tones

Convergence

Abs. error: 1e-9 Rel. error: 1e-5

Linearization

No linearization (dropdown)
Normal tolerance (dropdown)

Harmonic Limiting

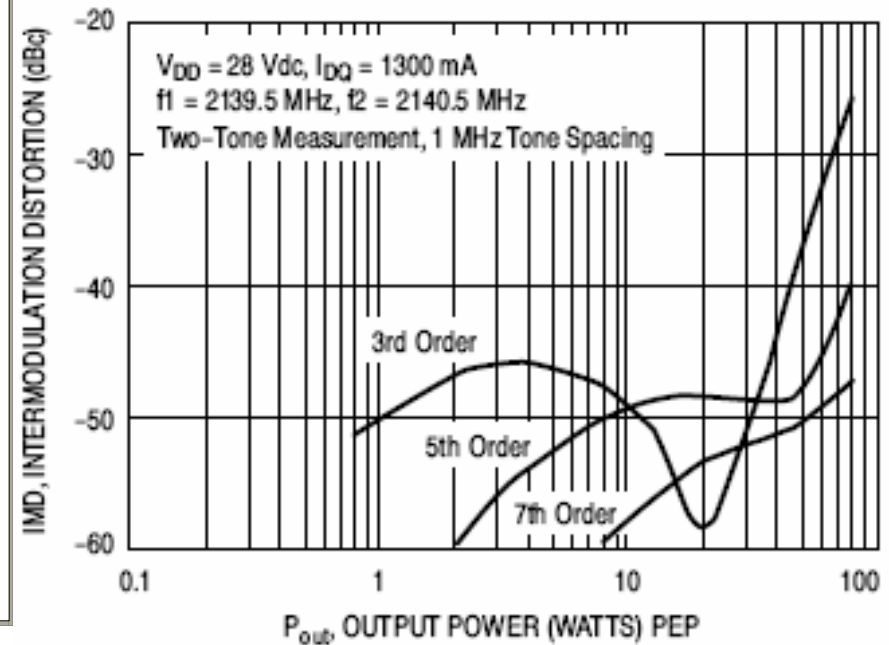
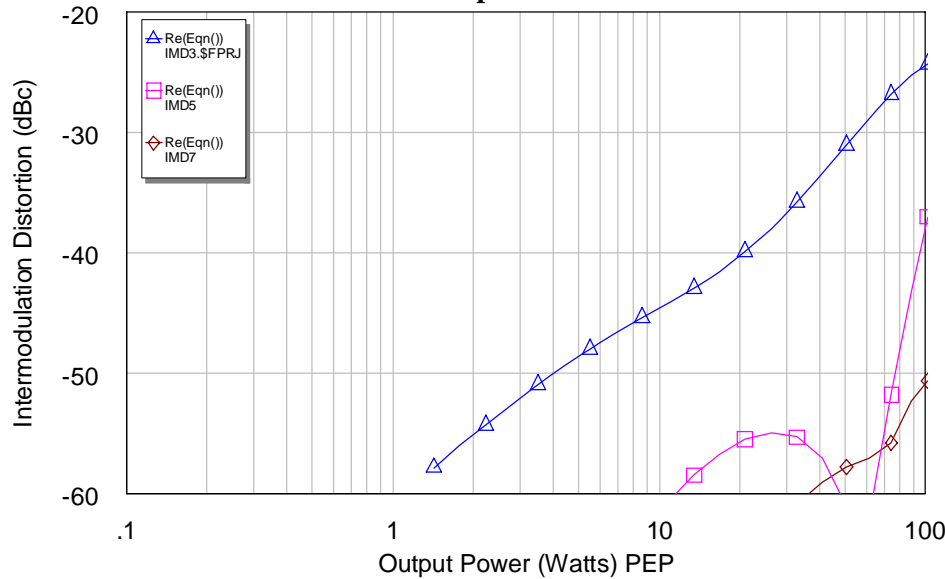
☒ Limit harmonic order Max order: 9
☒ Apply Max order to intermods only

Iteration settings

Max number of iterations: 25

OK Cancel Help

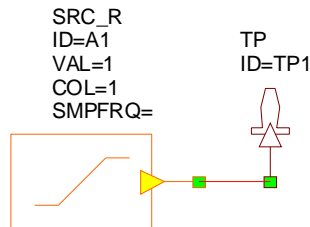
Amplifier IMD



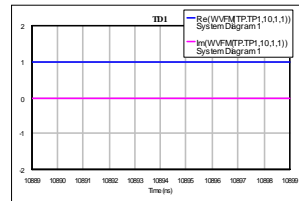
Visual System Simulator

- VSS is a time domain simulator that uses both real samples and complex envelope representation.
- Complex envelope representation allows baseband waveforms to be up-converted and simulated at RF.
- VSS is very powerful however the manual should be read to fully understand it.
- See example vss_data.emp for examples of sources.

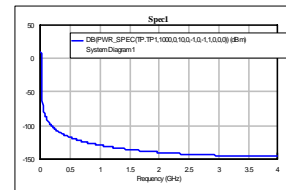
A real signal $x(t)=1$



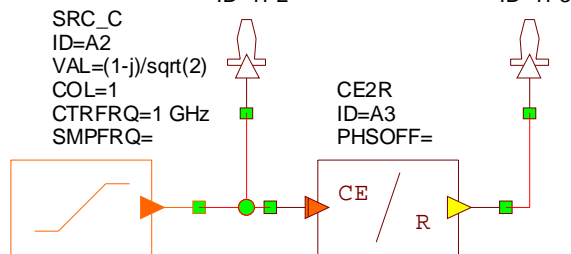
$\text{Re}[x(t)] = xc(t) = 1$
 $\text{Im}[x(t)] = xs(t) = 0$



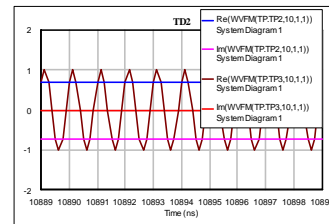
Power at DC



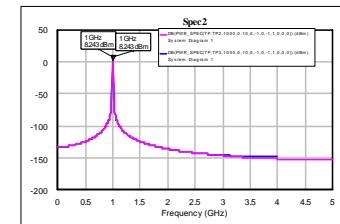
$c(t) = (1-j)/\sqrt{2}$, $f_c = 1$ GHz



$\text{TP2 } \text{Re}[c(t)] = xc(t) = 1/\sqrt{2}$
 $\text{TP2 } \text{Im}[c(t)] = xs(t) = -1/\sqrt{2}$
 $\text{TP3 } \text{Re}[x(t)] = \cos(A + \pi/4)$
 $\text{TP3 } \text{Im}[x(t)] = 0$

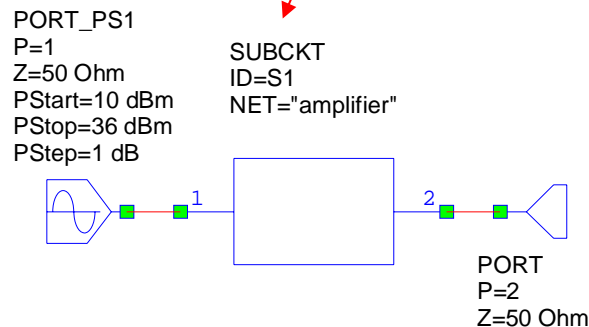


TP2 $c(t)$ is shifted by F_c
 TP3 $x(t)$ is a real spectrum

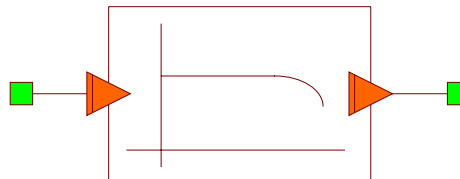


Behavioral Modeling

- VSS uses AM-AM and AM-PM results from HB analysis to obtain a behavioral model.
- Strong nonlinearities may not model well.
- HB simulation document “Bmodel_TB” must be configured for power ranges and frequencies desired in system simulation.



NL_S
ID=S1
NET="Bmodel_TB"
NOISE=Freq analysis only
PSWP=""
FSWP=""



Behavioral system model

Options

Schematic Symbol Locking Hspice Options

Frequencies Harmonic Balance Circuit Solvers Modeling Options

☐ Use project defaults

Current Range

2.04
2.05
2.06
2.07
2.08
2.09
2.1
2.11
2.12
2.13
2.14
2.15

Modify Range

Start (GHz) 2.04

Stop (GHz) 2.24

Step (GHz) 0.01

☐ Single point

☐ Add

☐ Delete

☒ Replace

Apply

Sweep Type

☒ Linear

☐ Exponential

Data Entry Units

GHz

Does not affect global units

Delete Selected

OK Cancel Help

Options

Schematic Symbol Locking Hspice Options

Frequencies Harmonic Balance Circuit Solvers Modeling Options

☐ Use project defaults

Configuration Wizard... Advanced...

Tone Harmonics (individual sources may override)

	Number of harmonics	Oversample factor
Tone 1	5	3
Tone 2	2	1
Tone 3	2	1

Use source elements to setup more than 3 tones

Convergence

Abs. error 1e-9

Rel. error 1e-5

Linearization

No linearization

Normal tolerance

Harmonic Limiting

☒ Limit harmonic order

Max order 9

☒ Apply Max order to intermods only

Iteration settings

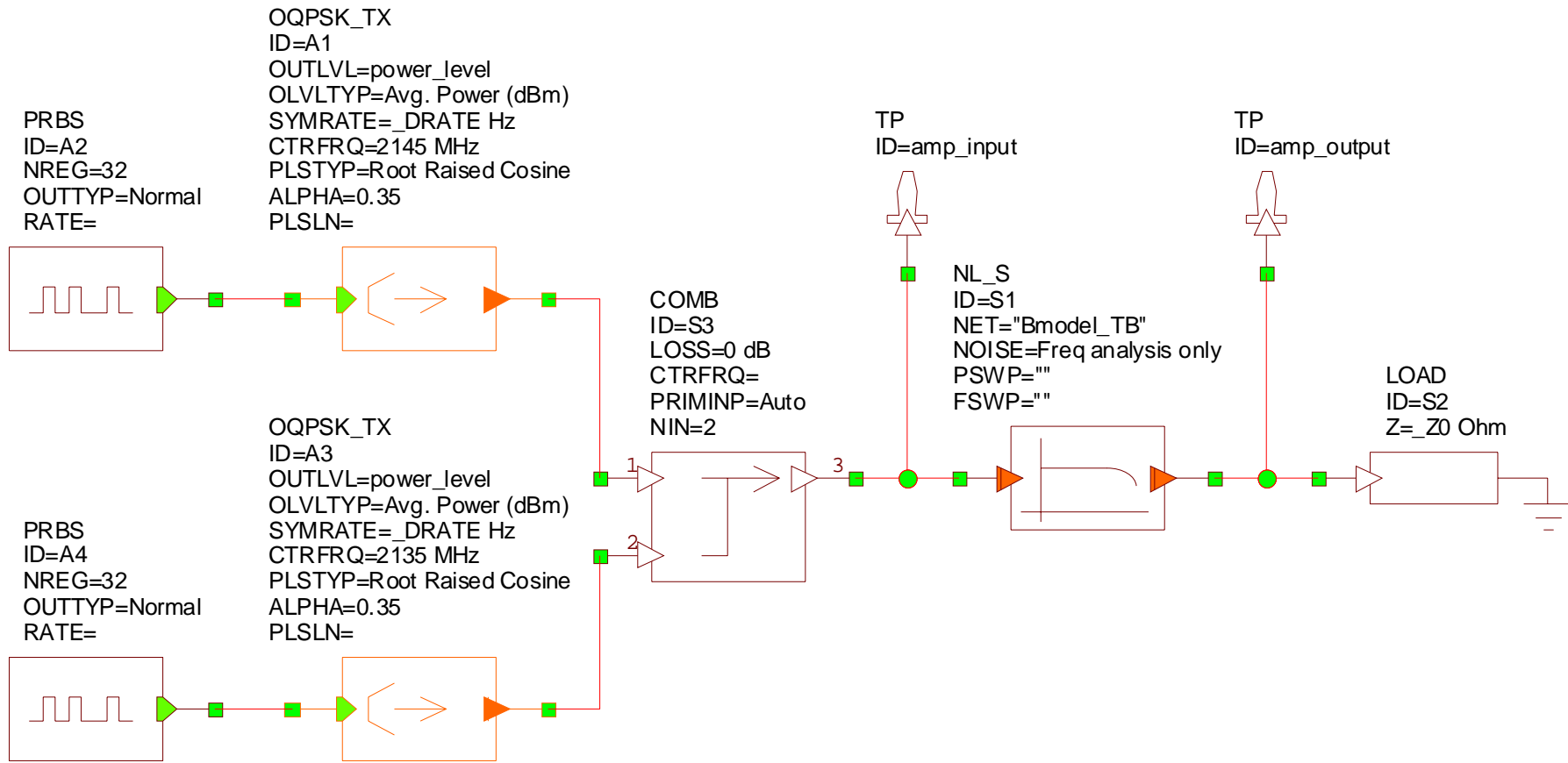
Max number of iterations 25

OK Cancel Help

OQPSK Transmitter

power_level=0 +12 dBm power_level yields +30 dBm out of amplifier

Add $10 \cdot \log(3.84 \text{ MHz}) = 65.8 \text{ dBHz}$ to power spectral density graph for total power



Default System Options

The screenshot shows the 'System Simulator Options' dialog box with the 'Frequency Analysis' tab selected. The dialog has a blue title bar and several tabs: 'Simulator', 'RF Settings', 'Frequency Analysis', 'Advanced', and 'Result Display'. The 'Frequency Analysis' tab contains the following settings:

- Simulation Control:** A group box containing two radio buttons: 'Run continuously' (selected) and 'Stop after: 260417 ns'. A red arrow points from a text box to the 'Stop after' value.
- *Sampling Frequencies/Data Rates:** A group box containing two dropdown menus: 'Data rate [_DRATE]:' set to '3.84 MHz' and 'Oversampling [_SMPSYM]:' set to '16'. A red arrow points from a text box to the 'Data rate' dropdown.
- Footer:** Three buttons: 'OK', 'Cancel', and 'Help'.

Annotations (text boxes with red arrows pointing to specific settings):

- Used for budget analysis of RX & TX chains:** Points to the 'Frequency Analysis' tab.
- Stop after 1000 bits (not enough for smooth spectrum):** Points to the 'Stop after: 260417 ns' value.
- Max data rate in system:** Points to the 'Data rate [_DRATE]:' dropdown.
- Increase oversampling to allow for wider bandwidth analysis:** Points to the 'Oversampling [_SMPSYM]:' dropdown.

* Indicates default model values that can be overridden

Simulation Control

☒ Run continuously ☐ Stop after: 260417 ns

*Sampling Frequencies/Data Rates

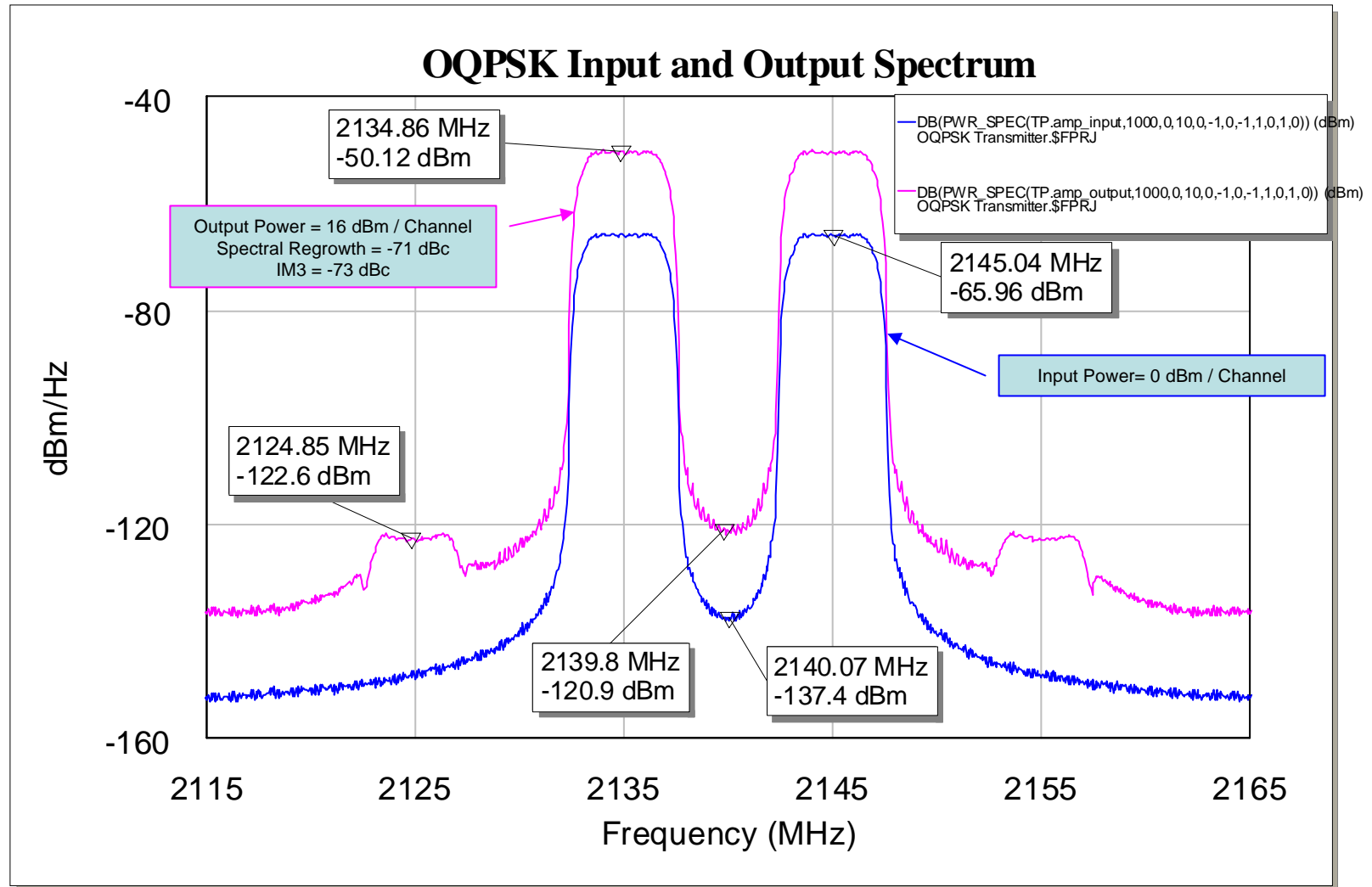
Data rate [_DRATE]: 3.84 MHz

Oversampling [_SMPSYM]: 16

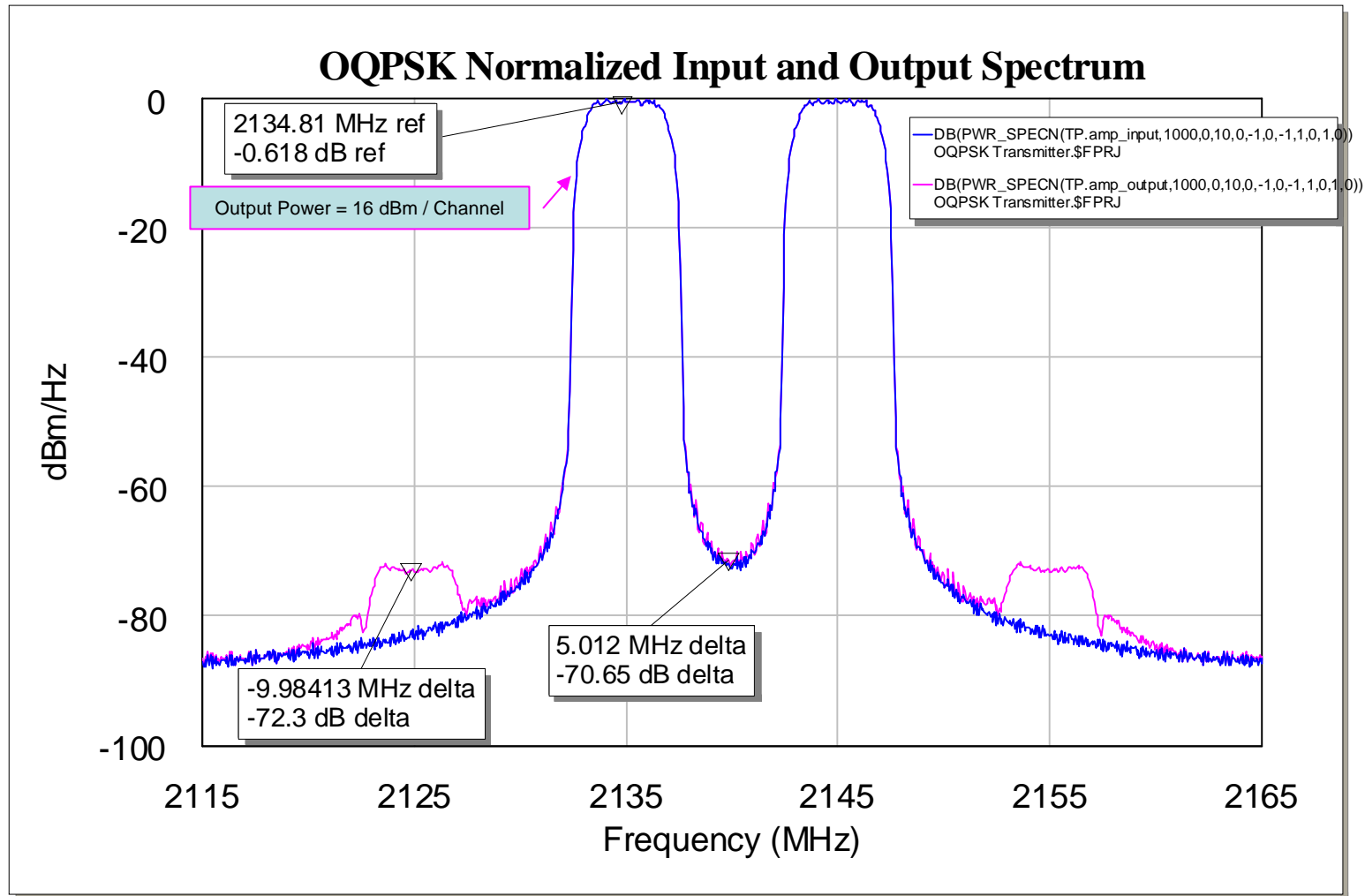
(Symbol period = 260.417 ns, Sampling frequency = 61.44 MHz)

OK Cancel Help

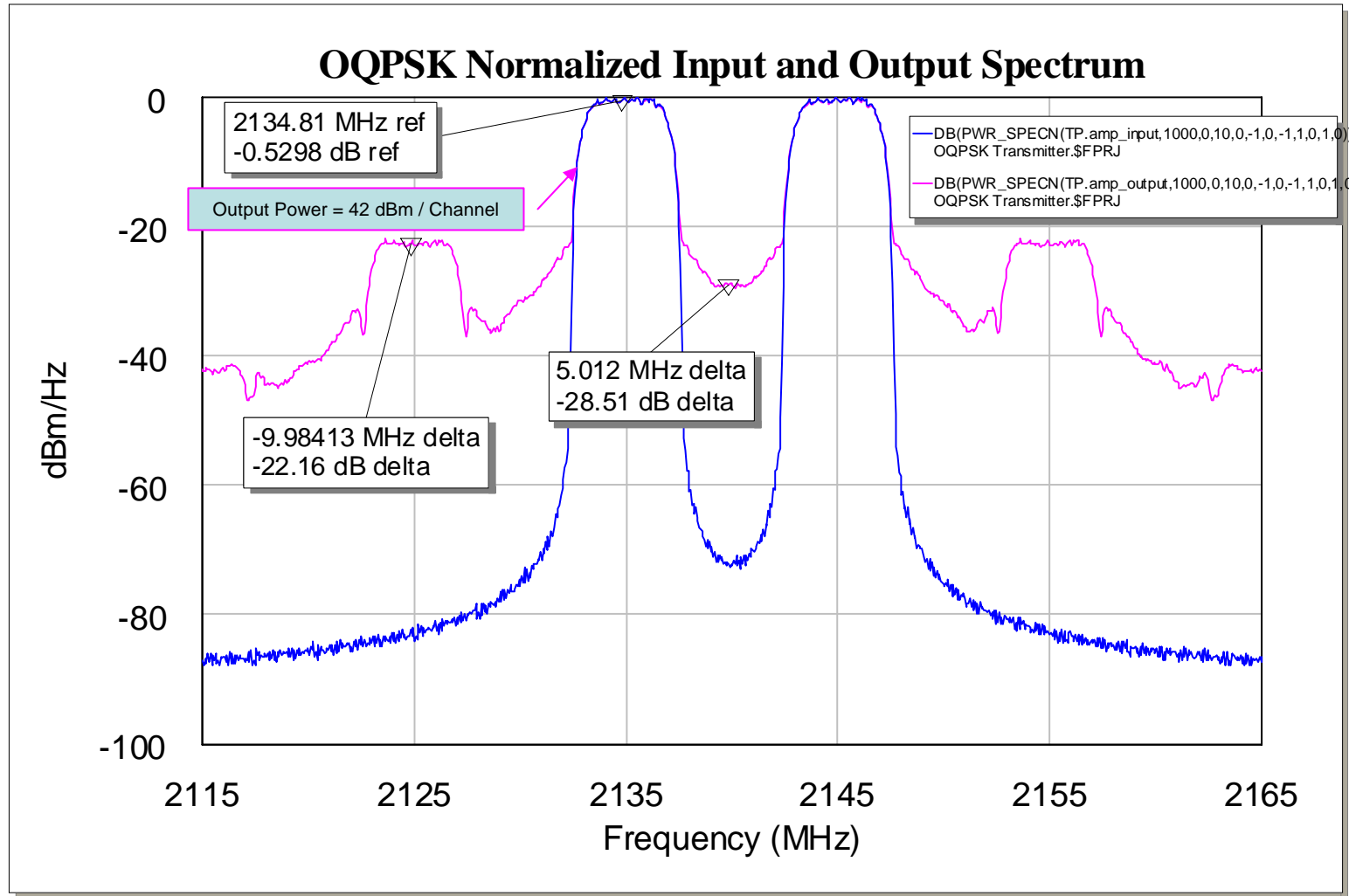
OQPSK Spectrum (PSD)



OQPSK Normalized Spectrum (PSD)



OQPSK Normalized Spectrum (PSD)



References

MRF5S21150H Reference Design

http://www.freescale.com/files/rf_if/hardware_tools/printed_circuit_boards_for_reference_designs/MRF5S21150H_UMTS_RD.pdf

MRF5S21150H Data Sheet

http://www.freescale.com/files/rf_if/doc/data_sheet/MRF5S21150H.pdf

Applied Wave Research's Microwave Office & Visual System Simulator

<http://www.appwave.com/>