Communication System Design using Keysight SystemVue

Dan Nakahara
Keysight Technologies Japan

2018.8.9

KEYSIGHT TECHNOLOGIES

Agenda

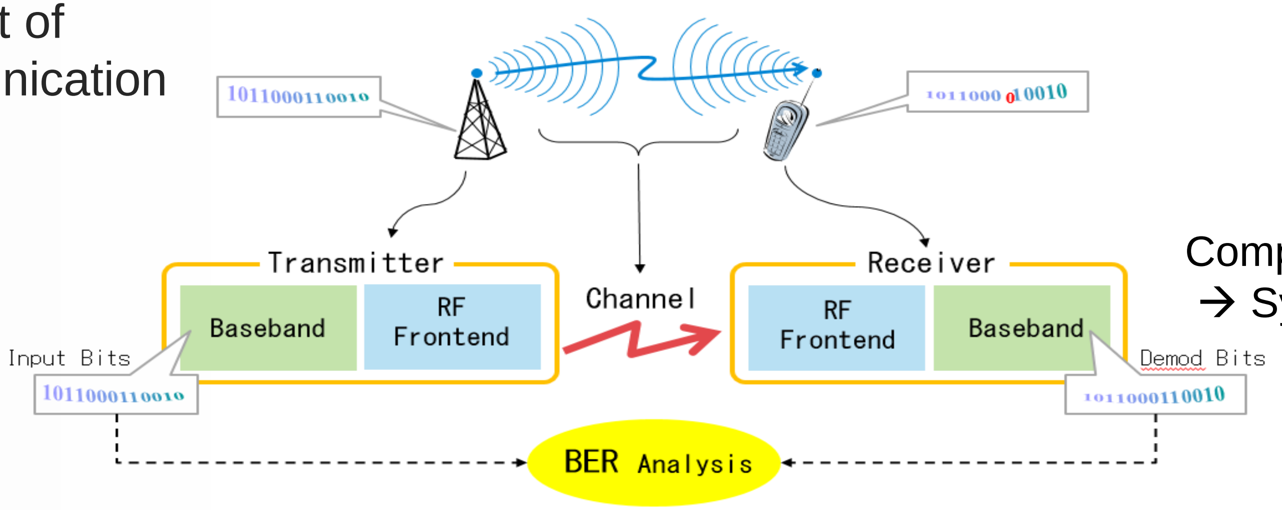
10:00～12:00

- SystemVue overview
- Lab 1: Getting Started with SystemVue
- Lab 2: QPSK Modulator Design
- Lab 3: RF design in LTE system

What is “System Simulation” ?

PHY of Wireless system simulation

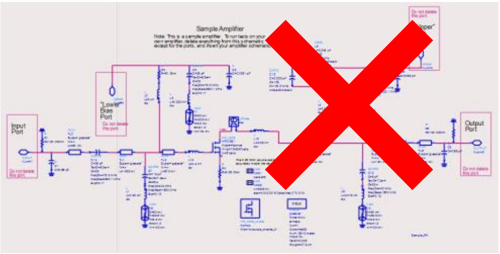
General concept of wireless communication



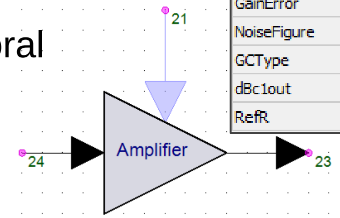
Compare Tx Bits and Rx Bits
→ System performance (Bit Error Rate)

SystemVue uses “Behavioral model”

Transistor level circuit



Behavioral model



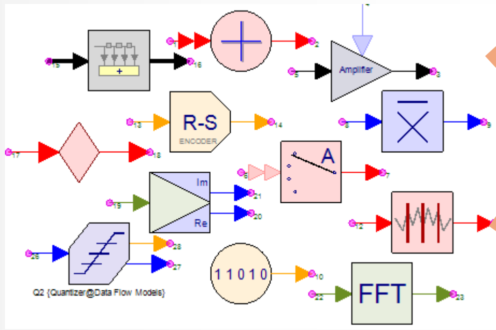
Name	Value	Units
GainUnit	1:dB	
Gain	20	
Quantization	0:NO	
GainError	0:None	
NoiseFigure	3	
GCType	2:dBc1	
dBc1out	15.5	(dBm)
RefR	50	Ohm

KEYSIGHT TECHNOLOGIES

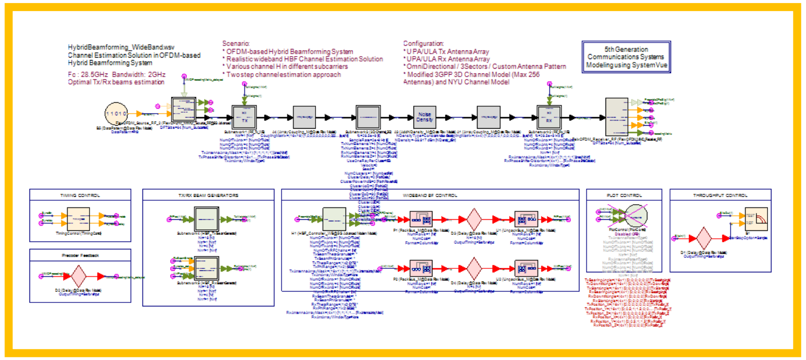
SystemVue : Model Based Simulator

SystemVue

Behavioral Model

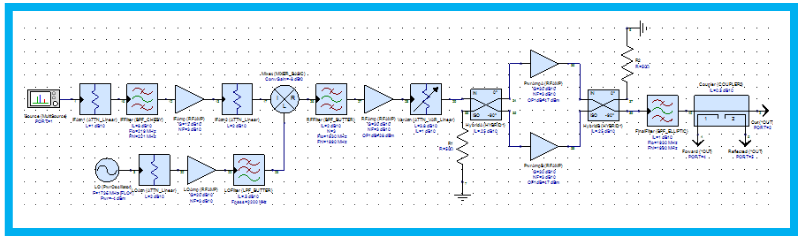


Data Flow



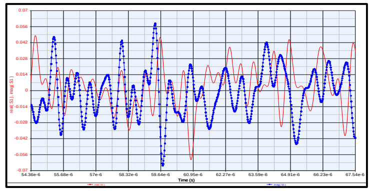
Baseband（Coding, Modem）

RF-front end block

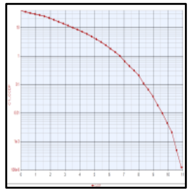


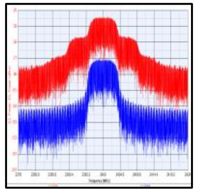
Spectrasys

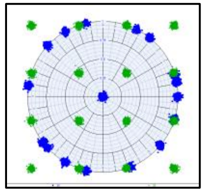
Modulated waveform,
Modulation scheme



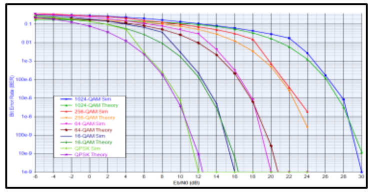
Transmitter Eval.
・Power, CCDF
・Spectrum
・Modulation
Accuracy (EVM)



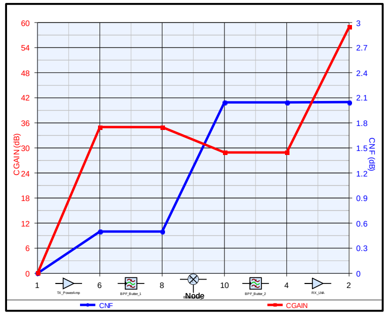




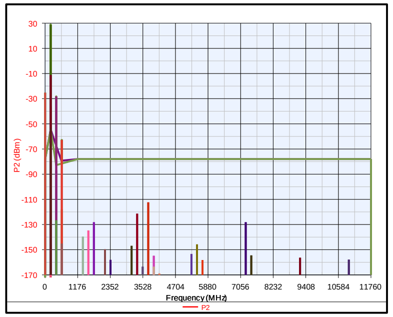
Receiver Eval.
・BER, Throughput



Level Diagram
(e.g. CGain, CNF)



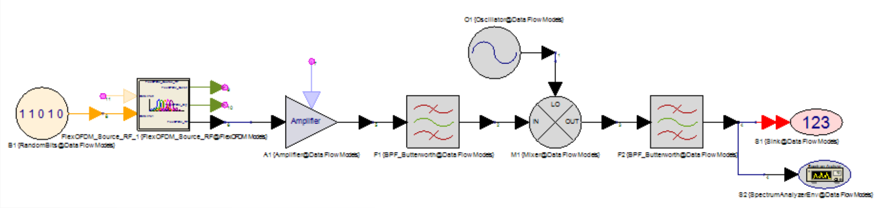
Spurious



Time Domain and Frequency Domain

Two types of simulation

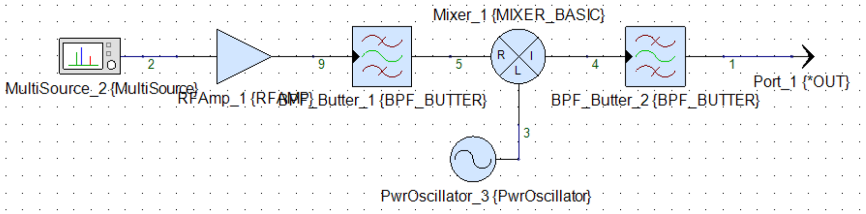
Data Flow



- Time domain simulation
- “Sink” captures simulation result
- Transfer data to only forward direction without reflection

For baseband algorithm including MODEM

Spectrasys



- Frequency domain simulation
- Each nodes has simulation results
- Consider mismatch and reflection

For RF verification

Phased Array Beamforming Kit (W1720)

- Array/matrix model for Data Flow was added
- Dedicated models and simulation controller for Phased Array Simulation

IP Reference Libraries

Use libraries as reference to faster development

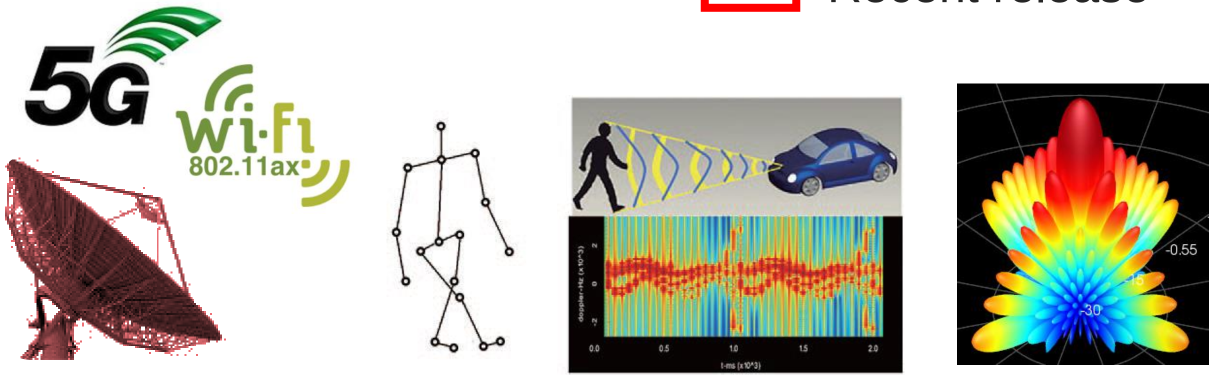
Reference IP

Radar	5G	WLAN
Auto Radar	LTE-A & LTE	WPAN, 11ad
Digital Modem	LTE	DVB-S2/T2
WiMAX	MIMO Chan	*DOCSIS*
GNSS	2G/3G	*DVB-S2X*
Zigbee		

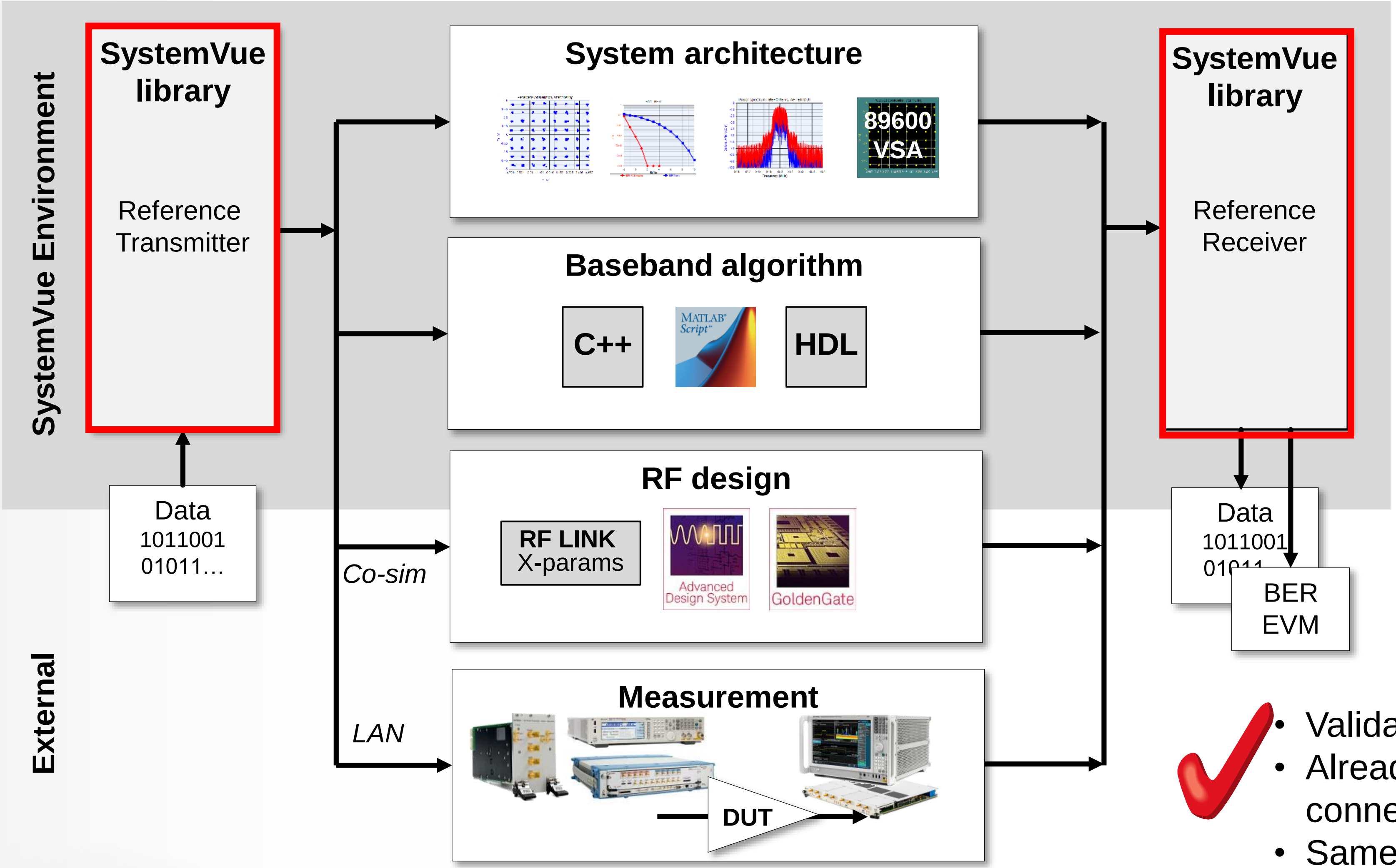
Design Kit

MIMO	RF Design Kit	DPD
Phased Array		

Recent release



IP Reference Libraries

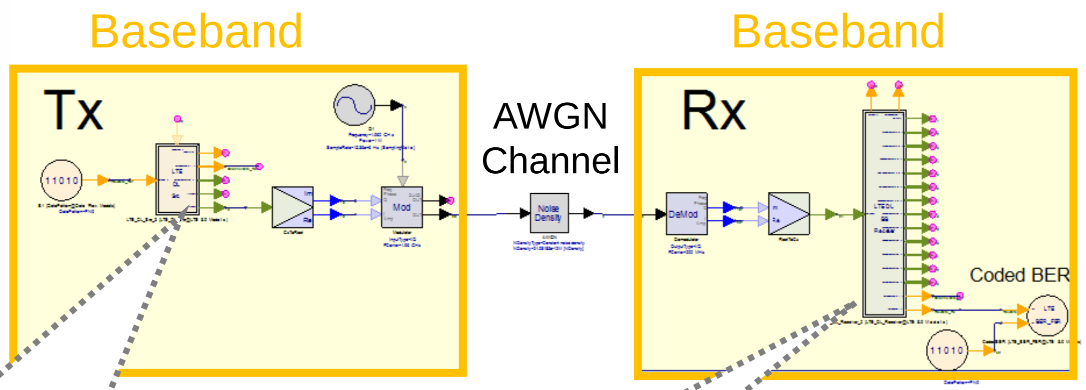


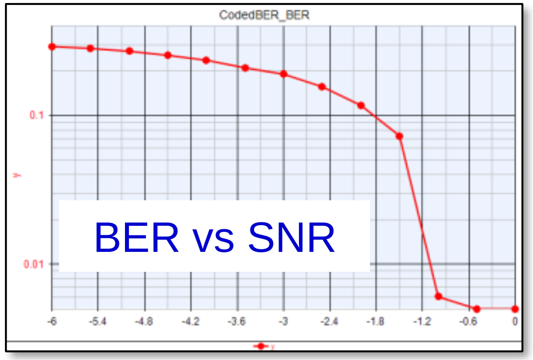
- Validate anywhere in the R&D lifecycle
- Already using MATLAB? Want to connect to the RF/ADS teams? Use SV!
- Same measurement IP throughout
- Unified, cross-platform approach

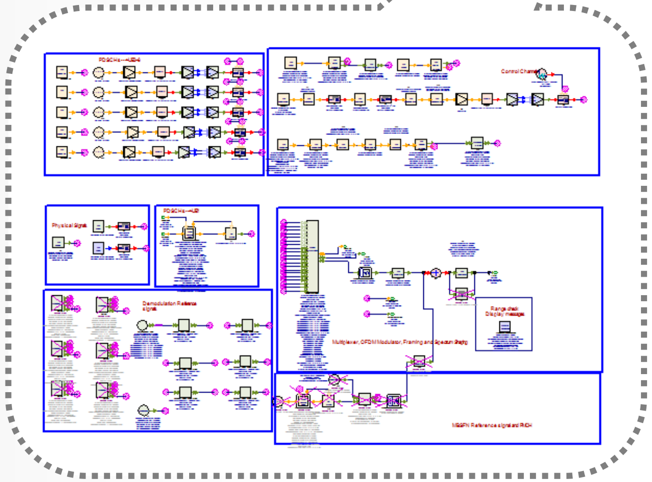
System verification

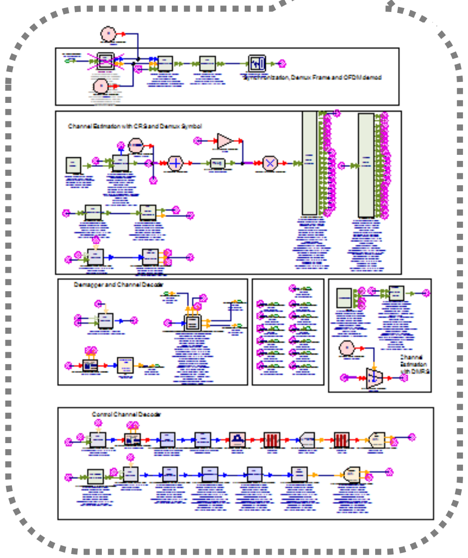
Baseband algorithm validation

Data Flow



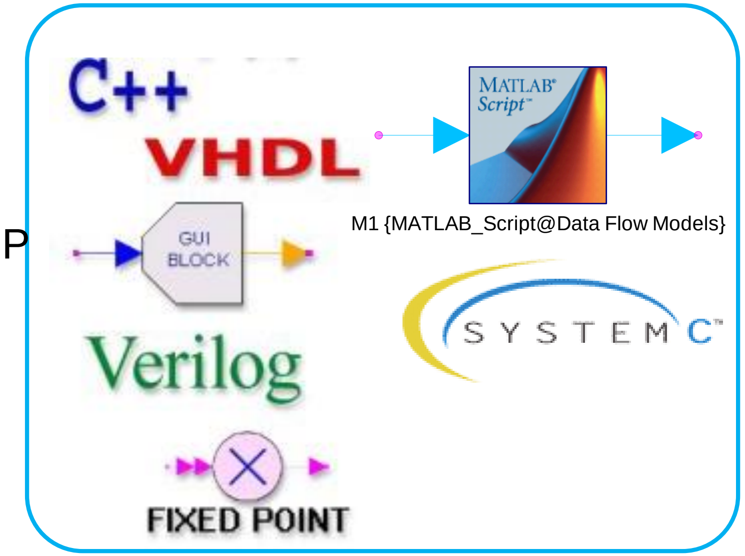






Source, Receiver sub circuit

Replace IP

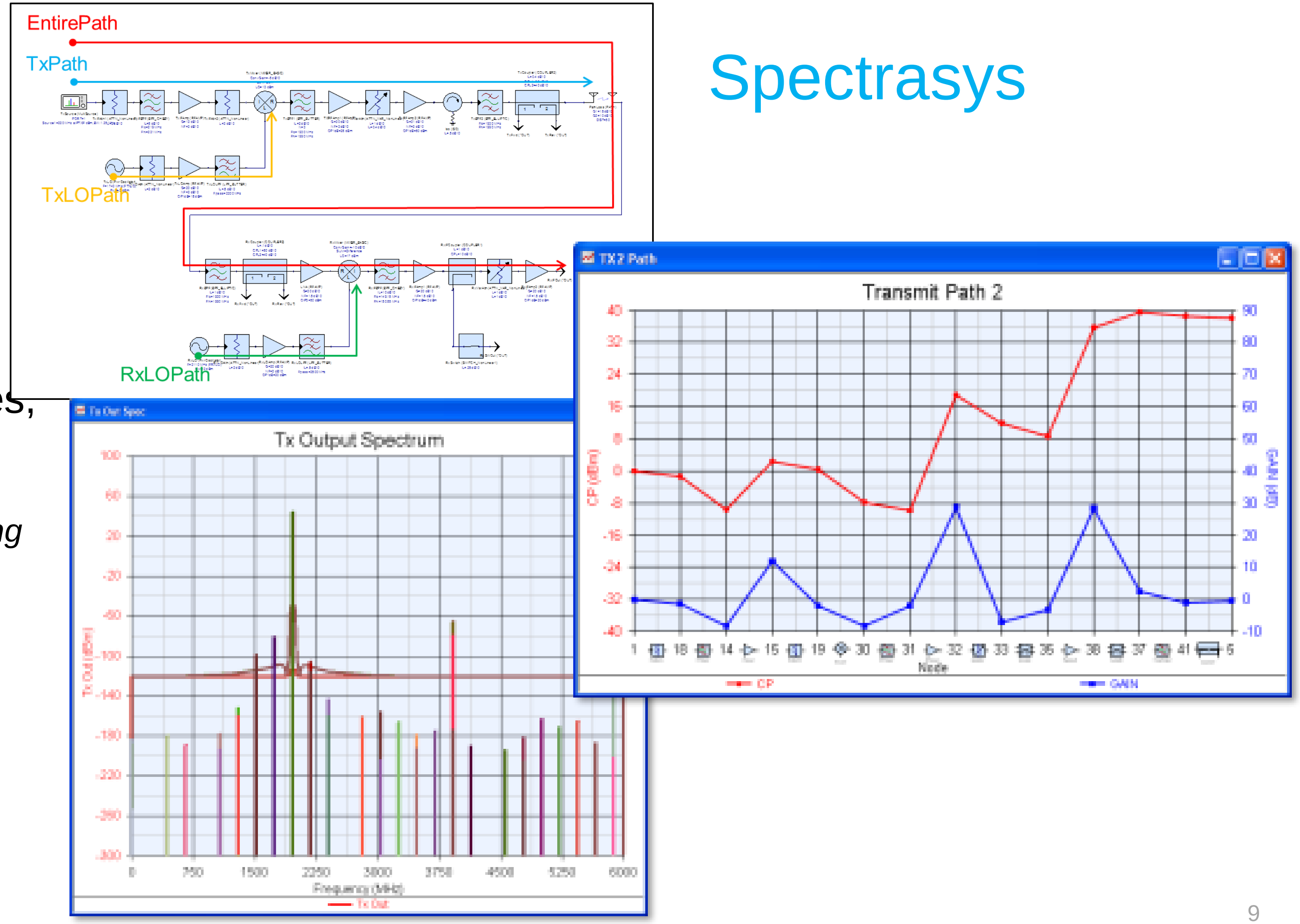


RF design verification

"Spectrasys" continuous spectrum simulation

- Full continuous spectrum at every node (unique RF simulator)
- Track origin and propagation of spurs, noise, mismatch, leakage paths
- Easily plot RF Budget & Spectral info
- Create and refine RF System Architectures, then use them in System-level Dataflow

A base-station user says . . .*"After briefly using SPECTRASYS on a 'must work right the first time' RF design, I identified several issues that were undetected in our spreadsheets and current simulation tools. I'm absolutely positive that it has saved us at least one board spin! "*



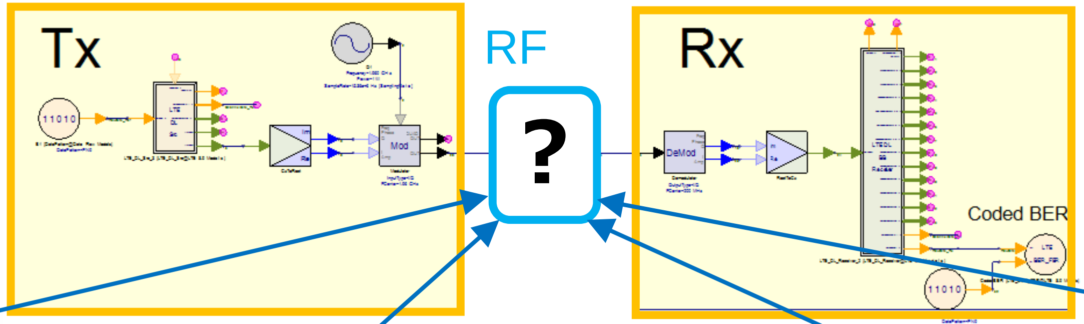
Spectrasys

RF design verification

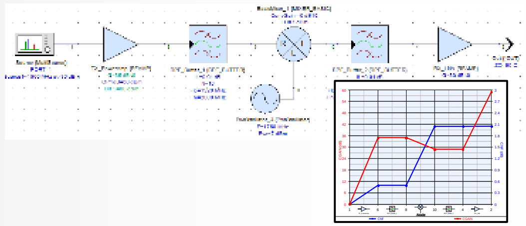
Use various level of RF model

Baseband

Baseband

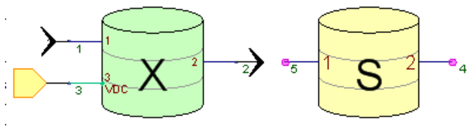


RF Link



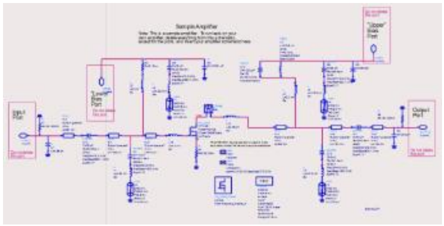
RF design in SystemVue

Xparam, Sparam



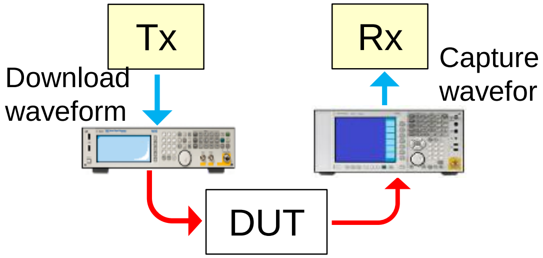
Xparameter and Sparameter by simulation or measurement

VTB, FCE



VTB : Co-simulation with ADS or GG circuit simulator
FCE : extracted RF model by GG

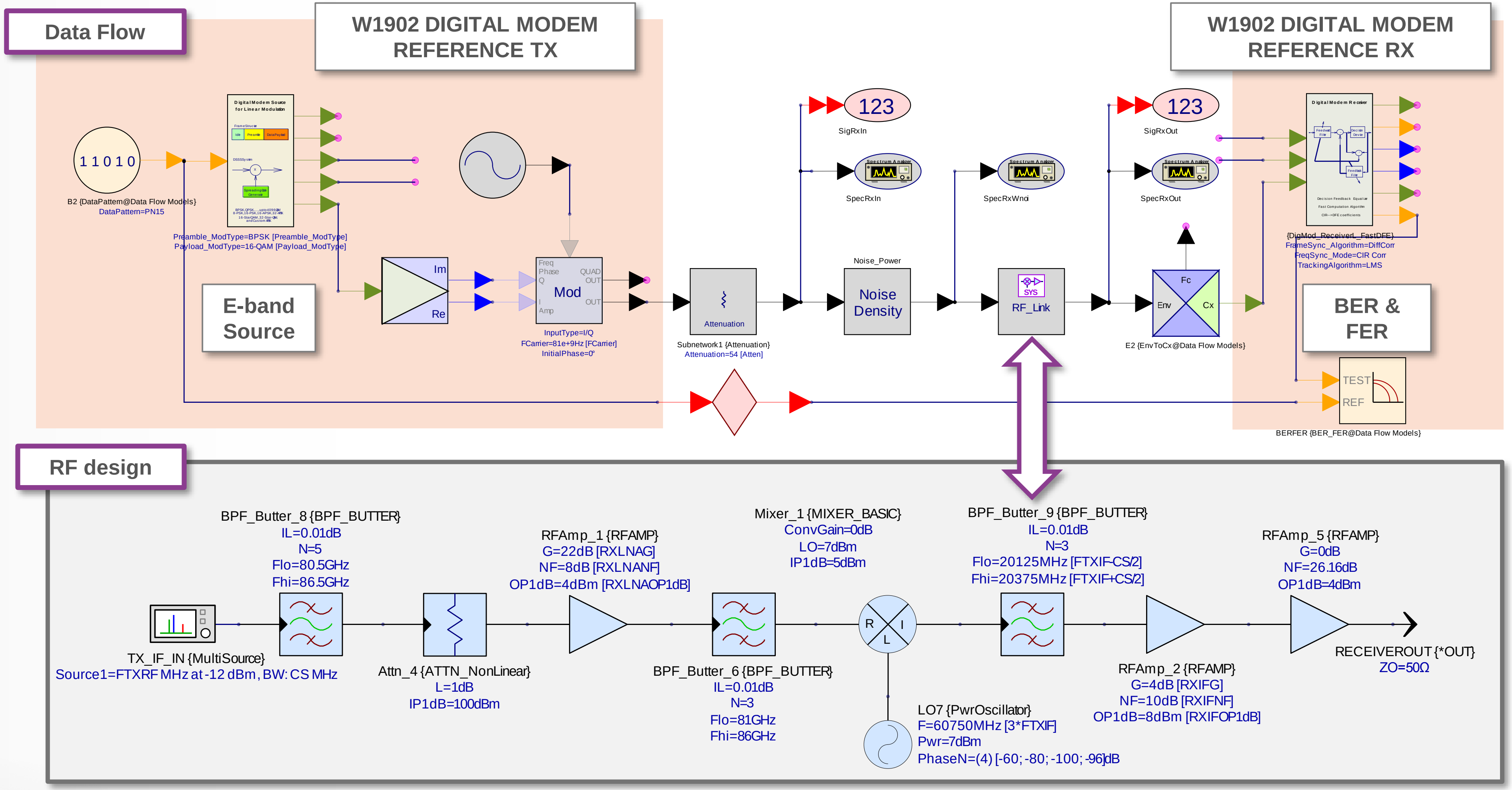
Instrument link



Download waveform to Signal Generator
Capture waveform from Analyzer

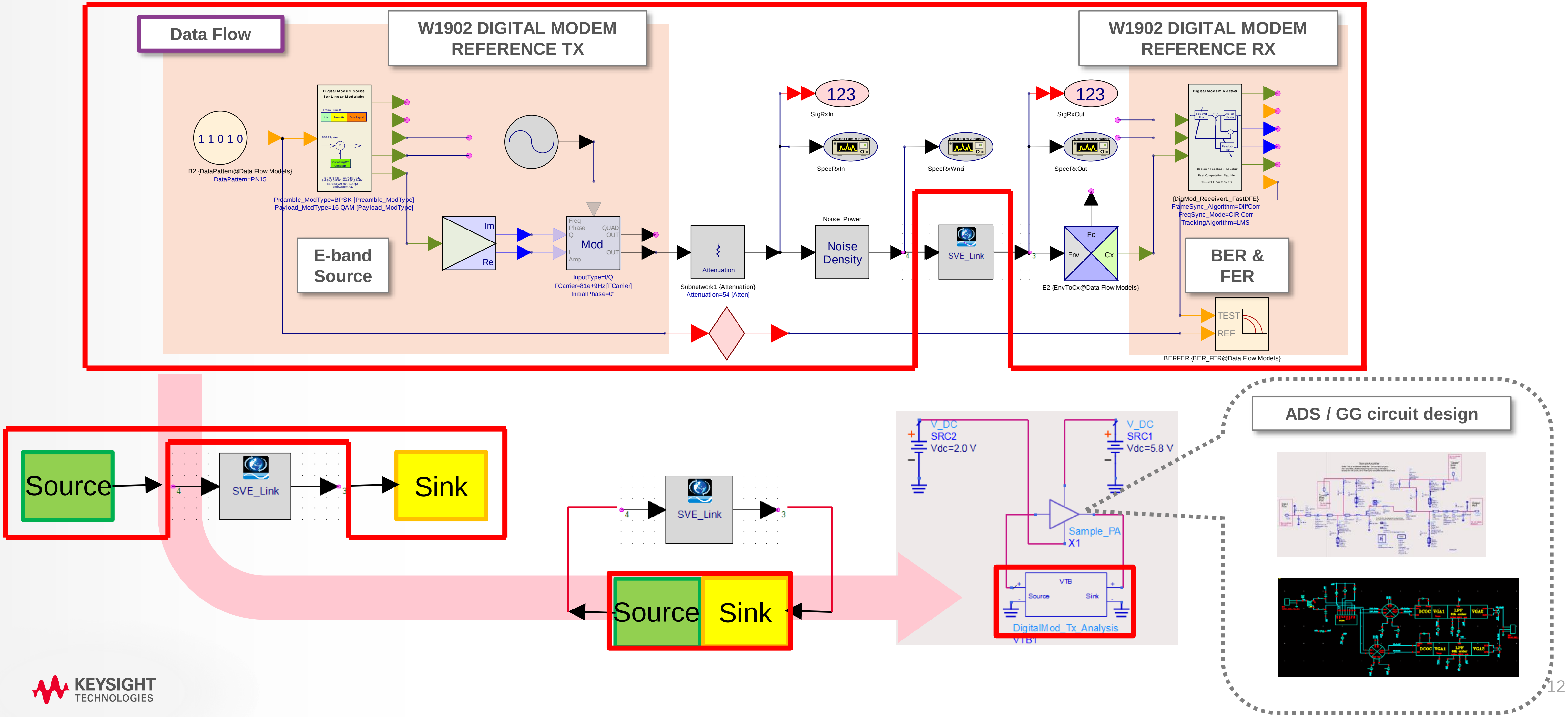
RF Link

Co-simulate with RF design



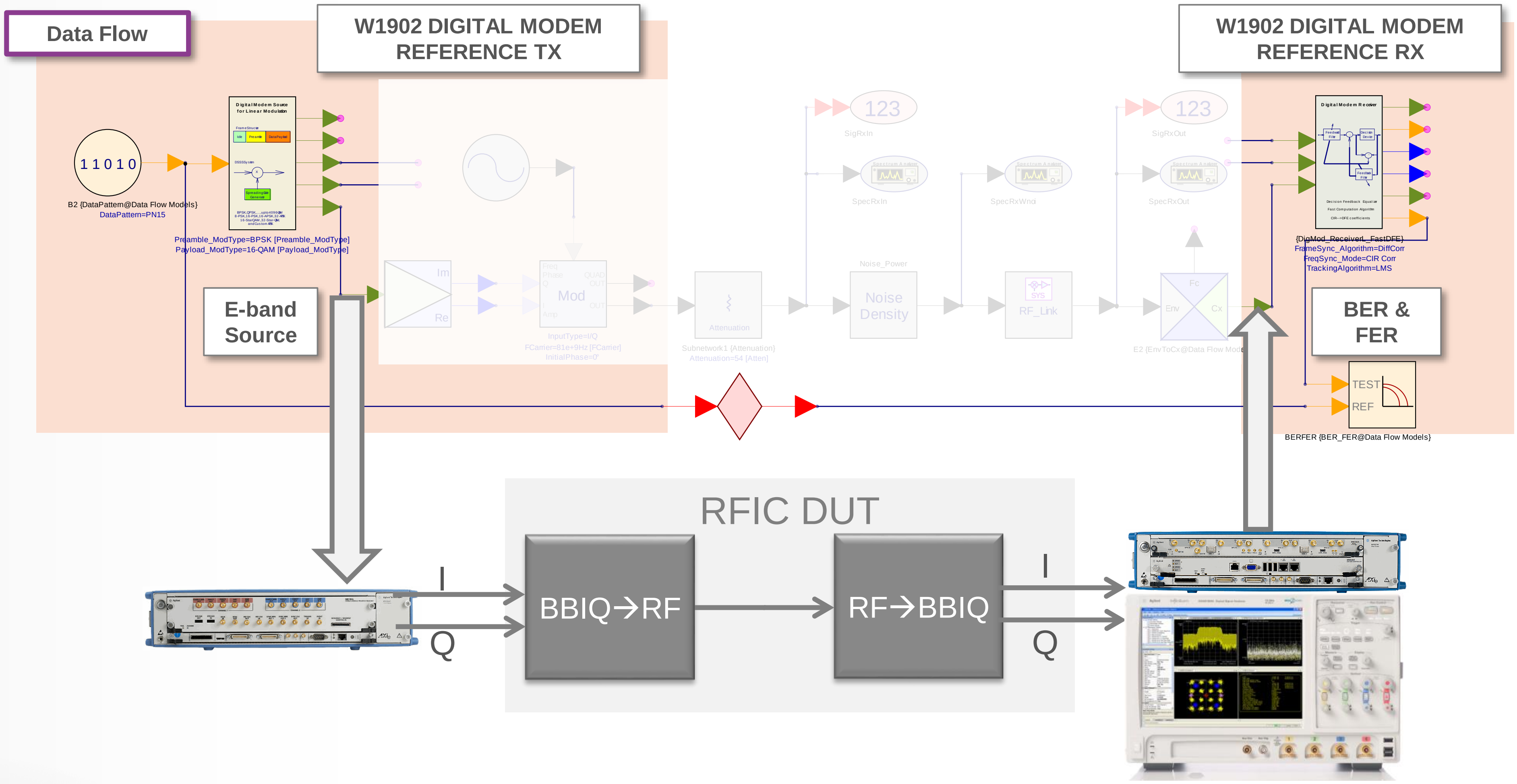
Virtual Test Bench

Verify circuit level design



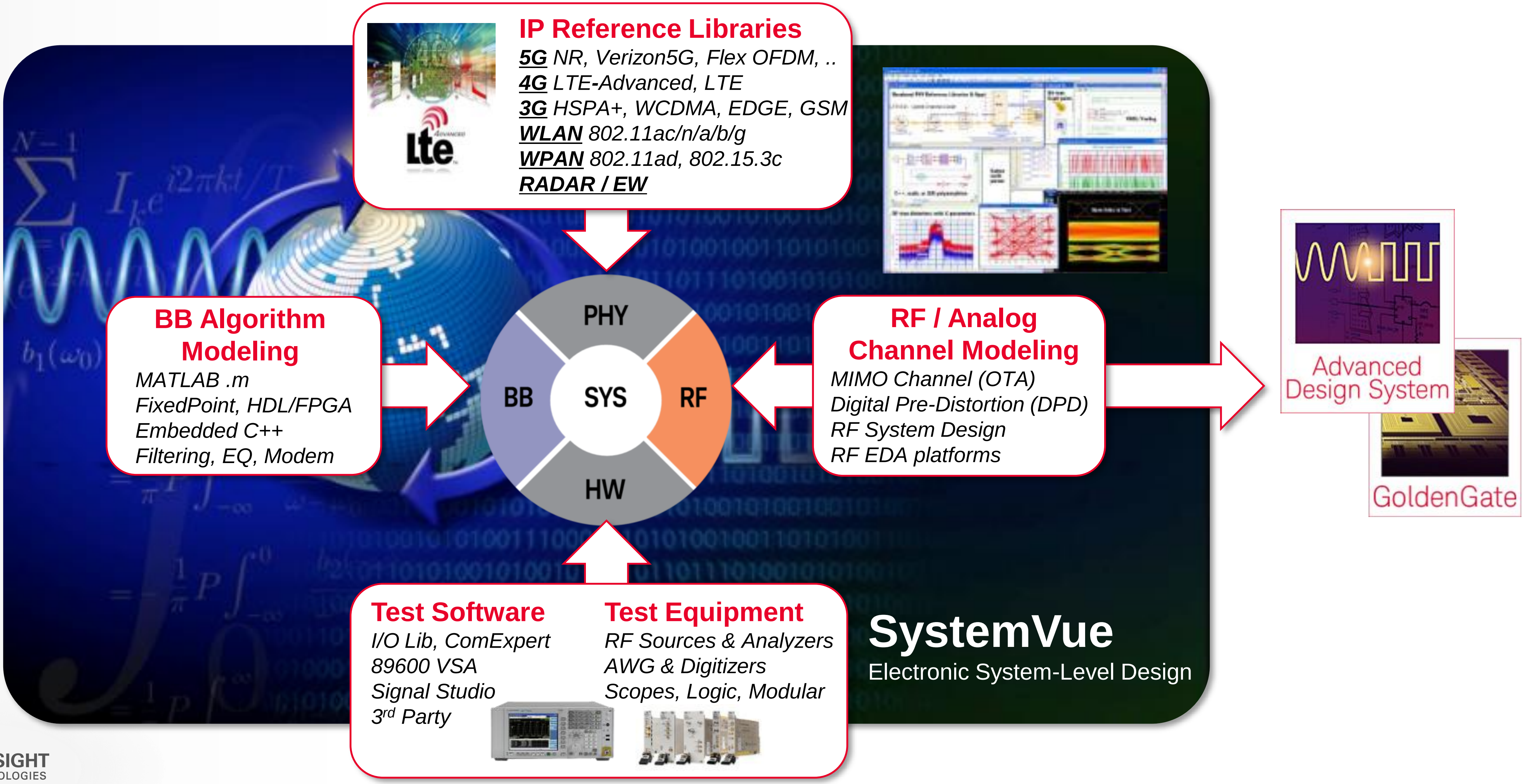
Instrument Link

Test real device



Integrated Design and Test solution

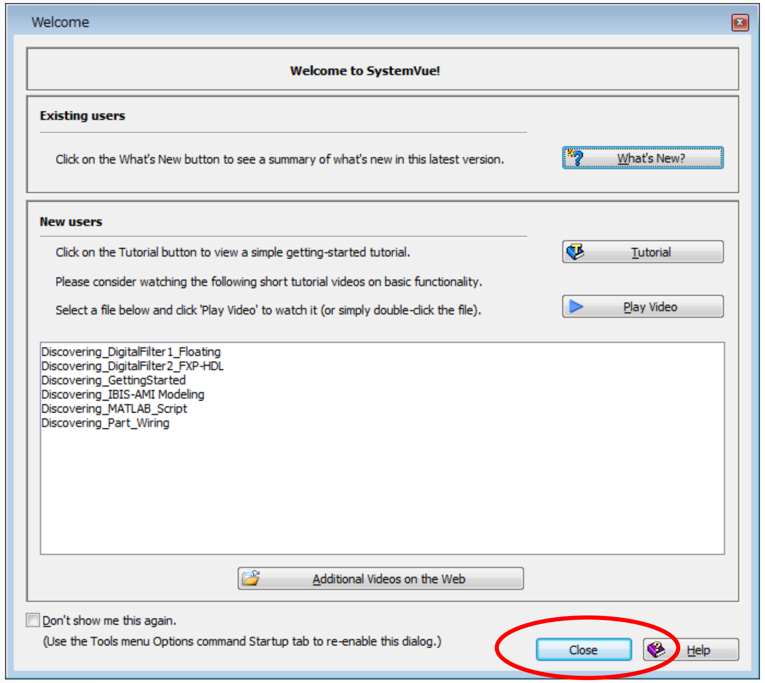
Transition naturally from Design to Test with a single "cockpit"

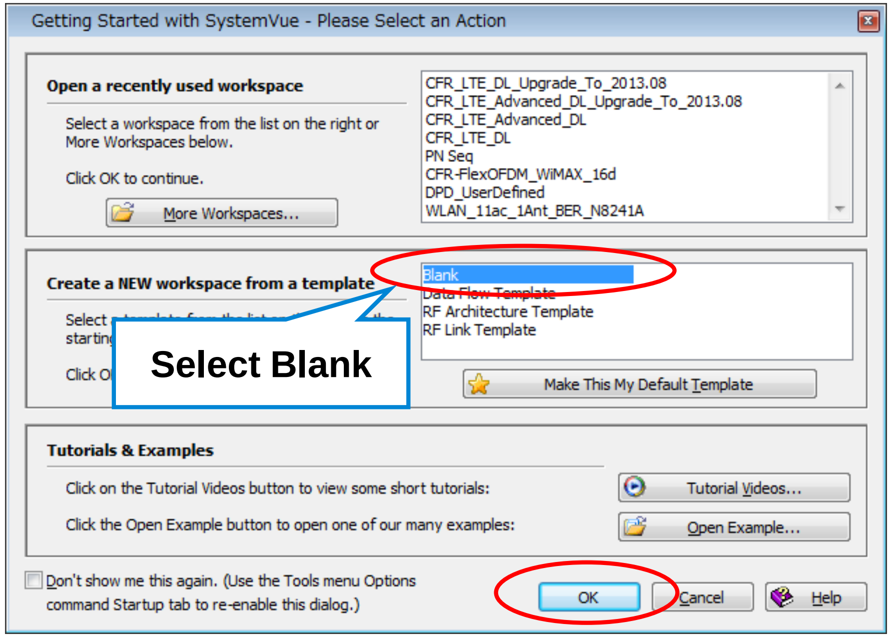


Lab 1 : Getting Started with SystemVue

Lab 1: Getting Started with SystemVue

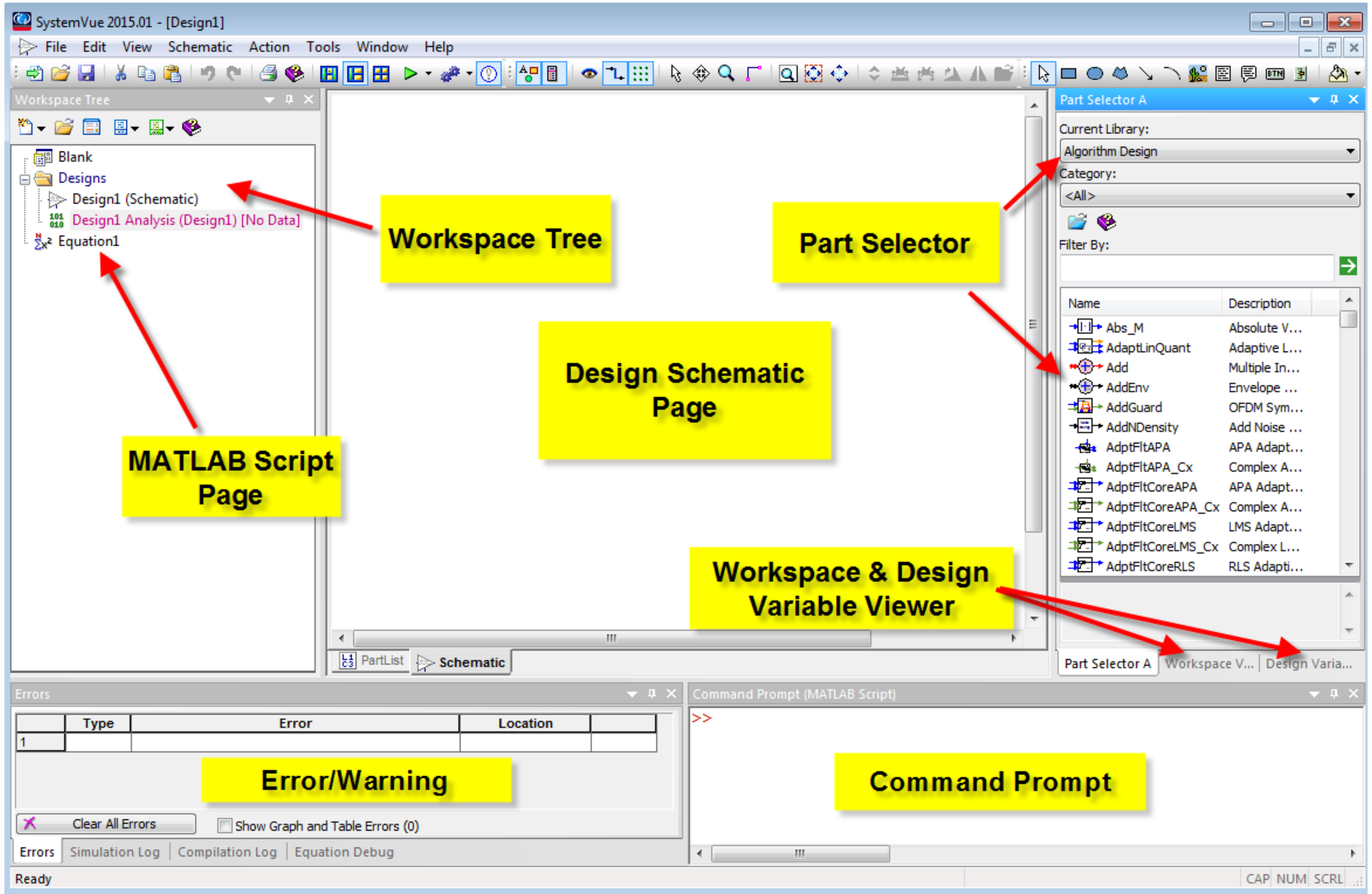
- Start SystemVue and **close the initial window** to see Getting Started Window. We can use the predefined templates to start our design or double click on recently used workspaces to open the same
- **Select "Blank" Template** from "Create a NEW Workspace from a template" section and **click OK**





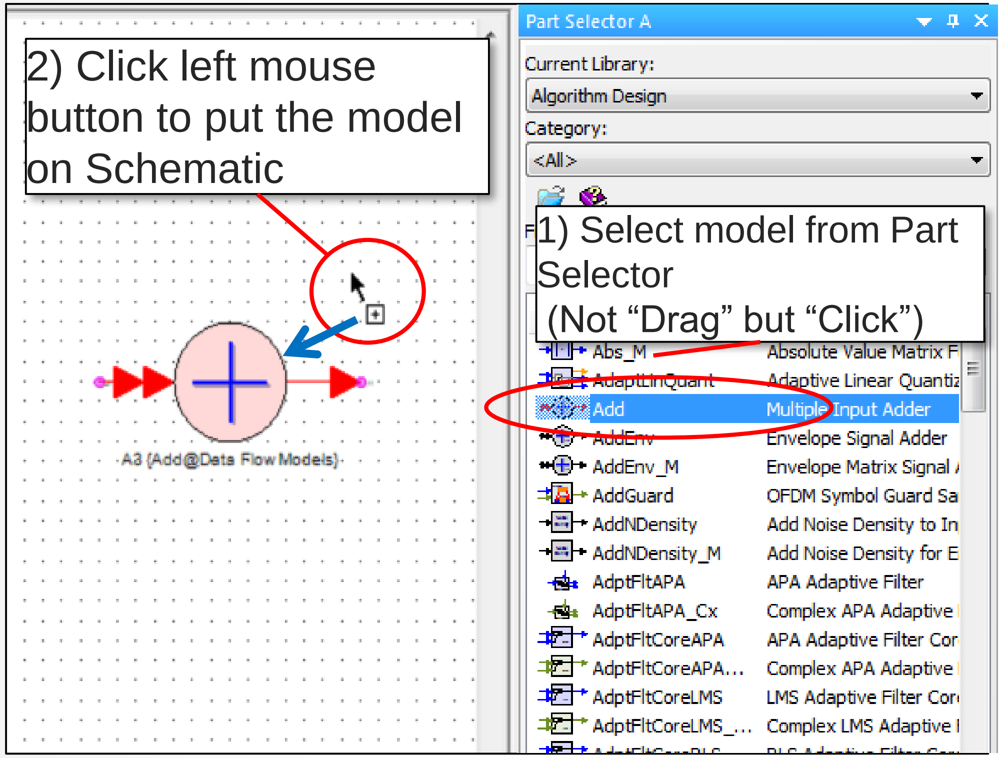
Lab 1: Getting Started with SystemVue

➢ Spend some time and explore different parts of the GUI as shown below

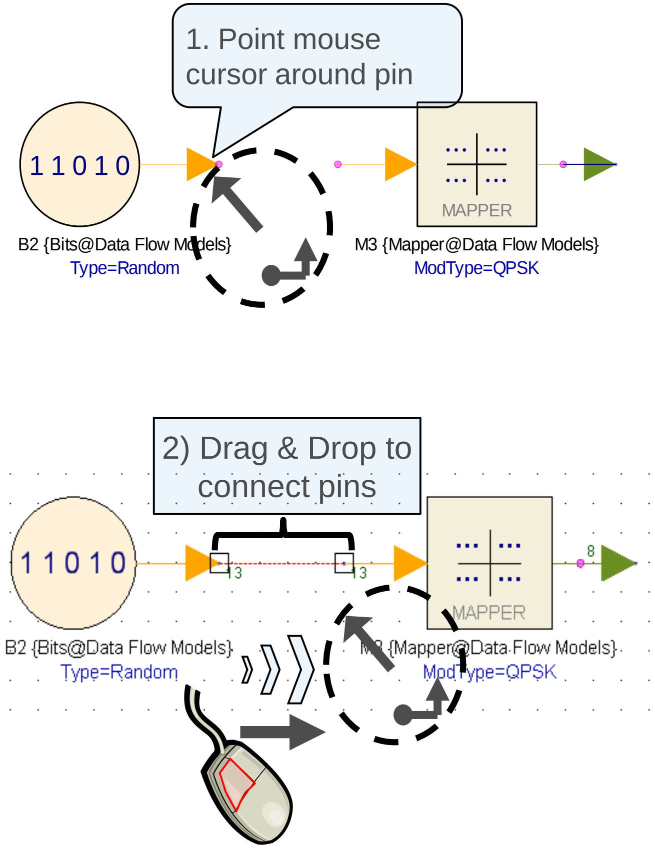


Exercise: (#DO NOT run simulation)

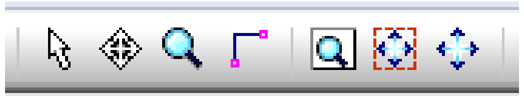
- Select model from Part Selector and put it on Schematic



- How to connect models



- Adjust window size & position



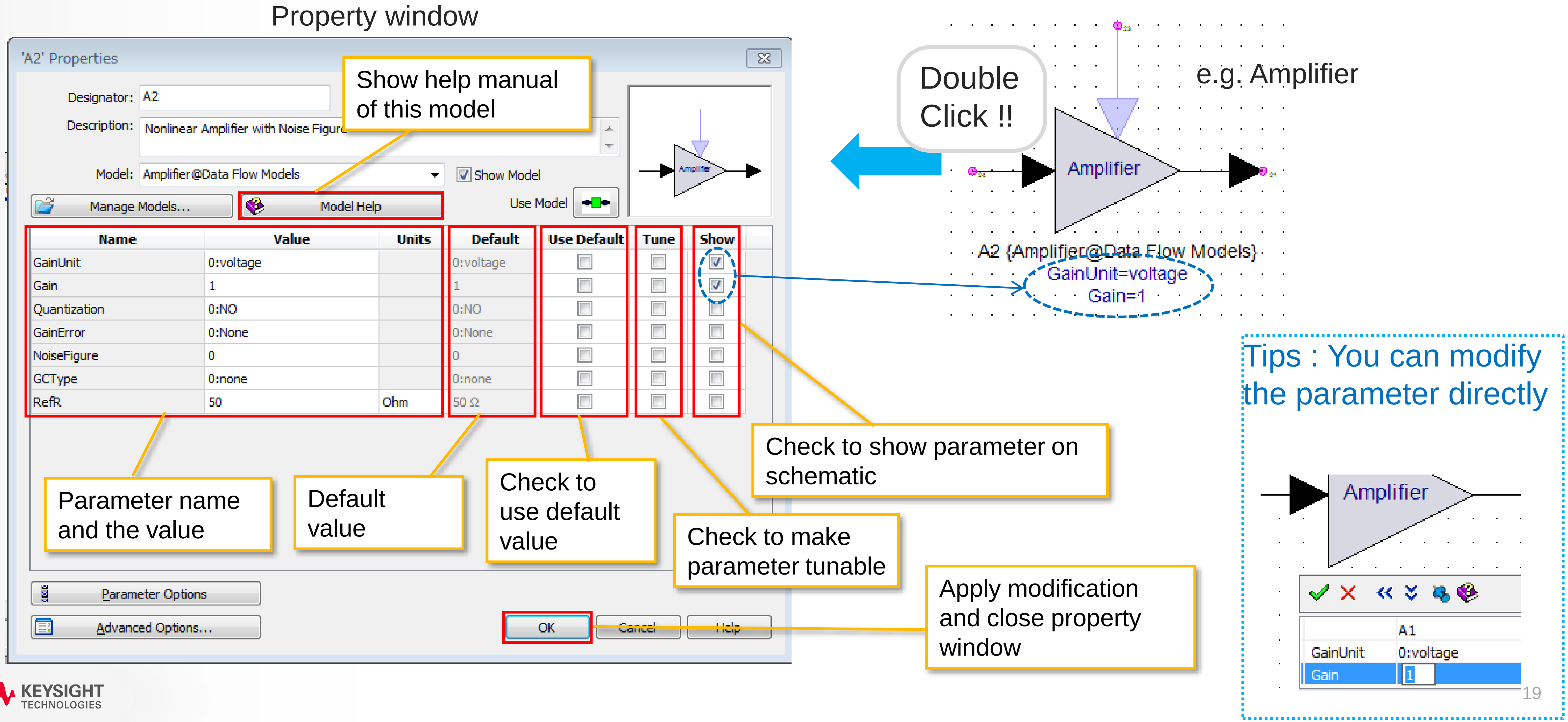
Zoom In & Out : Mouse wheel

PAN : Right Mouse Click

Zoom to fit all : "Z" key

Exercise: Model property and parameter

- Double click on model instance and open property window



【Ref.】Data Flow simulation

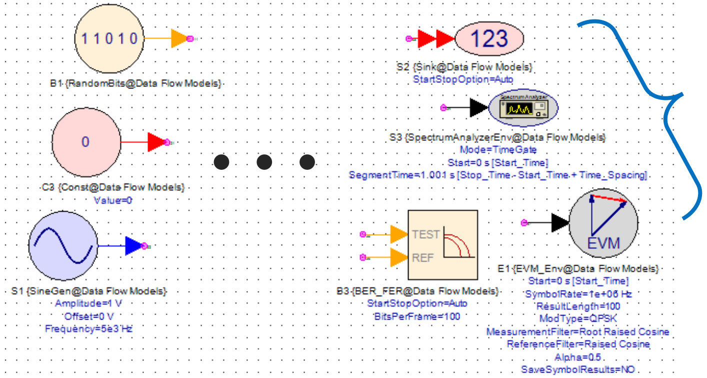
SystemVue's core engine is time domain simulator based on sampled data called **Data Flow**

DataFlow is…

- Data-driven simulator
  - Component turns ON when it receives data
  - FIFO（first-in-first-out） data process
  - Data is transferred from each component to each synchronously
- Each model requires some number of input data to output one data
- Each "sink" component decides the total simulation schedule by backward order
- Schedule means 1. order / 2. number of "turn ON" each components
- "Sink" controls the simulation time

Source

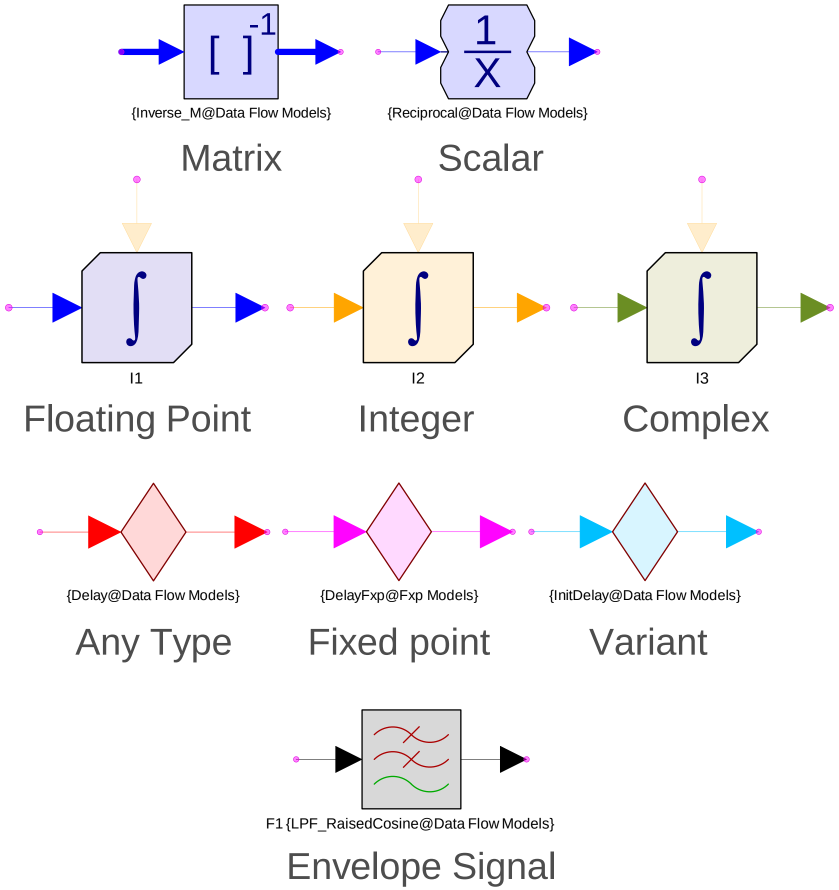
Sink



Capture data and save as Dataset

【Ref.】Data type and color in Data Flow simulation

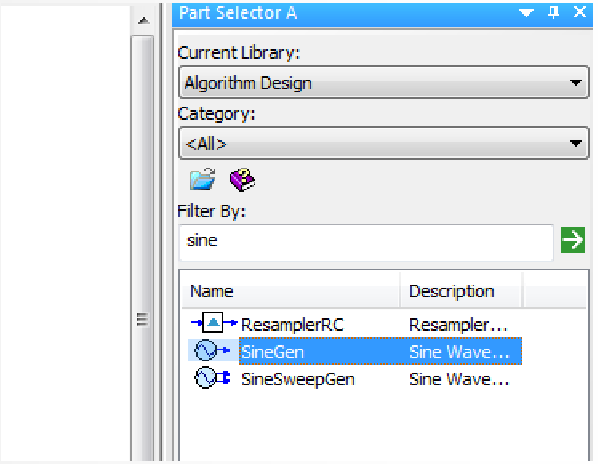
Data Type	Port Color	Port Thickness	Model Suffix
Scalar Integer	Orange	Thin	Int
Scalar Floating Point (Real)	Blue	Thin	
Scalar Complex	Green	Thin	Cx
Scalar Fixed Point	Magenta	Thin	Fxp
Integer Matrix	Orange	**Thick**	Int_M
Floating Point (Real) Matrix	Blue	**Thick**	_M
Complex Matrix	Green	**Thick**	Cx_M
Envelope Signal	Black	Thin	Env
Any Type	Red	Thin	
Variant	Cyan	Thin	

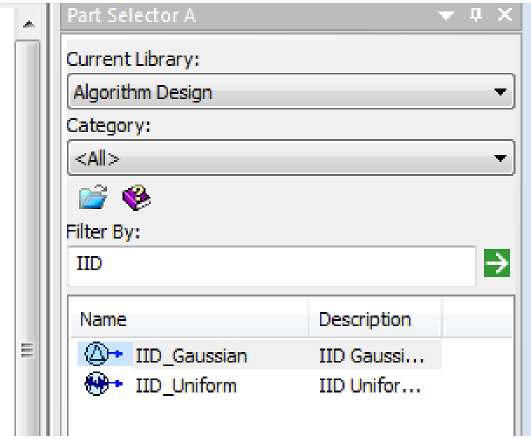


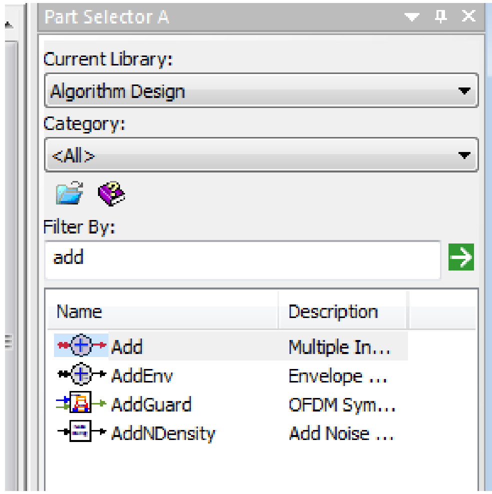
- Need to connect same color pins except “Any Type”
- Use “Type Converters” to connect different color pins
- Take care in case auto-converting data among Integer/Fixed Point/Floating Point types

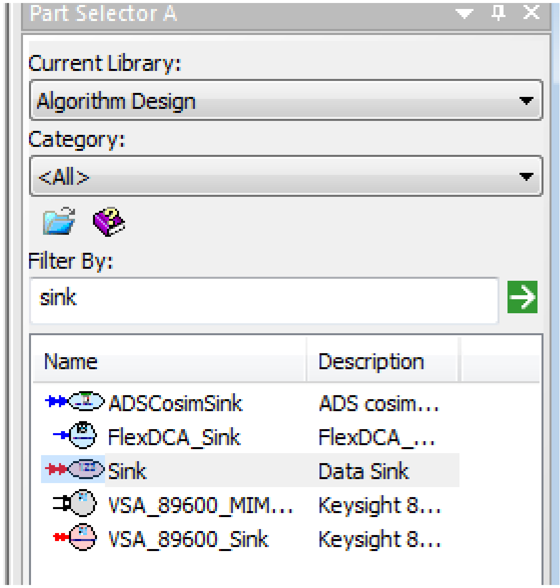
Lab 1: SineGen, IID_Gaussian, Add and Sink

- Let’s create a simple schematic to get started with SystemVue software…….
- **From Algorithm Design library,** place **SineGen**, **IID_Gaussian, Add** and **Sink** components onto the schematic page
- **Double click on IID_Gaussian** component and define **StdDev = 0.1 V**



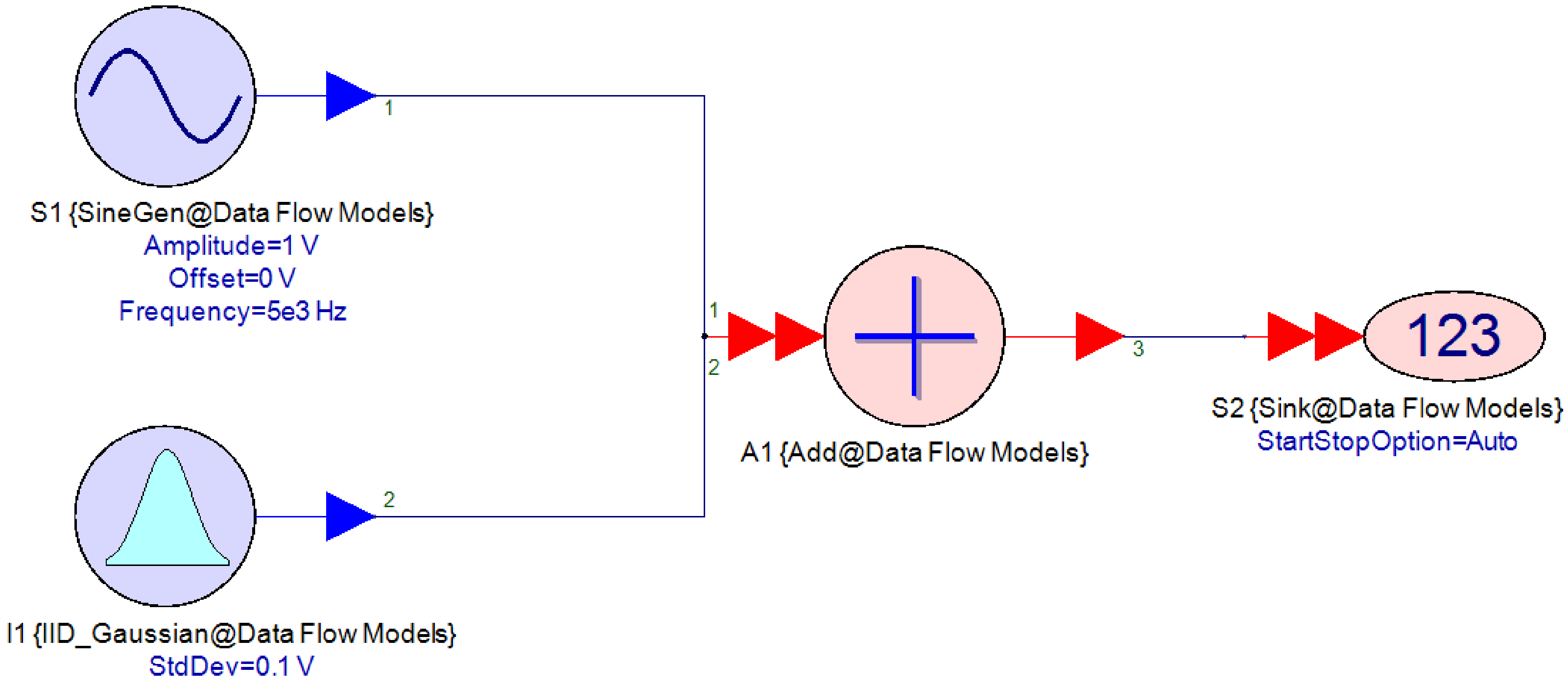






Lab 1: SineGen, IID_Gaussian, Add and Sink

- Connect the components as shown here……
- Note that default frequency in SineGen is 5KHz and we shall use the same
- Save the workspace and give name as **Lab1_GettingStarted**

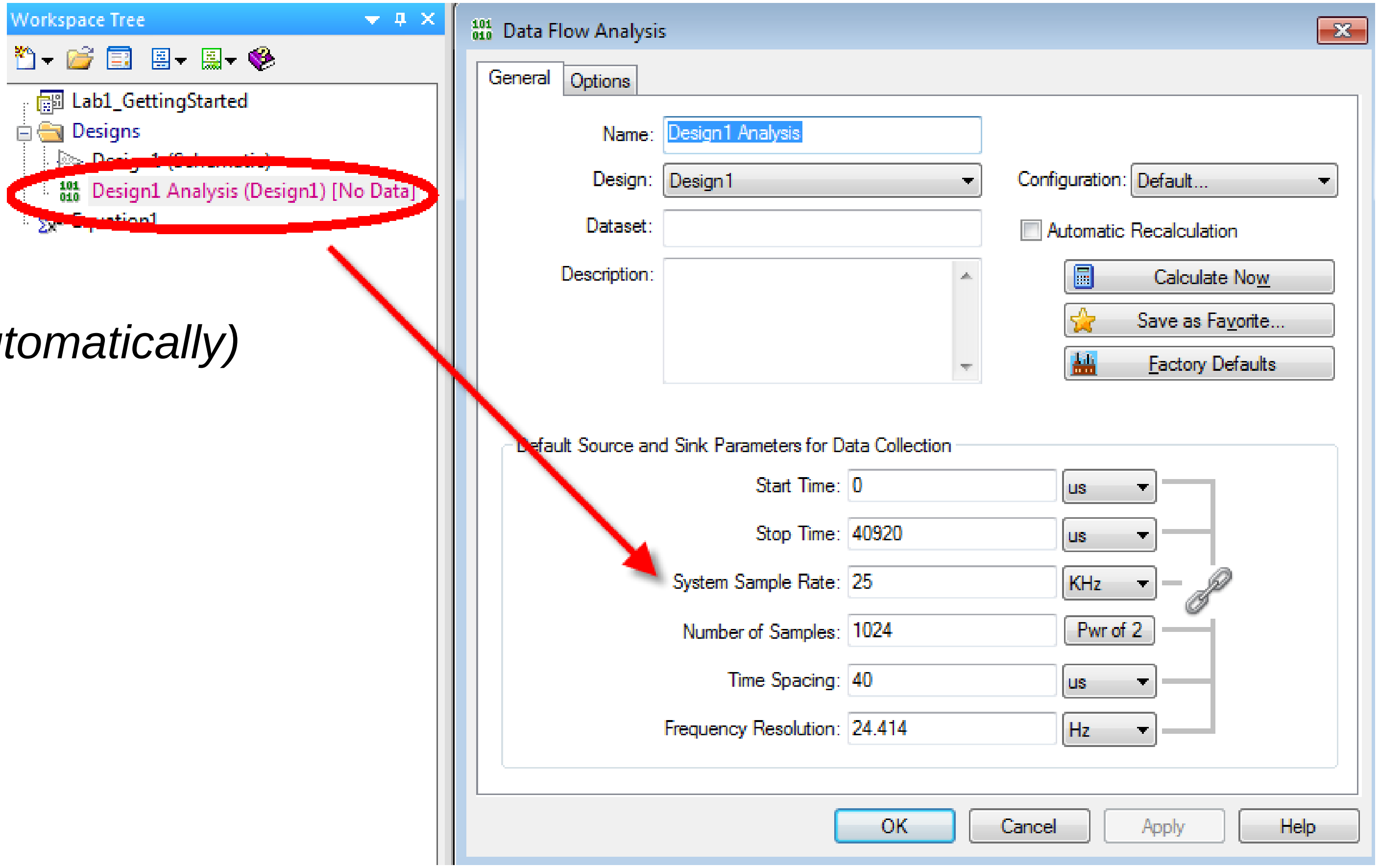


KEYSIGHT TECHNOLOGIES

Lab 1: DF Simulation Controller

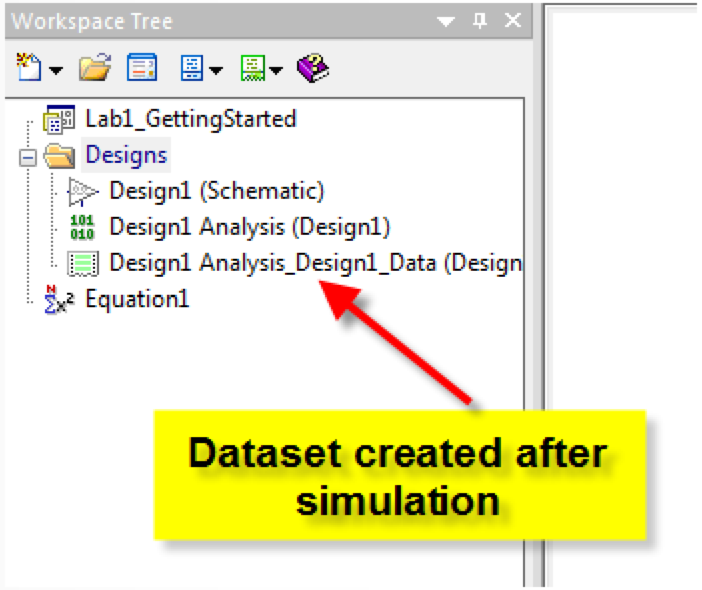
- Double click on Design1 Analysis in the workspace tree and define the two main parameters as:
  - **System Sample Rate = 25 KHz**
  - **Number of Samples = 1024**

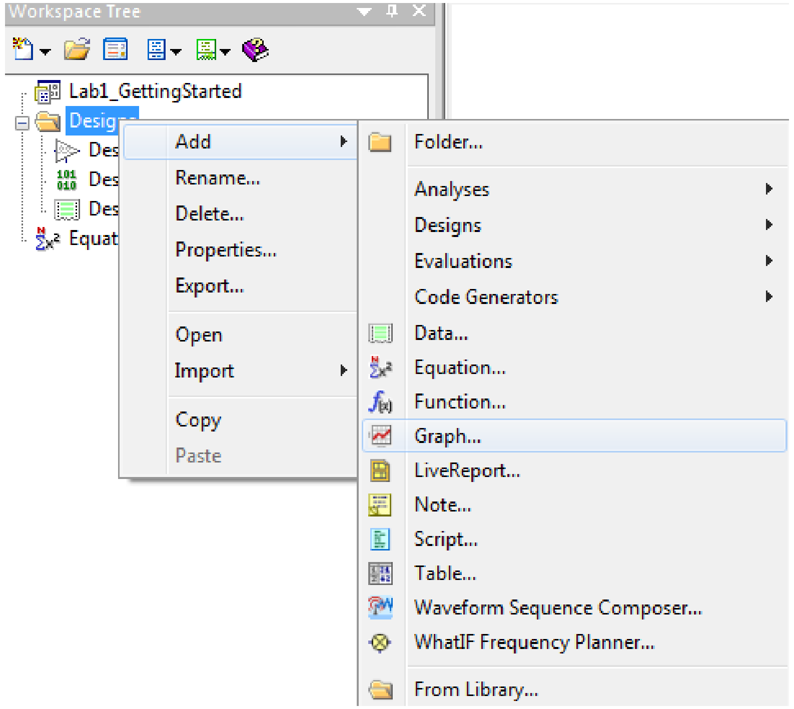
(rest of the parameters will update automatically)



Lab 1: Run simulation and show result

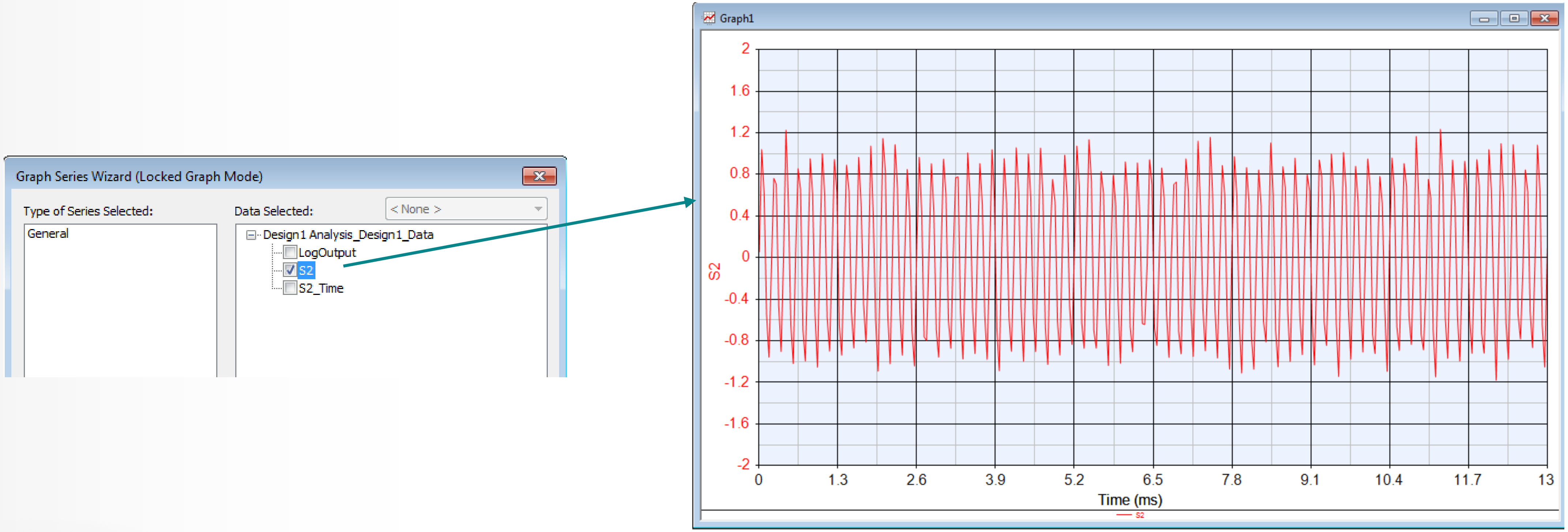
- Run Simulation (right click on Design1 Analysis and select Run (Calculate Now) and notice a dataset containing our simulation results created under workspace tree
- **Right click on Designs folder under workspace tree, select Add->Graph**





Lab 1: Plot Waveform

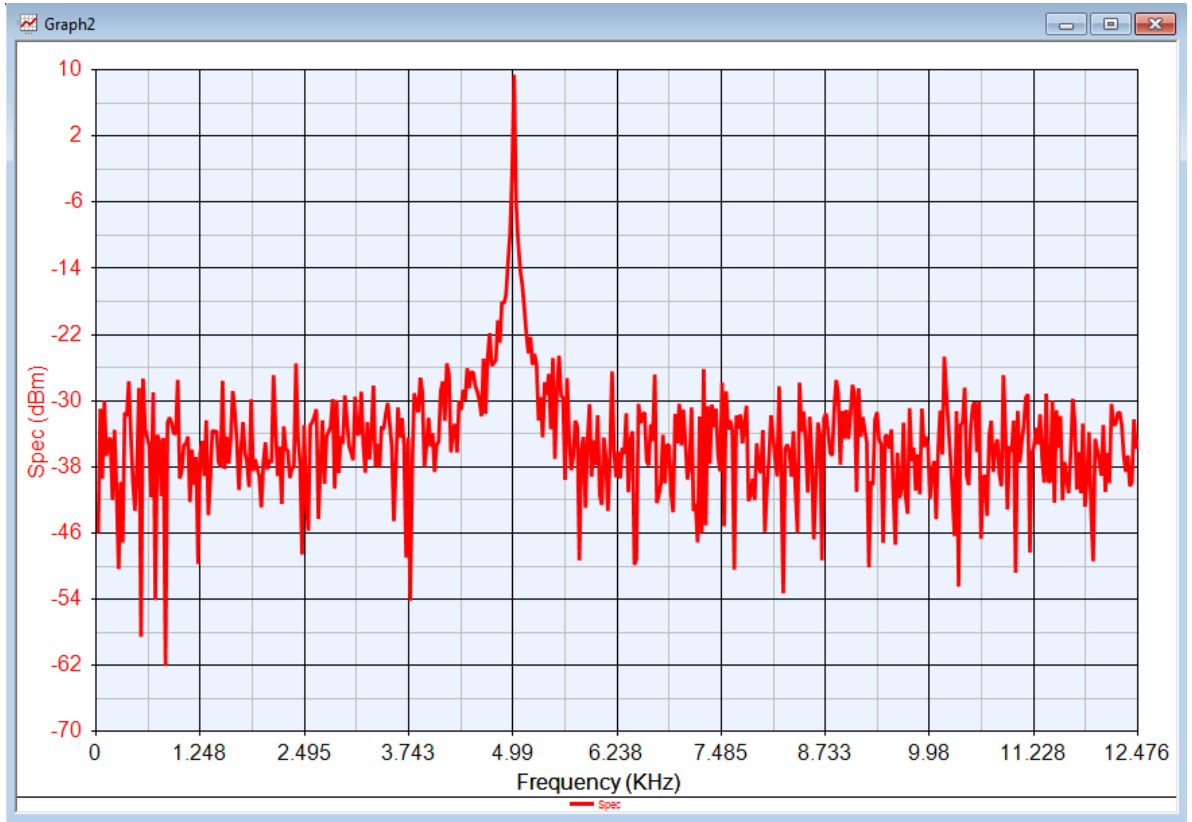
- Select **Series Type = General** and select **S2** (name of the sink in schematic) sink from the available sinks under Dataset to plot time domain waveform. Click OK and select X-axis unit as “ms” in the graph wizard window
- Use Scroll wheel of mouse to Zoom-In to the desired section of the waveform in the plot to observe Sine Wave with Noise as shown here….



KEYSIGHT TECHNOLOGIES

Lab 1: Plot Spectrum

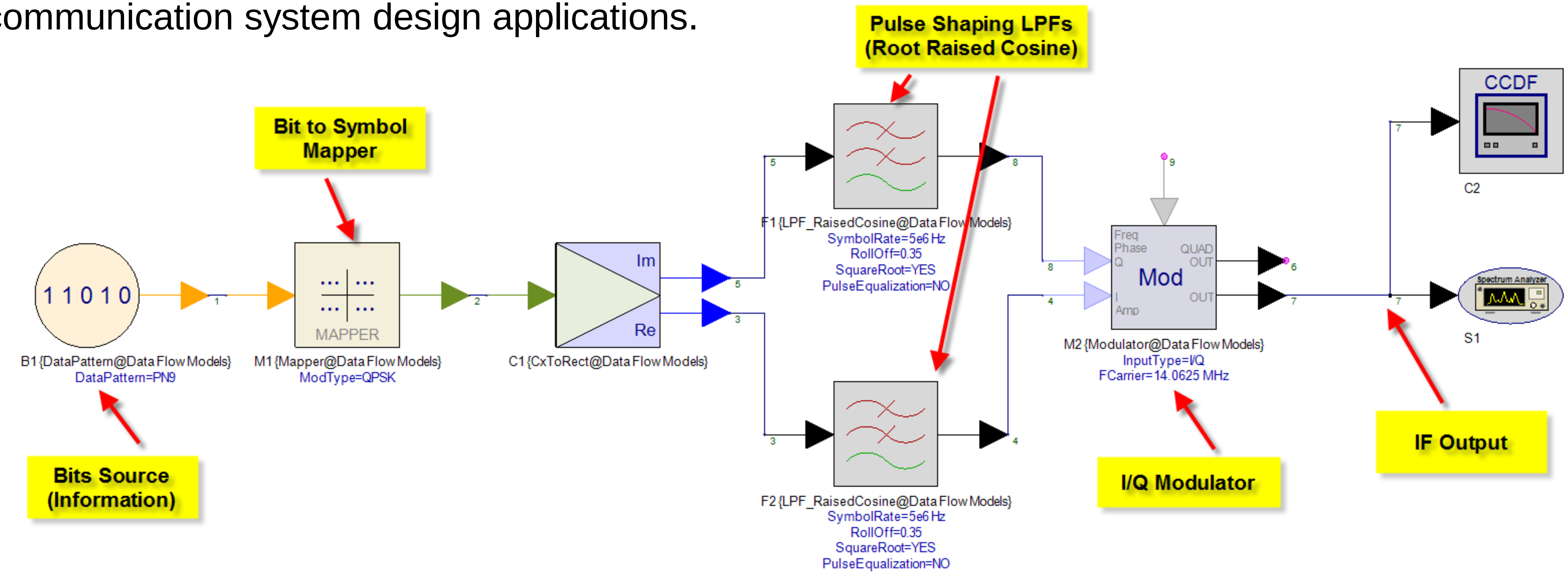
- **Add a new graph**, select **Series Type as Spectrum** and **select S2** from the available sink name under dataset
- Click OK to close the graph wizard window and observe the Spectrum plot



Lab 2: QPSK Modulator Design

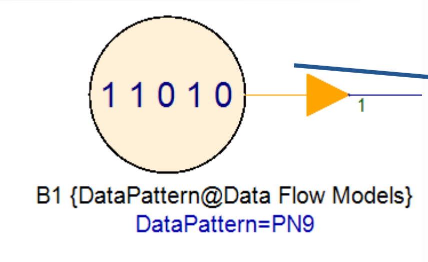
Lab 2: QPSK Modulator Design

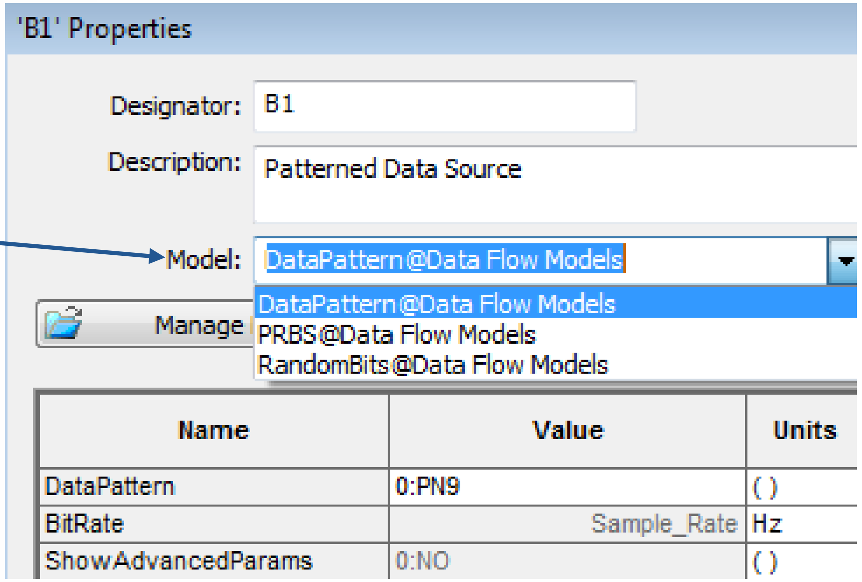
- Snapshot below shows a simple block diagram of a typical QPSK modulator.
- We shall implement this modulator in Lab 2 to increase our understanding of using SystemVue for communication system design applications.

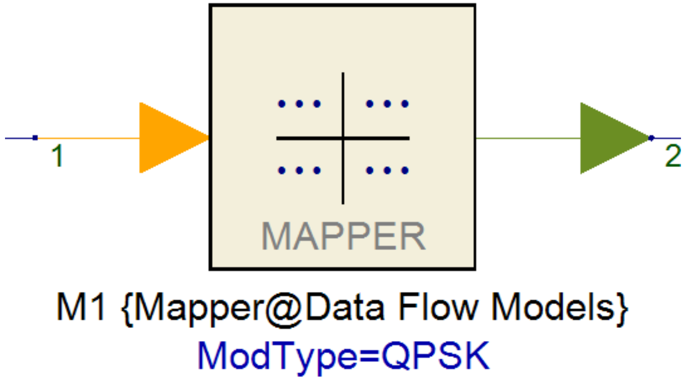


Lab 2: Bits and QPSK mapper

- **Create a new Workspace** and **save it as "Lab2_QPSK_Design"**
- Place **Bits, Mapper and Sink** from Algorithm Design library onto the default Design1 schematic.
- Double click on Bits components and select **Model = DataPatterns** and select **DataPattern as PN9**
- **Double click on Mapper component** (explore various Digital modulations possible) and ensure ModType is set to "QPSK"

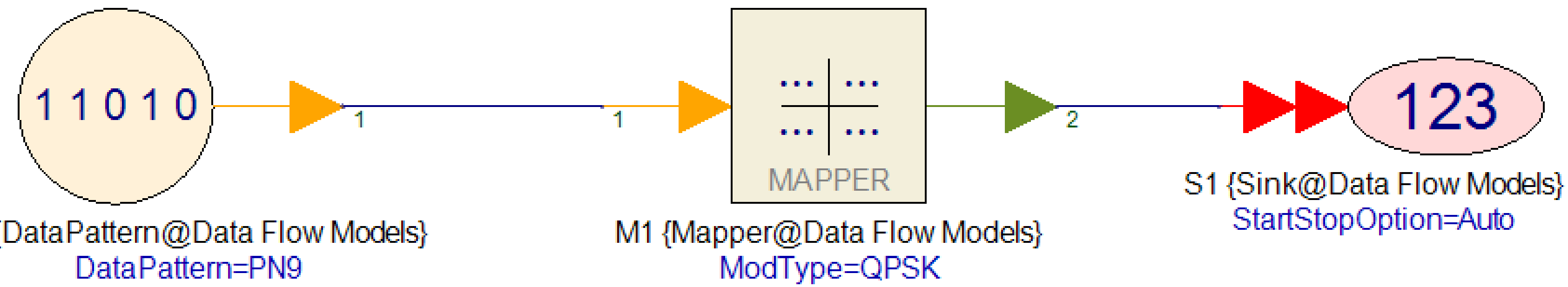






Lab 2: Bits and QPSK mapper

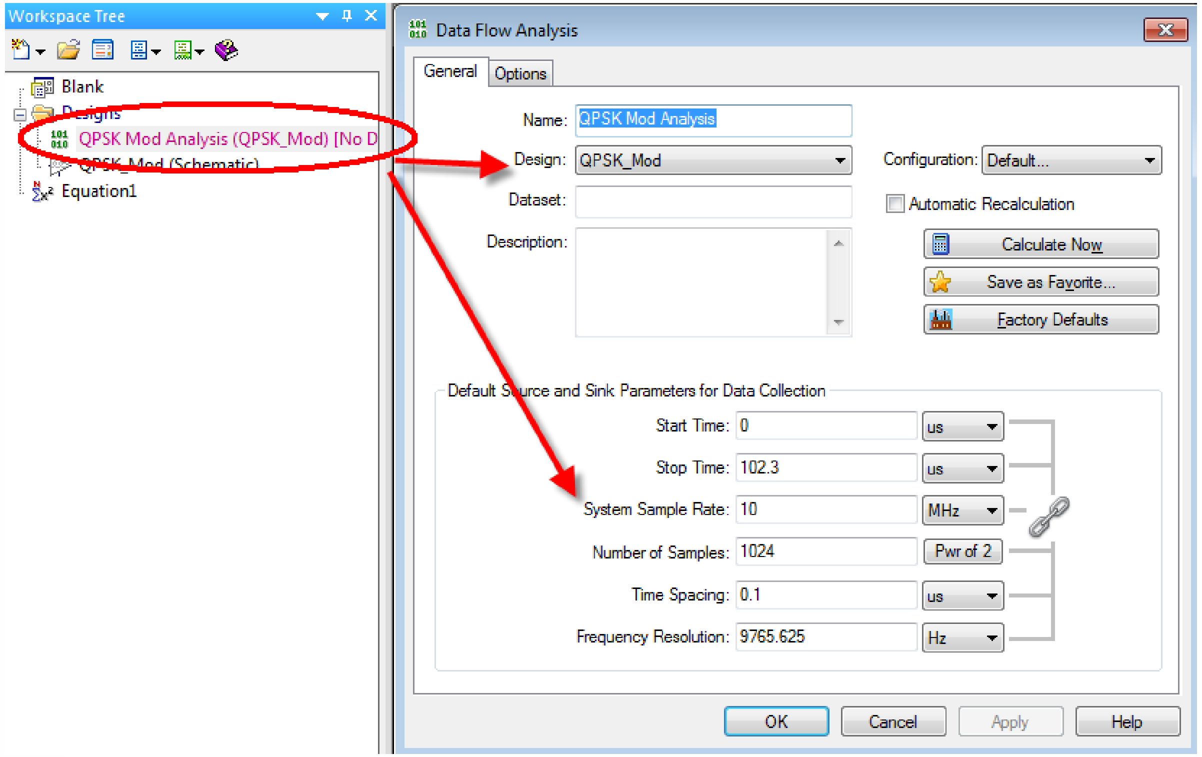
- Connect the components as shown here



- From the workspace tree:
  - **Rename Design1 as “QPSK_Mod”**
  - **Rename Design1 Analysis as “QPSK Mod Analysis”**

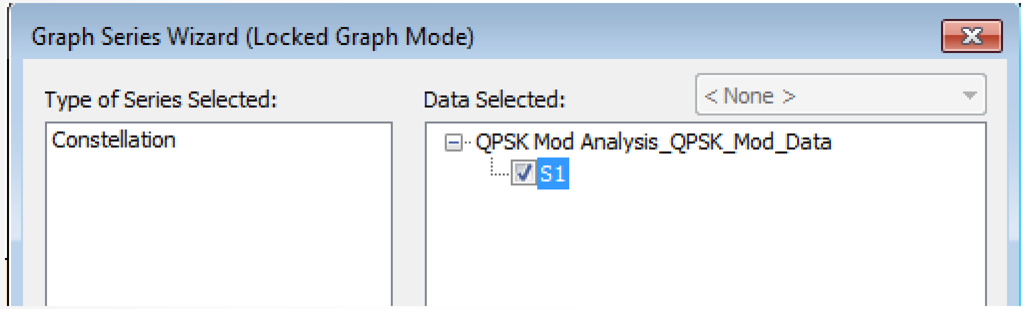
Lab 2: Run simulation

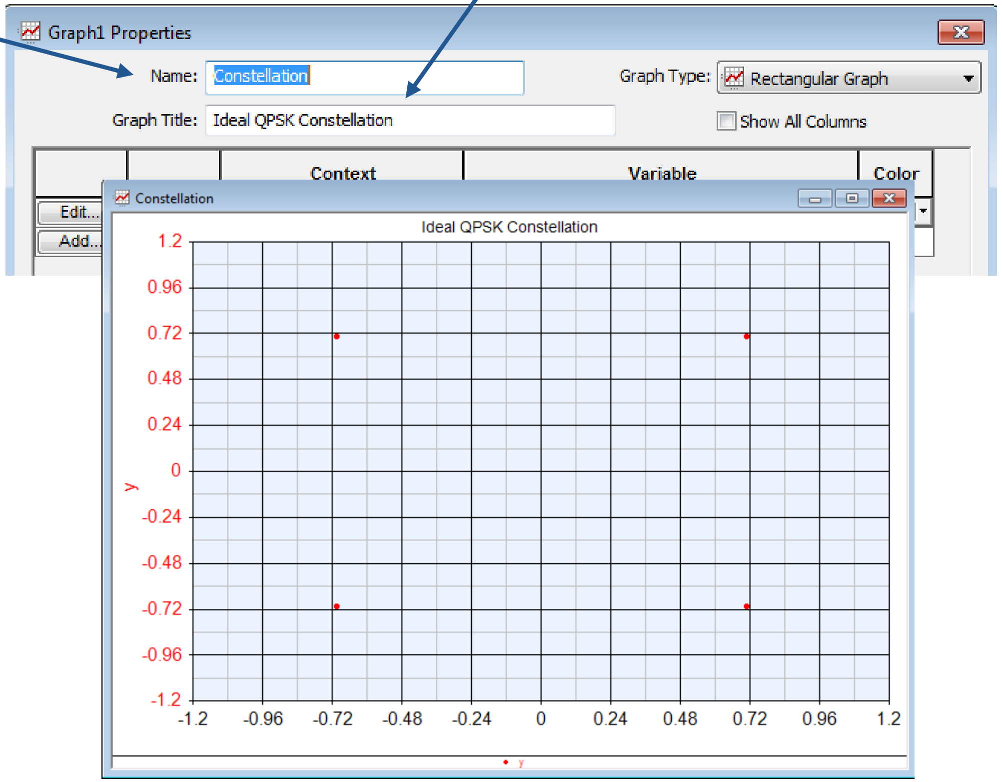
- **Double Click on Simulation Controller**: QPSK Mod Analysis and **select Design as "QPSK_Mod"**
- Set **Sample Rate = 10 MHz** and **Number of Samples = 1024**. Click OK and run simulation



Lab 2: Plot QPSK Constellation

- Add a new graph, select **Series Type = Constellation**. **Select S1** (name of the sink). Click OK
- **Set X and Y-axis limits** as: Min = -1.2, Max = 1.2.
- Give graph name as **Constellation** and Graph Title as **Ideal QPSK Constellation**. Click OK to observe QPSK constellation diagram as shown here

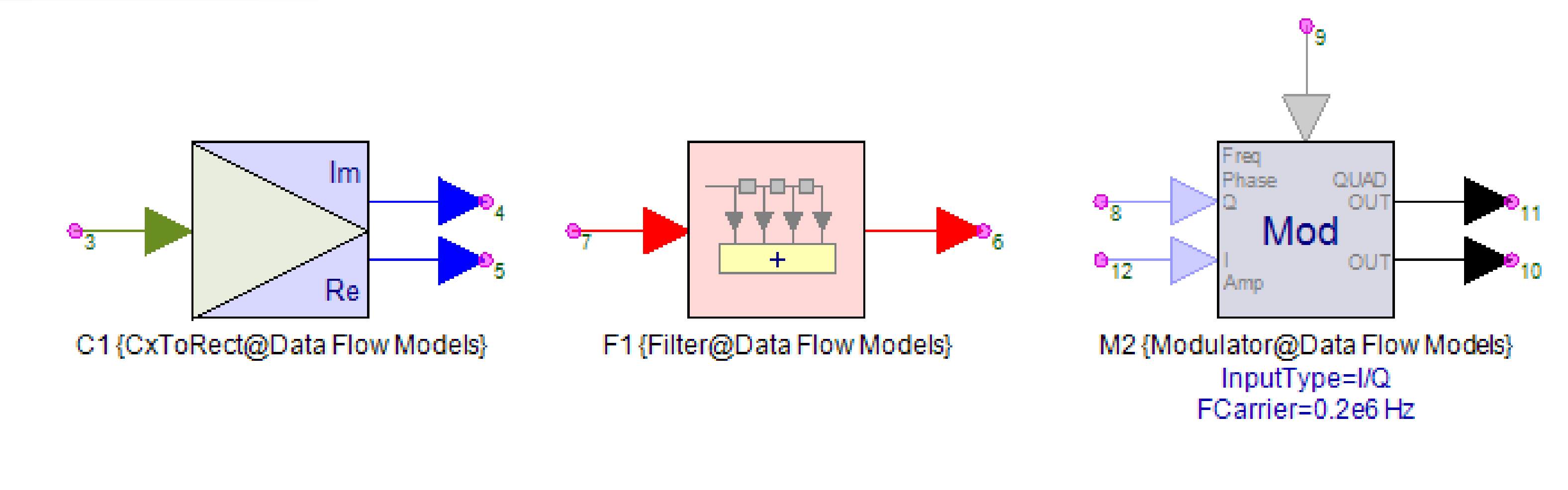


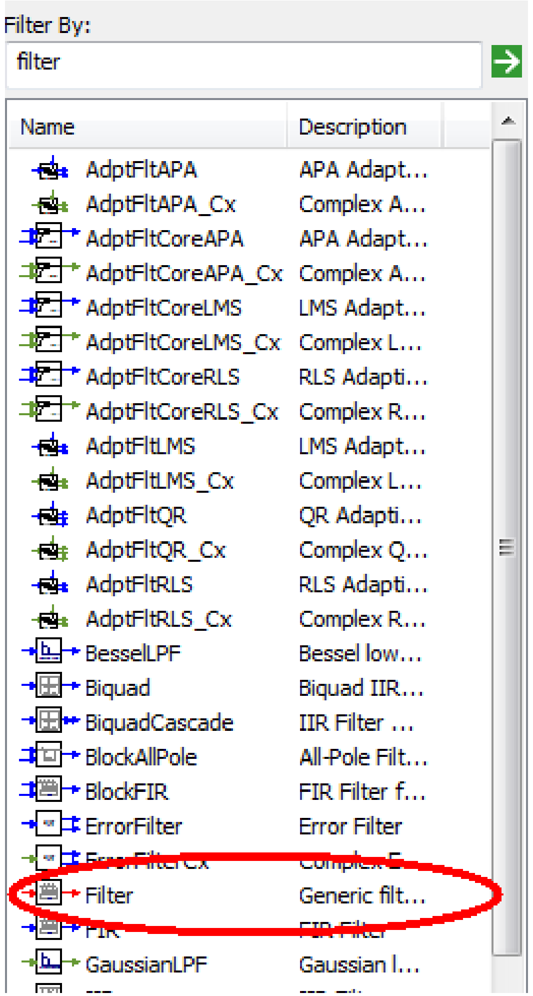


Hint: Right Click on X & Y axis of the graph and select "Axis Font" to increase/decrease the font size

Lab 2: Filter and I/Q modulator

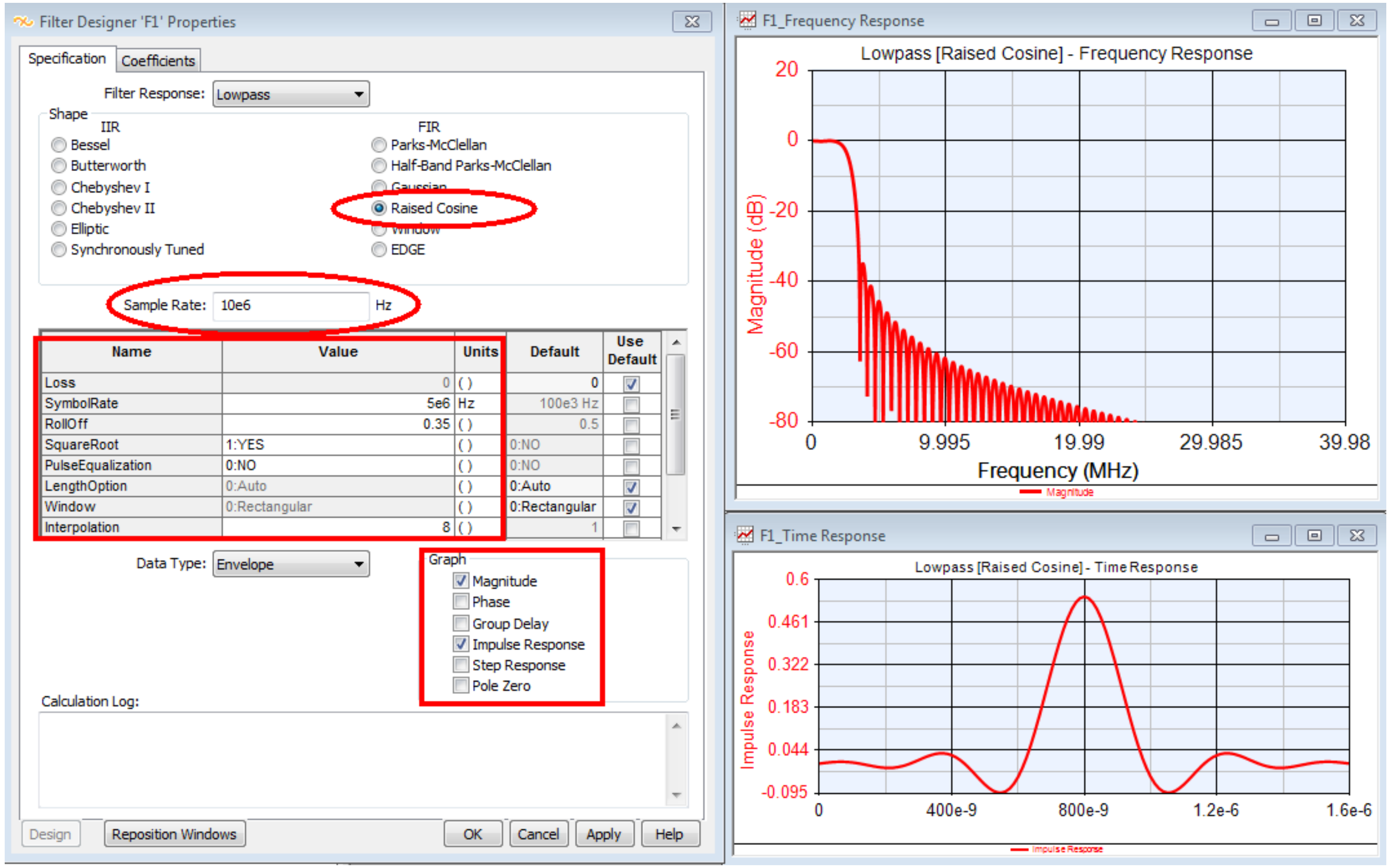
- ➢ Now, we have QPSK symbols available….we shall proceed ahead with rest of the Modulator design.
- ➢ Place **CxToRect** (Complex to Rectangular), **Filter** and **Modulator** from Algorithm Design library as shown here.
- ➢ There will be many filter components in the library, **select "Filter – Generic Filter"** from the list





Lab 2: LPF Design (using Filter Designer)

- Double click on Filter component and set following:
  - FIR: **Raised Cosine**
  - Sample Rate: **10e6**
  - Symbol Rate: **5e6**
  - RollOff: **0.35**
  - SquareRoot: **YES**
  - Interpolation: **8**
- Select **Magnitude** and **Impulse Response** under Graphs
- **Click on "Design" to design baseband RRC LPF and observe results of designed filter**

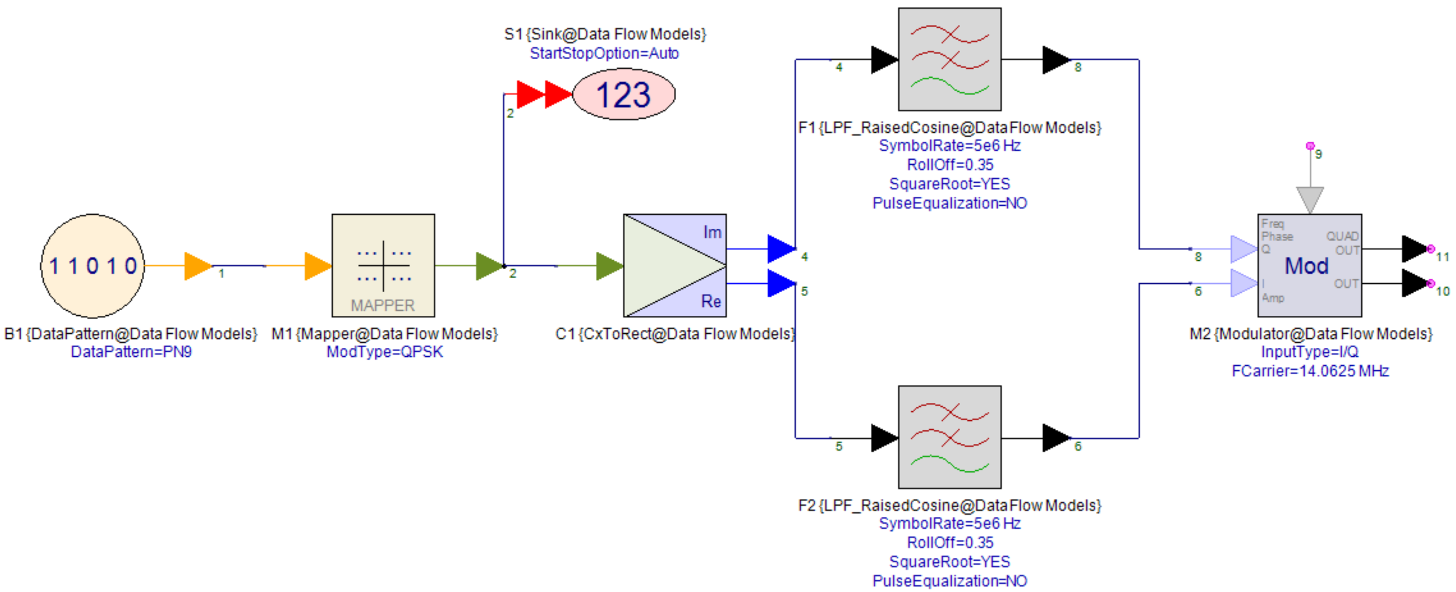


Click OK once you are satisfied with the filter design

Lab 2: I/Q Modulator

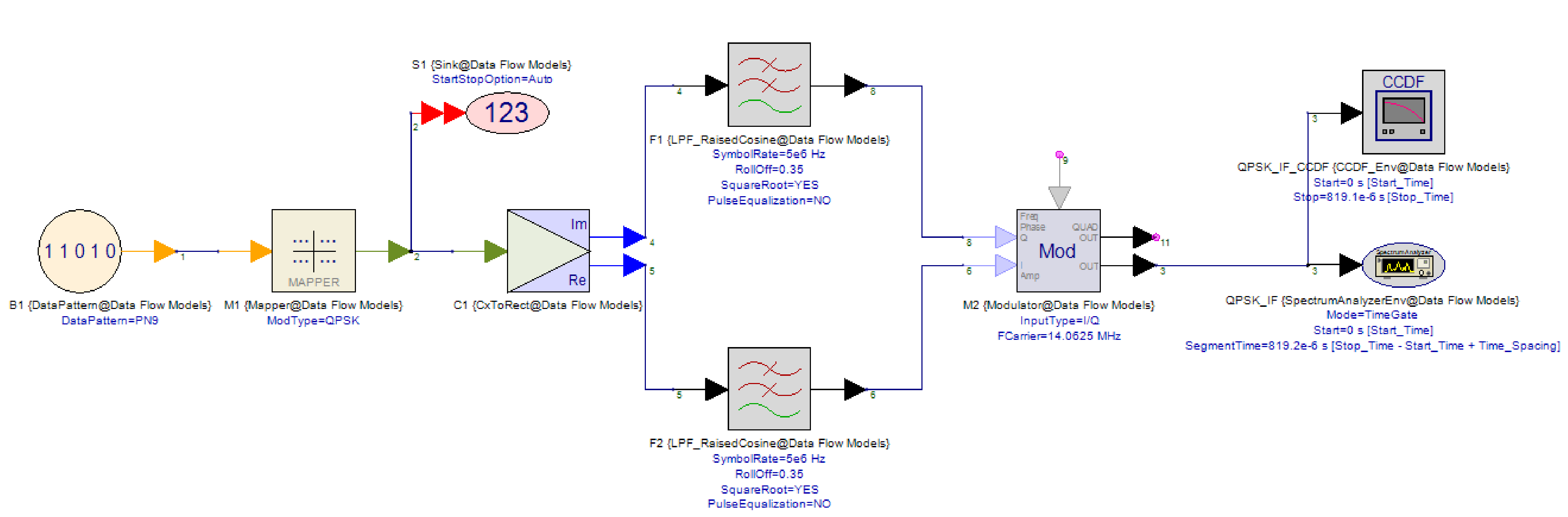
- Double click on Modulator component as define following specs:
  - InputType: **I/Q**
  - Fcarrier: **14.0625 MHz**
  - AmpSensitivity: **sqrt(2)** **<<this will set O/P power = 13dBm>>**
- **Copy/Paste LPF** for other branch and connect components as shown here:

Name	Value	Units
InputType	0:I/Q	( )
FCarrier	14.0625	MHz
InitialPhase	0	deg
AmpSensitivity	sqrt(2)	( )



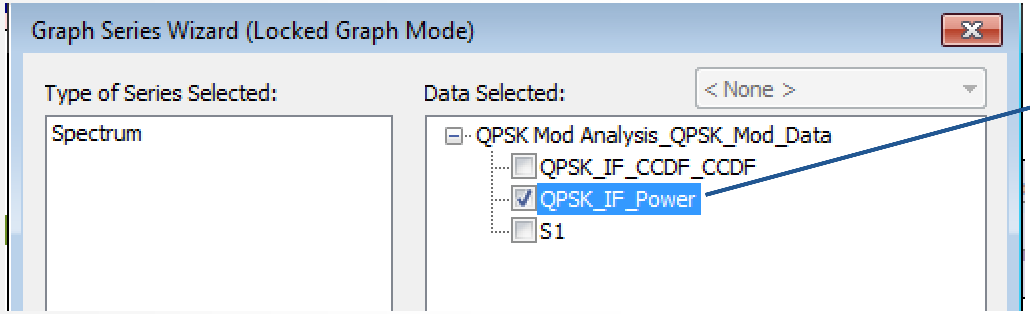
Lab 2: CCDF and Spectrum Analyzer

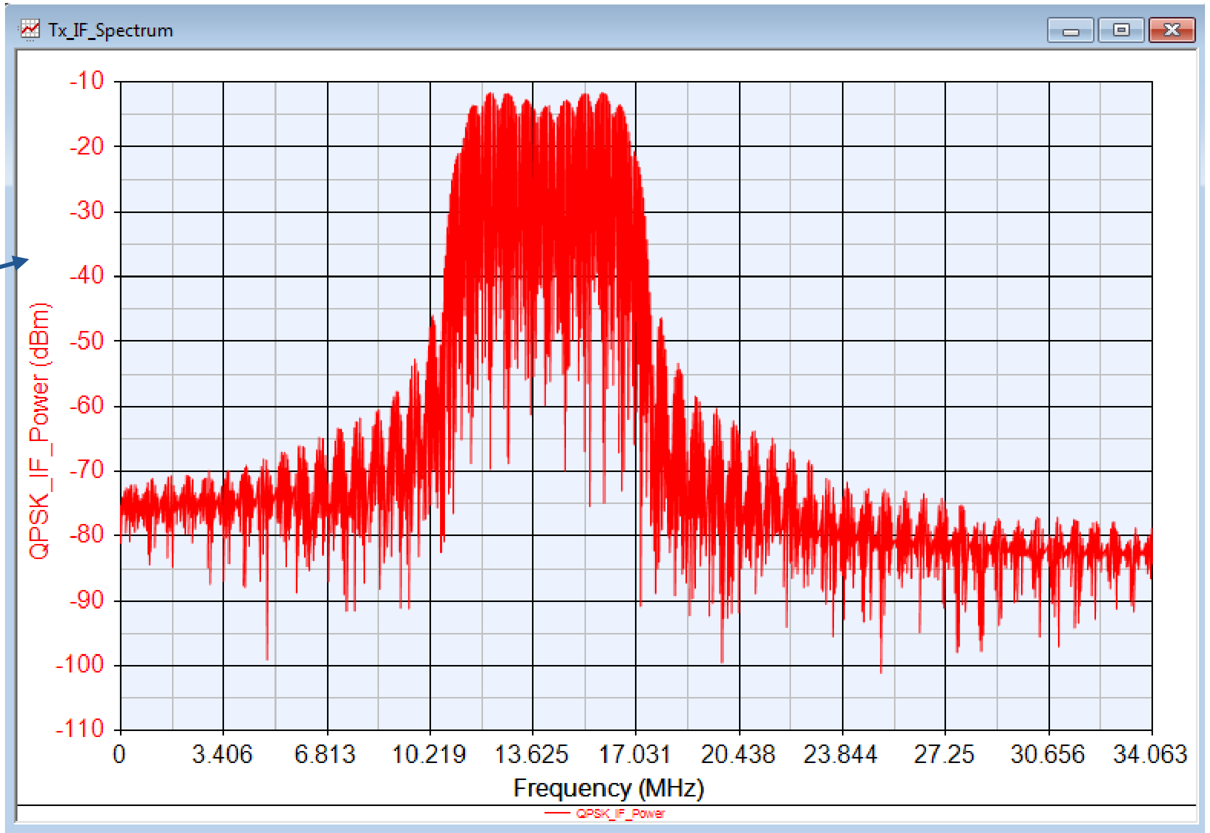
- **Place Spectrum Analyzer and CCDF sink** at the output of the modulator.
- **Double click on Spectrum Analyzer** sink and **name it as QPSK_IF**
- **Double click on the CCDF** sink and set **OutputPeakMean = YES** and name it as **QPSK_IF_CCDF**
- **Double click on Simulation Controller** and set **Number of Samples = 8192** (to get better resolution on the Output Spectrum plot)



Lab 2: Plot spectrum

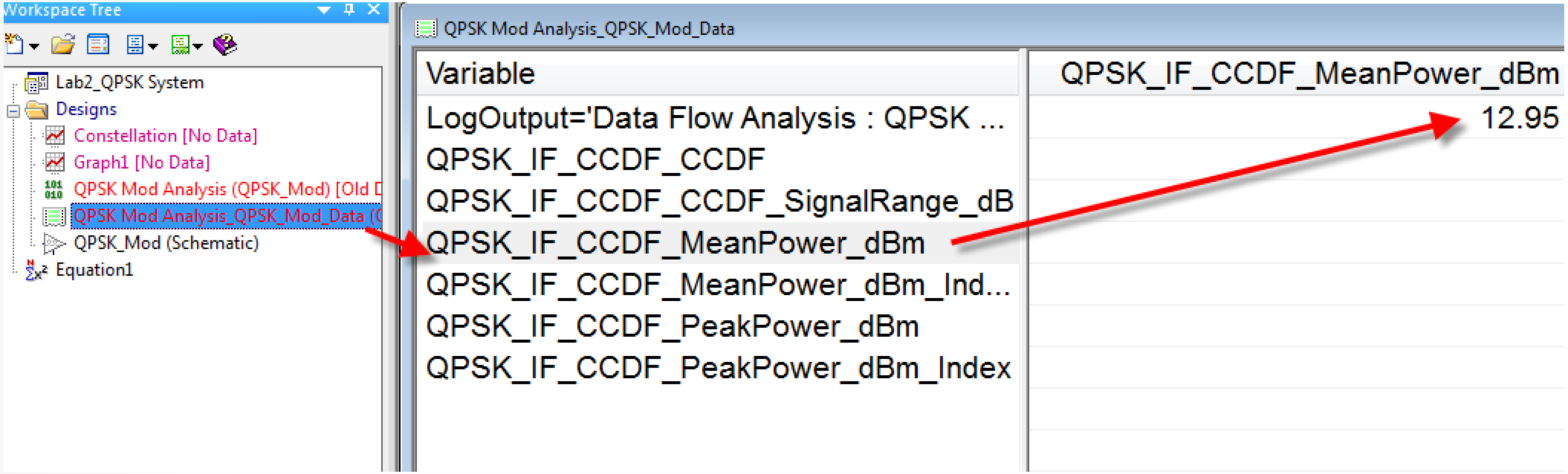
- Run Simulation. Add a new graph, select **Series Type = Spectrum** and select **"QPSK_IF_Power"** from the available measurements
- **Give the Graph name as Tx_IF_Spectrum**





Lab 2: Check MeanPower

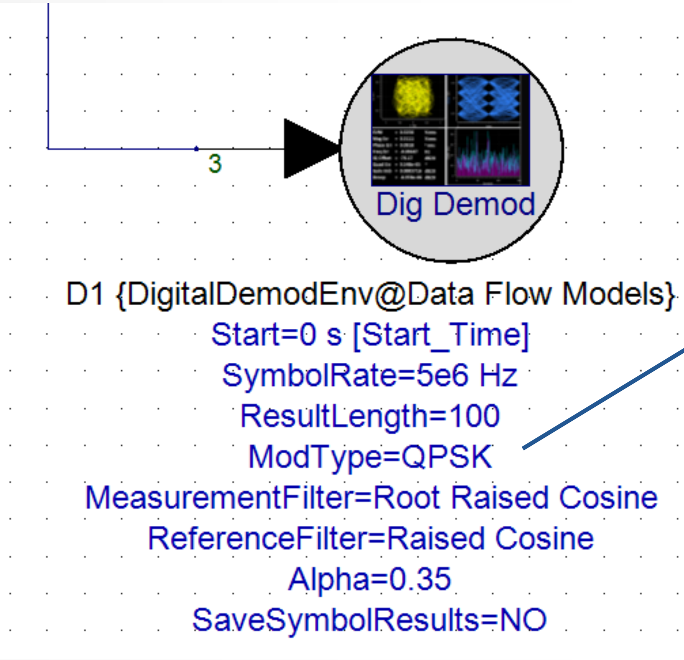
- Check Modulator output power, double click on dataset and look for MeanPower_dBm of the CCDF sink and notice output power is ~13dBm as needed.

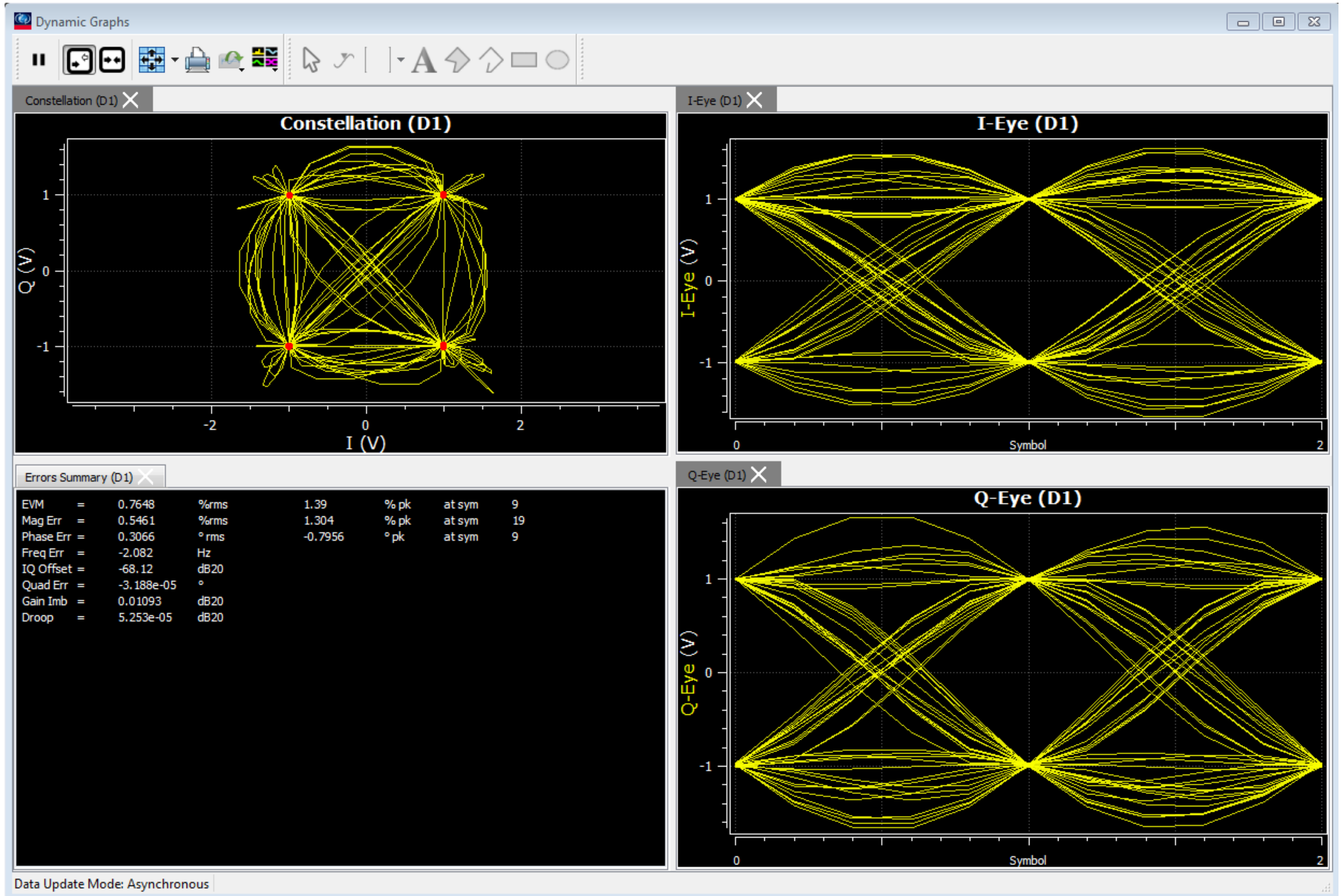


Save the workspace….

Lab 2: Real Time Demod

- Place **Digital Demod** component from Algorithm Design library and connect it to the output of the IQ Modulator.
- Set the parameters as shown here. Run Simulation and observe the results.
- **We can see that our modulator is providing good results with EVM = ~0.75%**

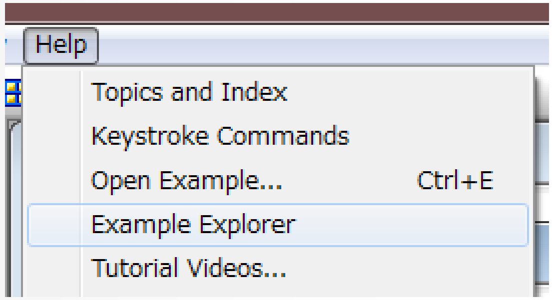


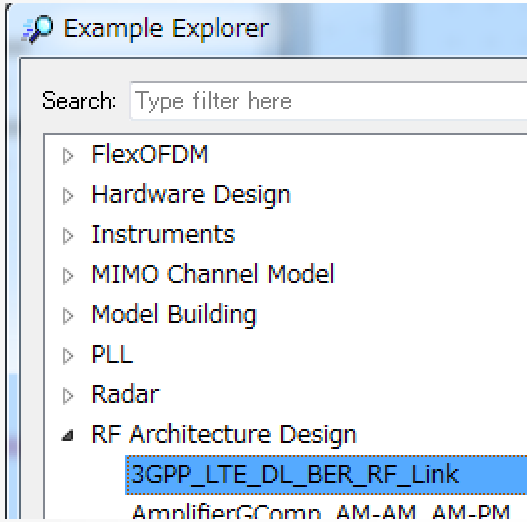


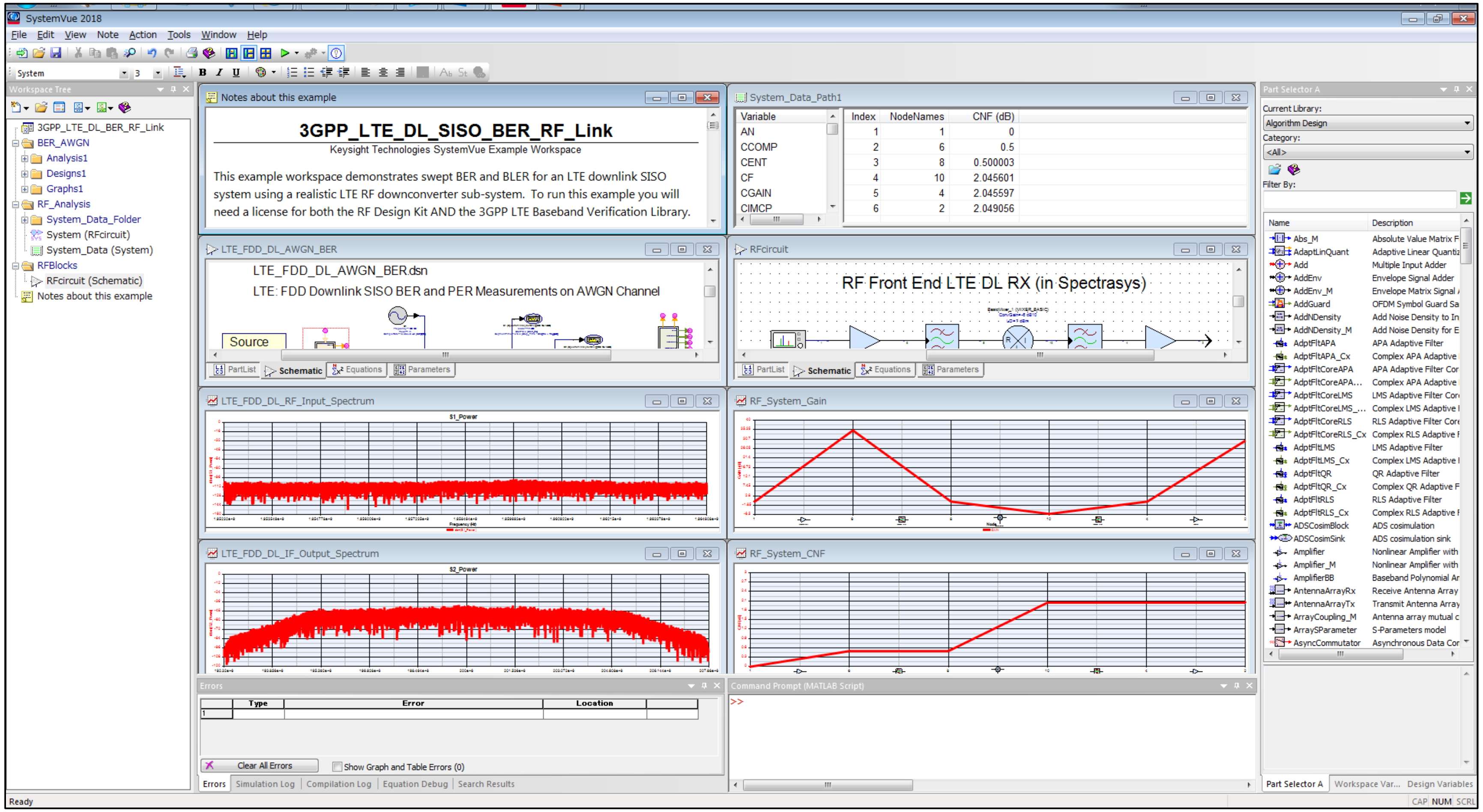
Lab 3: RF design in LTE system

Lab 3: Open 3GPP_LTE_DL_BER_RF_Link.wsv

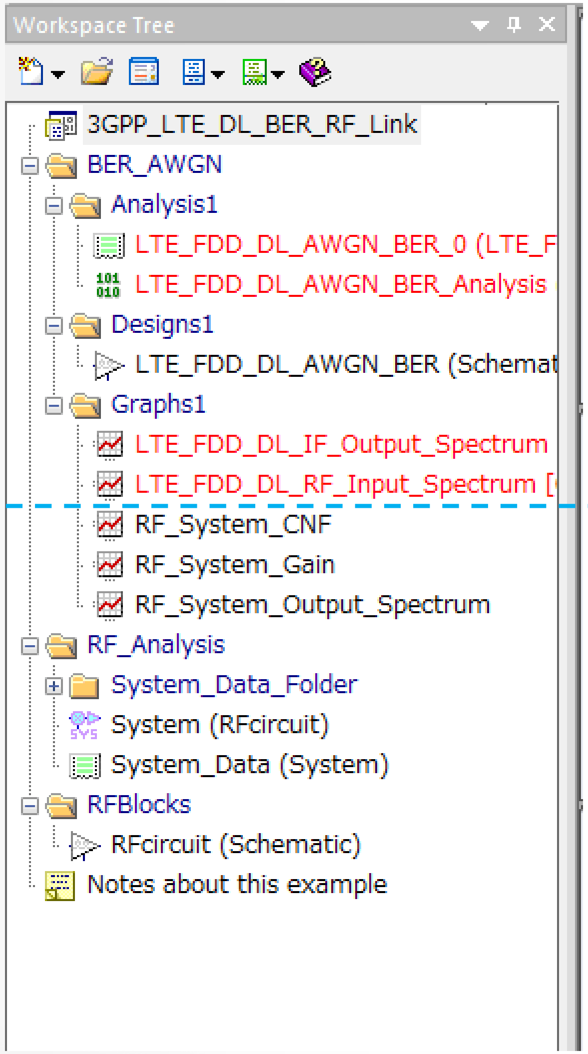
➢ Open "RF Architecture Design > 3GPP_LTE_DL_BER_RF_Link" from Example Explorer







Lab 3: Workspace overview



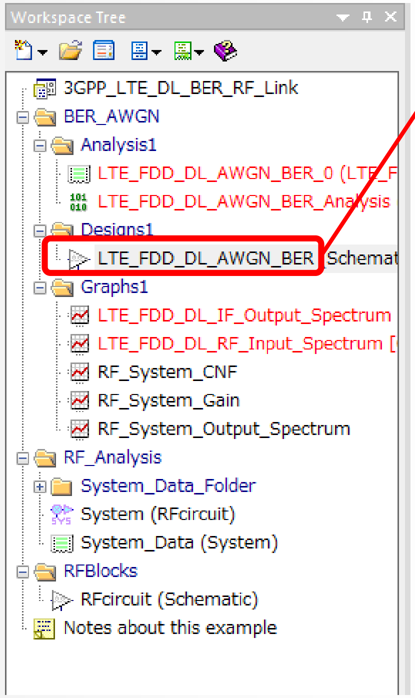
LTE DL BER by
Data Flow simulation

Receiver RF front-end
design by RF simulation

This Lab explains

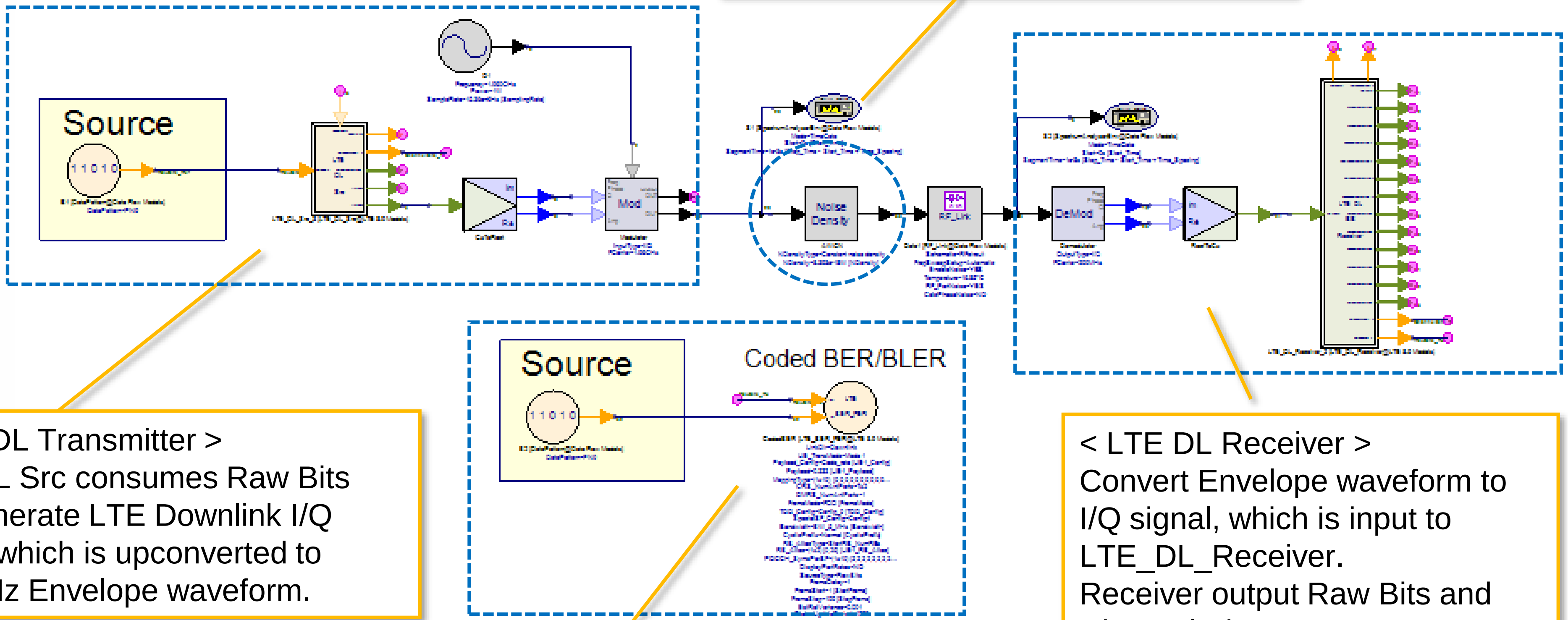
1. Overview of LTE system test bench
2. RF simulation (Spectrasys)
3. LTE system simulation with RF design

Lab 3: LTE_FDD_DL_AWGN_BER



① Open LTE_FDD_DL_AWGN_BER

< Noise >
User can specify SNR (Ndensity)



< LTE DL Transmitter >
LTE_DL Src consumes Raw Bits and generate LTE Downlink I/Q signal, which is upconverted to 1.96GHz Envelope waveform.

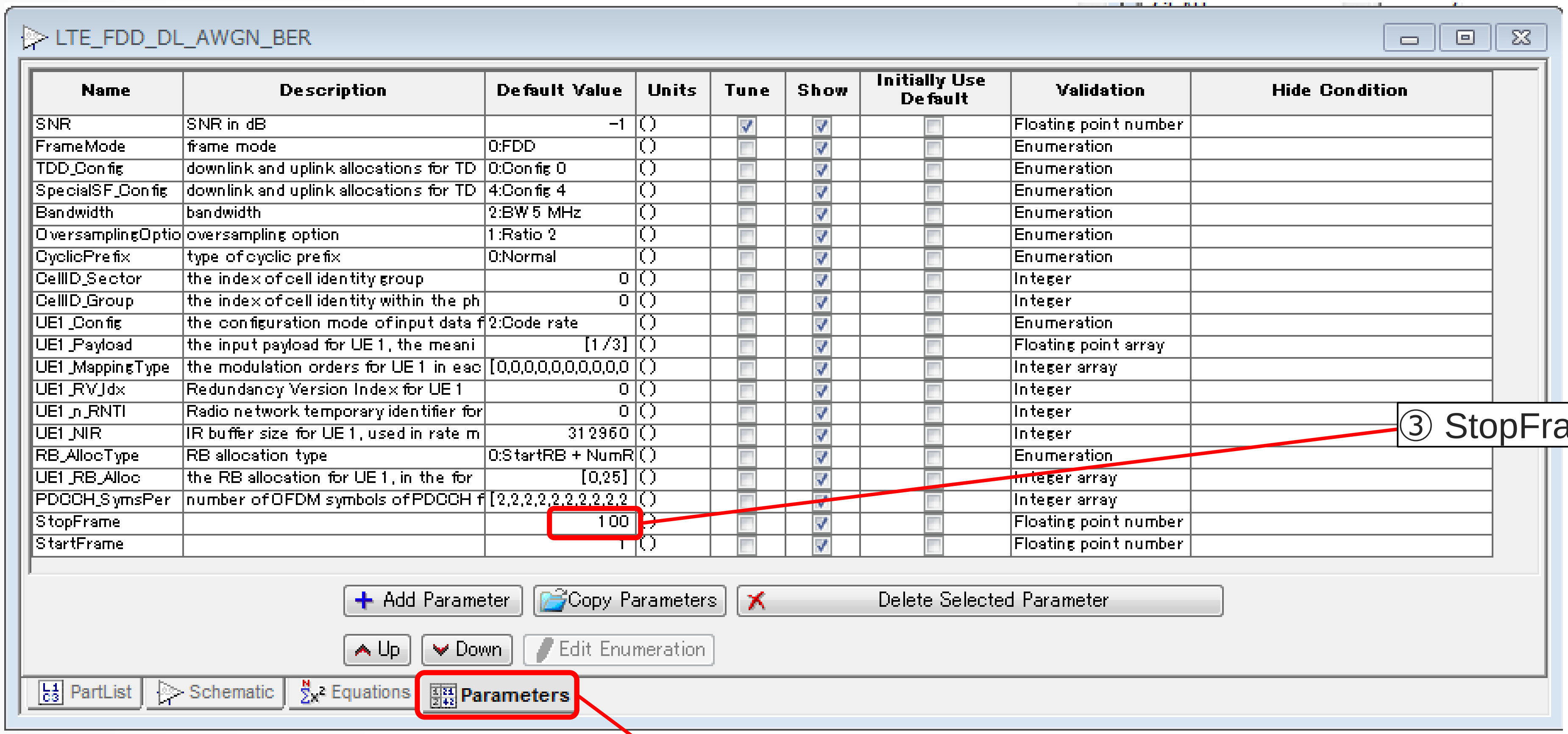
< BER calculation >
Compare Src and Receiver Bits to calculate BER / BLER

< LTE DL Receiver >
Convert Envelope waveform to I/Q signal, which is input to LTE_DL_Receiver.
Receiver output Raw Bits and Channel Bits

KEYSIGHT TECHNOLOGIES

Lab 3: LTE_FDD_DL_AWGN_BER : Parameters

➢ Change the value of “StopFrame” to reduce simulation time.



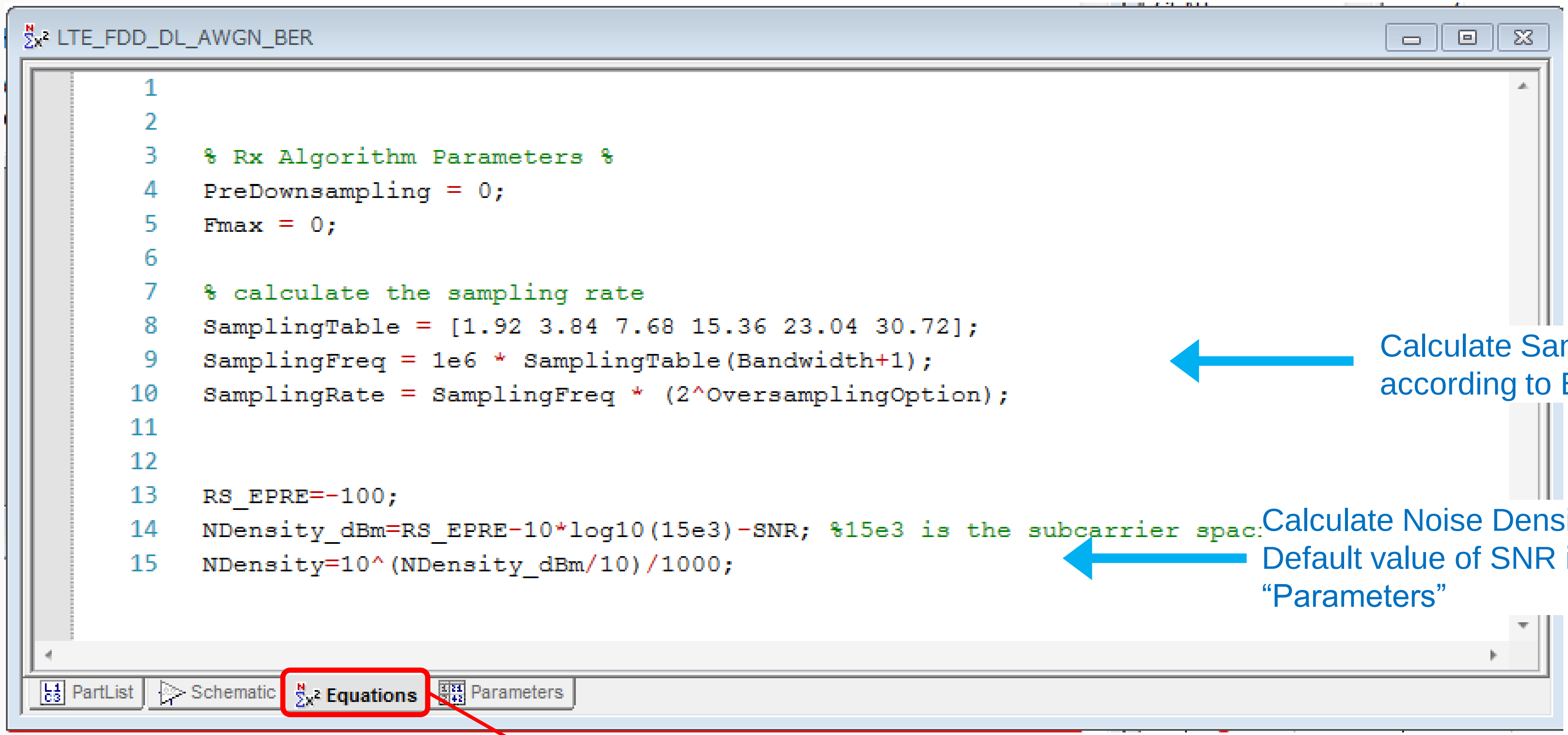
Name	Description	Default Value	Units	Tune	Show	Initially Use Default	Validation	Hide Condition
SNR	SNR in dB	-1	()	☑	☑	☐	Floating point number	
FrameMode	frame mode	0:FDD	()	☐	☑	☐	Enumeration	
TDD_Config	downlink and uplink allocations for TD	0:Config 0	()	☐	☑	☐	Enumeration	
SpecialSF_Config	downlink and uplink allocations for TD	4:Config 4	()	☐	☑	☐	Enumeration	
Bandwidth	bandwidth	2:BW 5 MHz	()	☐	☑	☐	Enumeration	
OversamplingOptio	oversampling option	1:Ratio 2	()	☐	☑	☐	Enumeration	
CyclicPrefix	type of cyclic prefix	0:Normal	()	☐	☑	☐	Enumeration	
CellID_Sector	the index of cell identity group	0	()	☐	☑	☐	Integer	
CellID_Group	the index of cell identity within the ph	0	()	☐	☑	☐	Integer	
UE1_Config	the configuration mode of input data f	2:Code rate	()	☐	☑	☐	Enumeration	
UE1_Payload	the input payload for UE 1, the meani	[1/3]	()	☐	☑	☐	Floating point array	
UE1_MappingType	the modulation orders for UE 1 in eac	[0,0,0,0,0,0,0,0,0,0	()	☐	☑	☐	Integer array	
UE1_RV_Idx	Redundancy Version Index for UE 1	0	()	☐	☑	☐	Integer	
UE1_n_RNTI	Radio network temporary identifier for	0	()	☐	☑	☐	Integer	
UE1_NIR	IR buffer size for UE 1, used in rate m	312960	()	☐	☑	☐	Integer	
RB_AllocType	RB allocation type	0:StartRB + NumR	()	☐	☑	☐	Enumeration	
UE1_RB_Alloc	the RB allocation for UE 1, in the for	[0,25]	()	☐	☑	☐	Integer array	
PDCCH_SymsPer	number of OFDM symbols of PDCCH f	[2,2,2,2,2,2,2,2,2,2	()	☐	☑	☐	Integer array	
StopFrame		100	()	☐	☑	☐	Floating point number	
StartFrame		1	()	☐	☑	☐	Floating point number	

③ StopFrame: 100 → 10

② Select “Parameters”

Lab 3: LTE_FDD_DL_AWGN_BER : Equations

- Check equations in “Equations” for SamplingRate or Ndensity
- User can use “Equations” to define parameters instead of “Parameters”



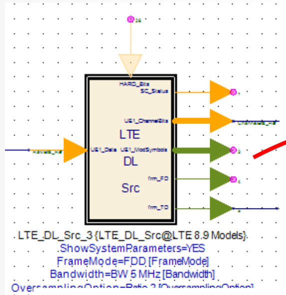
Calculate Sampling Rate according to Bandwidth

Calculate Noise Density by SNR
Default value of SNR is defined at “Parameters”

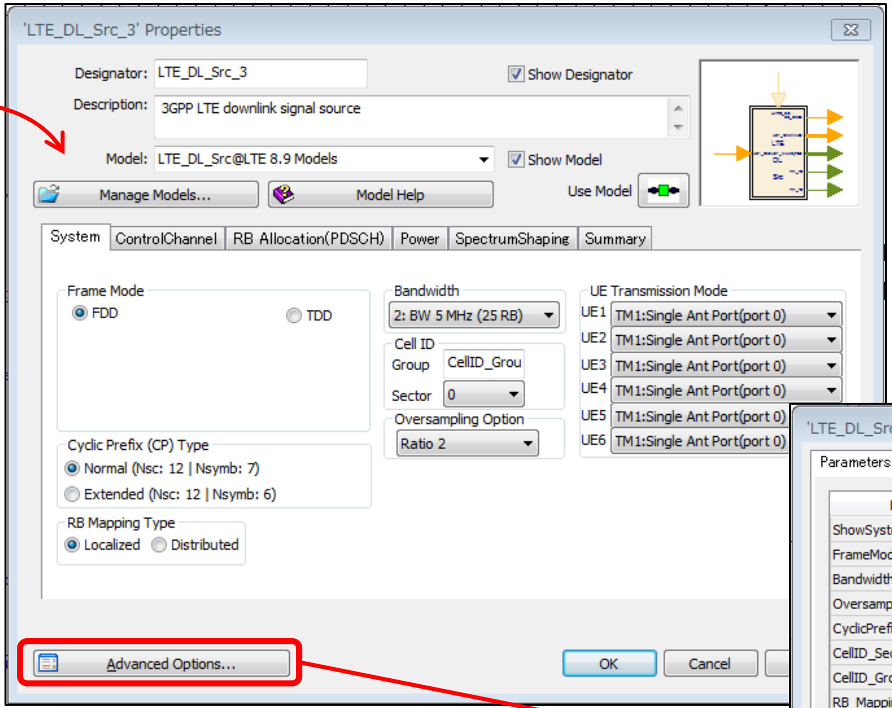
④ Equationsタブを開く

Lab 3: Source, Receiver model

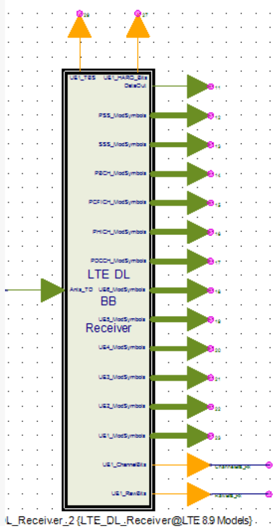
- This example uses Source/Receiver model in W1910 : LTE baseband verification library, which user can set parameters by GUI.



LTE_DL_Src

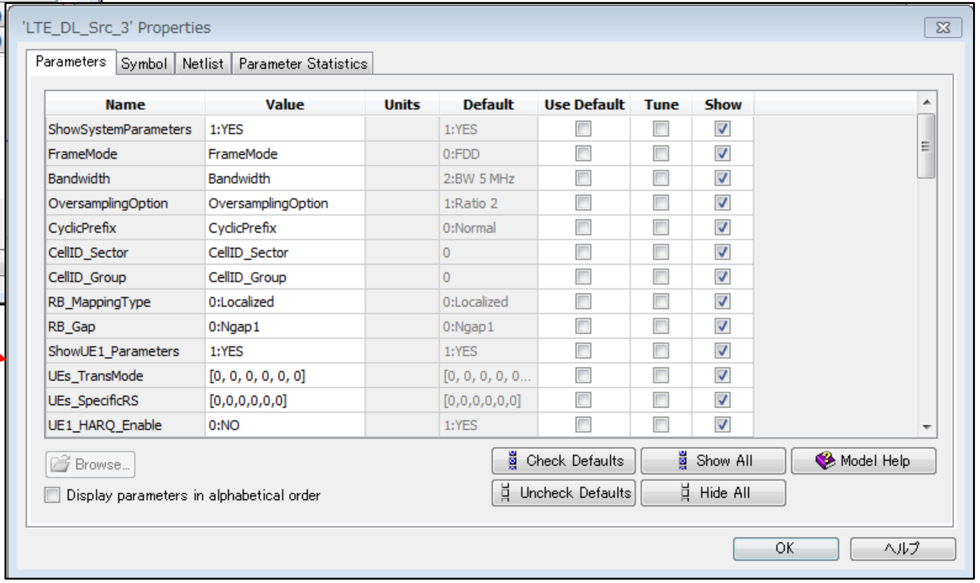


GUI has multiple tabs



LTE_DL_Receiver

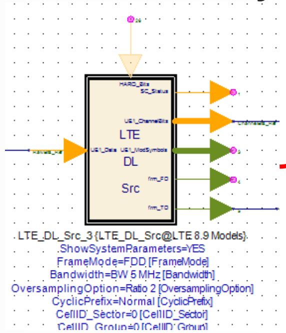
[Advanced Options..] button shows all parameters in a list



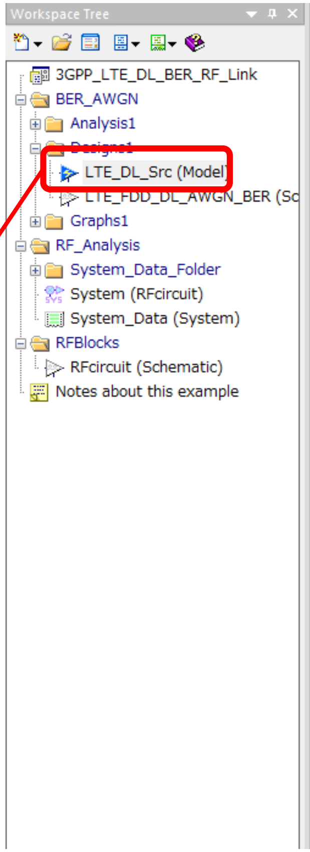
KEYSIGHT TECHNOLOGIES

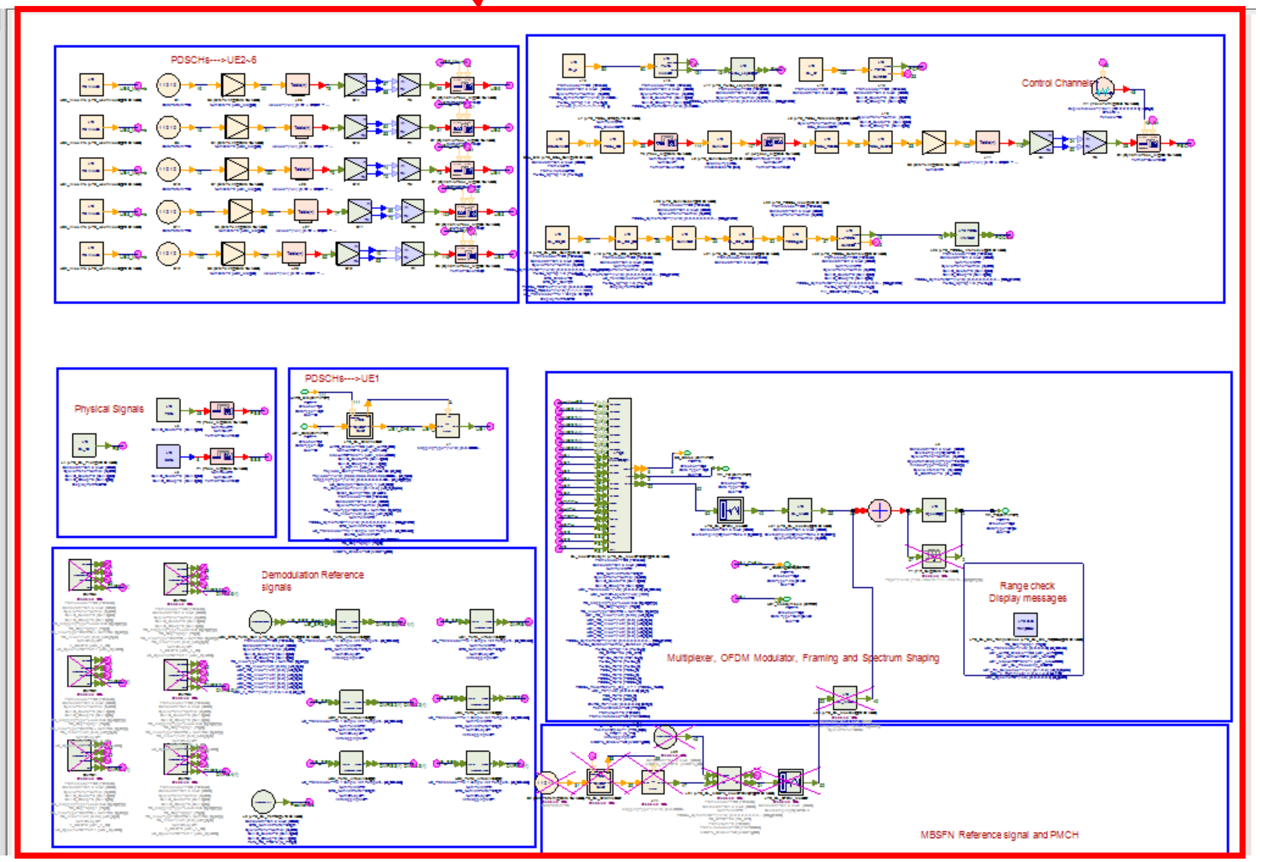
【Ref.】How to open sub-circuit

- If a model has sub-circuit, i.e. composed by other components, user can open the schematic by **"Right mouse click > Open > Model/Subcircuit"**



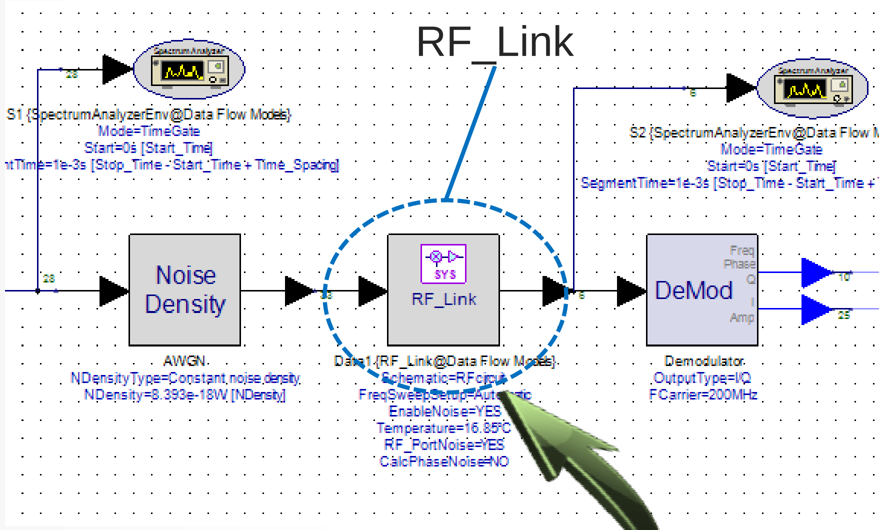
LTE_DL_Src



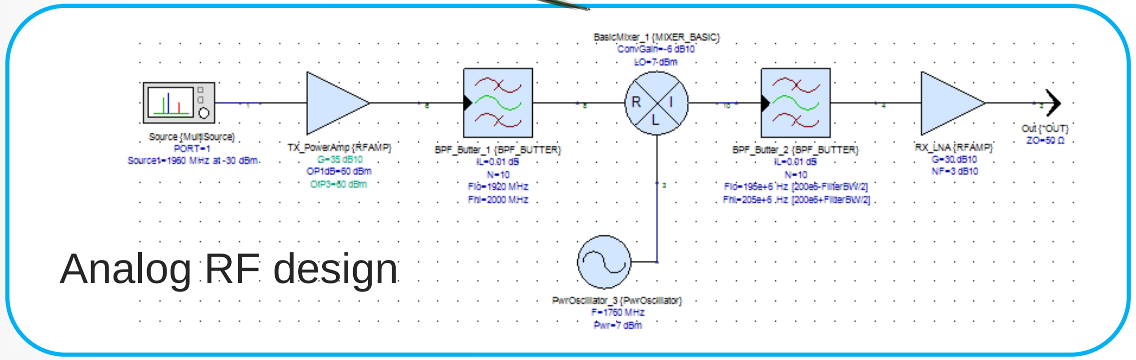


Add sub-circuit icon at workspace tree automatically

Lab 3: RF_Link component



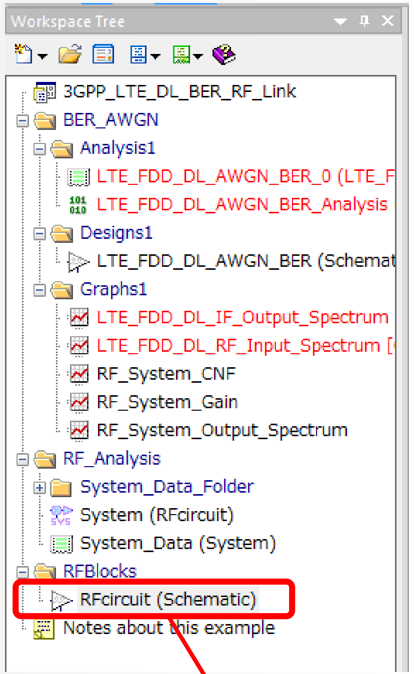
- ➢ RF_Link component connects Data Flow and RF front-end design that consists of parts in “RF Design” library.
- ➢ User can take RF model effect into account in LTE system .



Next page:
Learn how to simulate this RF design (Receiver block down converter)

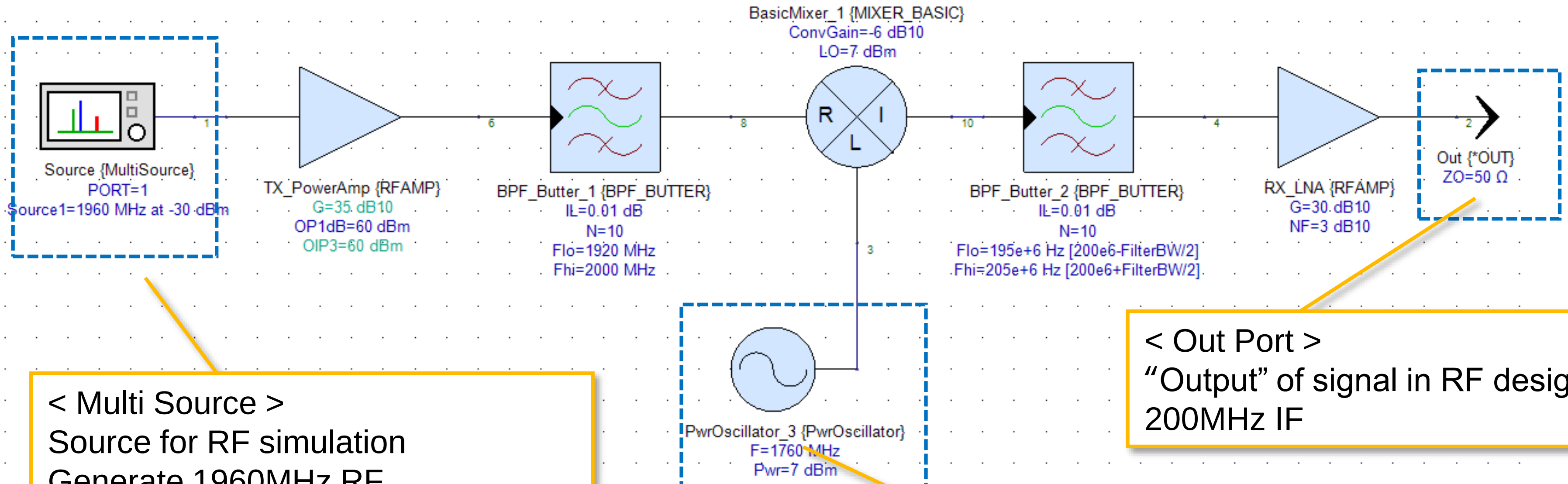
RF simulation (Spectrasys)

Lab 3: RFcircuit(design) overview



⑤ Open RFcircuit

➢ This design that consists of parts in “RF Design” library is block down converter from 1960MHz RF to 200MHz IF.



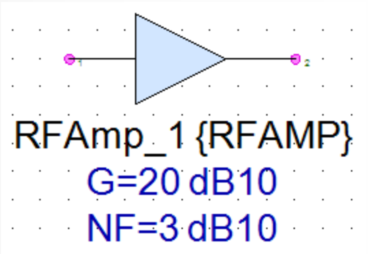
< Multi Source >
Source for RF simulation
Generate 1960MHz RF

< Power Oscillator >
1760MHz LO source
IF is LSB(200MHz) of Mixer output

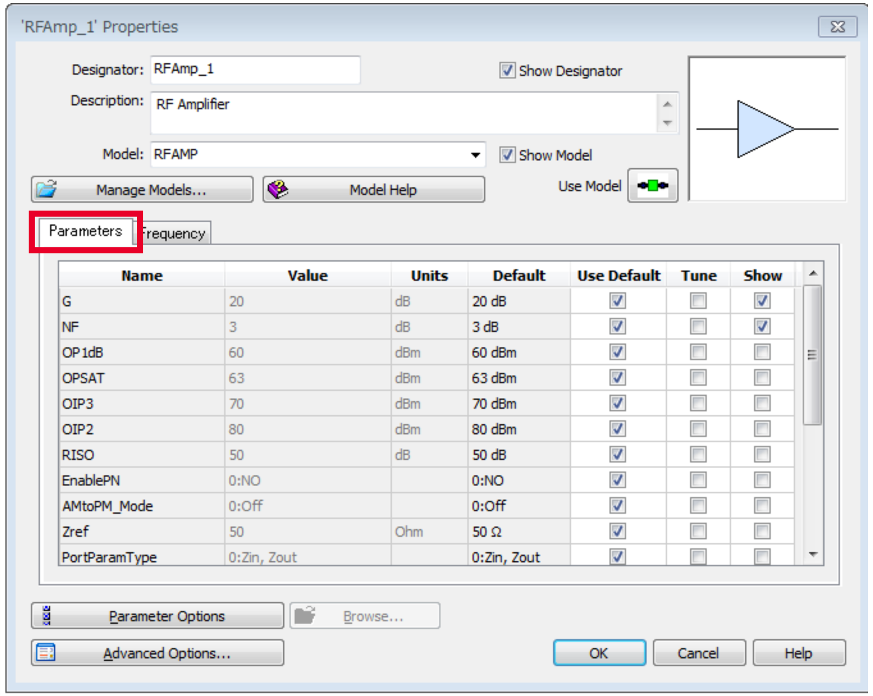
< Out Port >
“Output” of signal in RF design.
200MHz IF

Lab 3: Model in RF simulation vs Data Flow

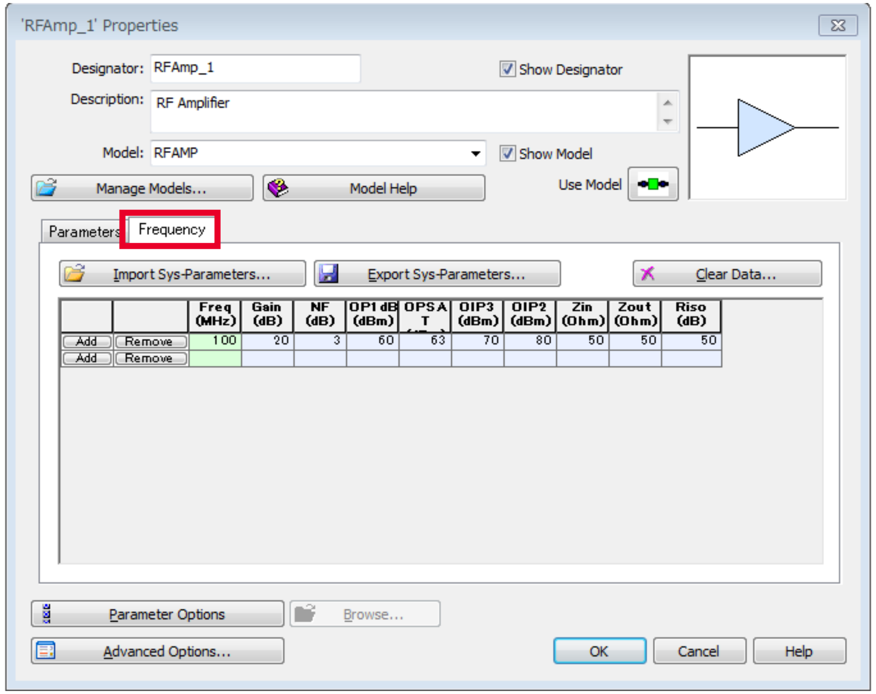
RF Amplifier



- Must set loss or nonlinearity (e.g. P1dB, Psaturation, OIP2/3, etc)
- Input/Output impedance to consider mismatch effect
- User can set frequency dependency at “Frequency” (can import/export CSV file)



RF Amplifier property



“Frequency” tab

KEYSIGHT TECHNOLOGIES

Lab 3: Node Measurements and Path Measurements

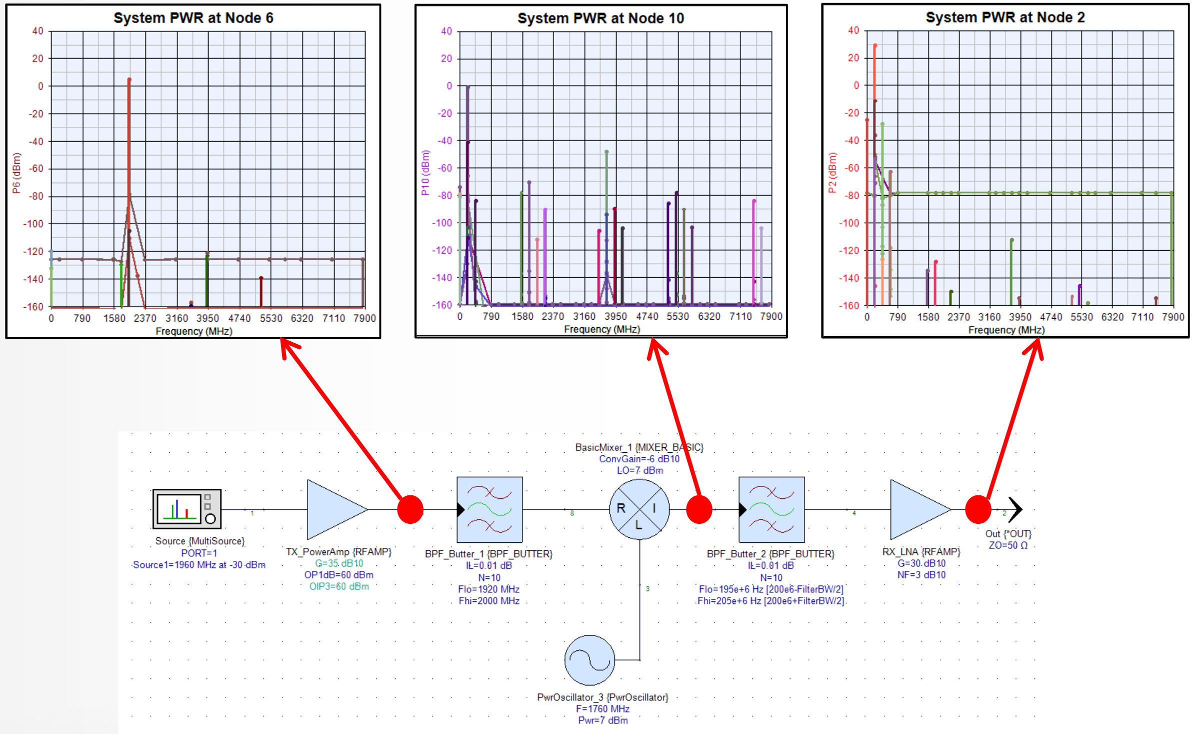
Node Measurements

- Power/Voltage vs Frequency at each nodes

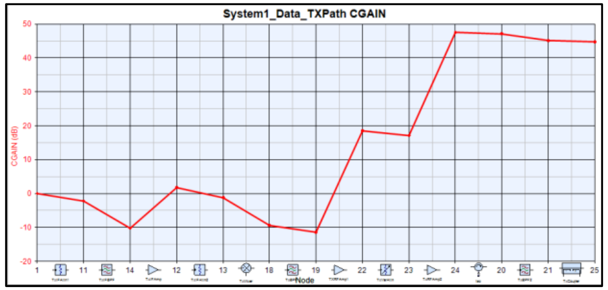
Path Measurements

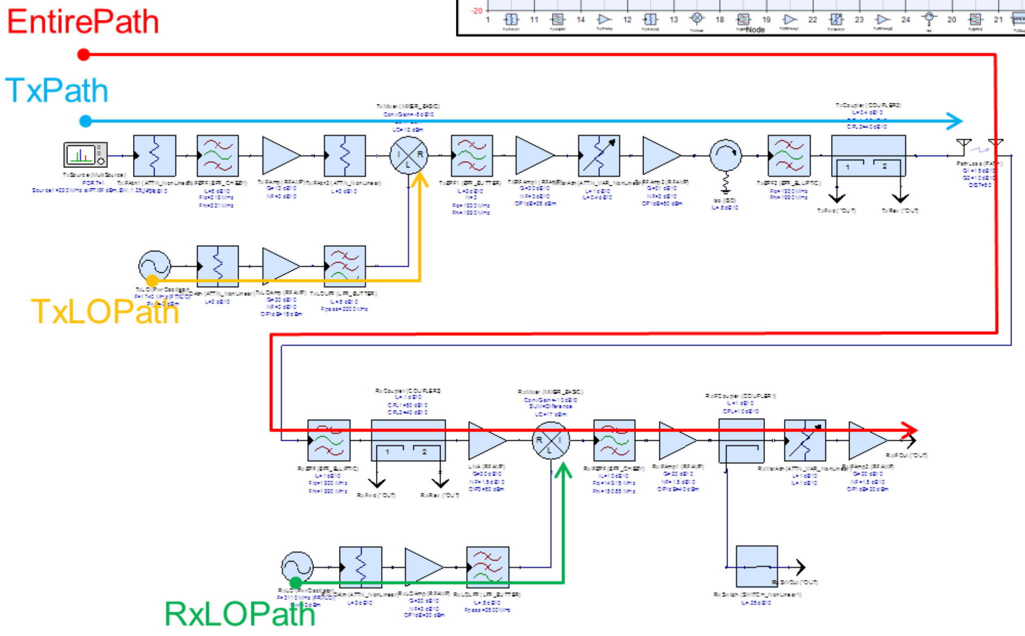
- Level diagram of Channel Power, Noise, InterMod, etc. at specified path.

Spurious analysis at each Node



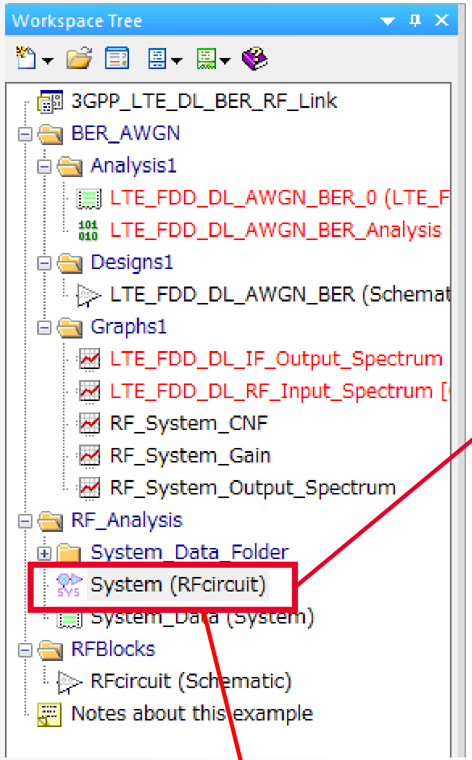
Level diagram of Tx Path (Cascaded Gain)



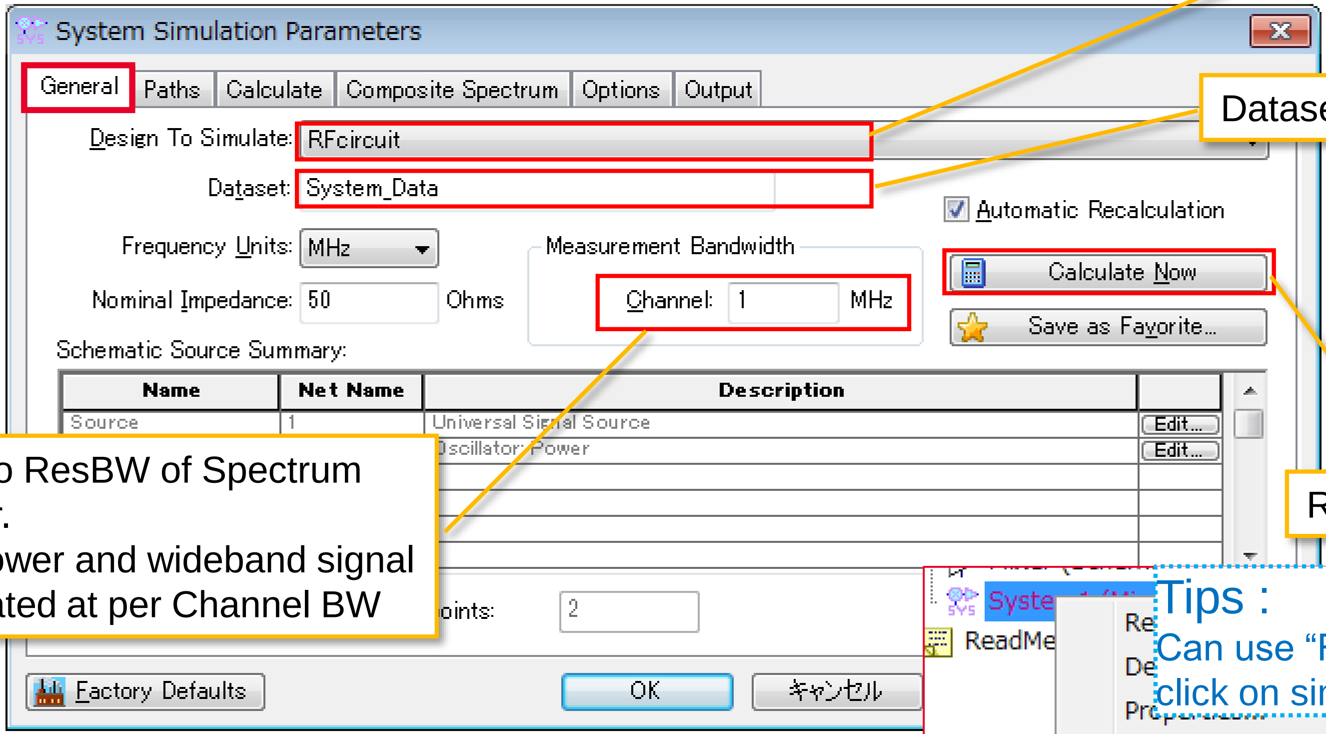


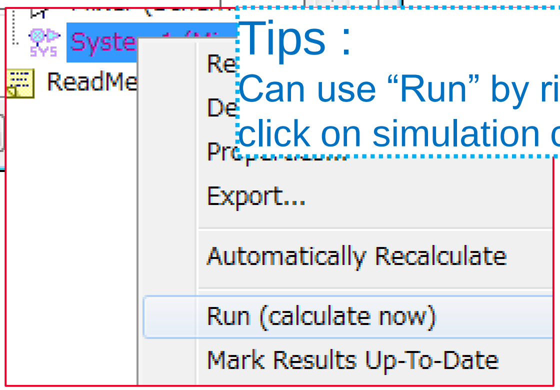
Lab 3: RF System Analysis controller : General

Set Design and Dataset



⑥ Open System controller





Select design(Schematic) to simulate

Dataset name

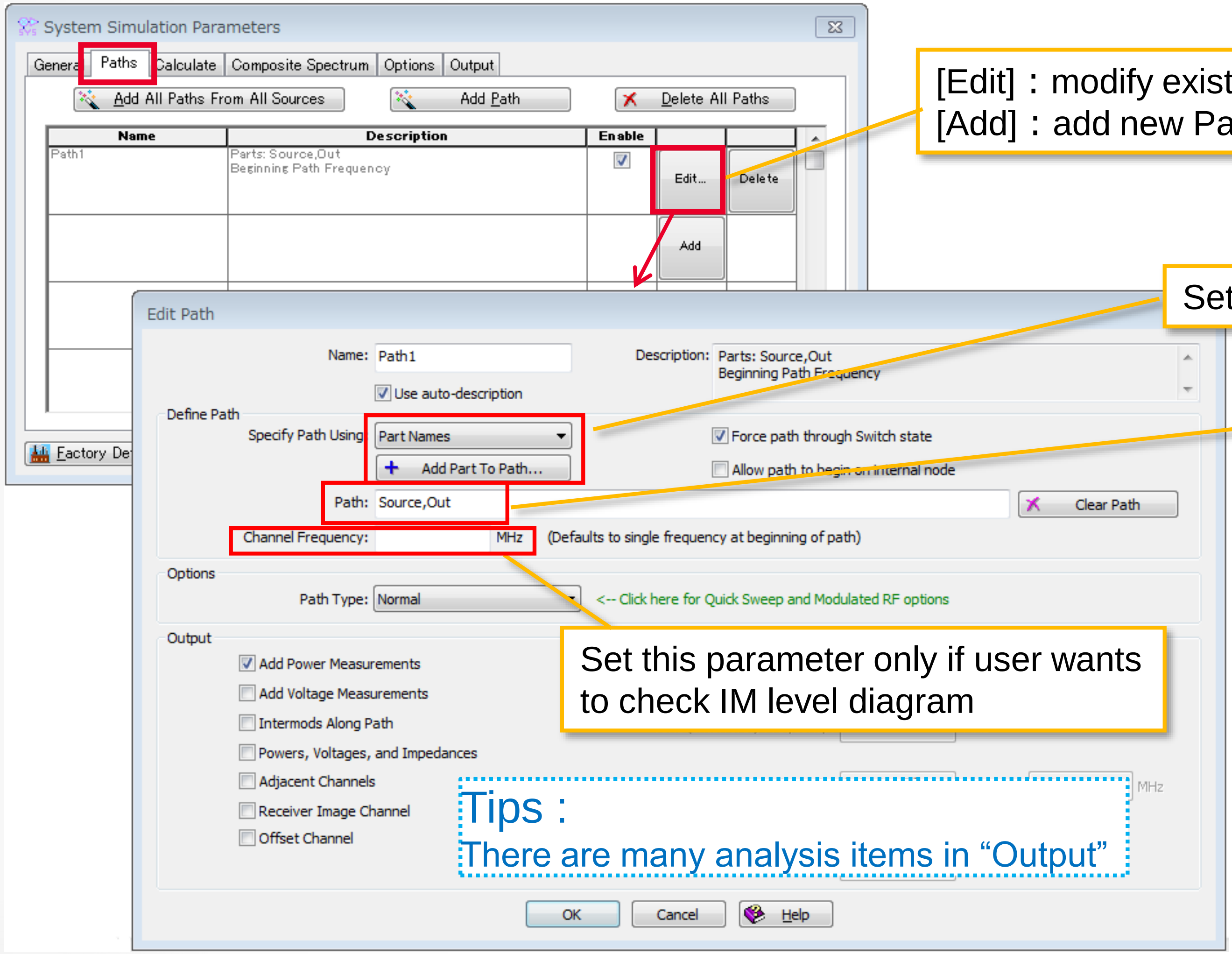
Run simulation

Similar to ResBW of Spectrum Analyzer.
Noise power and wideband signal is estimated at per Channel BW

Tips :
Can use “Run” by right mouse click on simulation controller

Lab 3: RF System Analysis Controller：Paths

Set Path to calculate Level Diagram



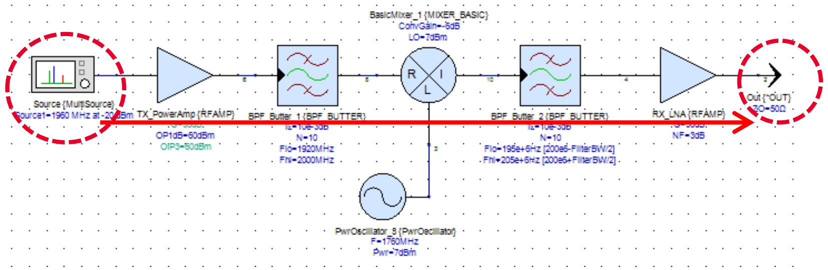
[Edit]：modify existed Path
[Add]：add new Path

Set the start/end point of path

Current Path : Source → Out

Set this parameter only if user wants to check IM level diagram

Tips :
There are many analysis items in “Output”



Lab 3: Run simulation and plot graph

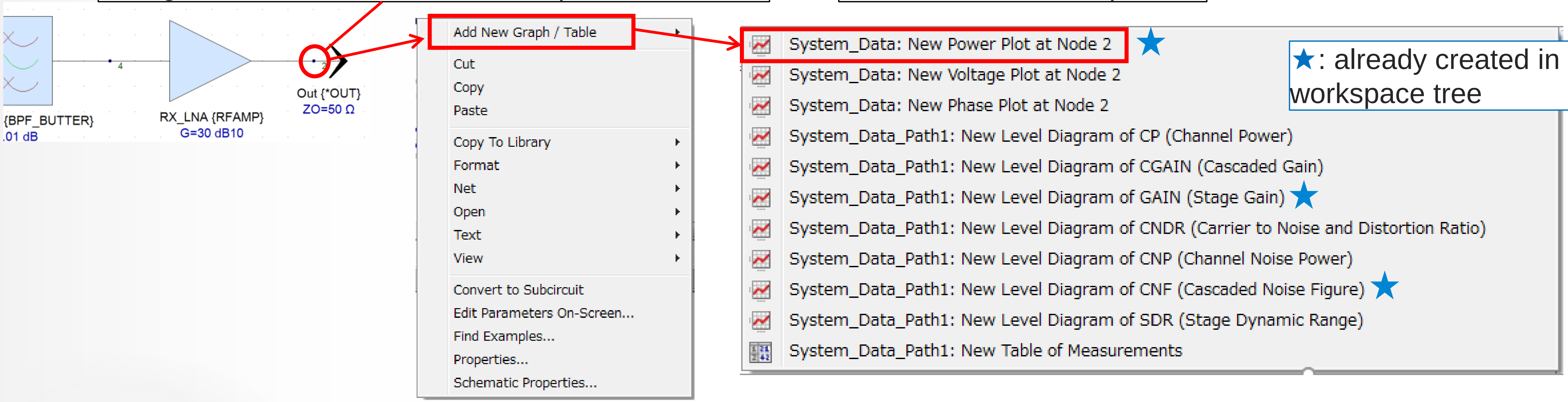
⑦ Run Simulation   Push [Calculate Now] button to run RF simulation

➢ Plot graph

After simulation finished, user can create plot from schematic node directly.
Mouse cursor is changed to ⌐ around the node, then Right mouse click can show the menu “Add New Graph/Table” and the items to be plotted.

⑧ Right mouse click > Add New Graph / Table

⑨ Select items to be plotted

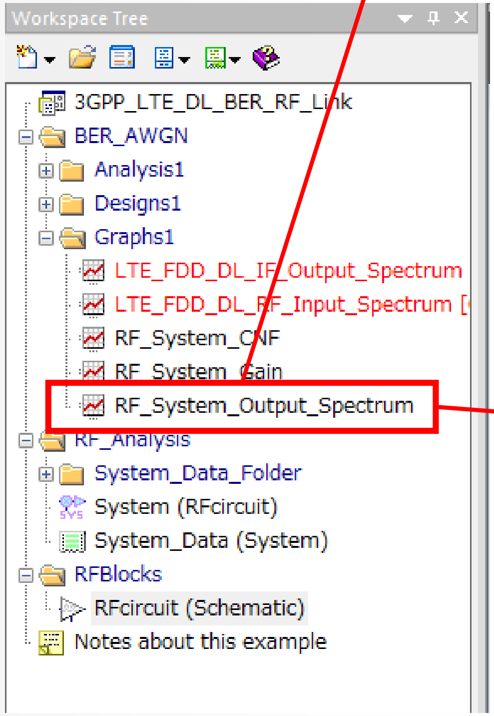


★: already created in workspace tree

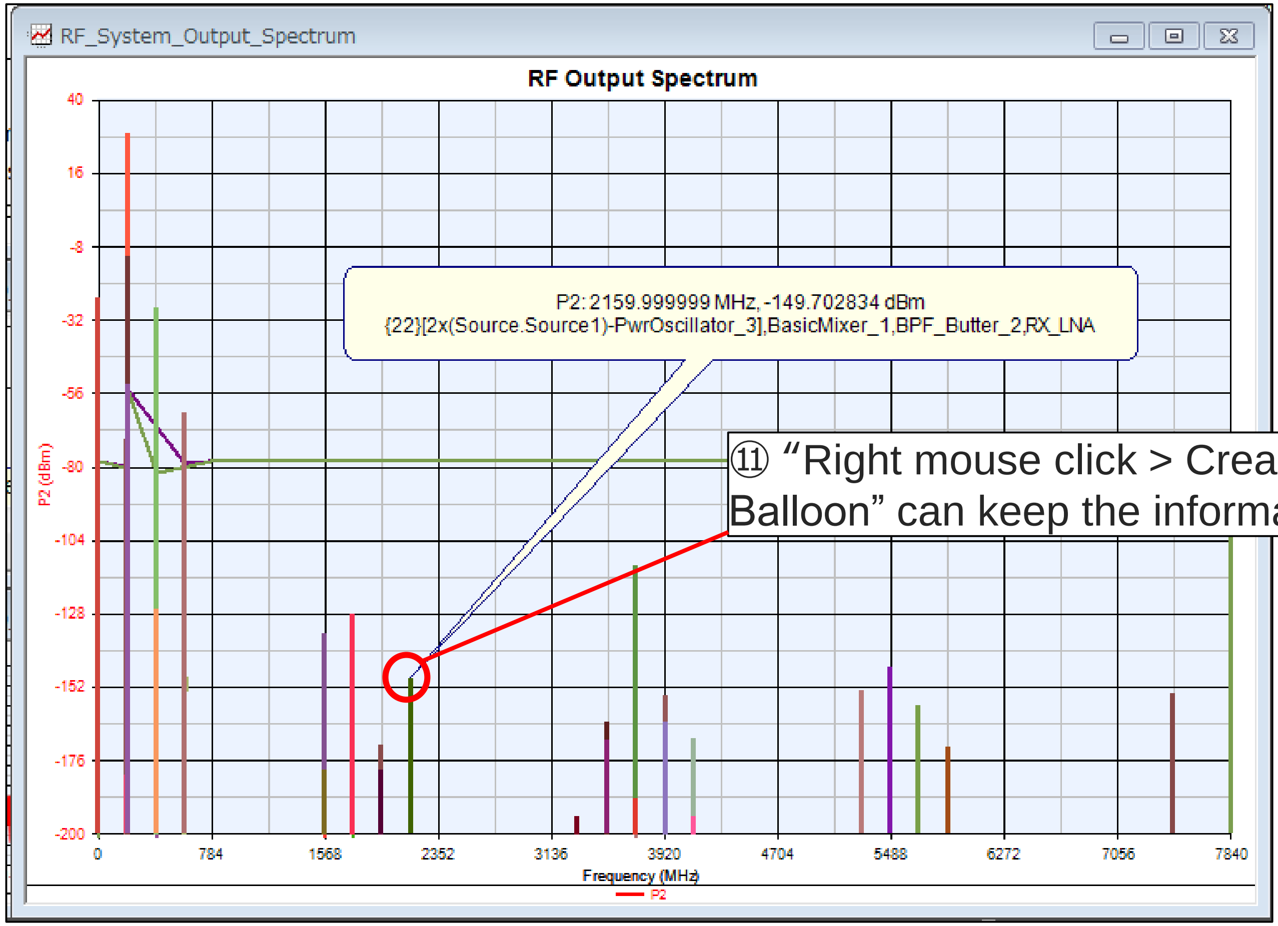
KEYSIGHT TECHNOLOGIES

Lab 3: Node Measurement : RF_System_Output_Spectrum

⑩ Open RF_System_Output_Spectrum



Point by mouse at each spectrum, then show the power and root-cause(path and the mixing order(Frequency Equation))



⑪ “Right mouse click > Create Info Balloon” can keep the information

【Ref.】Format of Root Cause analysis

Node Measurement shows root-cause of each spectrum.
This information is described by following format.

Format of Spectrum info.

{Coherency Number} Signal Type [Frequency Equation], Origin Part, Next Part, ..., Current Part

(#)

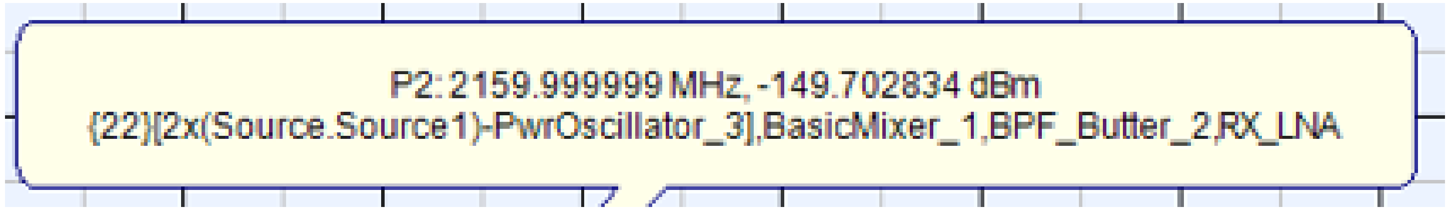
D: Desired Signal
None: others

[2x (MultiSource)]

Path of the spectrum generation

(#)All signal of Spectrasys is grouped by coherency number. Coherent means “using same reference clock”. Different reference clock defines non-coherent even if spectrums are same frequency.

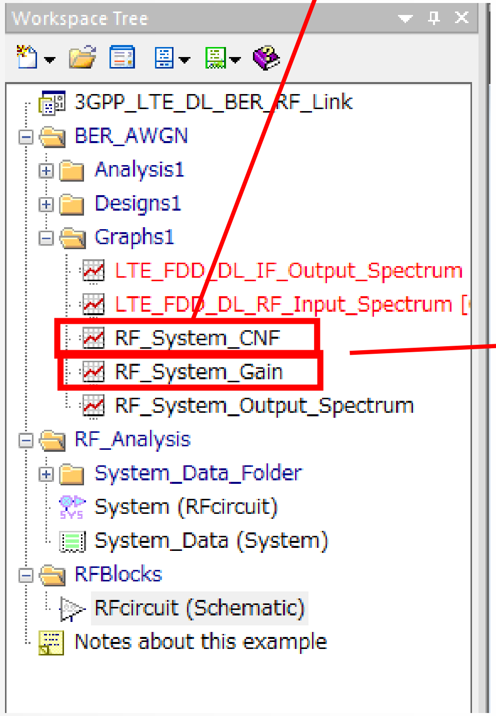
E.g. 2160MHz at previous page



Signal Type	: Not D → Spurious
Frequency Equation	: 2xRF - LO
Origin and route	: BasicMixer → BPF_Butter_2 → RX_LNA → Out

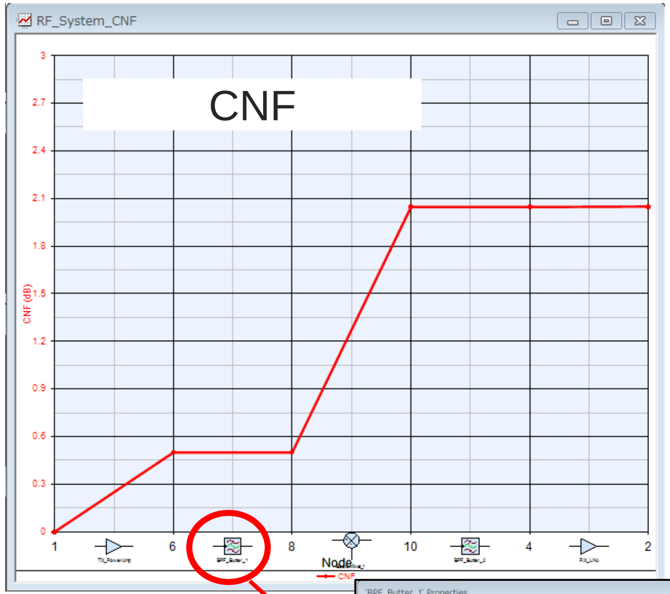
Lab 3: Path Measurement : System_CNF & System_Gain

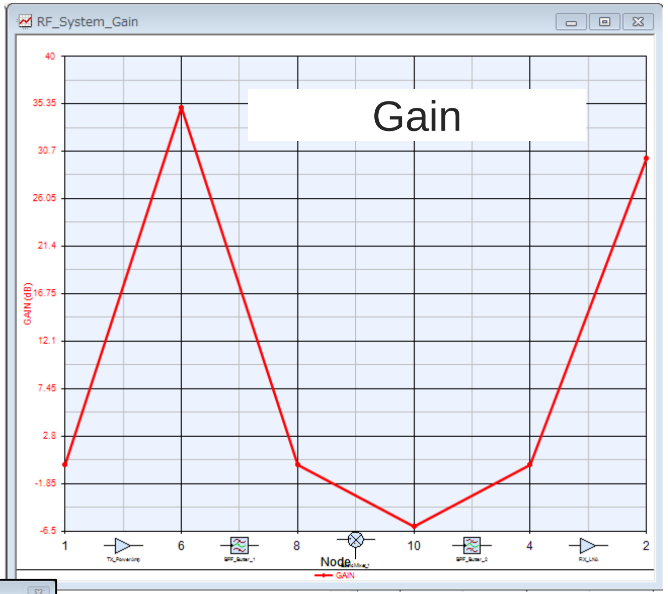
⑭ Open RF_System_CNF and RF_System_Gain



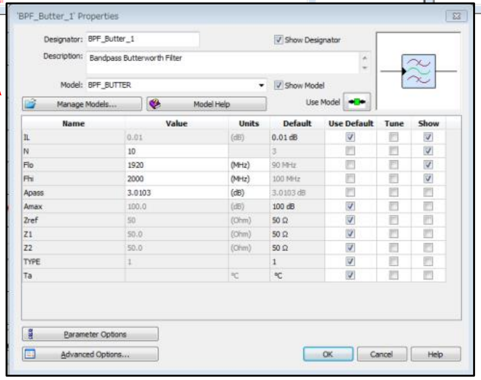
Path Measurement shows level diagram along the specified path: Source → Out

Followings are Cascaded Noise Figure and Stage Gain plot.



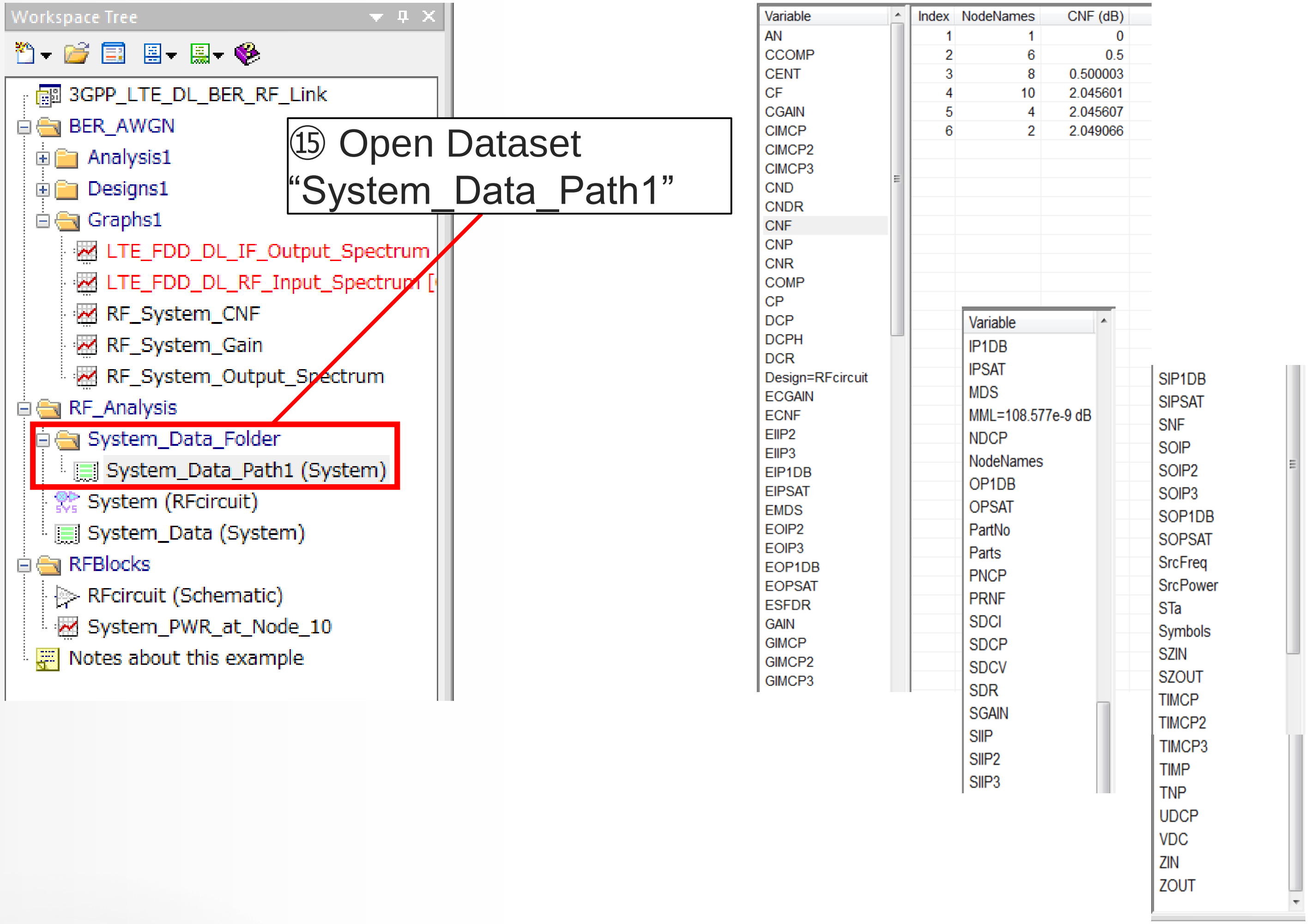


User can check/modify parameters of each component directly by double clicking the icon instead of returning to schematic.



Lab 3: Dataset of Path Measurements

Dataset of Path Measurements is created separately from the Dataset of Node Measurements



Examples of Variables included in Path Measurements

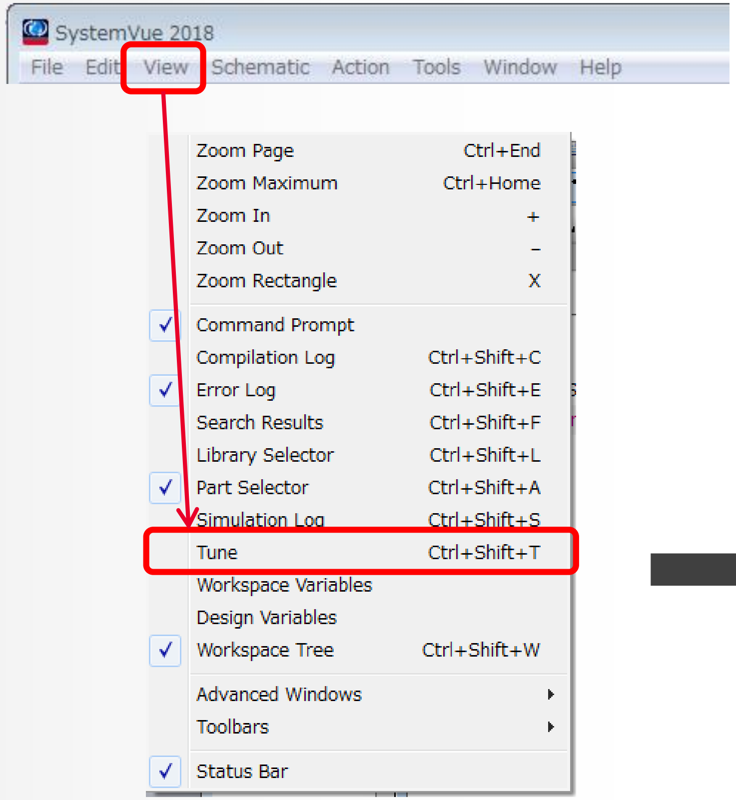
CGAIN	Cascaded Gain
CND	Channel Noise Density(1Hz当たりのCNP)
CNF	Cascaded Noise Figure
CNP	Channel Noise Power
CNR	Carrier to Noise Ratio
CP	Channel Power
DCP	Desired Channel Power
ECGAIN	Equation baseのCascaded Gain
ECNF	Equation baseのCascaded Noise Figure
PNCP	Phase Noise Channel Power
SDR	Stage Dynamic Range (SIP1DB - TNP)
SGAIN	Stage Gain
SIP1DB	Stage Input 1dB Compression Point
SNF	Stage Noise Figure
TNP	Total Node Power

User can check the Variables at Help manual.
by searching keyword “Path Measurement”

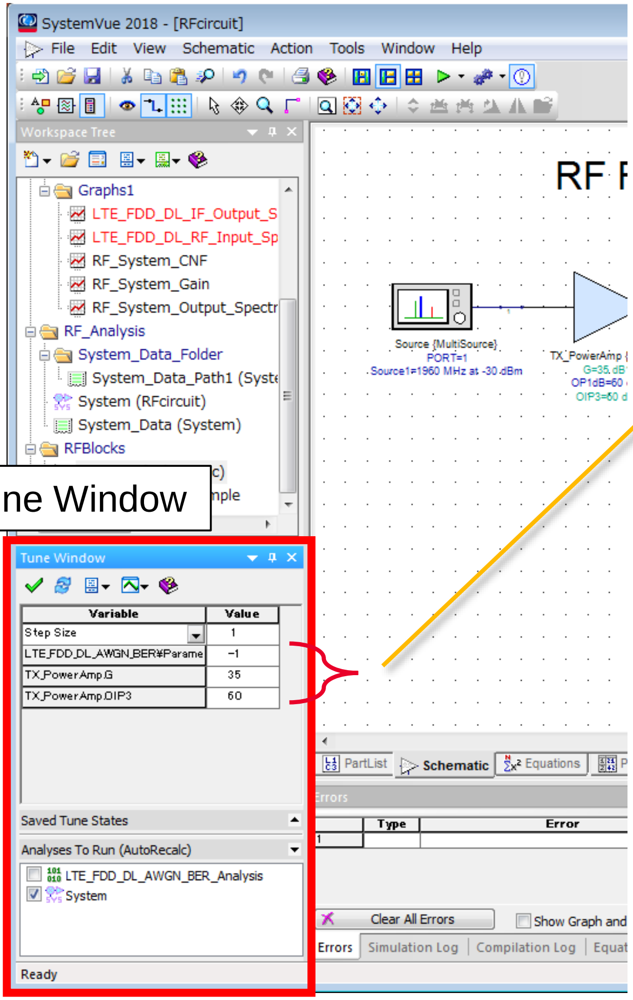
Lab 3: Tune parameter

- "Tune" parameter can be used for both Data Flow and RF simulation.
- "Tune" G and OIP3 in the RFcircuit, then observe CNF or Stage Gain

Check "View > Tune" to show Tune Window

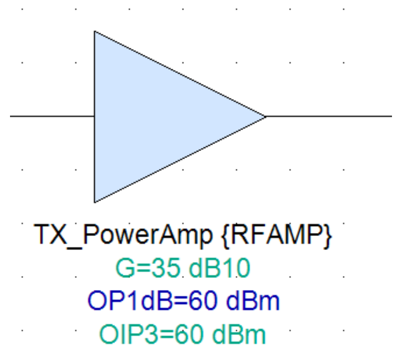


Tune Window



Tunable parameters in whole workspace are listed

SNR : N/A at this time (it is for Data Flow design)
G : TX_PowerAmp > Gain
OIP3 : TX_PowerAmp > Output Intercept Point



Continue to the next page

KEYSIGHT TECHNOLOGIES

Lab 3: Tune parameter

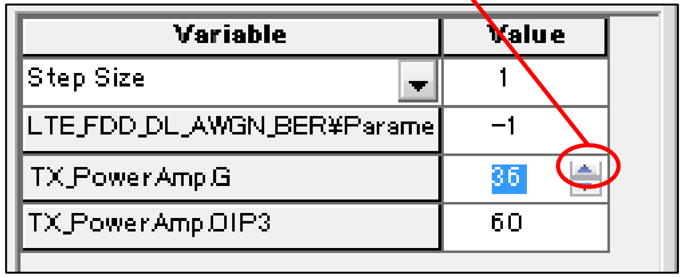
Apply tuned value to the plot

Refresh tuned value

Config menu of Tune Window

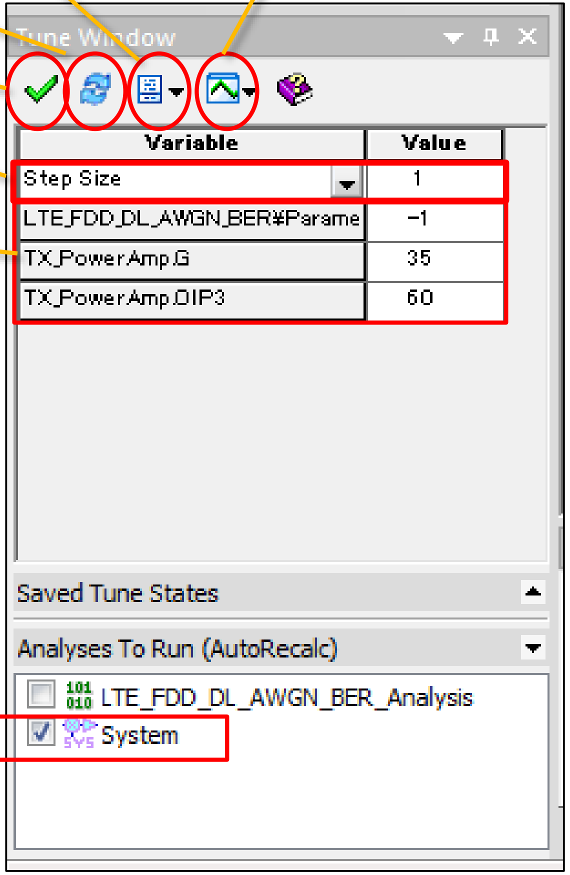
Select plot to show Check Point

- Input the value directly
- Click upper/lower icon (to change by Step Size)



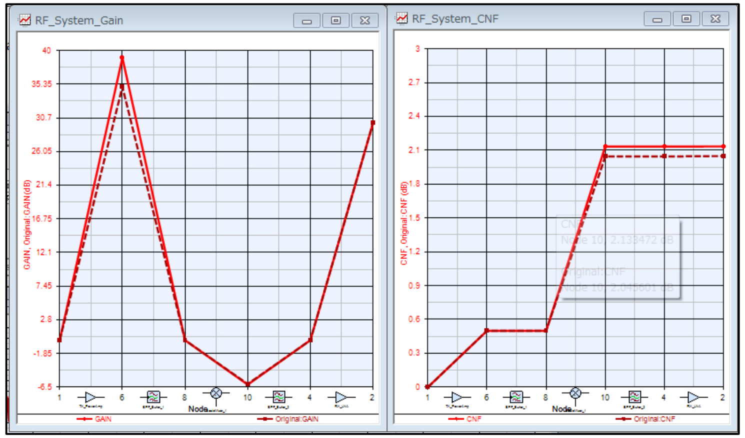
Step size of Tune

Parameter name and its value



User can add any parameters by checking “Tune” at component property or Config menu

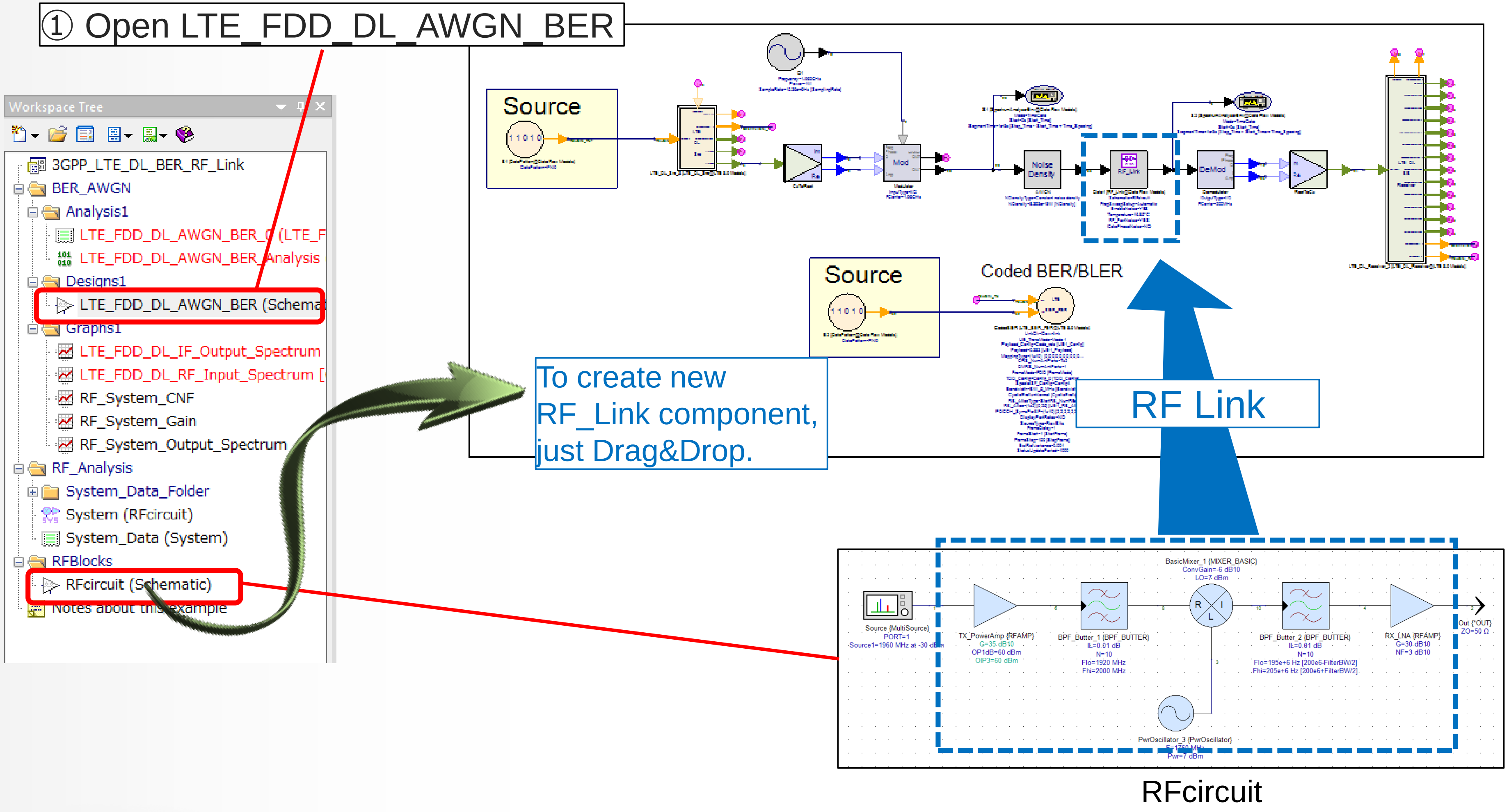
During Tune, level diagram shows “before/after” lines.



Check to run simulation automatically during Tune

RF Link

Lab 3: How to use “RF_Link”



To consider RF front-end design in the system, user can use RF_Link.

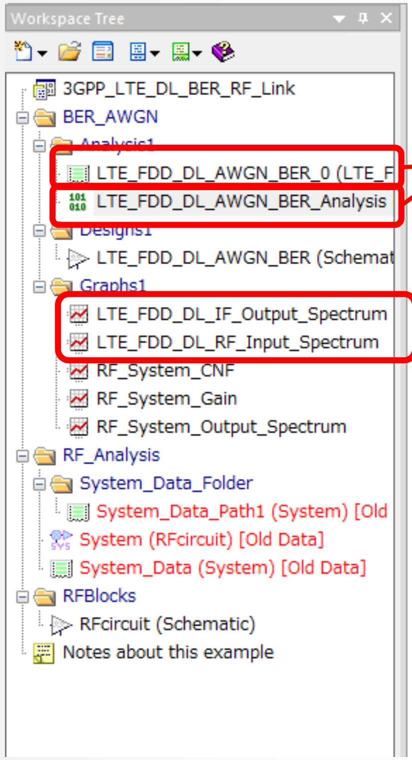
In this example, RF_Link has been already inserted in the system.

If user want to create RF_Link component, just Drag&Drop the RF design from workspace tree into Data Flow design.

KEYSIGHT TECHNOLOGIES

Lab 3: Run simulation and the result

② Right mouse click > Run (calculate now)

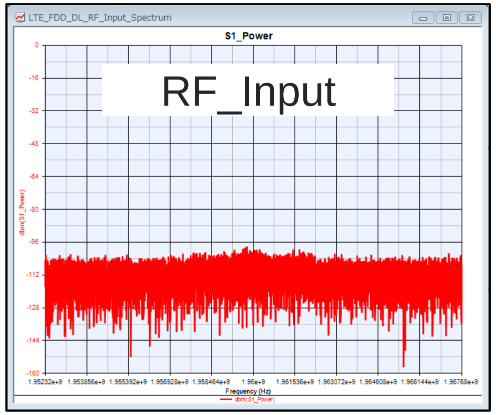


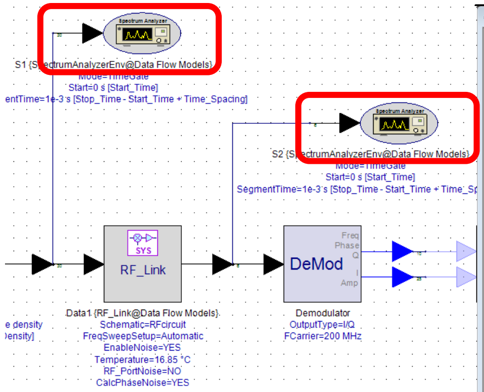
③ Open DataSet of LTE_FDD_DL_AWGN_BER_0 and check CodedBER_BER or CodedBER_FER

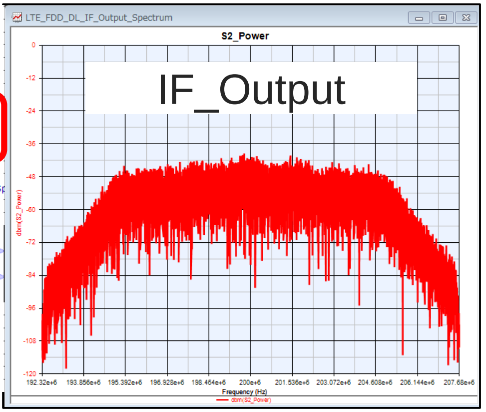
④ Open LTE_FDD_DL_RF_Input_Spectrum and LTE_FDD_DL_IF_Output_Spectrum to check RFcircuit effect (=filtered)

Variable	CodedBER_BER_Index	CodedBER_BER
CodedBER_BER	0	6.035106e-3
CodedBER_BER_Index		
CodedBER_FER		
CodedBER_FER_Index		

Variable	CodedBER_FER_Index	CodedBER_FER
CodedBER_BER	0	0.289474
CodedBER_BER_Index		
CodedBER_FER		
CodedBER_FER_Index		





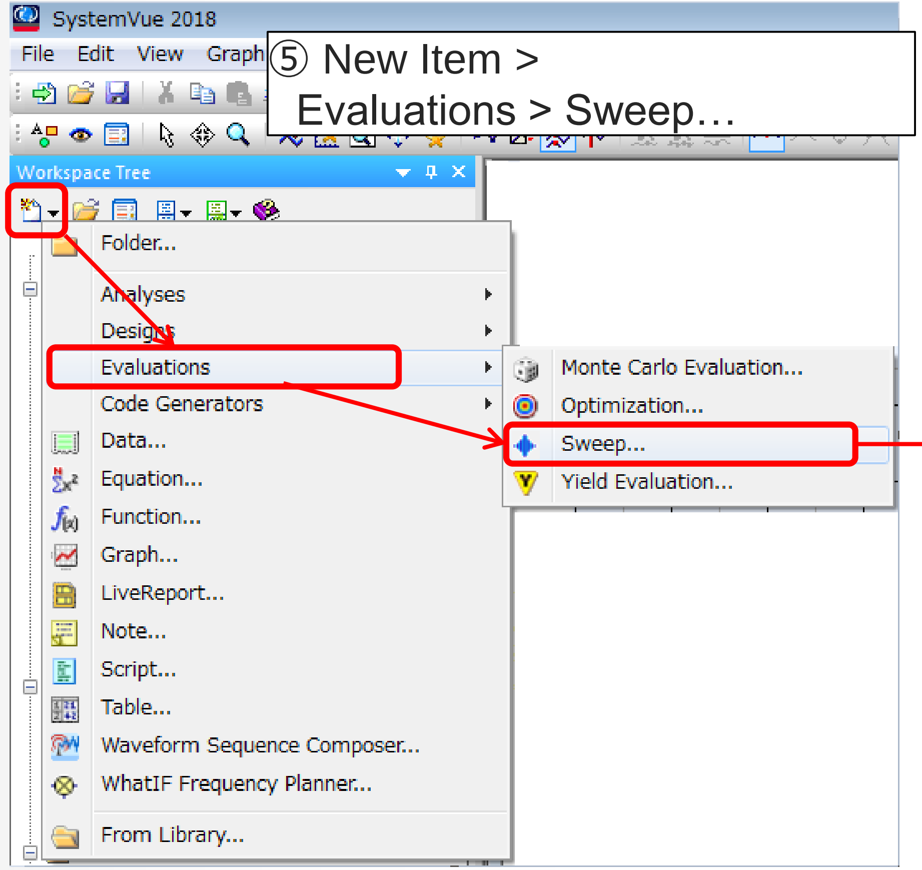


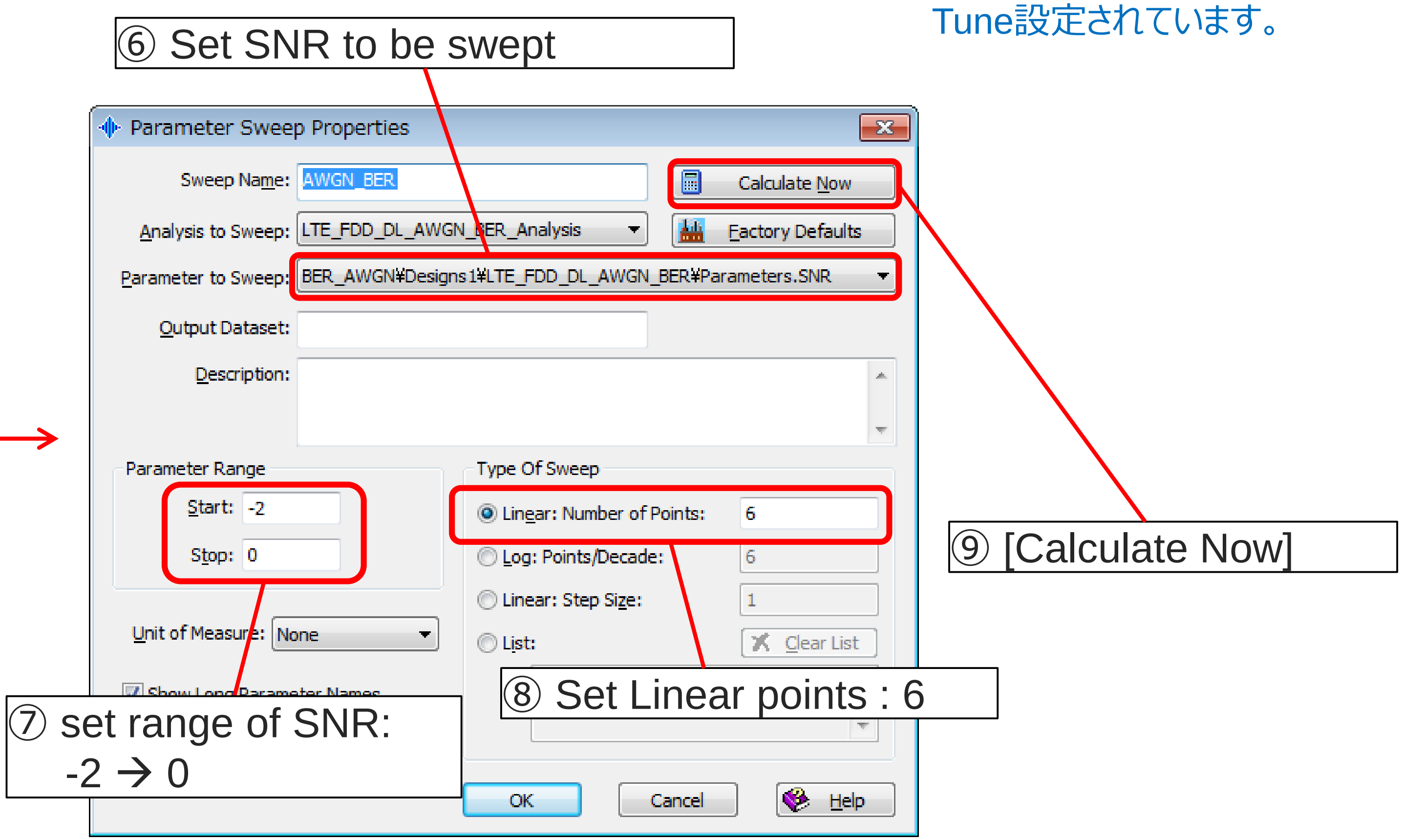
KEYSIGHT TECHNOLOGIES

Lab 3: Sweep SNR

- ➢ To get BER vs SNR plot, run simulation by sweeping SNR
- ➢ Add Sweep Controller and set parameter(SNR) range

※SNRはParametersタブでTune設定されています。



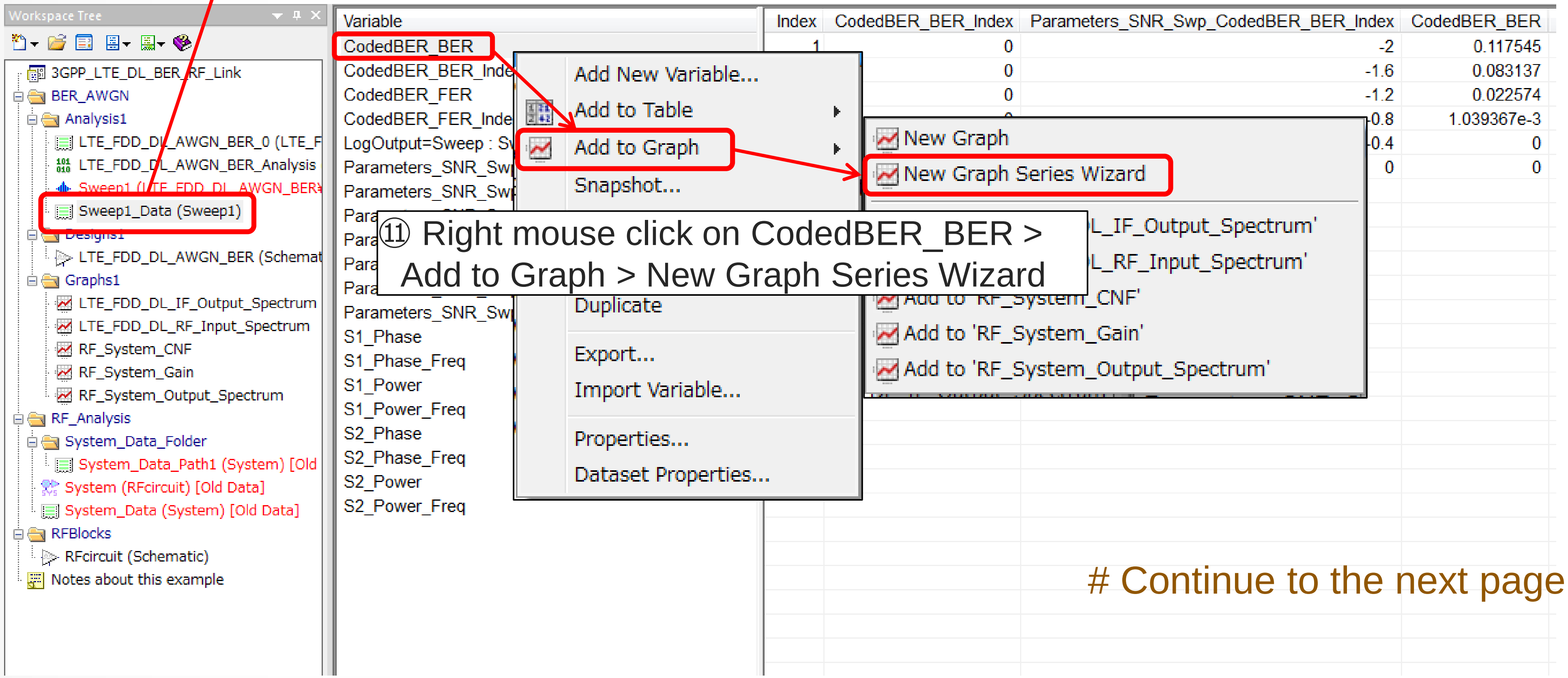


It will take several minutes to complete

KEYSIGHT TECHNOLOGIES

Lab 3: BER vs SNR

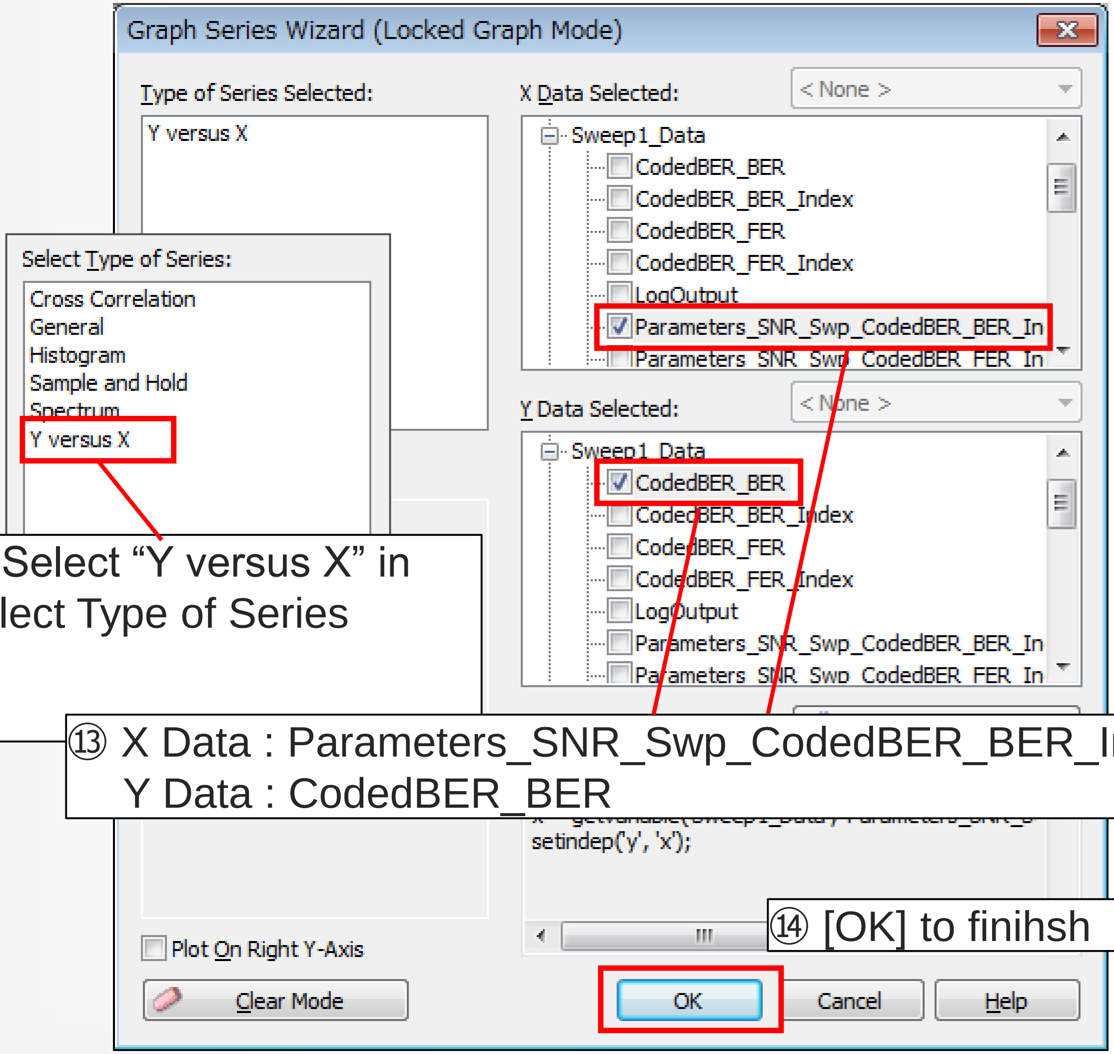
⑩ Open Sweep1_Data



⑪ Right mouse click on CodedBER_BER > Add to Graph > New Graph Series Wizard

Continue to the next page

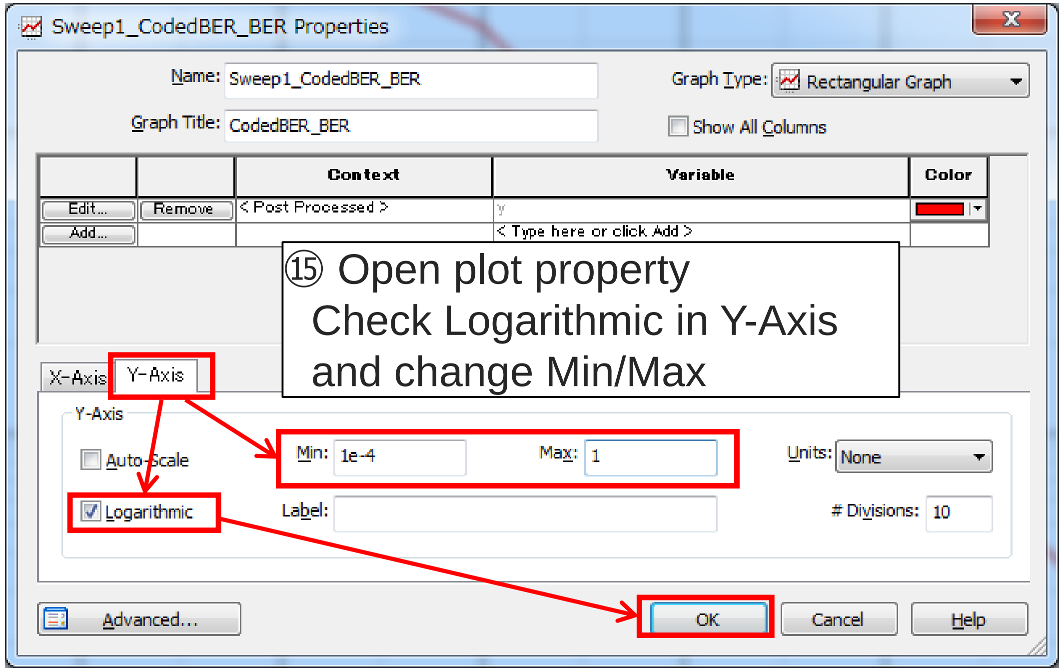
Lab 3: BER vs SNR (Cont’d)



⑫ Select “Y versus X” in Select Type of Series

⑬ X Data : Parameters_SNR_Swp_CodedBER_BER_Index
Y Data : CodedBER_BER

⑭ [OK] to finihsh

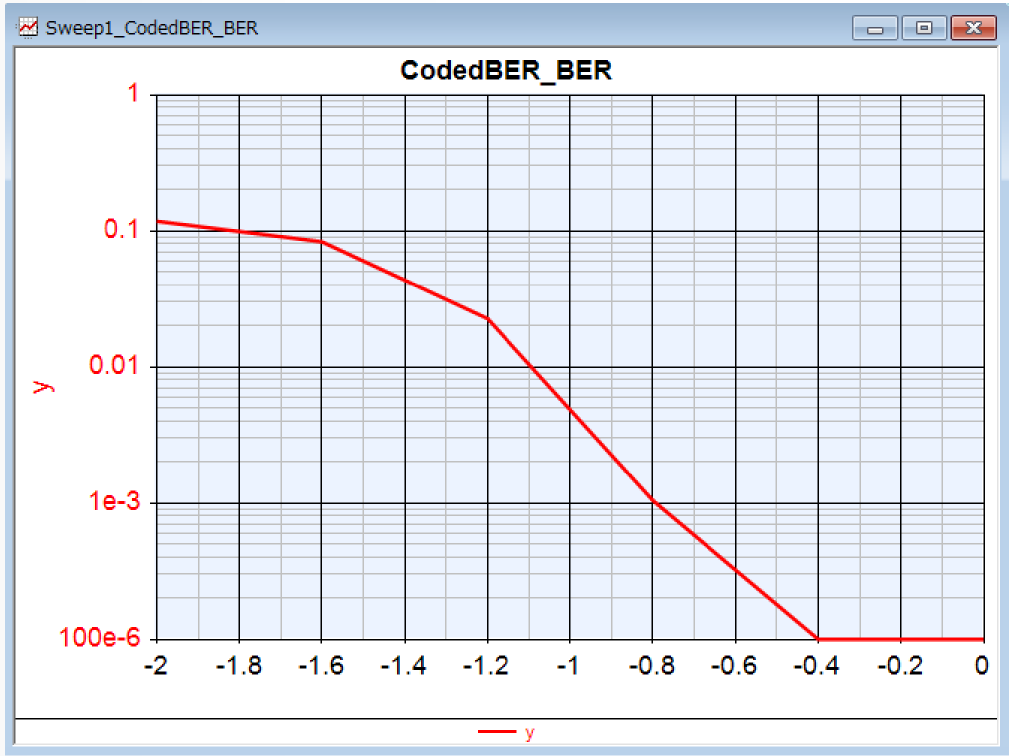


⑮ Open plot property
Check Logarithmic in Y-Axis
and change Min/Max

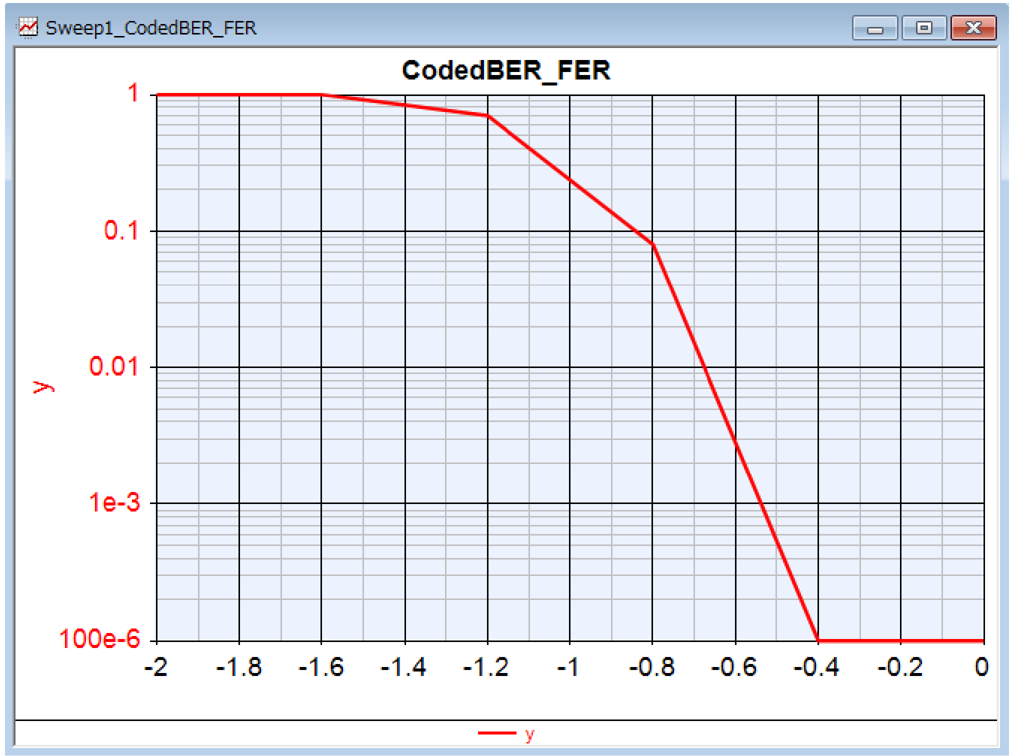
Lab 3: BER vs SNR (Cont'd)

- Create FER vs SNR as well to be familiar with "Y versus X" plot
  Just select variables for X axis and Y axis respectively

BER vs SNR



FER vs SNR



KEYSIGHT
TECHNOLOGIES