

Communication System Design using Keysight SystemVue

Dan Nakahara

2018.8.9

Keysight Technologies Japan



Agenda

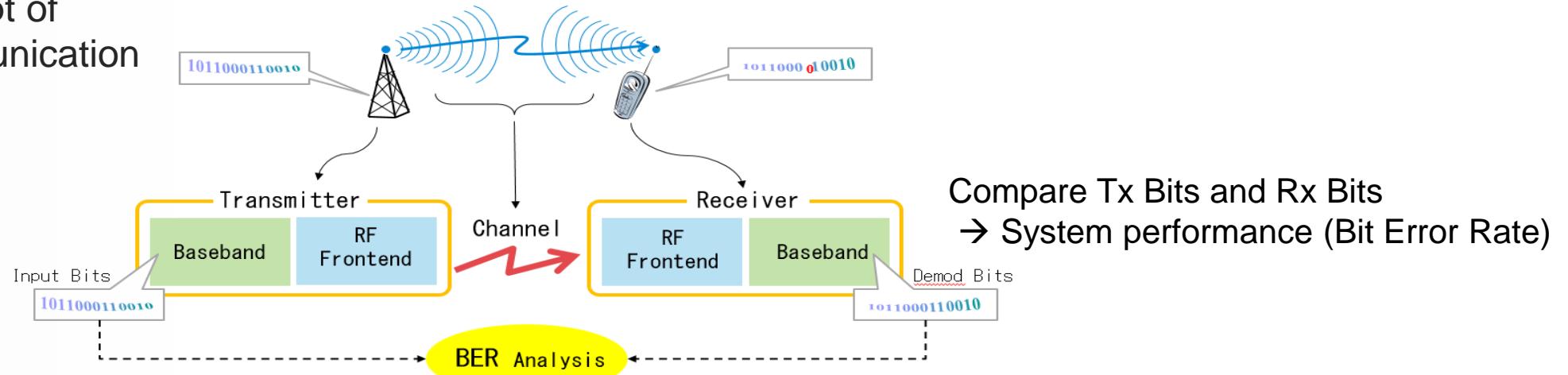
10:00~12:00

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

What is “System Simulation” ?

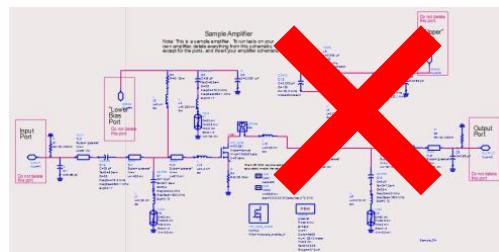
PHY of Wireless system simulation

General concept of wireless communication

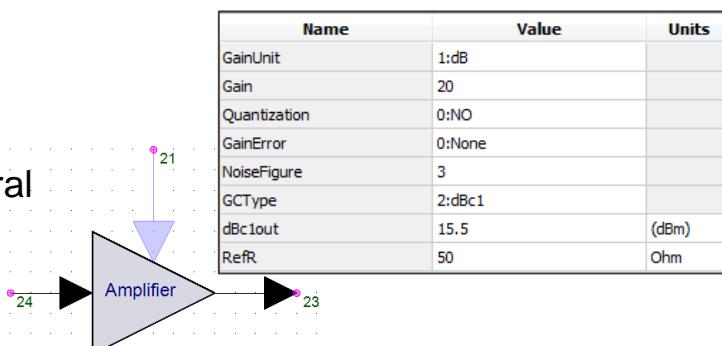


SystemVue uses “Behavioral model”

Transistor level circuit



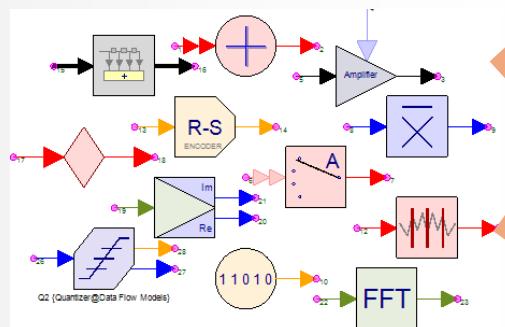
Behavioral model



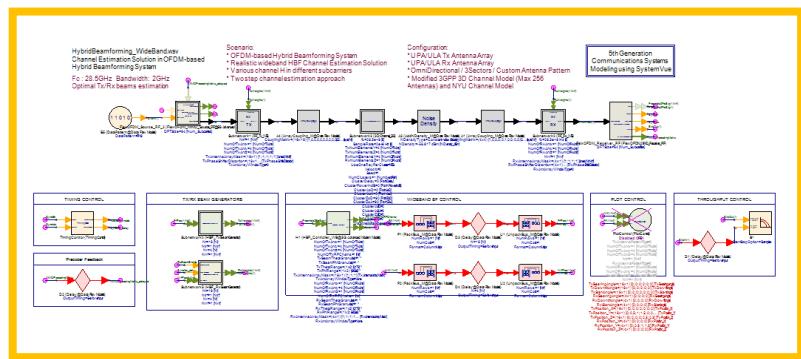
SystemVue : Model Based Simulator



Behavioral Model

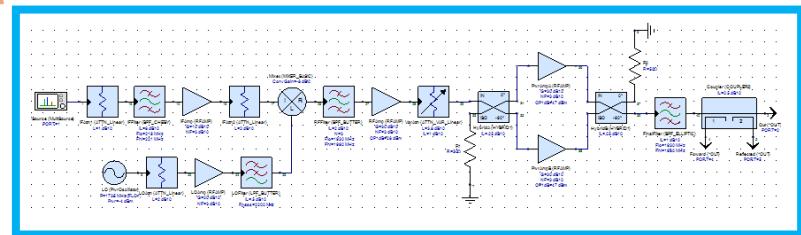


Data Flow



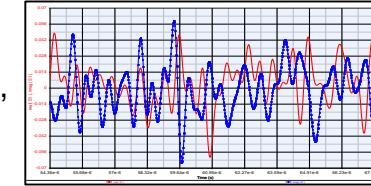
Baseband (Coding, Modem)

RF-front end block



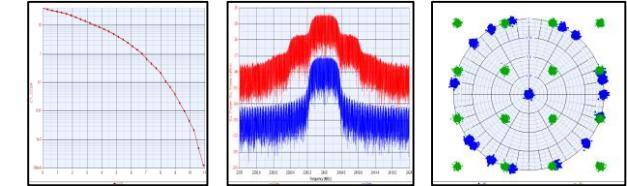
Spectrasys

Modulated waveform,
Modulation scheme



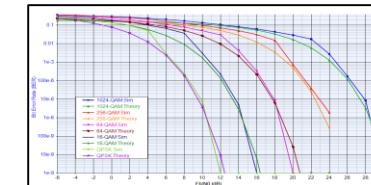
Transmitter Eval.

- Power, CCDF
- Spectrum
- Modulation Accuracy (EVM)



Receiver Eval.

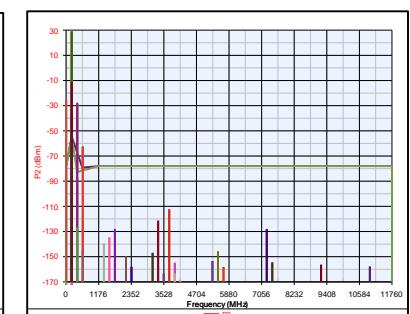
- BER, Throughput



Level Diagram
(e.g. CGain, CNF)



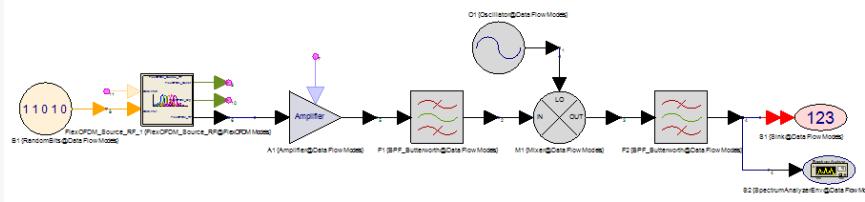
Spurious



Time Domain and Frequency Domain

Two types of simulation

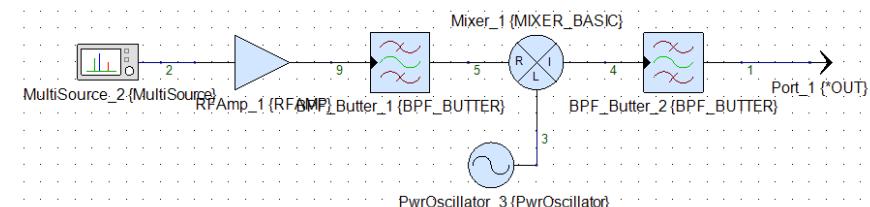
Data Flow



- 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 node 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

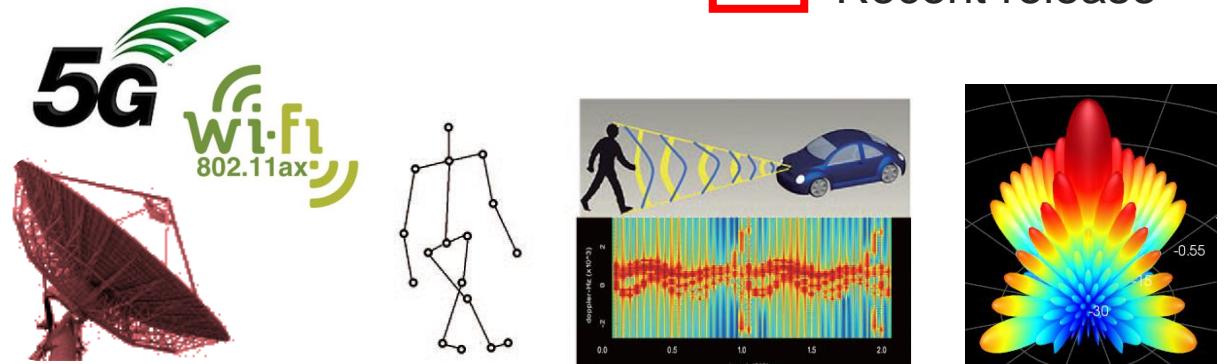
Reference IP

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		

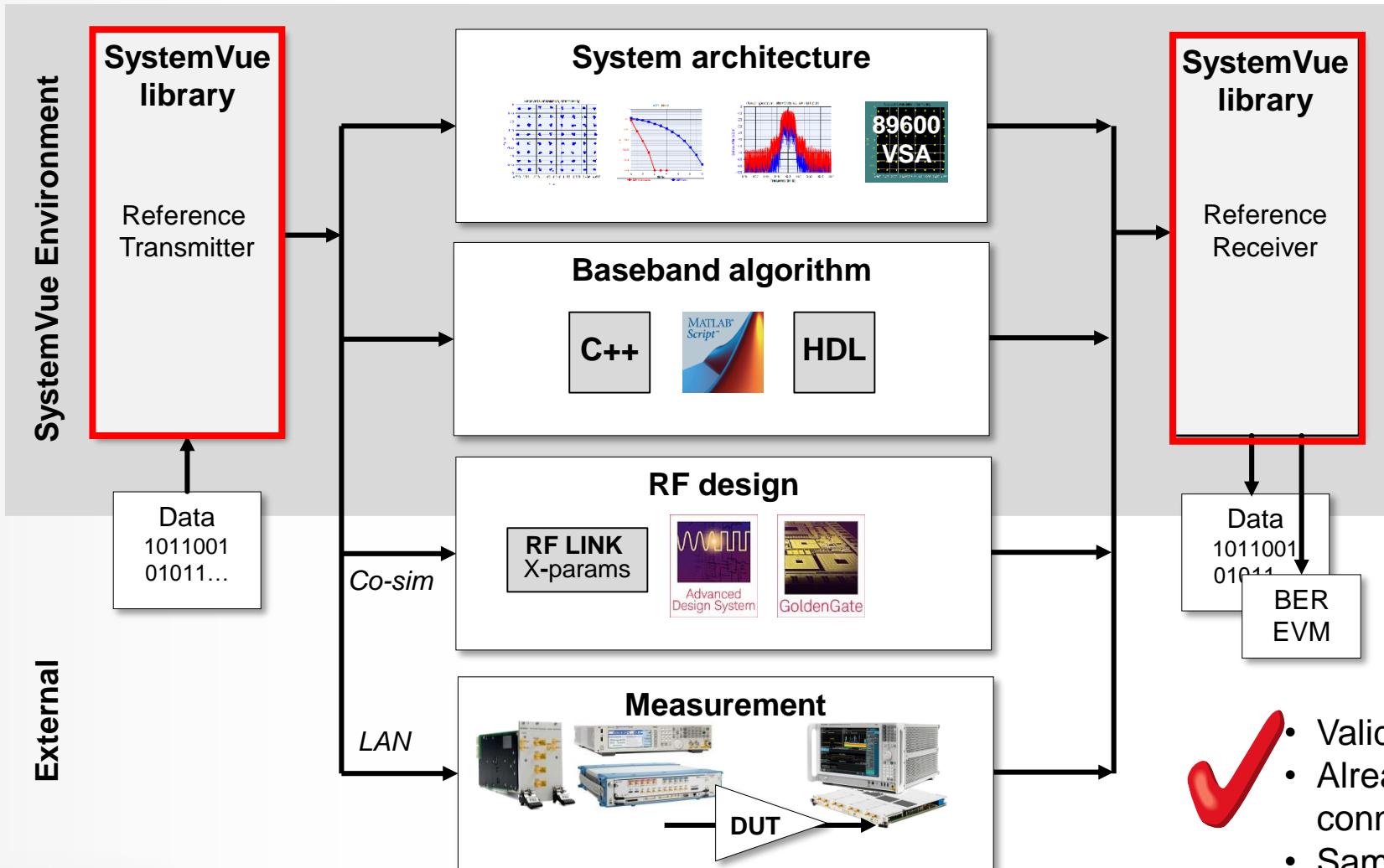
Design Kit

MIMO	RF Design Kit	DPD
Phased Array		

Recent release



IP Reference Libraries

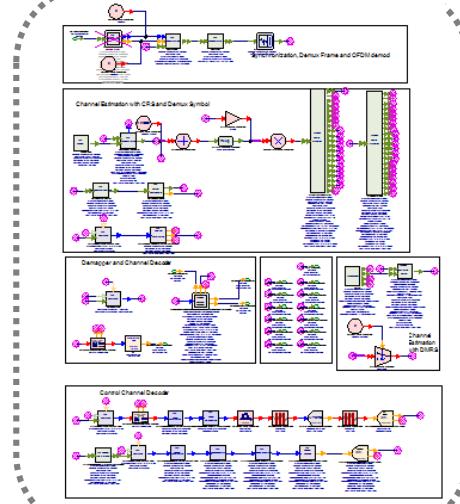
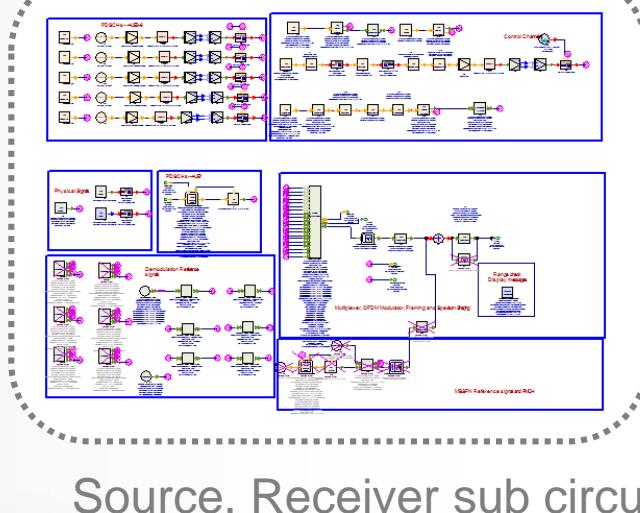
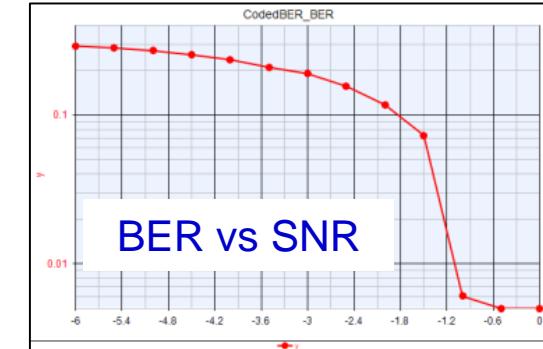
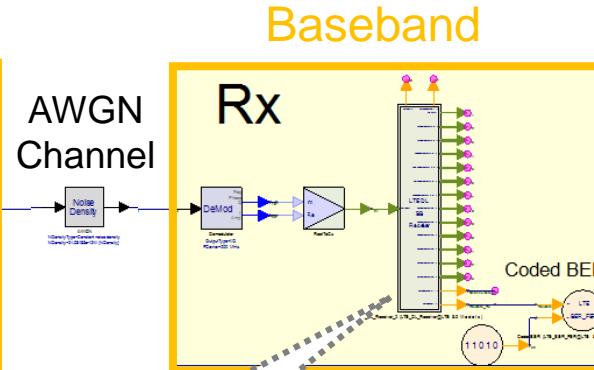
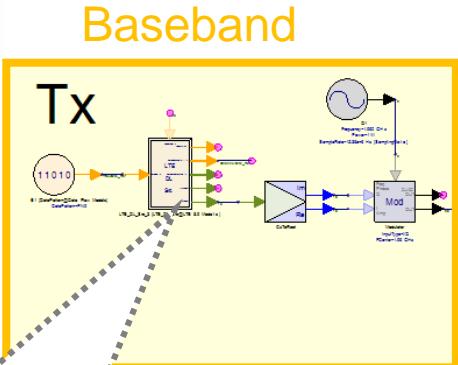


- Validate anywhere in the R&D lifecycle
- Already using MATLAB? Want to connect to the RF/ADS teams? Use SV!
- Same measurement IP throughout
- Unified, cross-platform approach

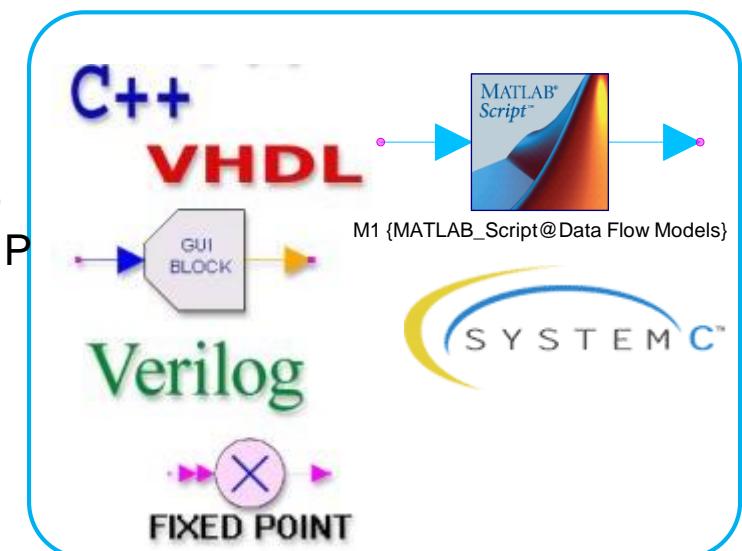
System verification

Baseband algorithm validation

Data Flow



Replace IP



RF design verification

“Spectrasys” continuous spectrum simulation

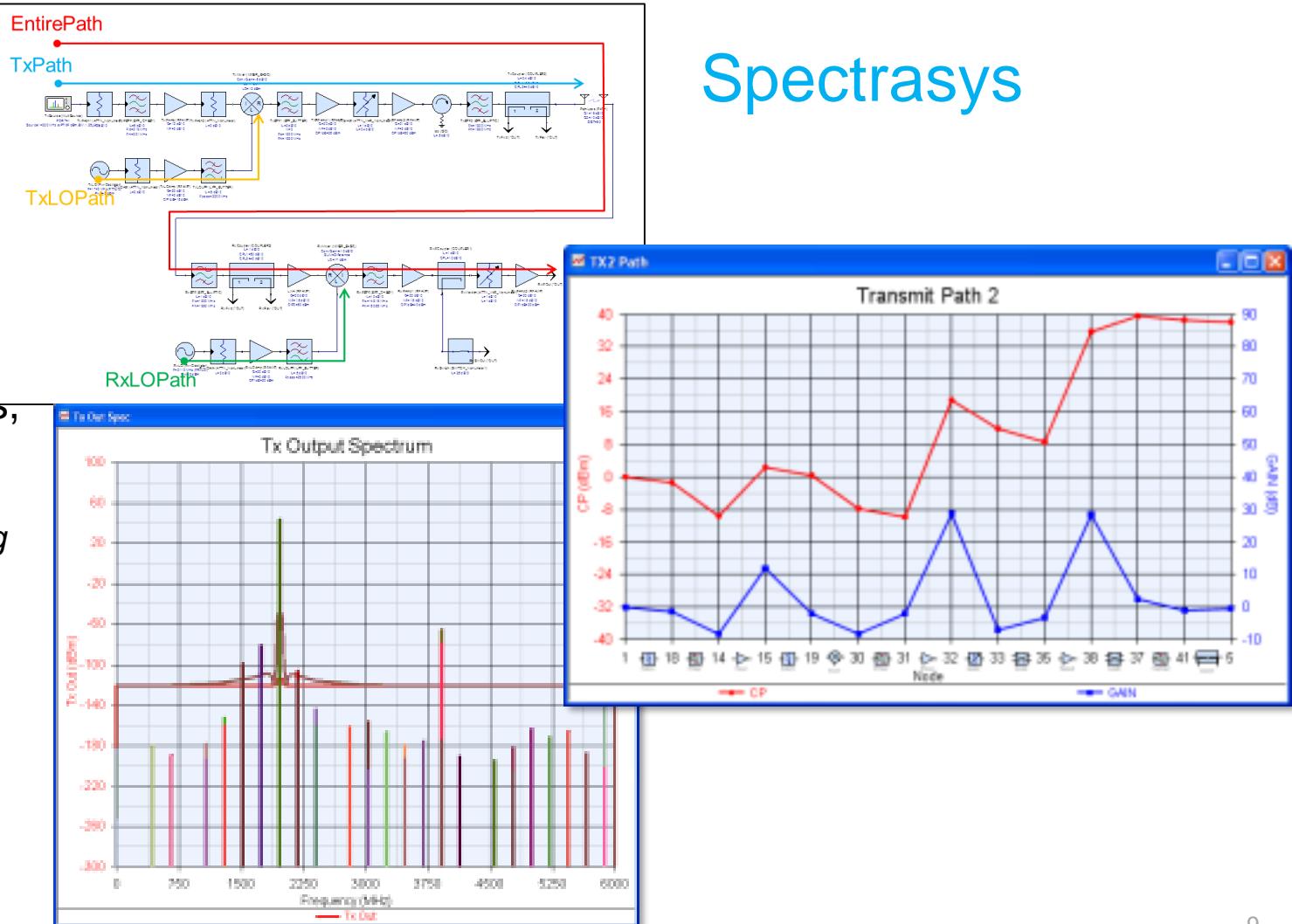
Full continuous spectrum at every node
(unique RF simulator)

Track origin and propagation of spurs,
noise, mismatch, leakage paths

Easily plot RF Budget & Spectral info

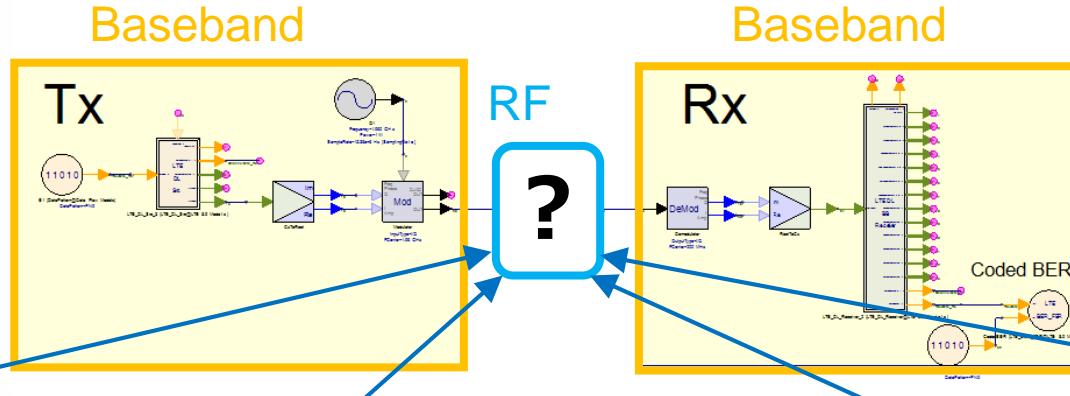
Create and refine RF System Architectures,
then use them in System-level Dataflow

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! ”

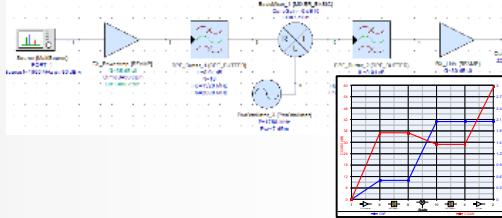


RF design verification

Use various level of RF model

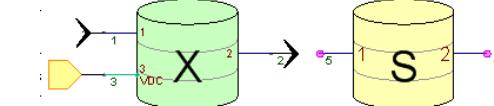


RF Link



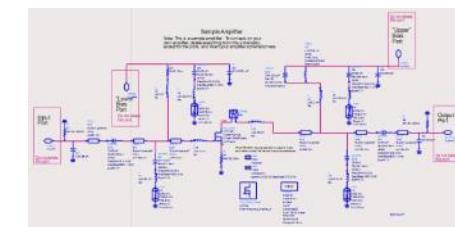
RF design in
SystemVue

Xparam, Sparam



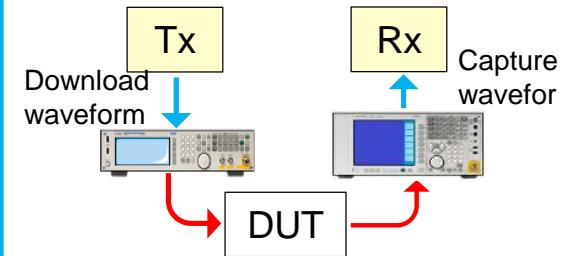
Xparameter and Sparameter
by simulation or
measurement

VTB, FCE



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

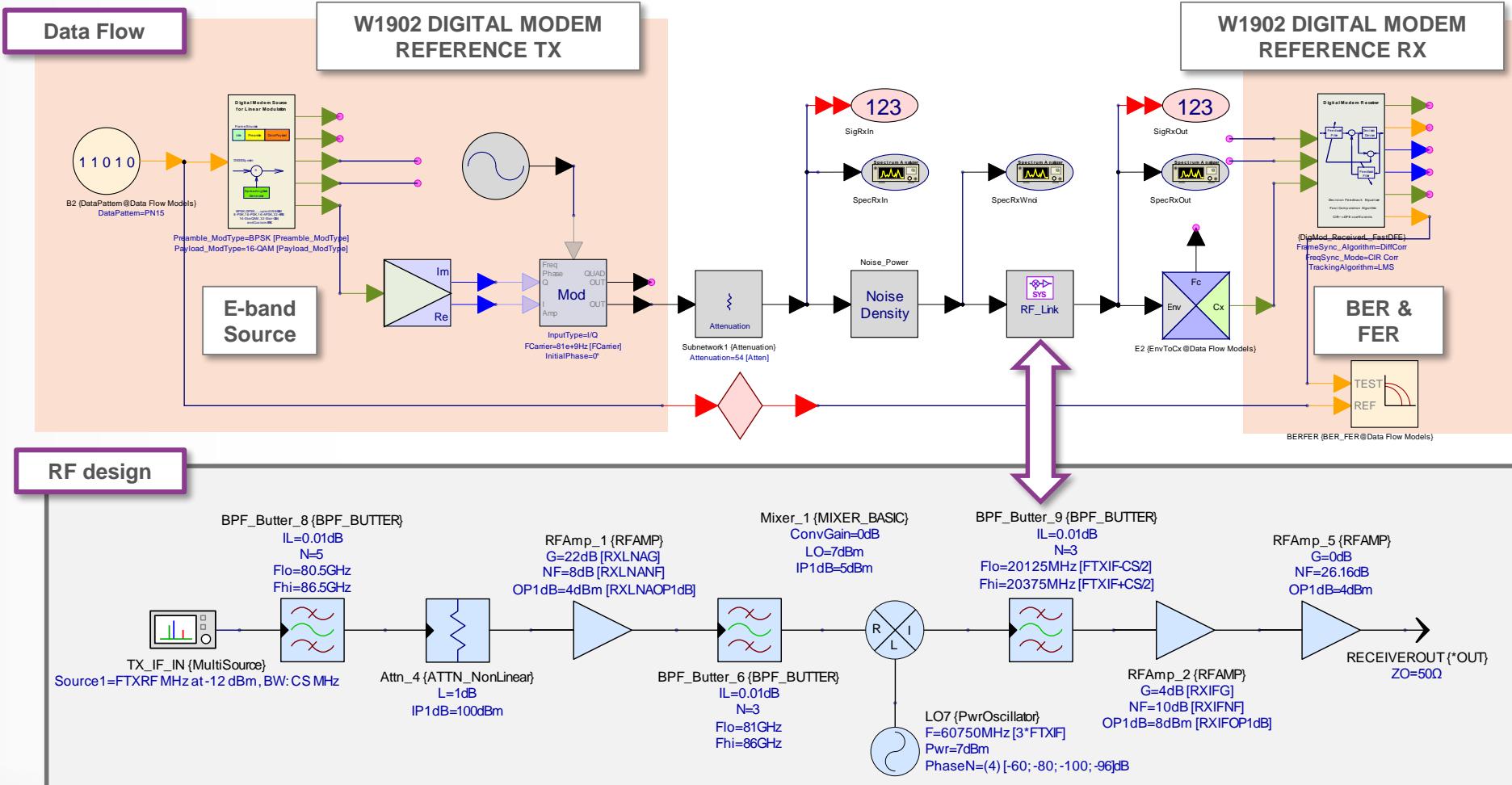
Instrument link



Download waveform to
Signal Generator
Capture waveform from
Analyzer

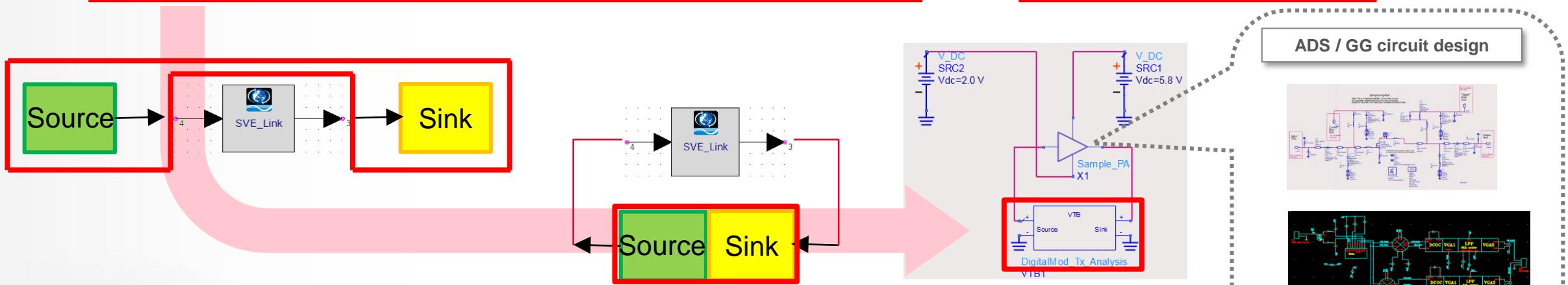
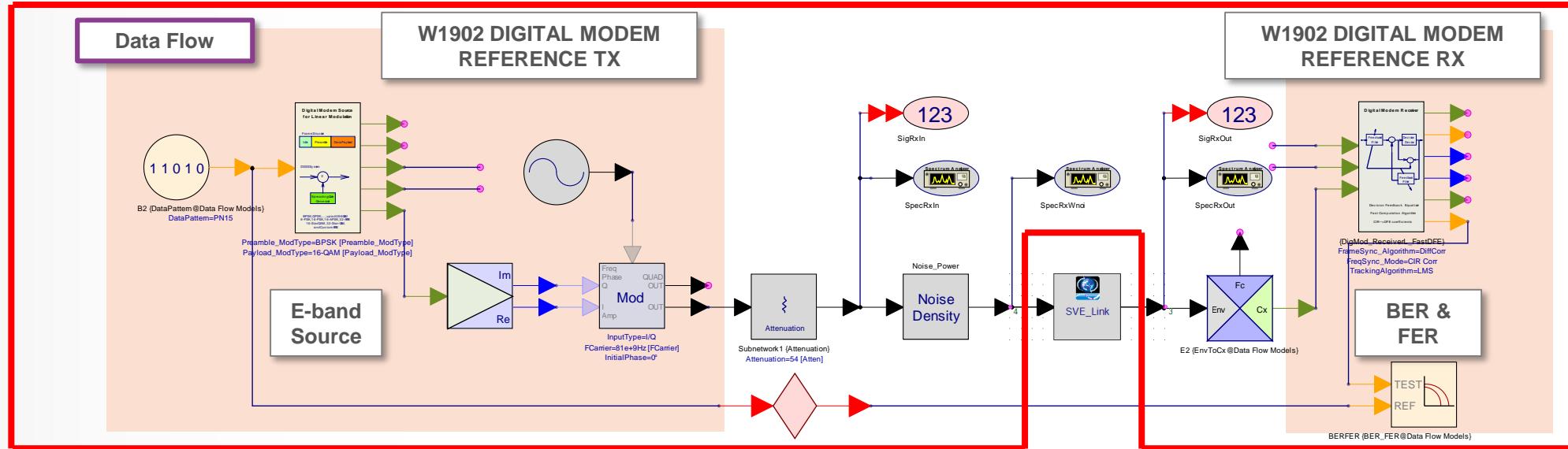
RF Link

Co-simulate with RF design



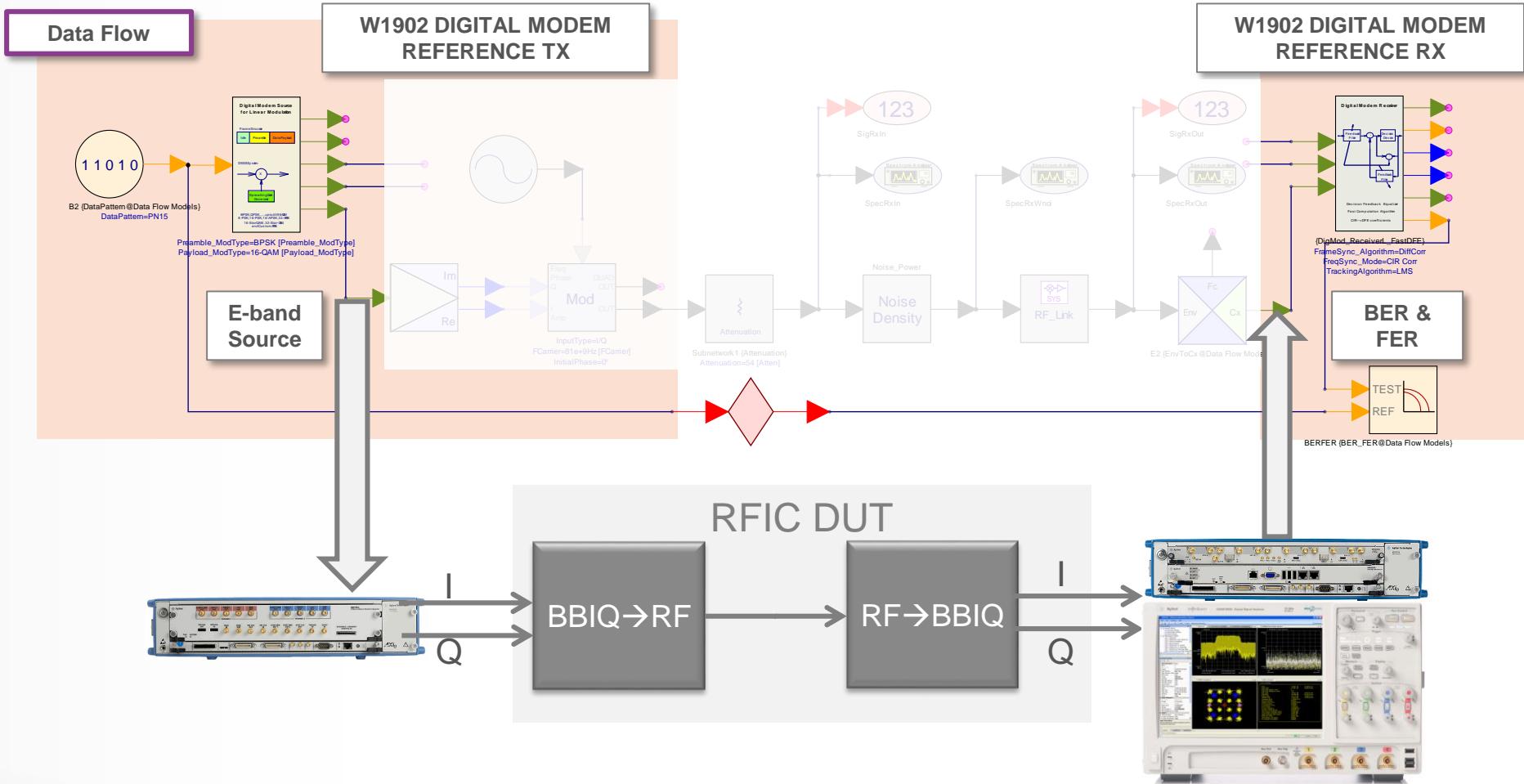
Virtual Test Bench

Verify circuit level design



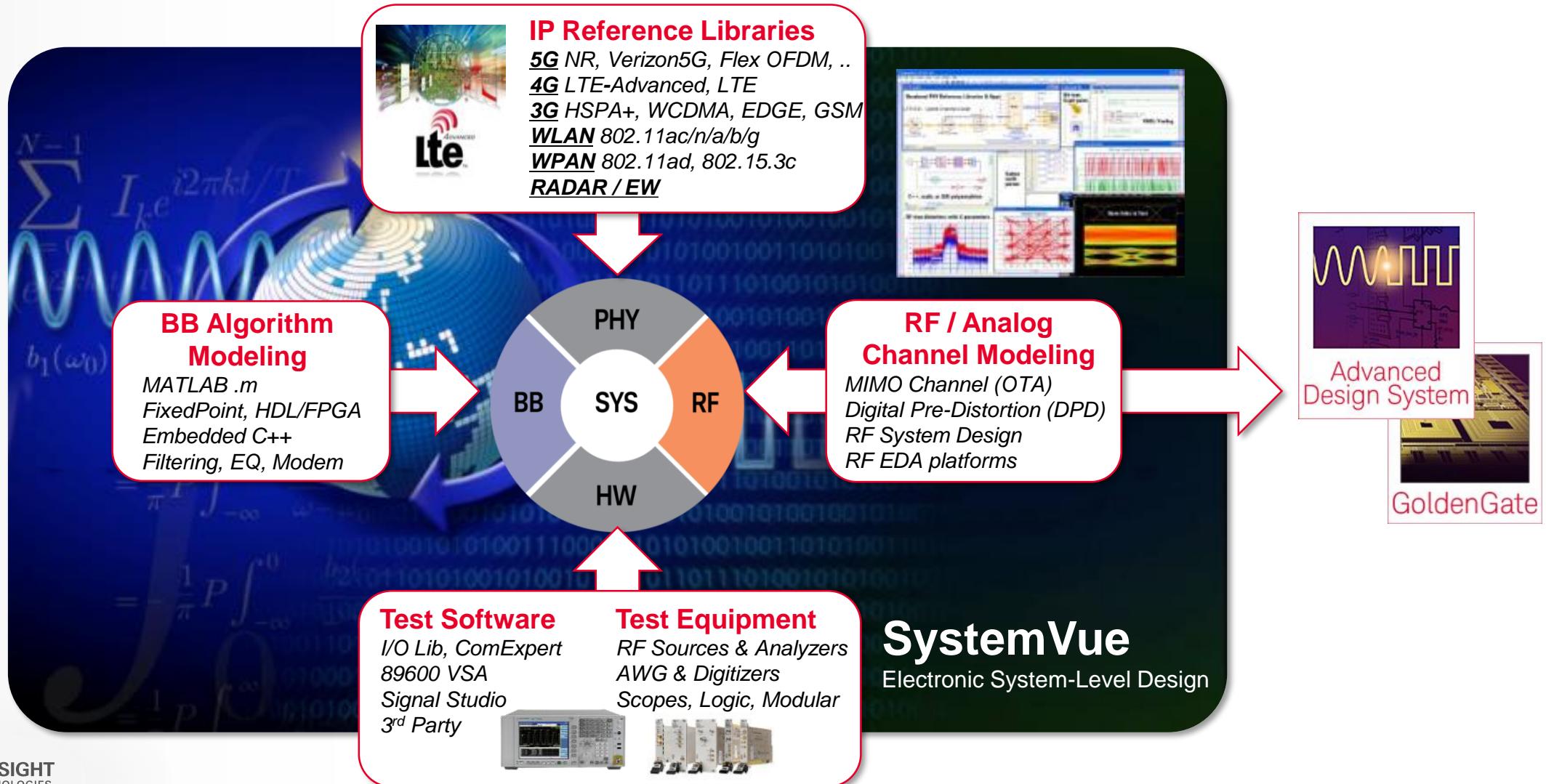
Instrument Link

Test real device



Integrated Design and Test solution

Transition naturally from Design to Test with a single “cockpit”

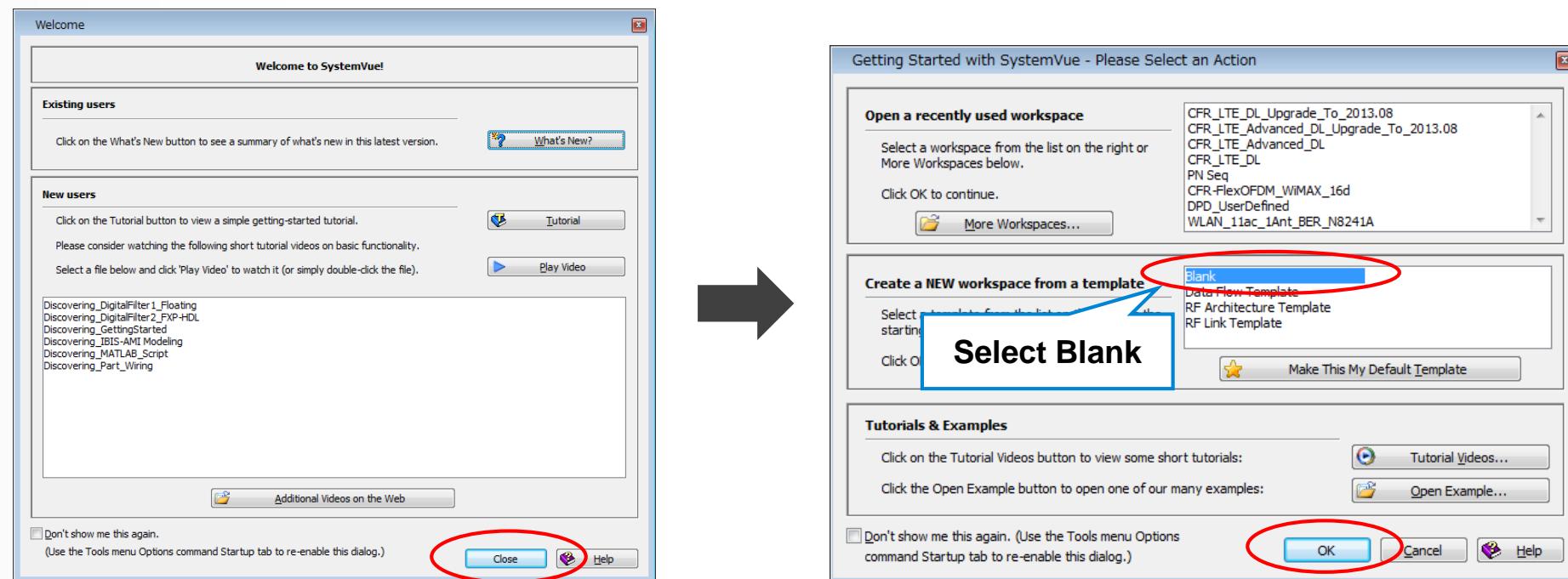


Lab 1 : Getting Started with SystemVue

Lab 1: Getting Started with SystemVue

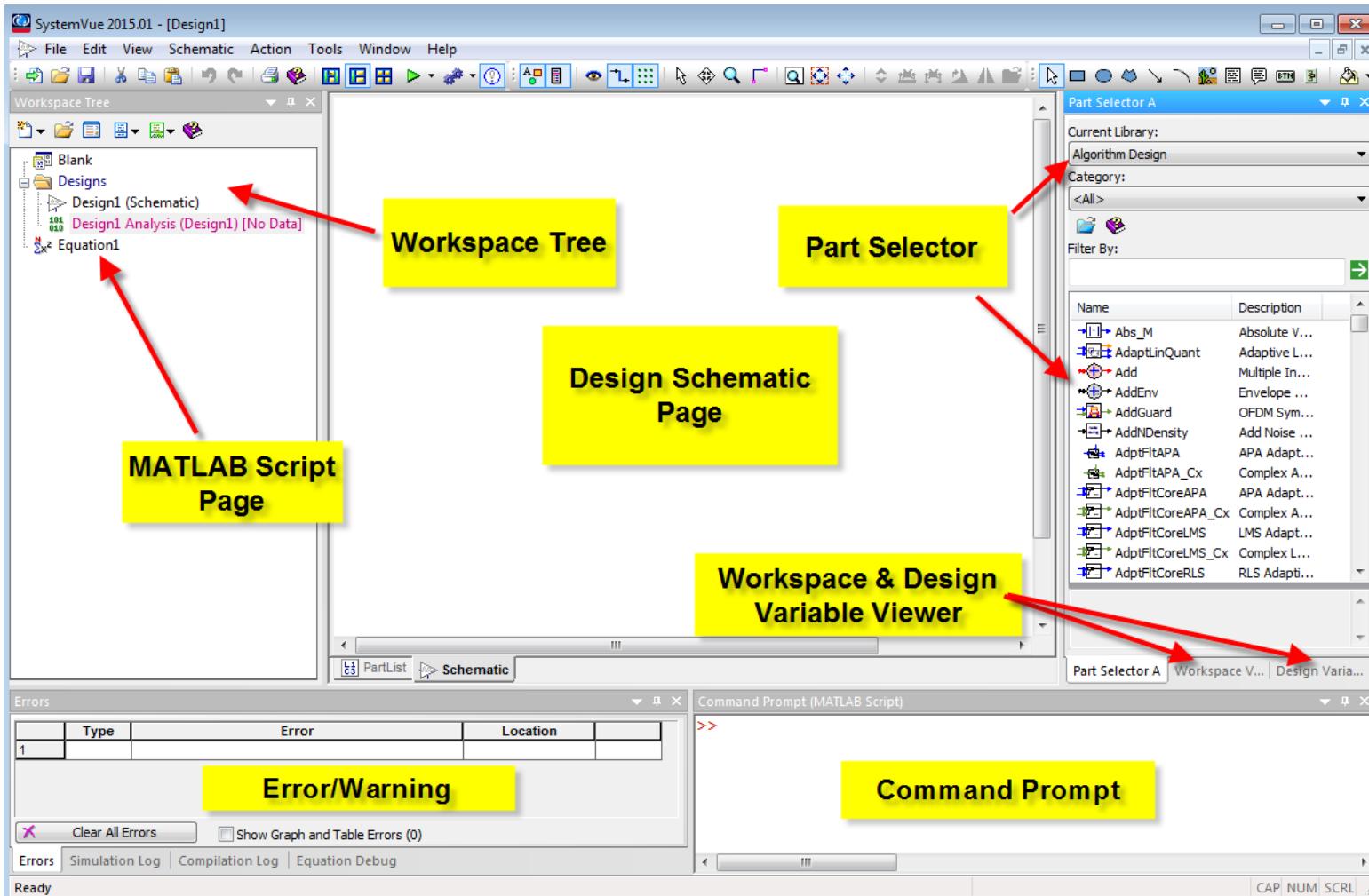


- Start SystemVue and **close the initial window** to see Getting Started Window. We can use the predefined templates to start our design or double click on recently used workspaces to open the same
- **Select “Blank” Template** from “Create a NEW Workspace from a template” section and **click OK**



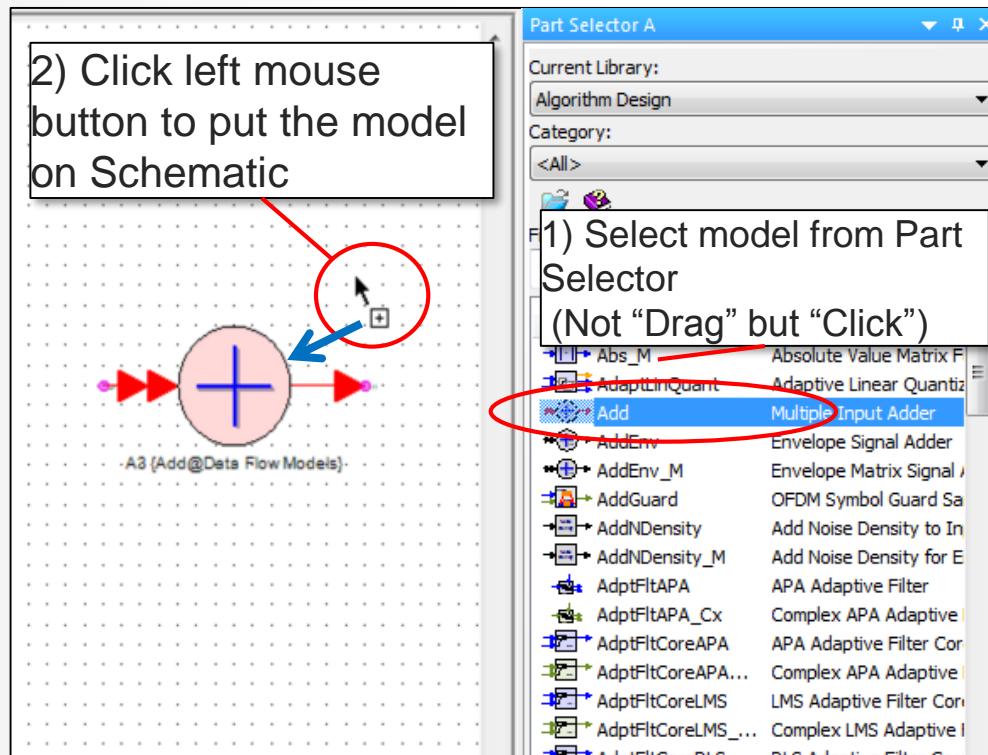
Lab 1: Getting Started with SystemVue

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

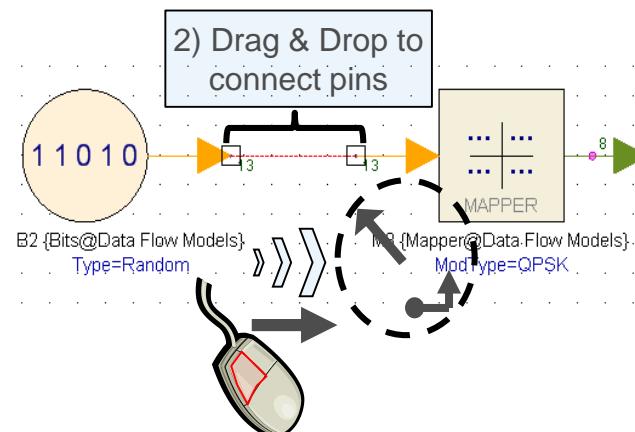
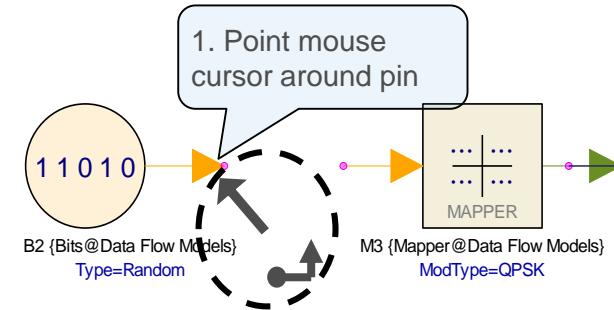


Exercise: (#DO NOT run simulation)

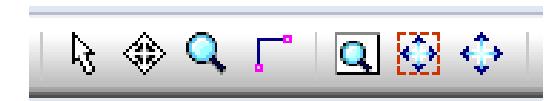
- Select model from Part Selector and put it on Schematic



- 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

Property window

The screenshot shows the 'A2' Properties dialog box with the following details:

Name	Value	Units
GainUnit	0:voltage	
Gain	1	
Quantization	0:NO	
GainError	0:None	
NoiseFigure	0	
GCType	0:none	
RefR	50	Ohm

Buttons and checkboxes in the dialog box include:

- Model Help (highlighted with a yellow box)
- Show help manual of this model (yellow callout)
- Manage Models...
- Show Model (checkbox checked)
- Use Model (button)
- OK (button highlighted with a red box)
- Cancel
- Help

The schematic diagram on the right shows an amplifier instance with the following properties:

- Designator: A2
- Model: Amplifier@Data Flow Models
- GainUnit: voltage
- Gain: 1

A blue arrow points from the 'Double Click !!' callout to the amplifier instance in the schematic.

Tips : You can modify the parameter directly

The modified schematic shows the following changes:

- Designator: A1
- Model: Amplifier@Data Flow Models
- GainUnit: 0:voltage
- Gain: 1

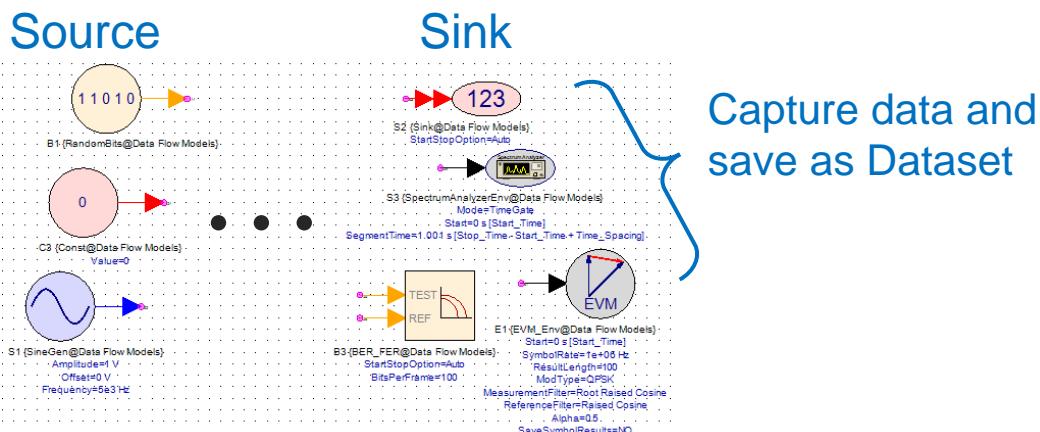
The parameter 'Gain' has been directly modified to '1'.

[Ref.] Data Flow simulation

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

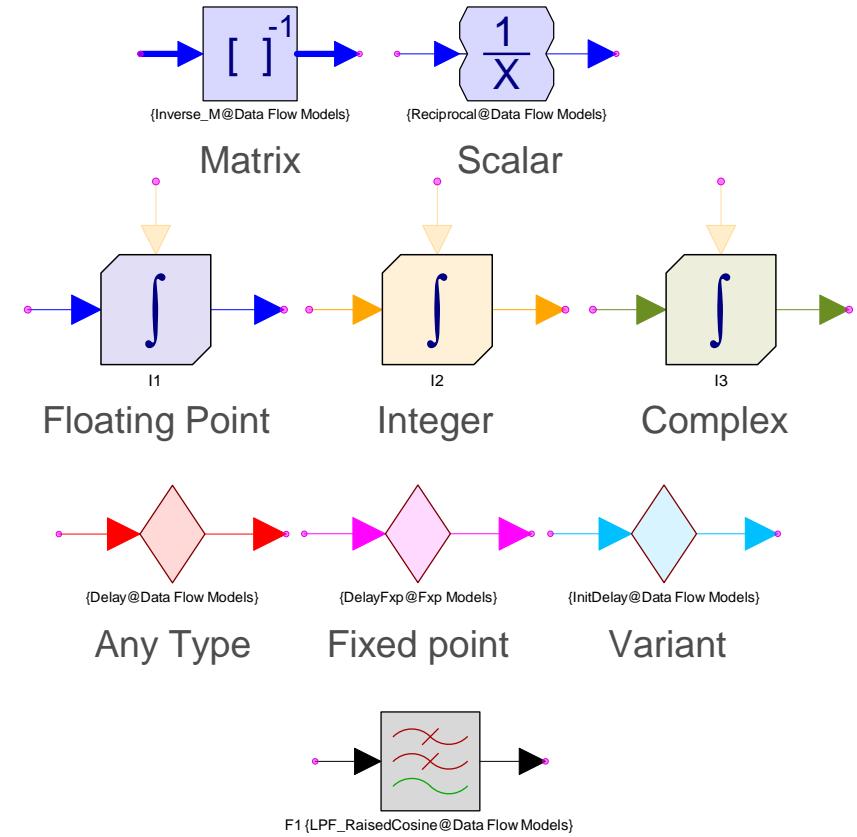
DataFlow is...

- 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



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

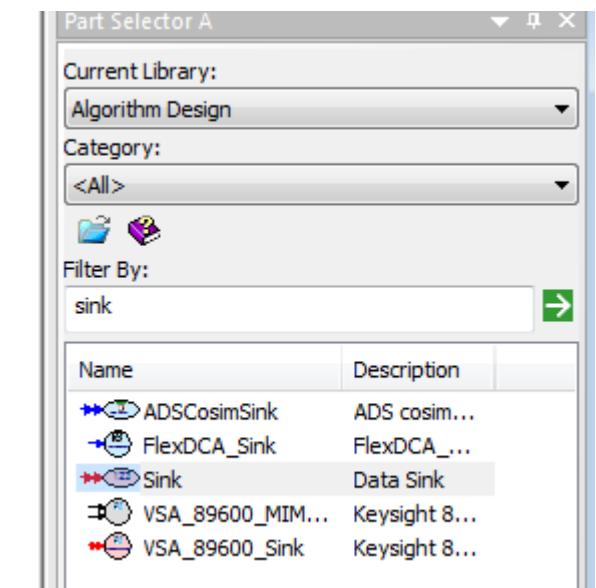
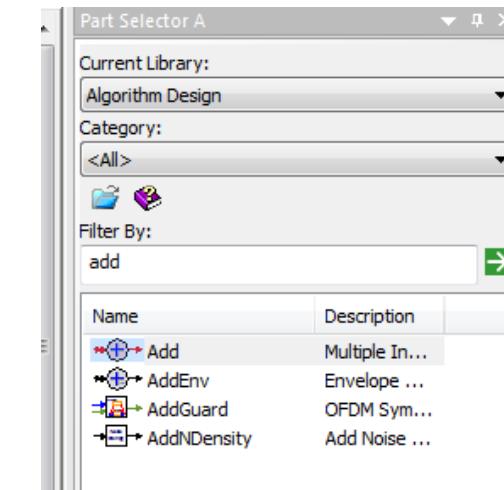
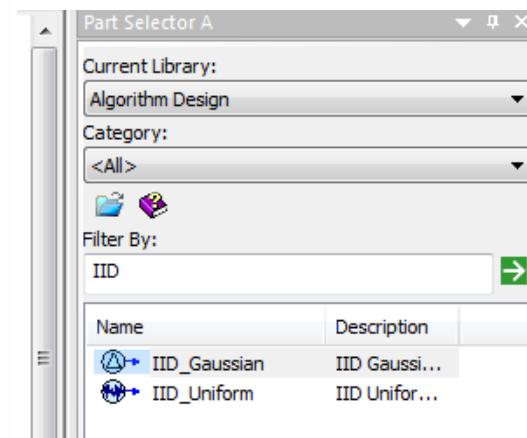
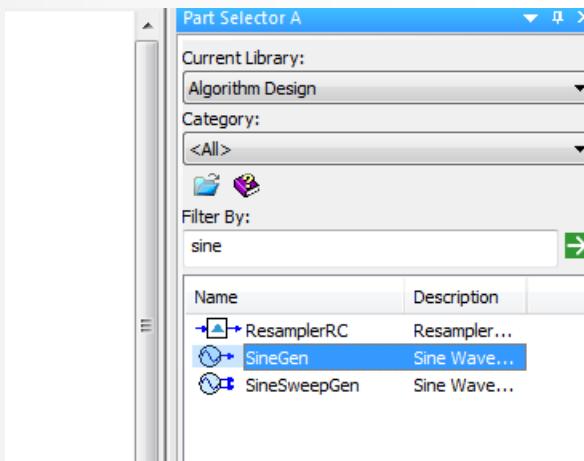
Data Type	Port Color	Port Thickness	Model Suffix
Scalar Integer	Orange	Thin	Int
Scalar Floating Point (Real)	Blue	Thin	
Scalar Complex	Green	Thin	Cx
Scalar Fixed Point	Magenta	Thin	Fxp
Integer Matrix	Orange	Thick	Int_M
Floating Point (Real) Matrix	Blue	Thick	_M
Complex Matrix	Green	Thick	Cx_M
Envelope 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

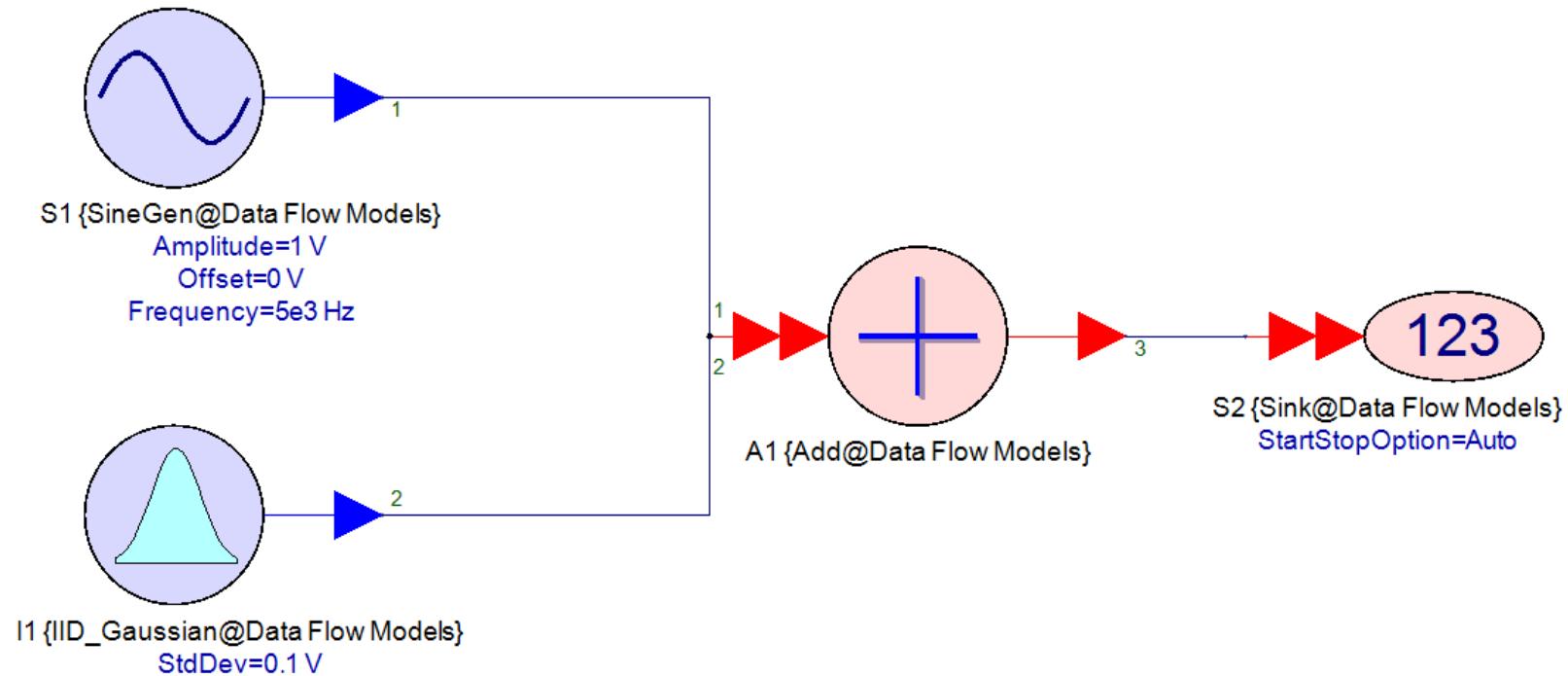
Lab 1: SineGen, IID_Gaussian, Add and Sink

- 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

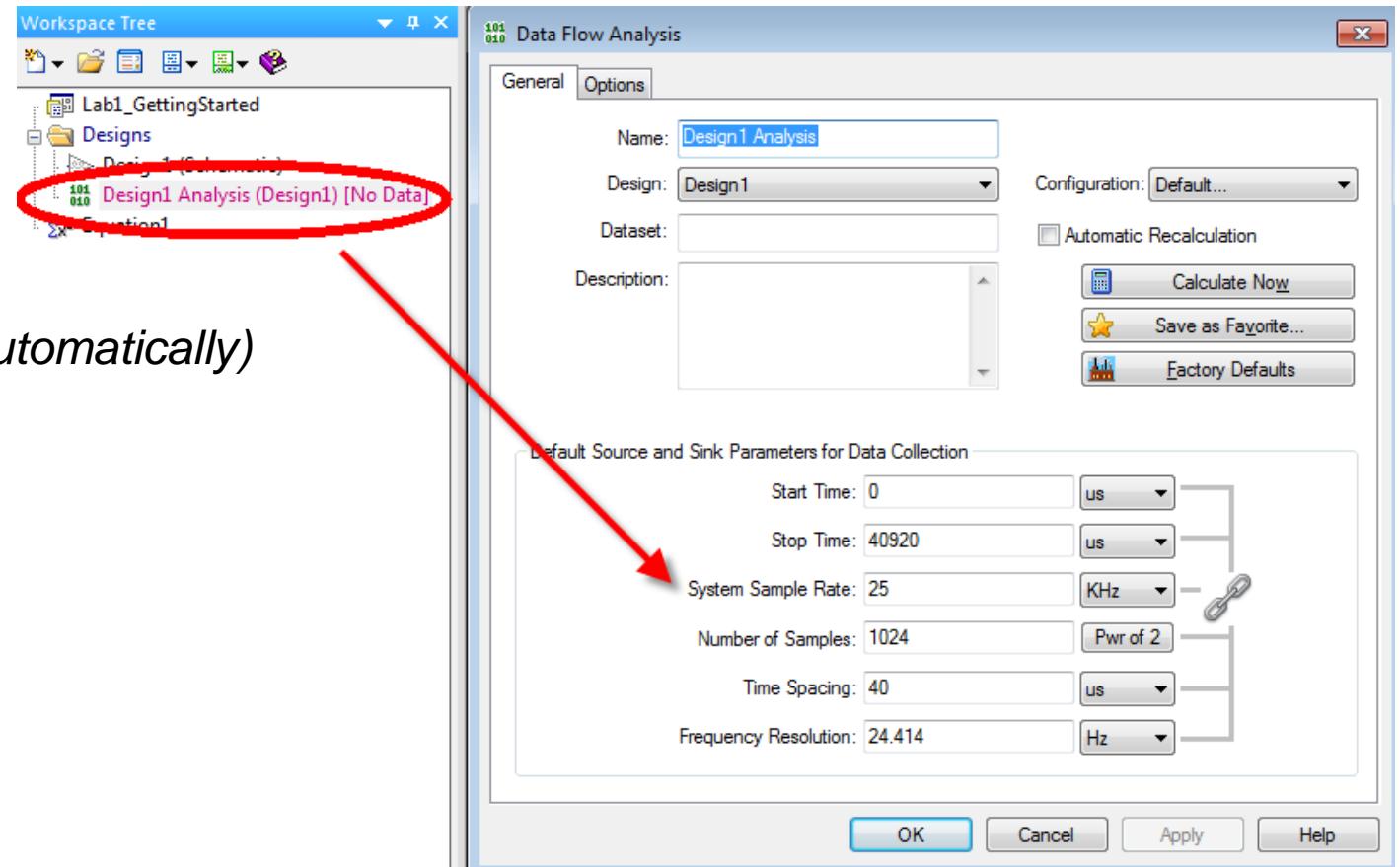
- Connect the components as shown here.....
- Note that default frequency in SineGen is 5KHz and we shall use the same
- Save the workspace and give name as **Lab1_GettingStarted**



Lab 1: DF Simulation Controller

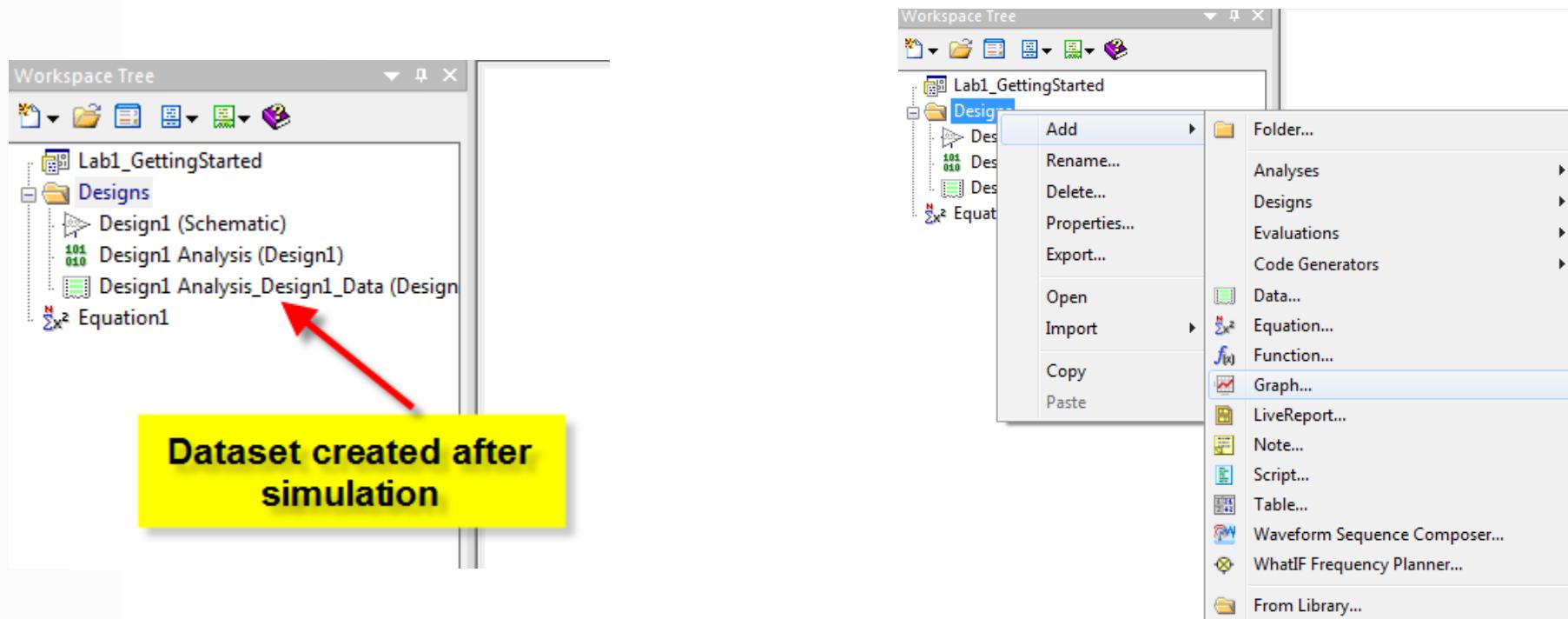
- Double click on Design1 Analysis in the workspace tree and define the two main parameters as:
 - System Sample Rate = 25 KHz
 - Number of Samples = 1024

(rest of the parameters will update automatically)



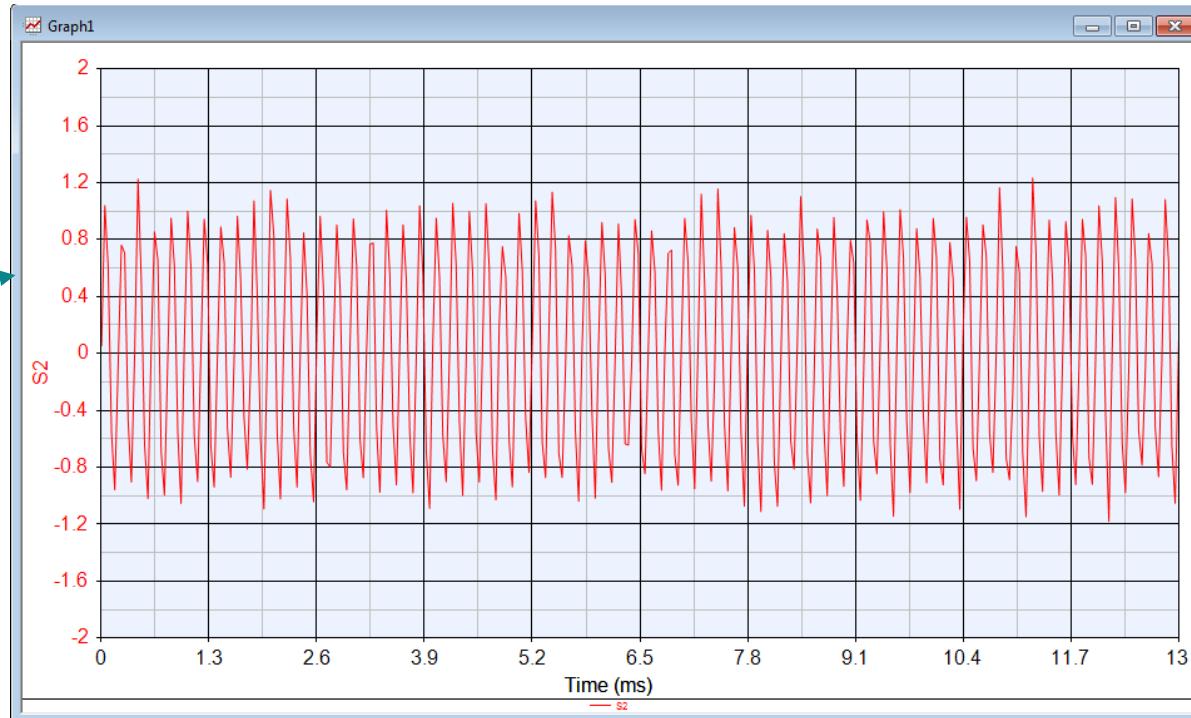
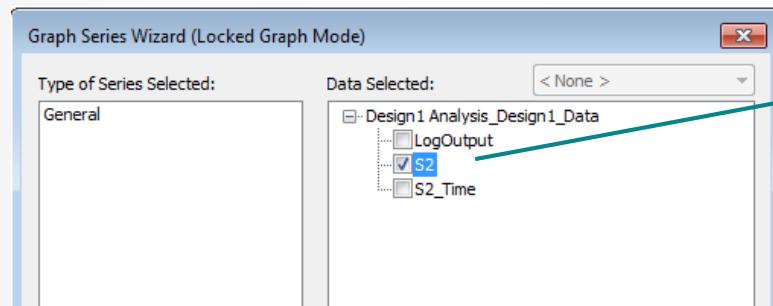
Lab 1: Run simulation and show result

- 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**



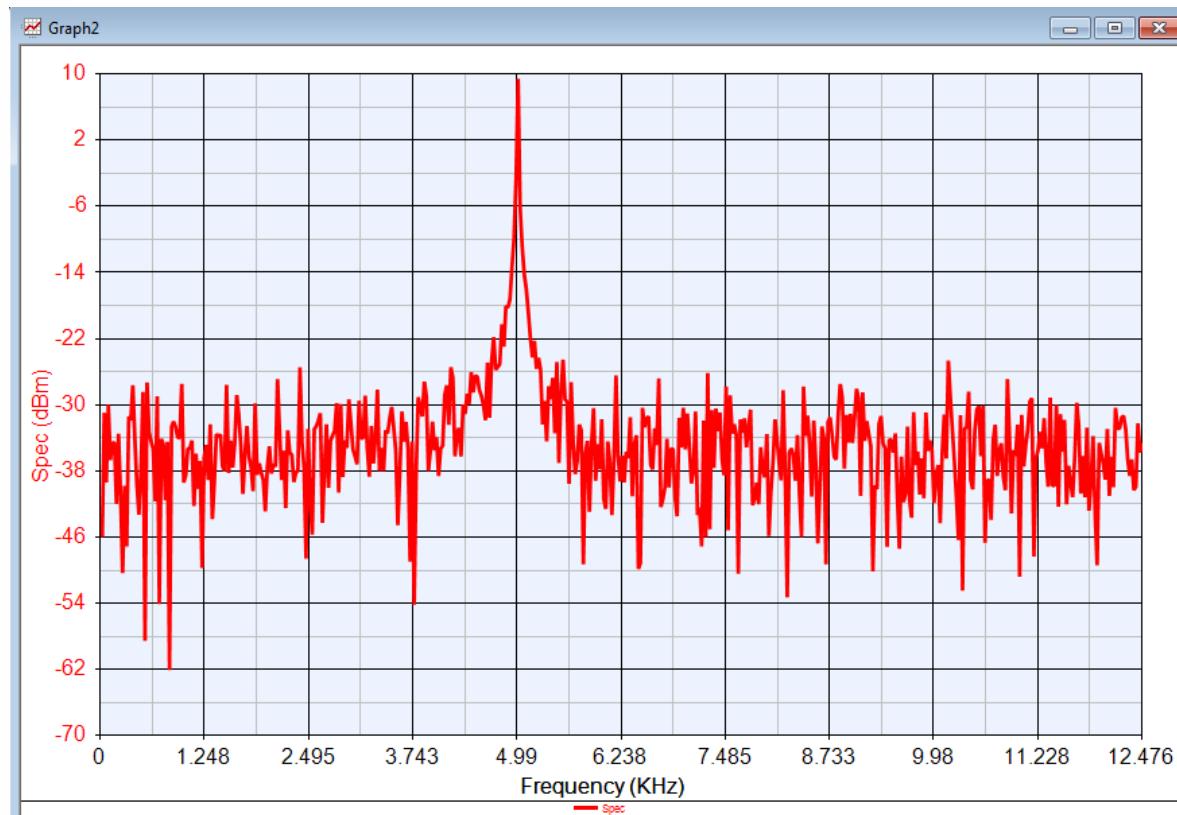
Lab 1: Plot Waveform

- 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....



Lab 1: Plot Spectrum

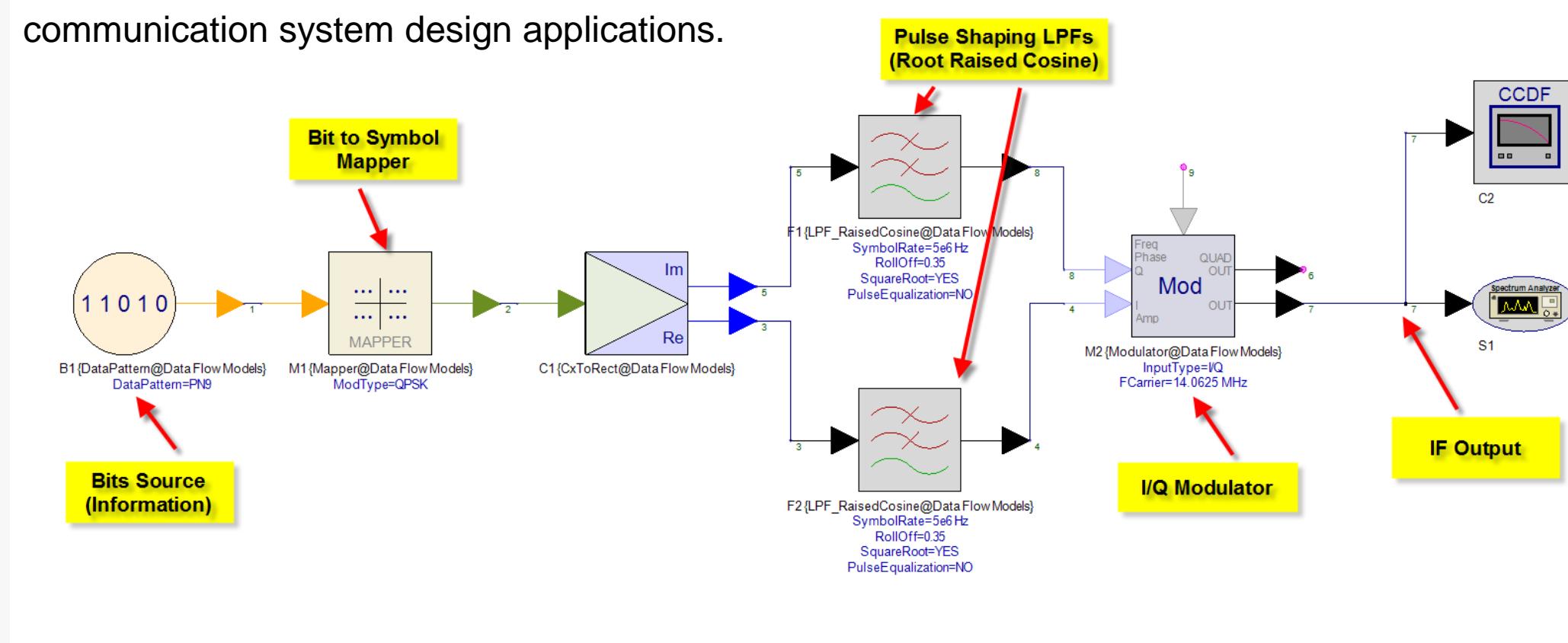
- Add a new graph, select **Series Type as Spectrum** and **select S2** from the available sink name under dataset
- Click OK to close the graph wizard window and observe the Spectrum plot



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.



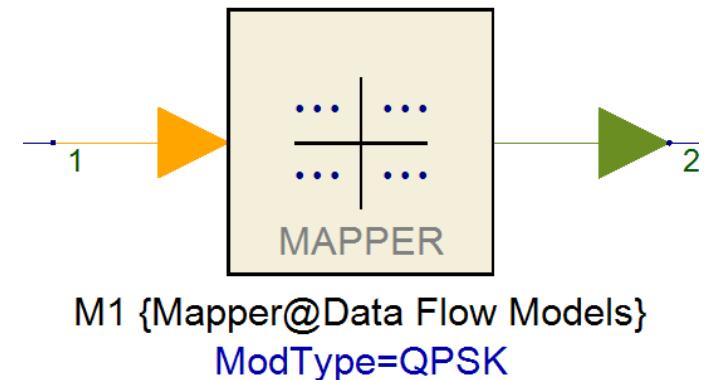
Lab 2: Bits and QPSK mapper

- 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”

B1 {DataPattern@Data Flow Models}
DataPattern=PN9

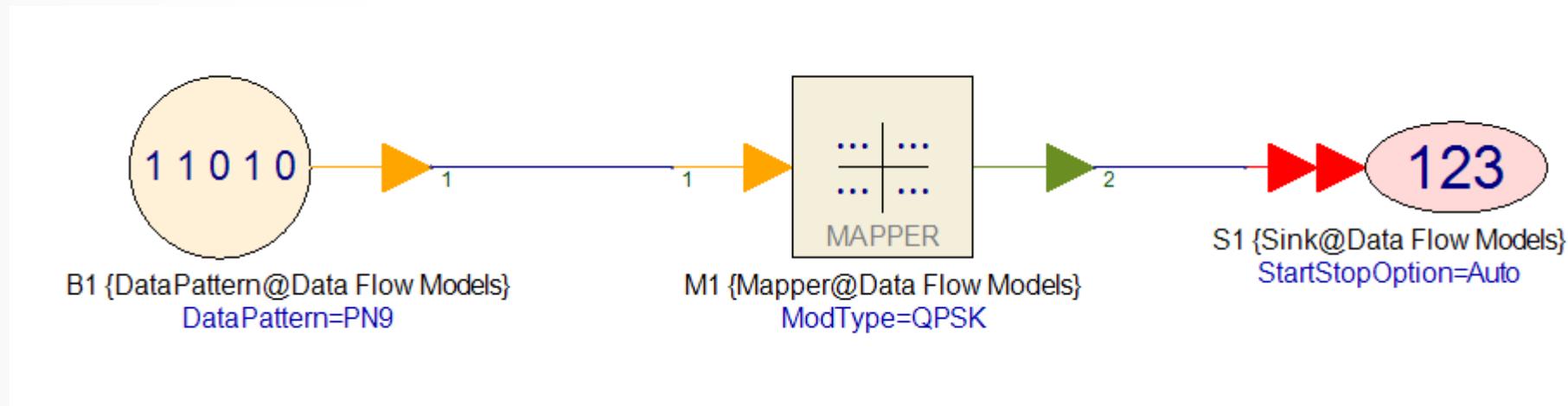
'B1' Properties

Designator:	B1	
Description:	Patterned Data Source	
Model:	DataPattern@Data Flow Models	
Manage		
DataPattern@Data Flow Models		
PRBS@Data Flow Models		
RandomBits@Data Flow Models		
Name	Value	Units
DataPattern	0:PN9	()
BitRate	Sample_Rate	Hz
ShowAdvancedParams	0:NO	()



Lab 2: Bits and QPSK mapper

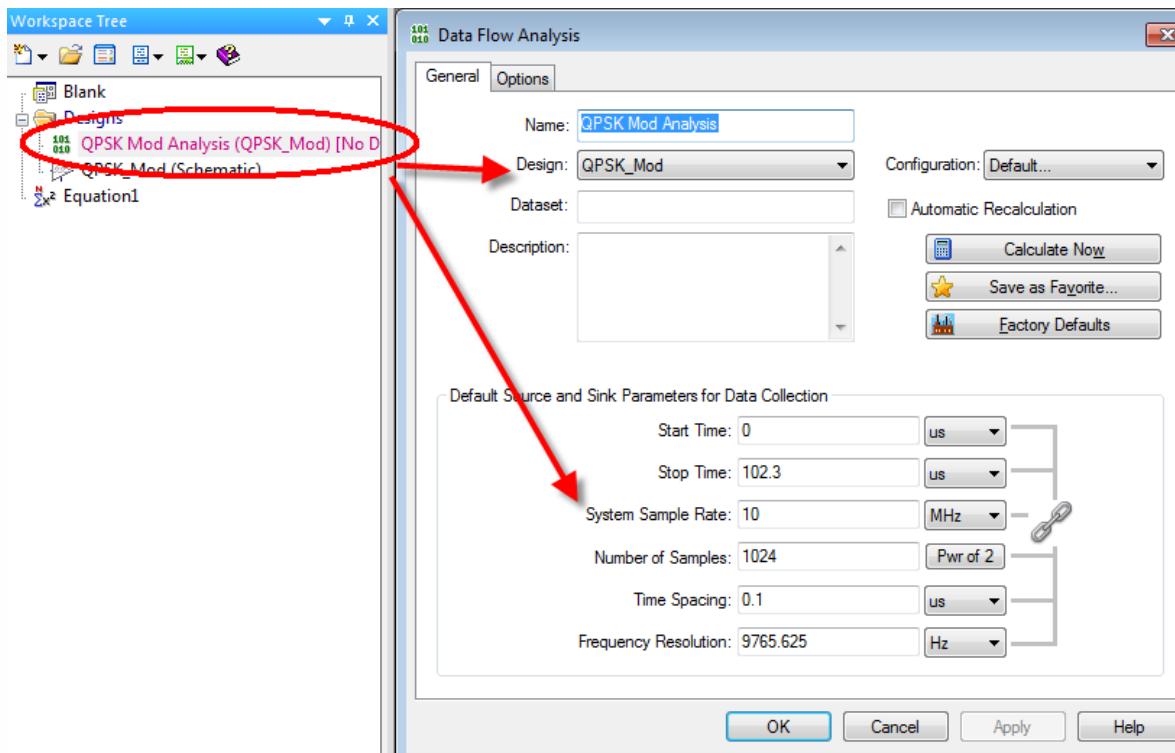
- Connect the components as shown here



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

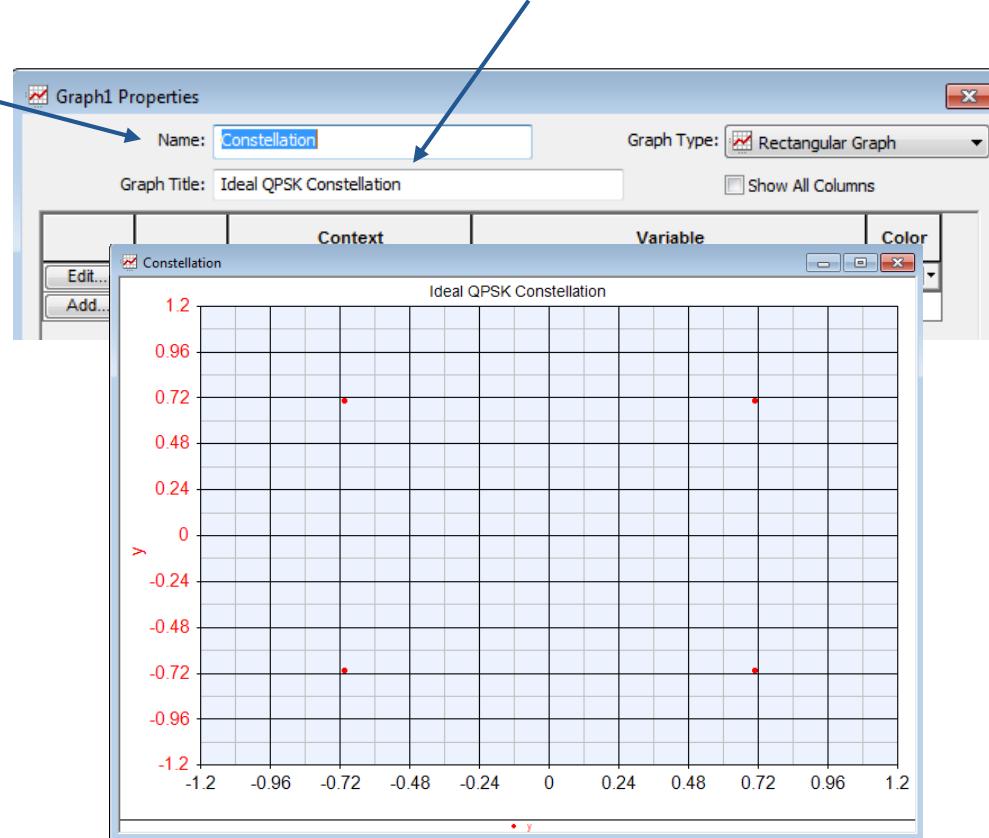
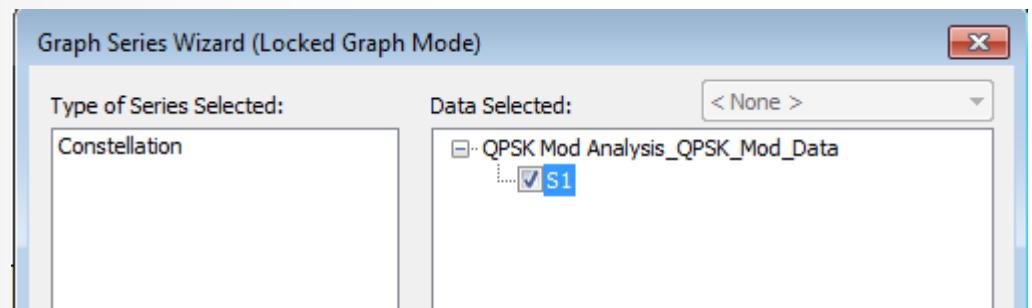
Lab 2: Run simulation

- 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

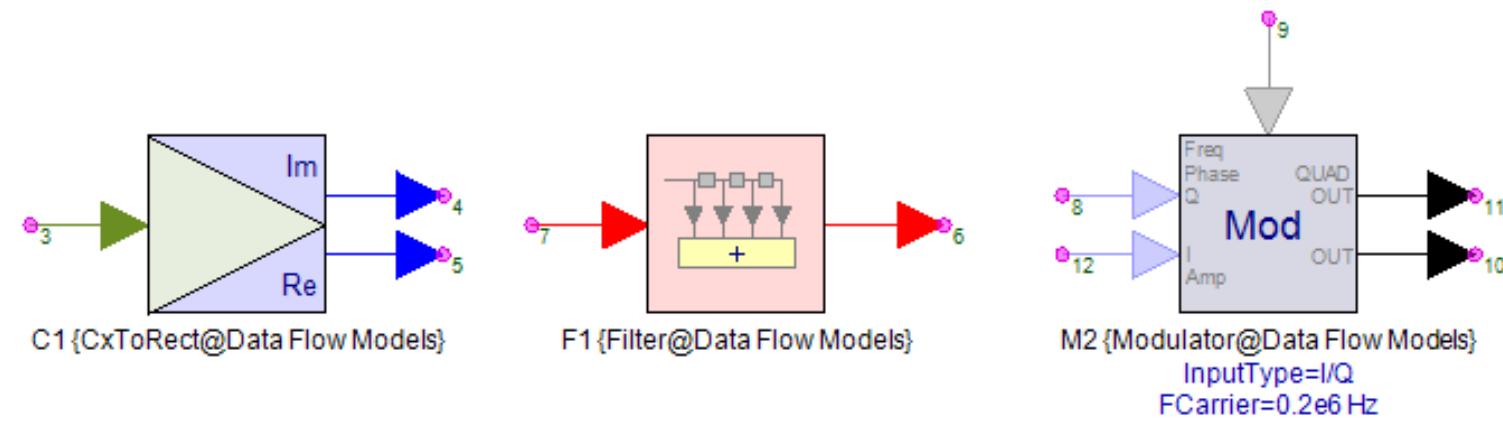
- Add a new graph, select **Series Type = Constellation**. Select **S1** (name of the sink). Click OK
- Set X and Y-axis limits as: Min = -1.2, Max = 1.2.
- Give graph name as **Constellation** and Graph Title as **Ideal QPSK Constellation**. Click OK to observe QPSK constellation diagram as shown here



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

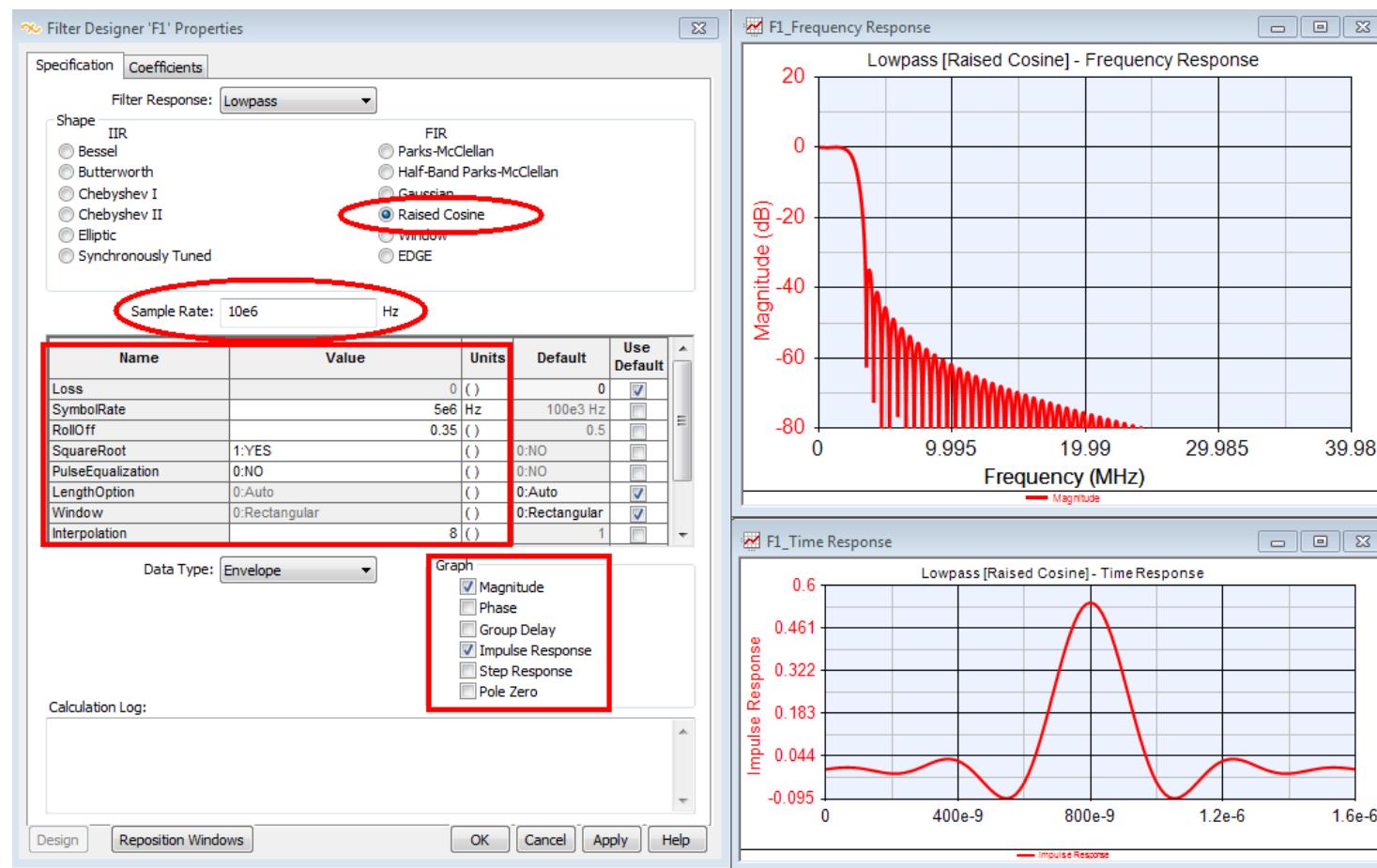
- Now, we have QPSK symbols available....we shall proceed ahead with rest of the Modulator design.
- Place **CxToRect** (Complex to Rectangular), **Filter** and **Modulator** from Algorithm Design library as shown here.
- There will be many filter components in the library, **select “Filter – Generic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



Name	Description
AdptFltAPA	APA Adapt...
AdptFltAPA_Cx	Complex A...
AdptFltCoreAPA	APA Adapt...
AdptFltCoreAPA_Cx	Complex A...
AdptFltCoreLMS	LMS Adapt...
AdptFltCoreLMS_Cx	Complex L...
AdptFltCoreRLS	RLS Adapti...
AdptFltCoreRLS_Cx	Complex R...
AdptFltLMS	LMS Adapt...
AdptFltLMS_Cx	Complex L...
AdptFltQR	QR Adapti...
AdptFltQR_Cx	Complex Q...
AdptFltRLS	RLS Adapti...
AdptFltRLS_Cx	Complex R...
BesselLPF	Bessel low...
Biquad	IIR Filter ...
BiquadCascade	IIR Filter ...
BlockAllPole	All-Pole Filt...
BlockFIR	FIR Filter f...
ErrorFilter	Error Filter
ErrorFilterCx	Complex E...
Filter	Generic filt...
FIR	FIR Filter
GaussianLPF	Gaussian I...

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

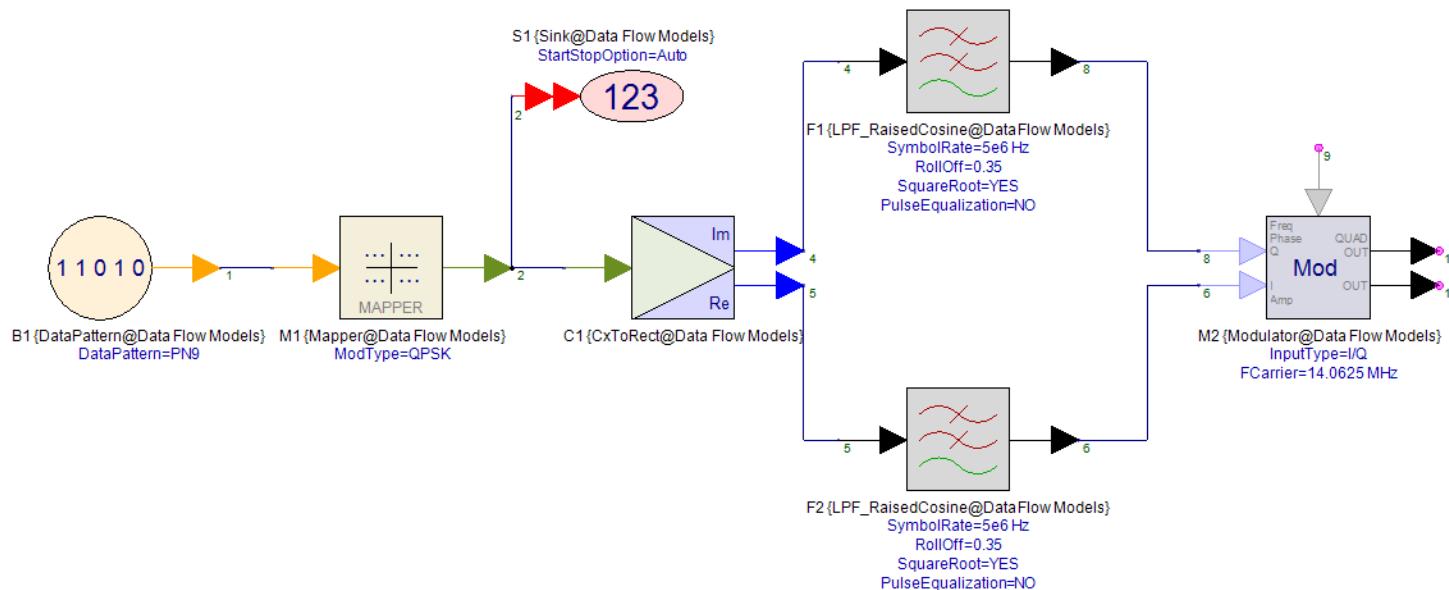
Lab 2: I/Q Modulator

- 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>>

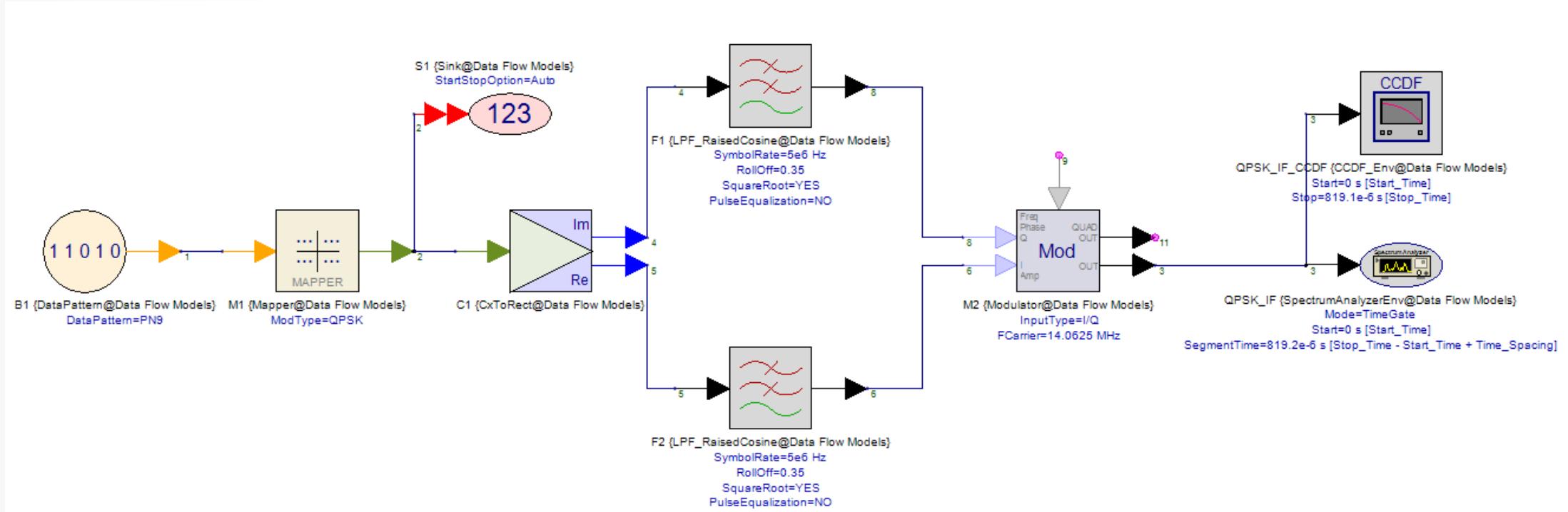
- **Copy/Paste LPF** for other branch and connect components as shown here:

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



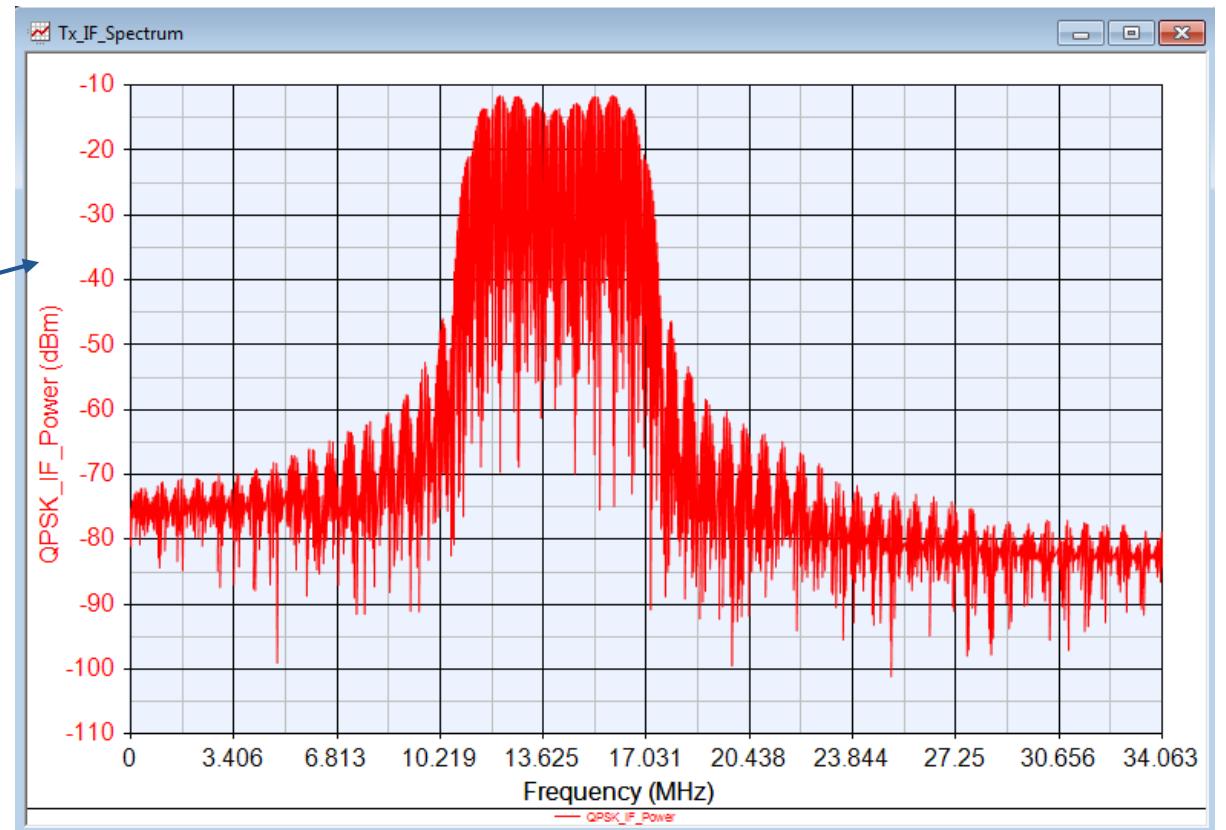
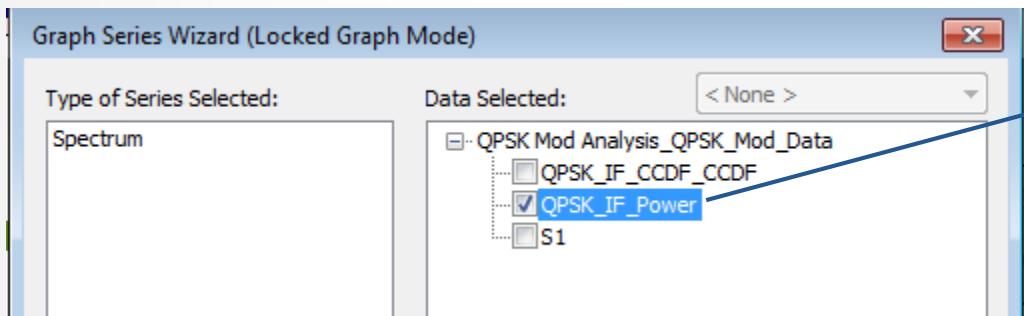
Lab 2: CCDF and Spectrum Analyzer

- 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)



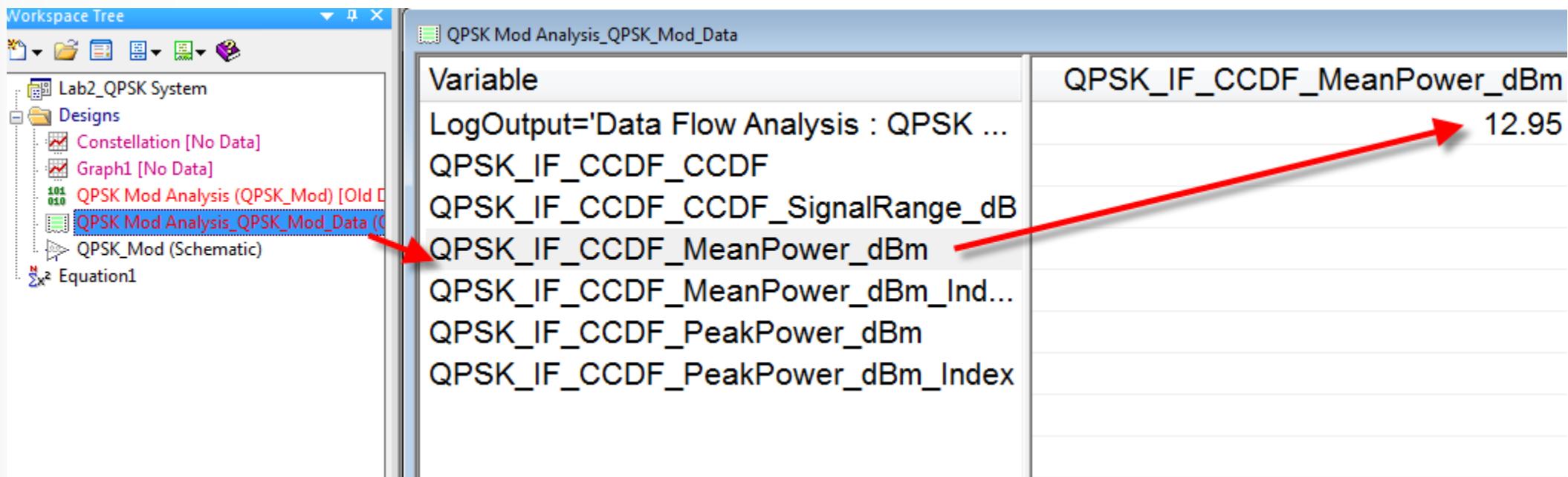
Lab 2: Plot spectrum

- 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

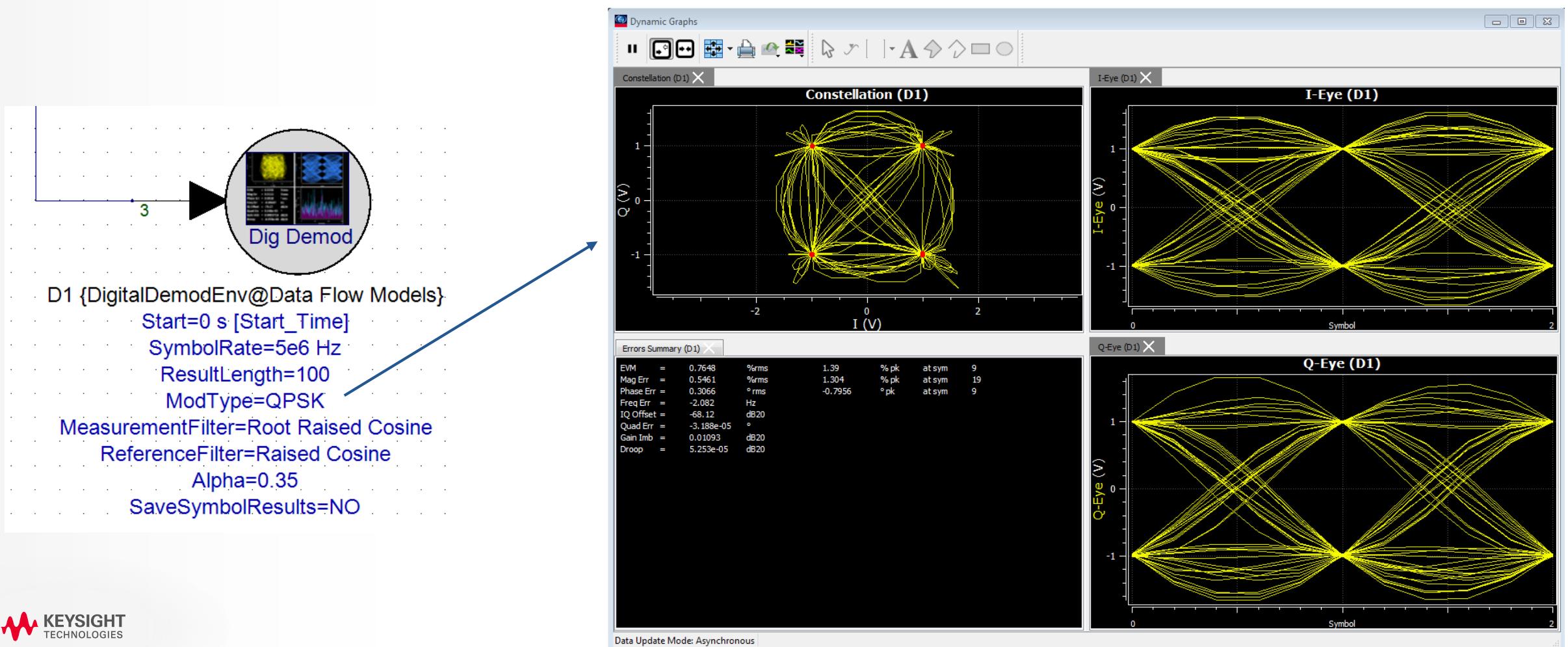
- Check Modulator output power, double click on dataset and look for MeanPower_dBm of the CCDF sink and notice output power is ~13dBm as needed.



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

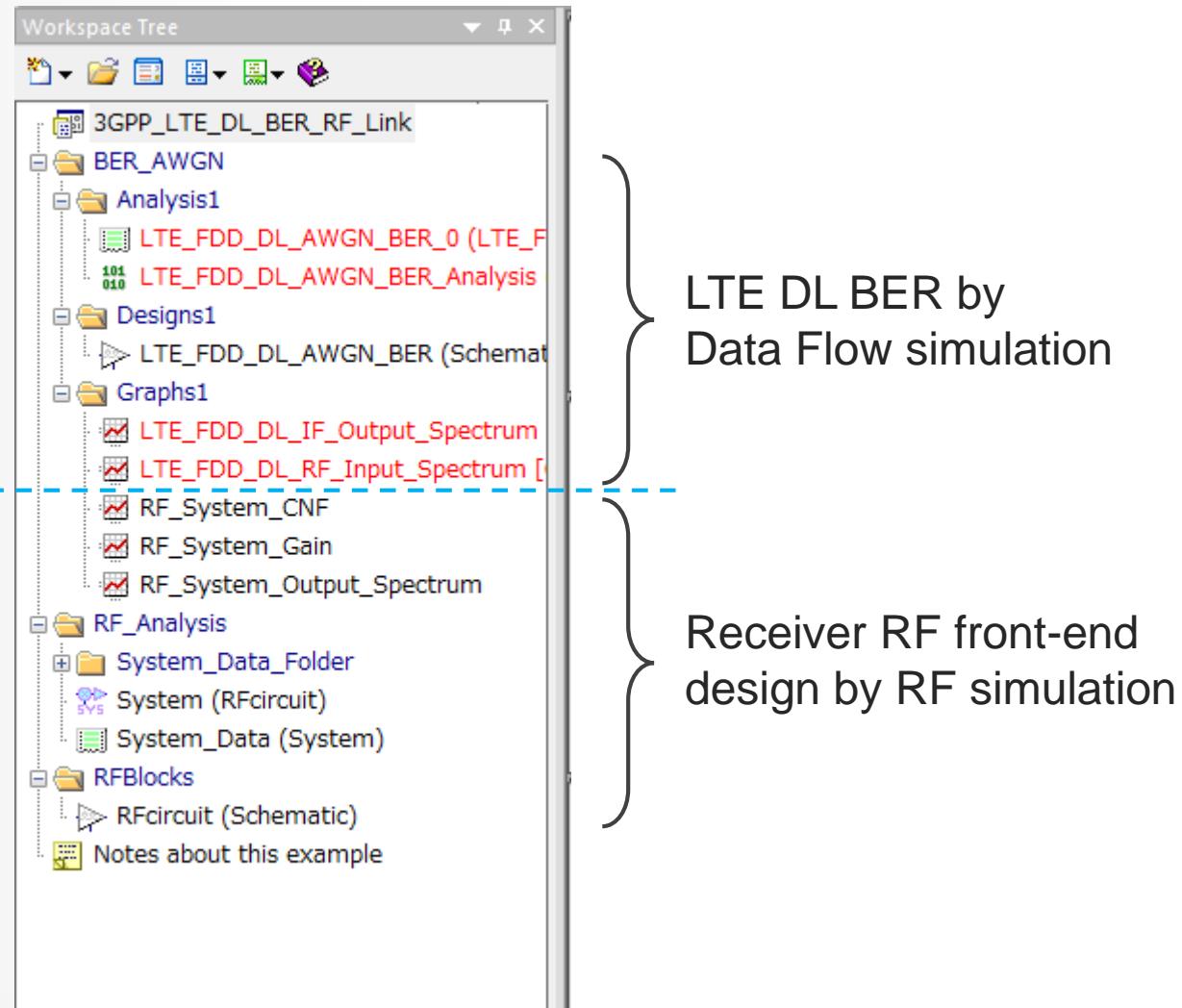
The screenshot shows the Keysight SystemVue 2018 software interface. On the left, the Example Explorer window is open, displaying a list of examples under the RF Architecture Design category. The '3GPP_LTE_DL_BER_RF_Link' example is selected and highlighted in blue. The main workspace contains several windows:

- Notes about this example**: A dialog box titled "3GPP_LTE_DL_SISO_BER_RF_Link" provides information about the example workspace.
- System_Data_Path1**: A table showing system data parameters:

Variable	Index	NodeNames	CNF (dB)
AN	1	1	0
CCOMP	2	6	0.5
CENT	3	8	0.500003
CF	4	10	2.045601
CGAIN	5	4	2.045597
CIMCP	6	2	2.049056

- LTE_FDD_DL_AWGN_BER**: A schematic diagram of the LTE FDD Downlink SISO BER and PER Measurements on AWGN Channel.
- RFcircuit**: A schematic diagram of the RF Front End LTE DL RX (in Spectrasys).
- LTE_FDD_DL_RF_Input_Spectrum**: A spectrum plot showing the input power spectrum.
- LTE_FDD_DL_IF_Output_Spectrum**: A spectrum plot showing the output power spectrum.
- RF_System_Gain**: A graph showing the system gain across frequency.
- RF_System_CNF**: A graph showing CNF (Carrier-to-Noise Ratio) across frequency.
- Errors**: A table showing errors and their locations.
- Command Prompt (MATLAB Script)**: A MATLAB command window.
- Part Selector A**: A panel listing various RF components with their descriptions.

Lab 3: Workspace overview



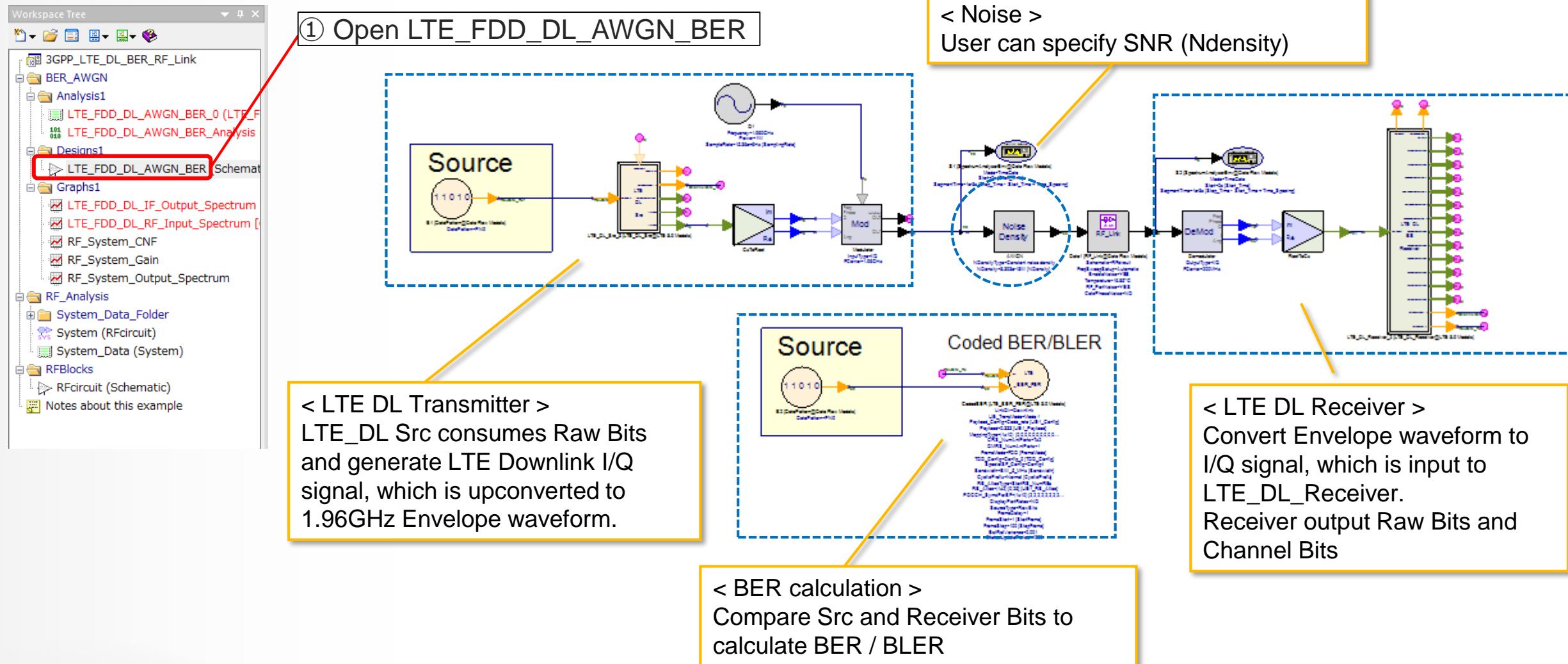
LTE DL BER by
Data Flow simulation

Receiver RF front-end
design by RF simulation

This Lab explains

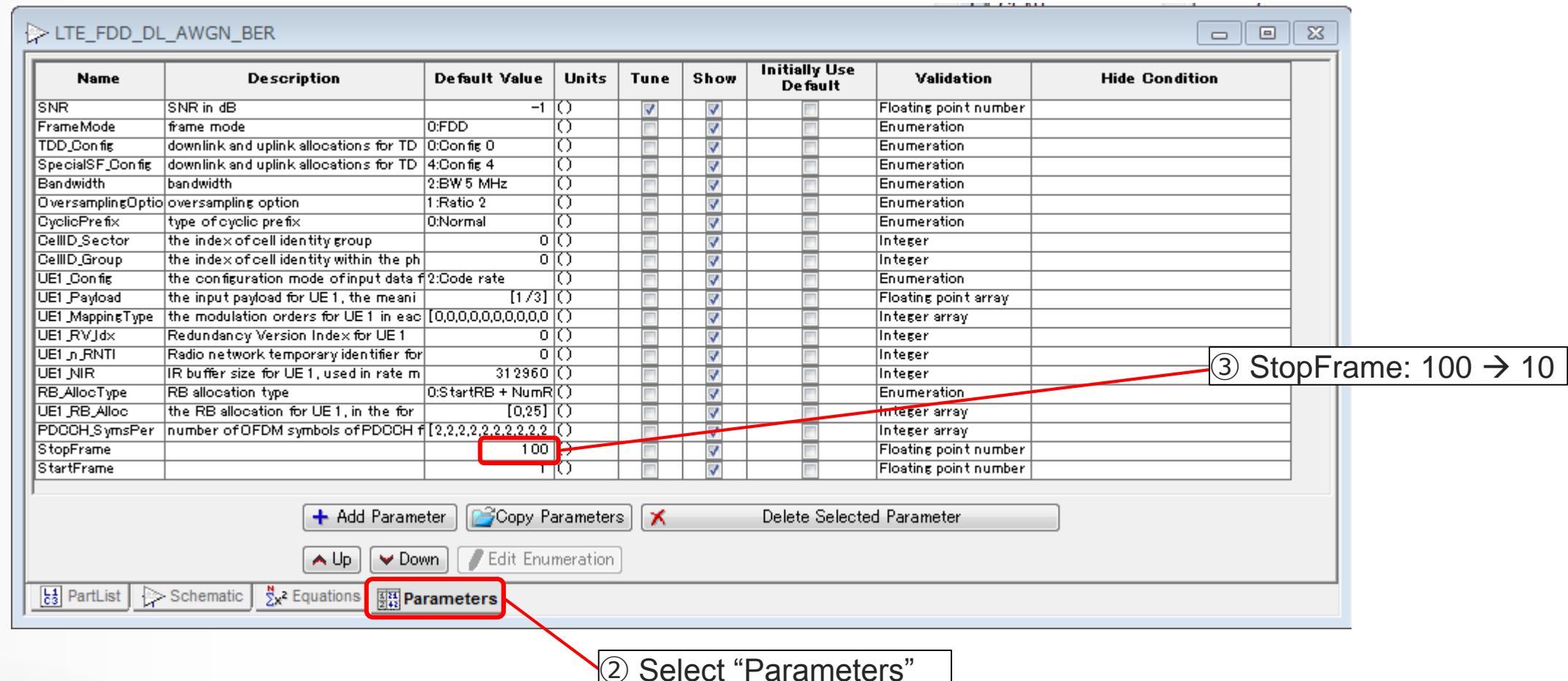
1. Overview of LTE system test bench
2. RF simulation (Spectrasys)
3. LTE system simulation with RF design

Lab 3: LTE_FDD_DL_AWGN_BER



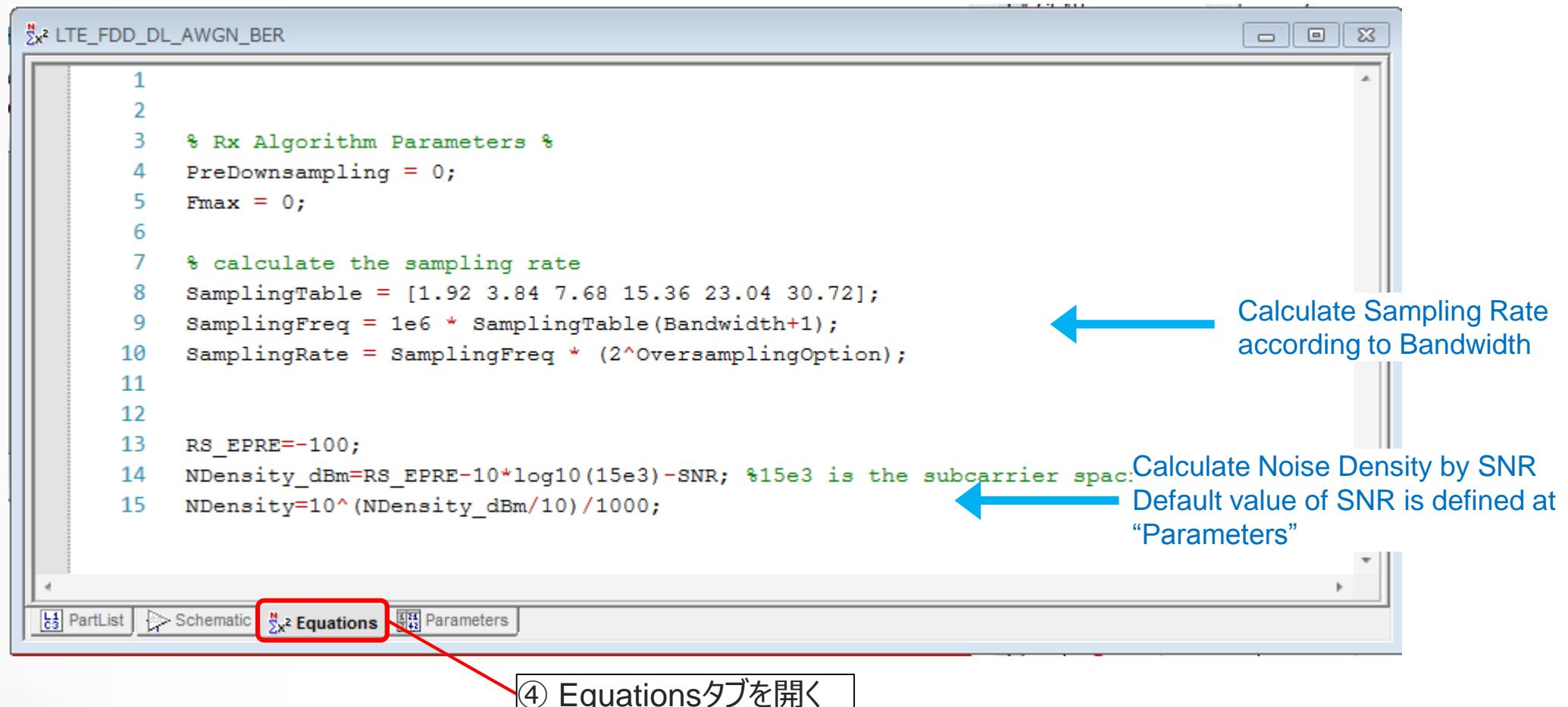
Lab 3: LTE_FDD_DL_AWGN_BER : Parameters

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



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”



```
Σx² LTE_FDD_DL_AWGN_BER
1
2
3 % Rx Algorithm Parameters %
4 PreDownsampling = 0;
5 Fmax = 0;
6
7 % calculate the sampling rate
8 SamplingTable = [1.92 3.84 7.68 15.36 23.04 30.72];
9 SamplingFreq = 1e6 * SamplingTable(Bandwidth+1);
10 SamplingRate = SamplingFreq * (2^OversamplingOption);
11
12
13 RS_EPRE=-100;
14 NDensity_dBm=RS_EPRE-10*log10(15e3)-SNR; %15e3 is the subcarrier spacing
15 NDensity=10^(NDensity_dBm/10)/1000;
```

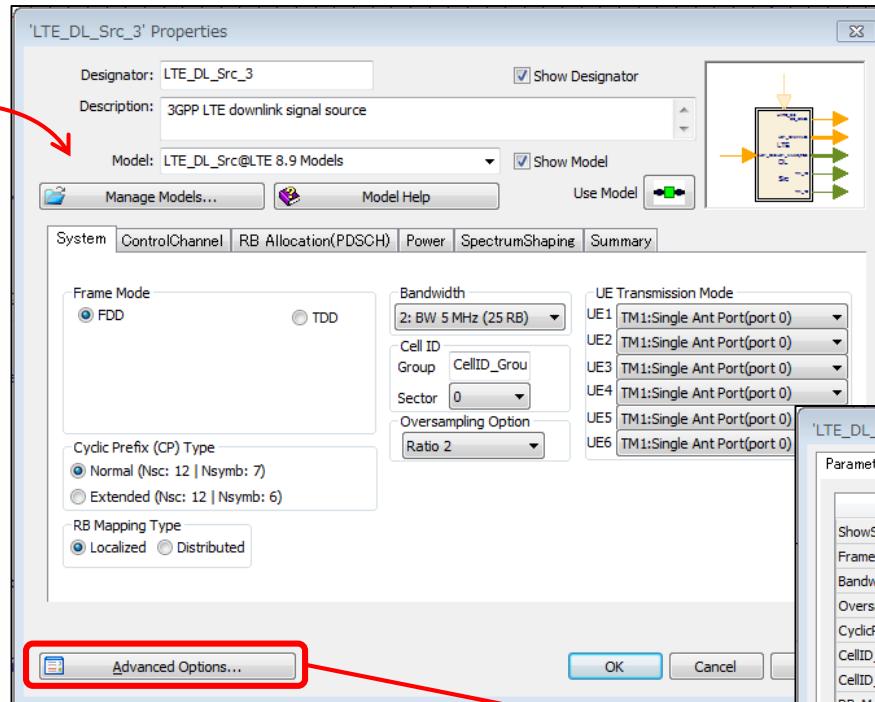
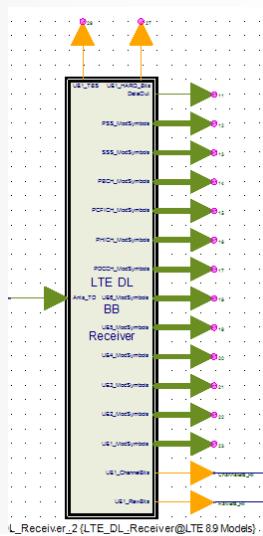
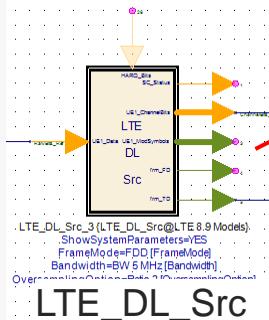
④ Equationsタブを開く

Calculate Sampling Rate according to Bandwidth

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

Lab 3: Source, Receiver model

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



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

'LTE_DL_Src_3' Properties

Parameters Symbol Netlist Parameter Statistics

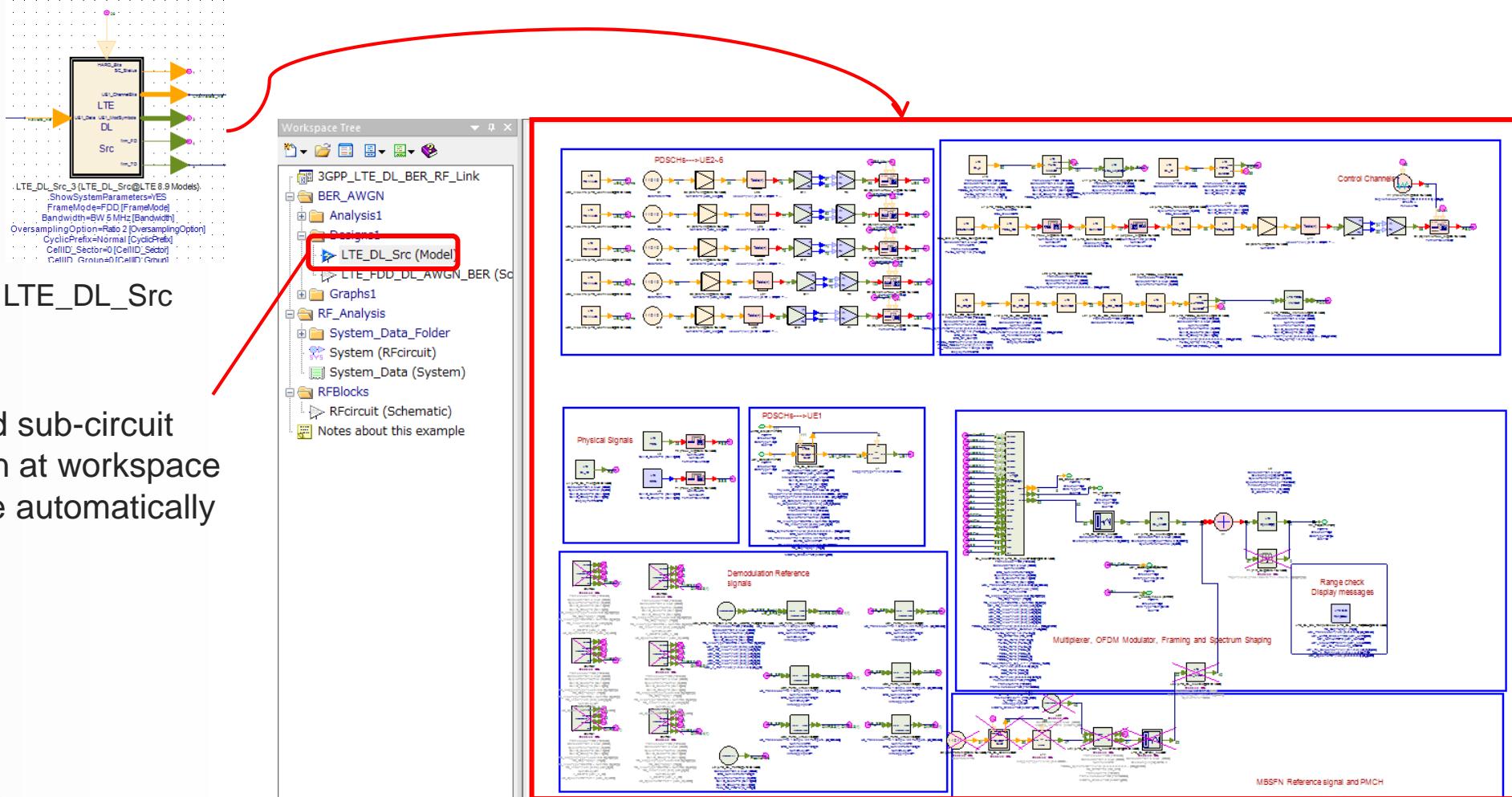
Name	Value	Units	Default	Use Default	Tune	Show
ShowSystemParameters	1: YES		1: YES	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
FrameMode	FrameMode		0:FDD	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Bandwidth	Bandwidth		2:BW 5 MHz	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
OversamplingOption	OversamplingOption		1:Ratio 2	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
CyclicPrefix	CyclicPrefix		0:Normal	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
CellID_Sector	CellID_Sector		0	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
CellID_Group	CellID_Group		0	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
RB_MappingType	0:Localized		0:Localized	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
RB_Gap	0:Ngap1		0:Ngap1	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
ShowUE1_Parameters	1: YES		1: YES	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
UE_TransMode	[0, 0, 0, 0, 0]		[0, 0, 0, 0, 0]	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
UE_SpecificRS	[0,0,0,0,0]		[0,0,0,0,0]	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>
UE1_HARQ_Enable	0: NO		1: YES	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>

Check Defaults Show All Model Help
Uncheck Defaults Hide All OK ヘルプ

GUI has multiple tabs

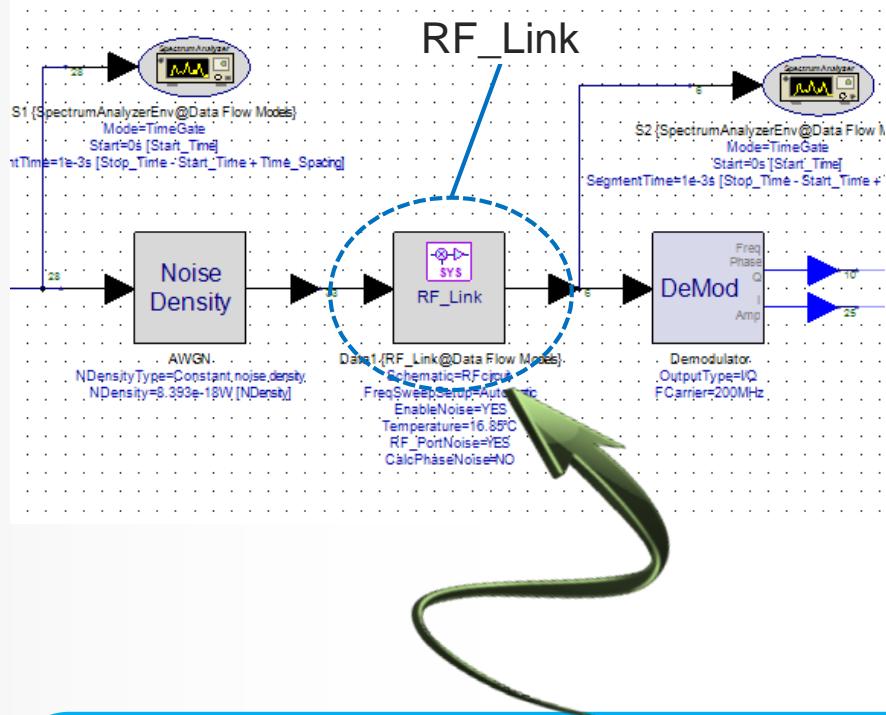
【Ref.】How to open sub-circuit

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

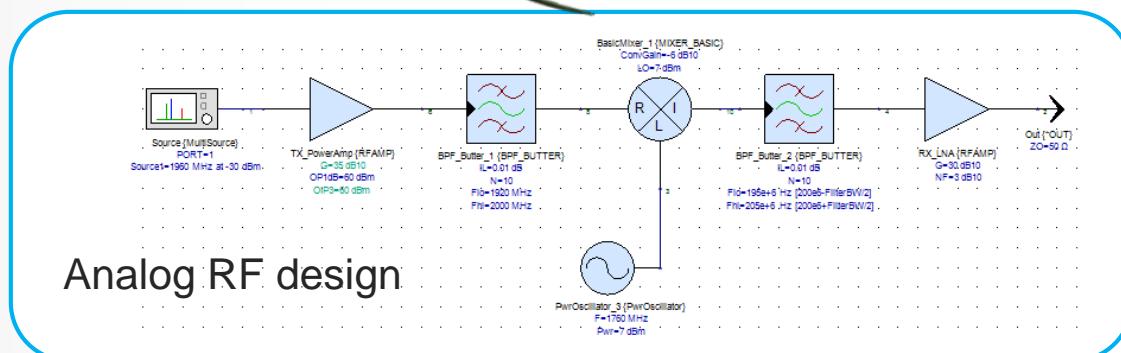


Add sub-circuit icon at workspace tree automatically

Lab 3: RF_Link component



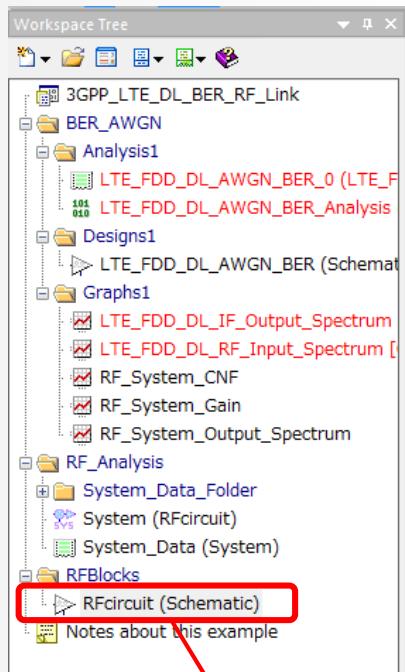
- RF_Link component connects Data Flow and RF front-end design that consists of parts in “RF Design” library.
- User can take RF model effect into account in LTE system .



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

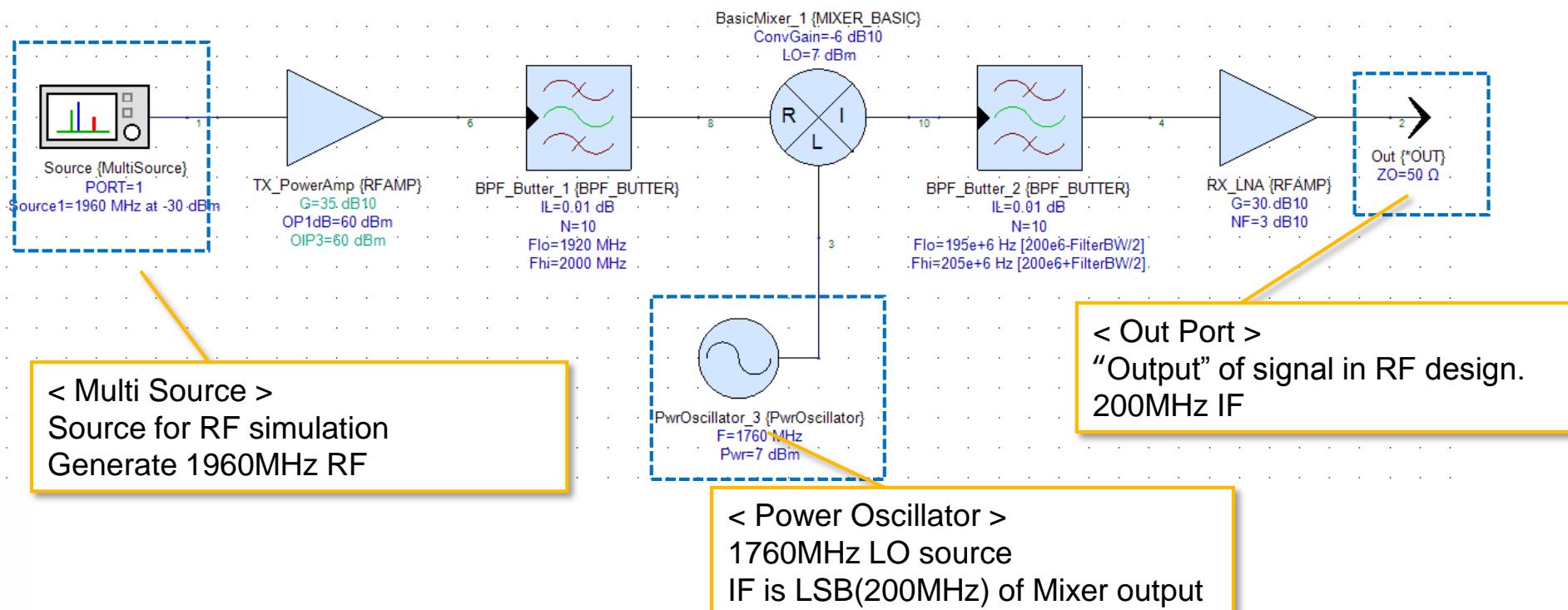
RF simulation (Spectrasys)

Lab 3: RFcircuit(design) overview



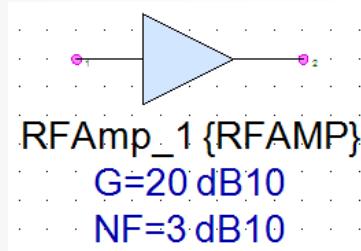
⑤ Open RFcircuit

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

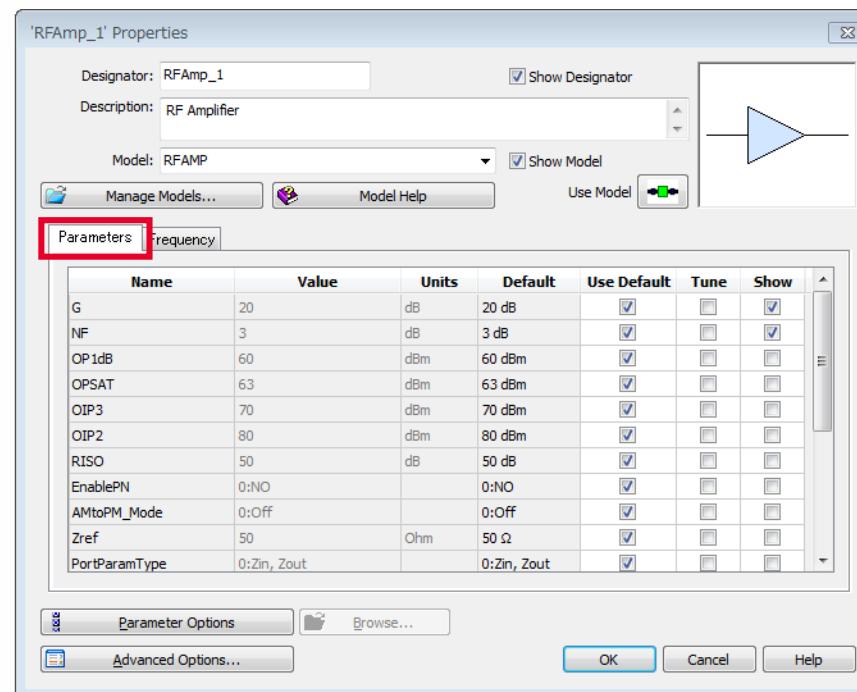


Lab 3: Model in RF simulation vs Data Flow

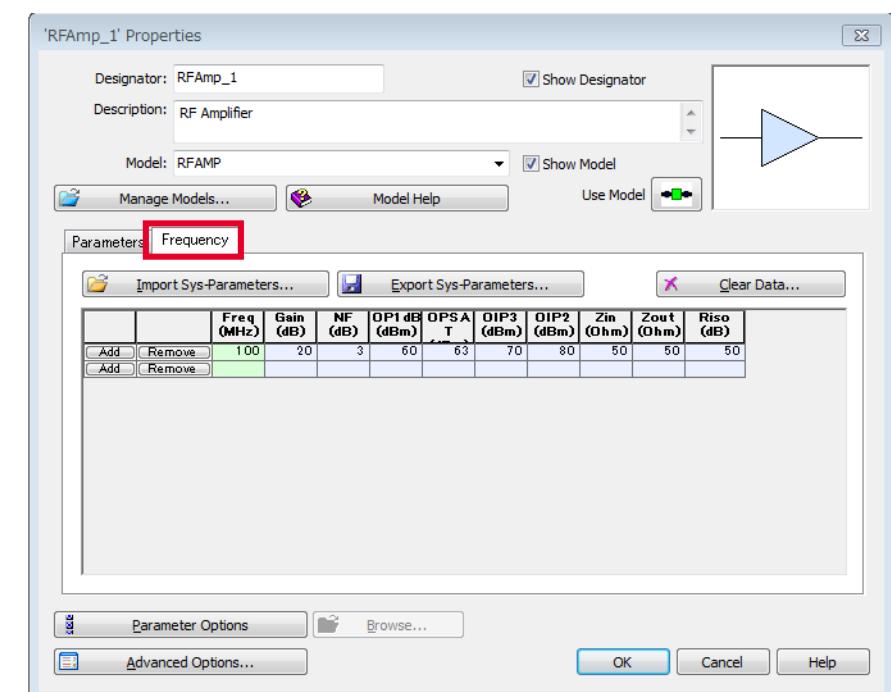
RF Amplifier



- 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



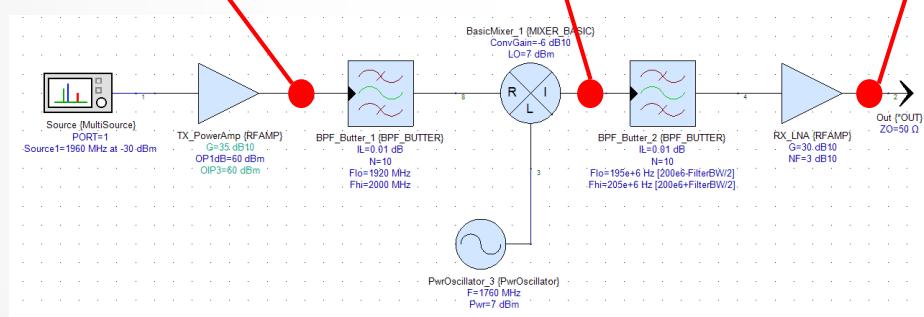
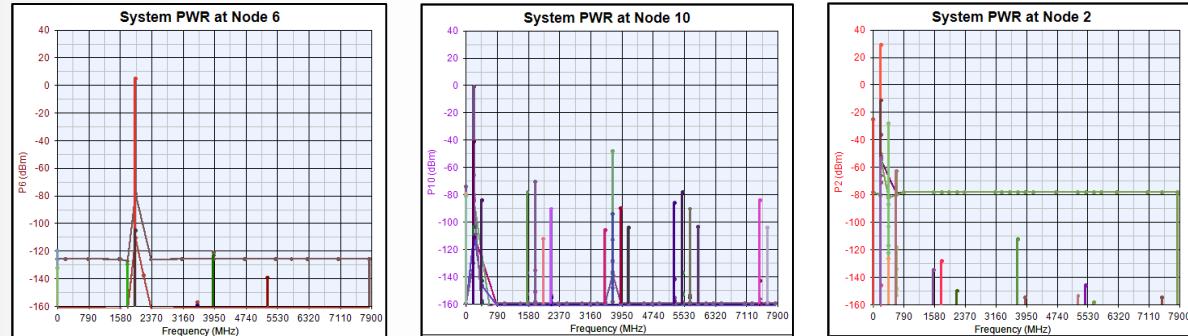
"Frequency" tab

Lab 3: Node Measurements and Path Measurements

Node Measurements

- Power/Voltage vs Frequency at each nodes

Spurious analysis at each Node



Path Measurements

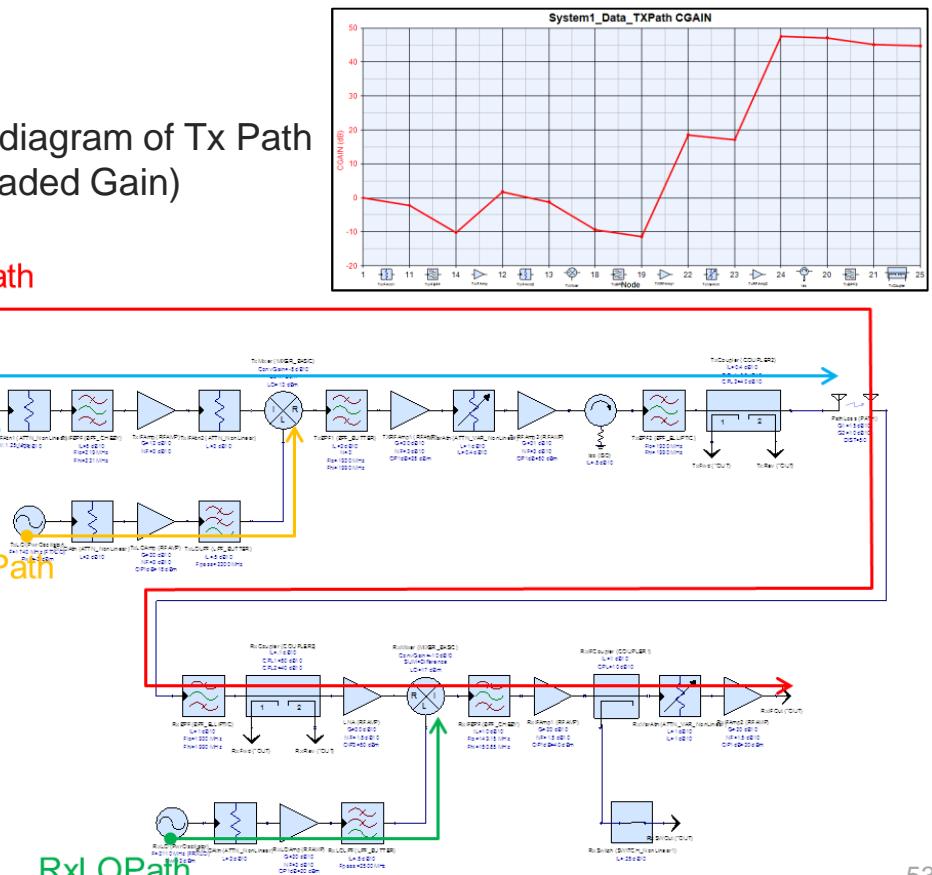
- Level diagram of Channel Power, Noise, InterMod, etc. at specified path.

Level diagram of Tx Path
(Cascaded Gain)

EntirePath

TxPath

TxLOPath



Lab 3: RF System Analysis controller : General

Set Design and Dataset

The screenshot shows the Keysight SystemVue workspace interface. On the left is the 'Workspace Tree' containing various project components like '3GPP_LTE_DL_BER_RF_Link', 'BER_AWGN', 'Analysis1', 'Designs1', 'Graphs1', 'RF_Analysis', and 'System_Data_Folder'. A red box highlights 'System (RFcircuit)' under 'System_Data_Folder'. A red arrow points from this highlighted item to the 'Design To Simulate' field in the 'System Simulation Parameters' dialog.

System Simulation Parameters Dialog:

- General Tab:** Contains fields for 'Design To Simulate' (set to 'RFcircuit'), 'Dataset' (set to 'System_Data'), 'Frequency Units' (MHz), 'Nominal Impedance' (50 Ohms), and 'Measurement Bandwidth' (Channel 1 MHz).
- Schematic Source Summary:** A table showing a single source named 'Source' with Net Name '1' and Description 'Universal Signal Source'.
- Buttons:** Includes 'Calculate Now' (highlighted with a red box), 'Save as Favorite...', 'Factory Defaults', 'OK', and 'キャンセル' (Cancel).

Context Menu (Run Simulation):

- Options include 'Run (calculate now)', 'Mark Results Up-To-Date', and 'Automatically Recalculate'.

Annotations:

- Select design(Schematic) to simulate:** Points to the 'Design To Simulate' field in the dialog.
- Dataset name:** Points to the 'Dataset' field in the dialog.
- Run simulation:** Points to the 'Calculate Now' button in the dialog.
- Similar to ResBW of Spectrum Analyzer.** Points to the 'Measurement Bandwidth' section in the dialog.
- Noise power and wideband signal is estimated at per Channel BW** Points to the 'Measurement Bandwidth' section in the dialog.
- Tips :** Points to the context menu with the text 'Can use "Run" by right mouse click on simulation controller'.

⑥ Open System controller

Lab 3: RF System Analysis Controller : Paths

Set Path to calculate Level Diagram

System Simulation Parameters

Paths

Name	Description	Enable
Path1	Parts: Source,Out Beginning Path Frequency	<input checked="" type="checkbox"/> Edit... <input type="button" value="Delete"/>
		<input type="button" value="Add"/>

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

Edit Path

Name: Path1
 Use auto-description

Specify Path Using: Part Names
+ Add Part To Path...
Path: Source,Out
Channel Frequency: MHz (Defaults to single frequency at beginning of path)

Force path through Switch state
 Allow path to begin on internal node

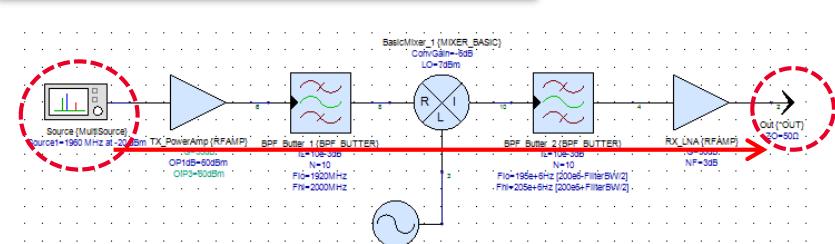
Set the start/end point of path

Current Path : Source → Out

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

Tips :
There are many analysis items in "Output"

OK Cancel Help



Lab 3: Run simulation and plot graph

⑦ Run Simulation

Push [Calculate Now] button to run RF simulation

➤ Plot graph

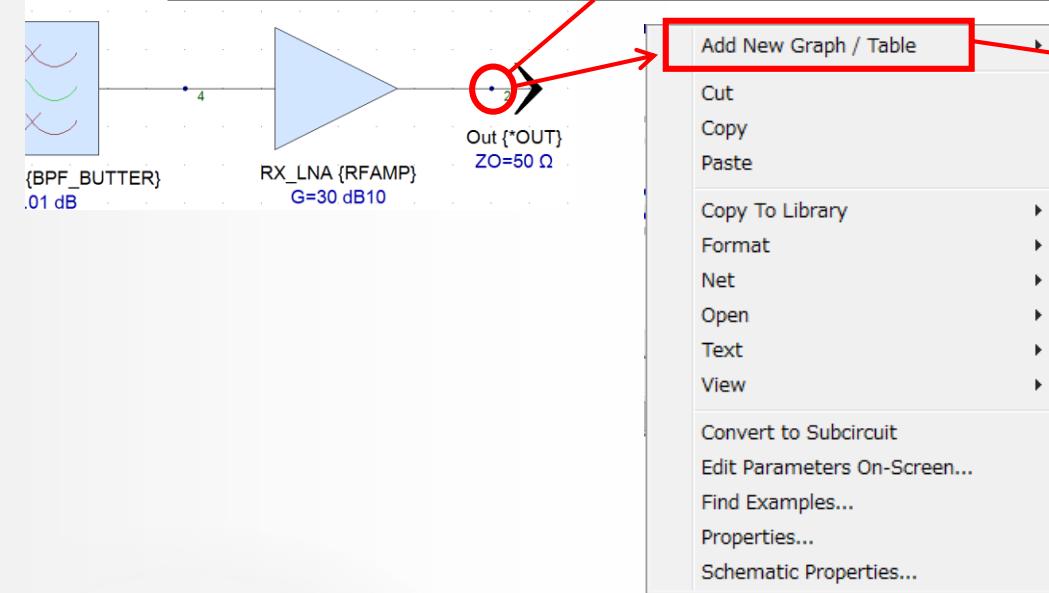
After simulation finished, user can create plot from schematic node directly.

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

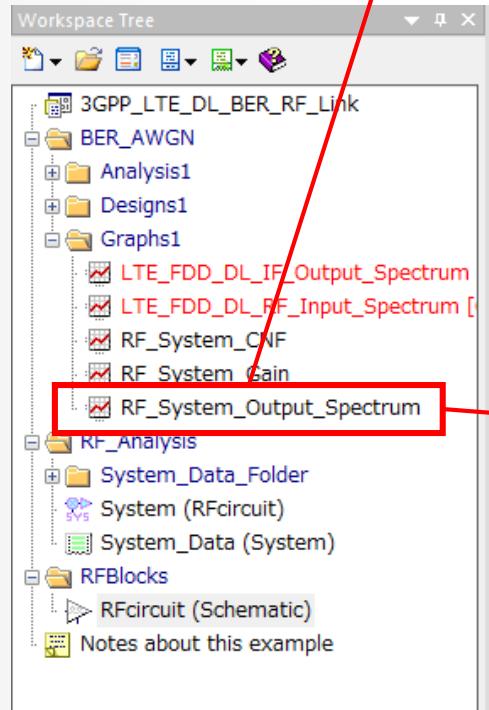
★: already created in workspace tree



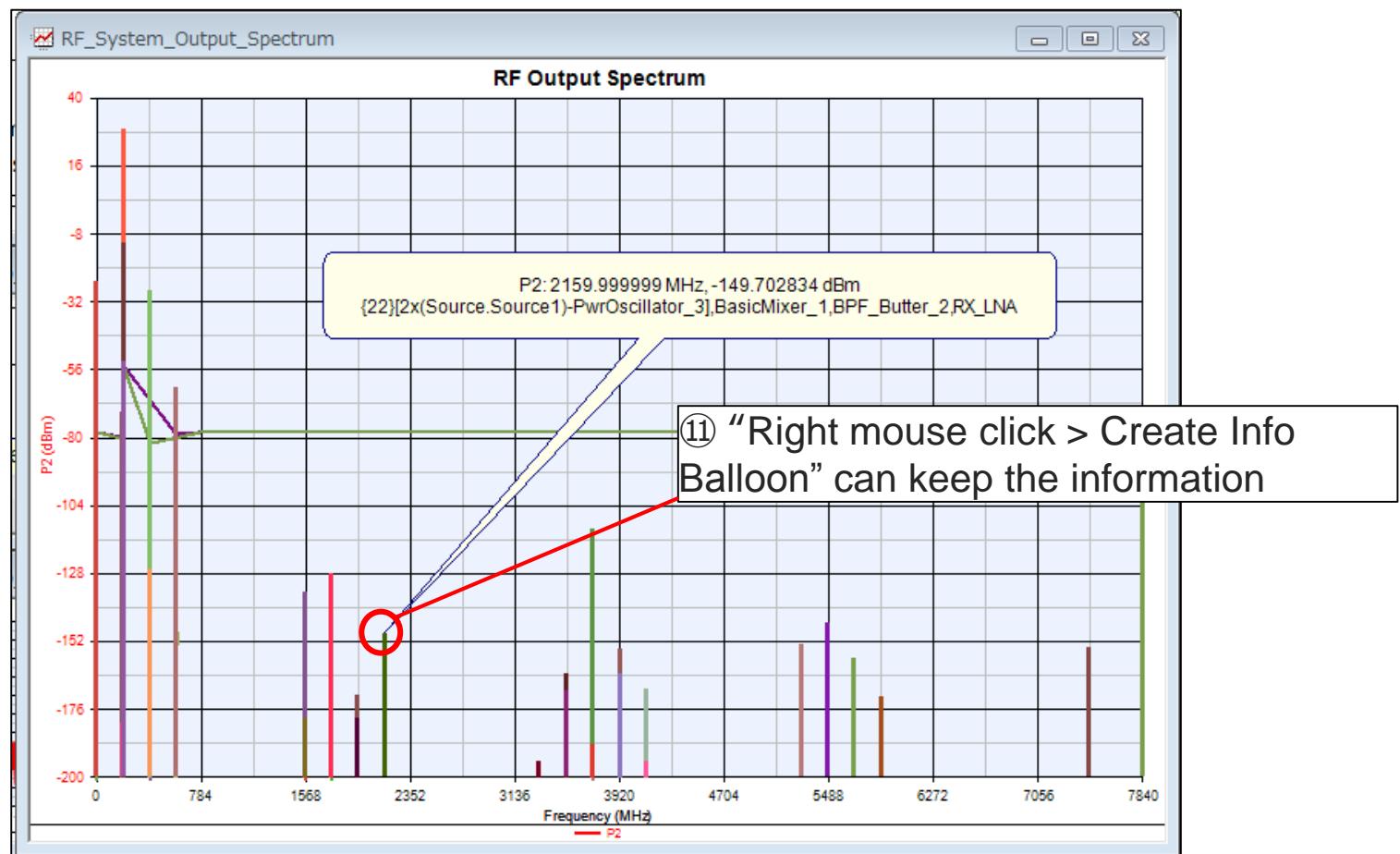
- System_Data: New Power Plot at Node 2 ★
- System_Data: New Voltage Plot at Node 2
- System_Data: New Phase Plot at Node 2
- System_Data_Path1: New Level Diagram of CP (Channel Power)
- System_Data_Path1: New Level Diagram of CGAIN (Cascaded Gain)
- System_Data_Path1: New Level Diagram of GAIN (Stage Gain) ★
- System_Data_Path1: New Level Diagram of CNDR (Carrier to Noise and Distortion Ratio)
- System_Data_Path1: New Level Diagram of CNP (Channel Noise Power)
- System_Data_Path1: New Level Diagram of CNF (Cascaded Noise Figure) ★
- System_Data_Path1: New Level Diagram of SDR (Stage Dynamic Range)
- System_Data_Path1: New Table of Measurements

Lab 3: Node Measurement : RF System Output Spectrum

⑩ Open RF_System_Output_Spectrum

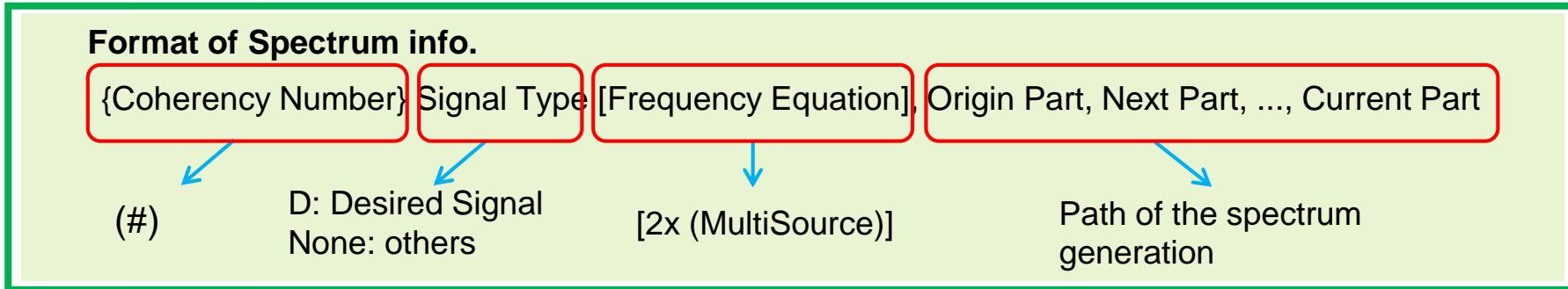


Point by mouse at each spectrum, then show the power and root-cause(path and the mixing order(Frequency Equation))



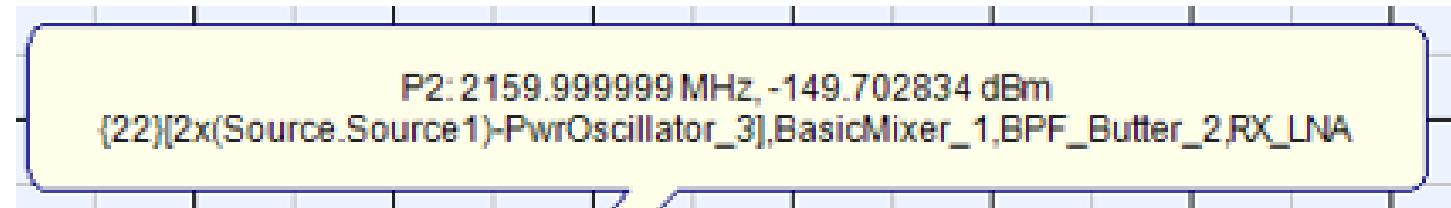
【Ref.】Format of Root Cause analysis

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



(#) 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.

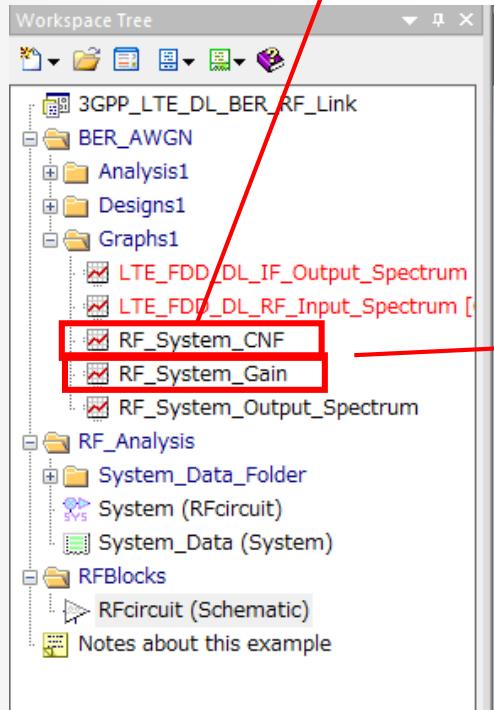
E.g. 2160MHz at previous page



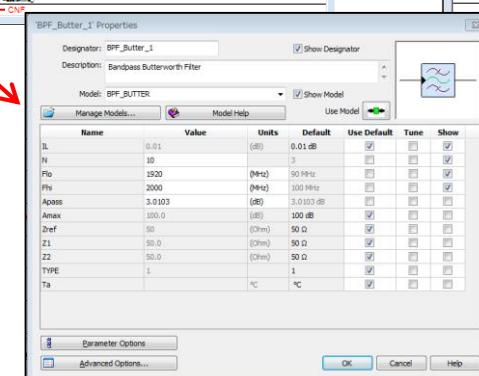
Signal Type	: Not D → Spurious
Frequency Equation	: 2xRF - LO
Origin and route	: BasicMixer → BPF_Butter_2 → RX_LNA → Out

Lab 3: Path Measurement : System_CNF & System_Gain

- ⑯ Open RF_System_CNF and RF_System_Gain



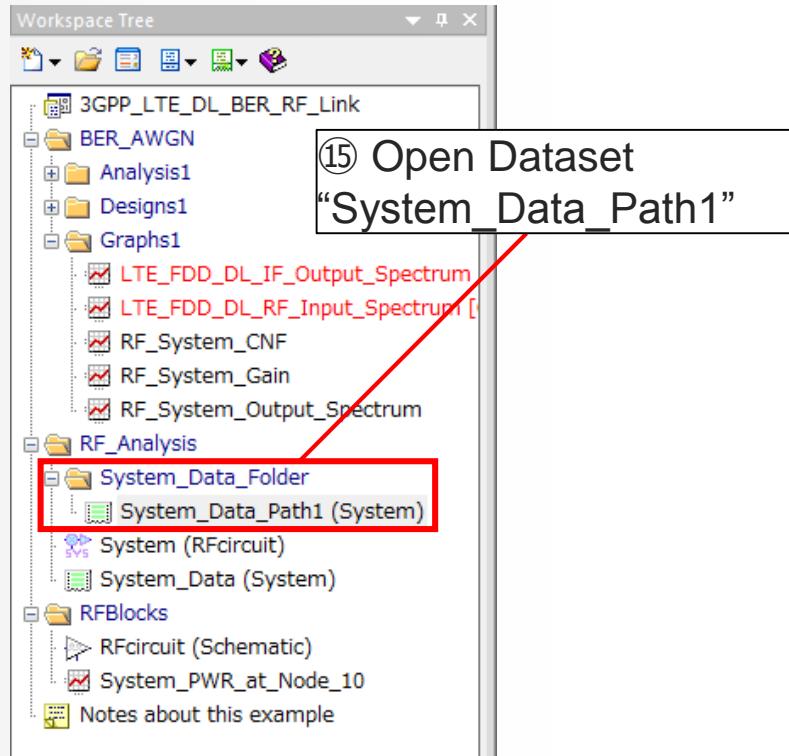
Path Measurement shows level diagram along the specified path: Source → Out
Followings are Cascaded Noise Figure and Stage Gain plot.



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



Variable	Index	NodeNames	CNF (dB)
AN	1	1	0
CCOMP	2	6	0.5
CENT	3	8	0.500003
CF	4	10	2.045601
CGAIN	5	4	2.045607
CIMCP	6	2	2.049066

Variable	Value
IP1DB	SIP1DB
IPSAT	SIPSAT
MDS	SNF
MML=108.577e-9 dB	SOIP
NDCP	SOIP2
NodeNames	OP1DB
OP1DB	SOIP3
EIIP2	SOP1DB
EIIP3	SOPSAT
EIP1DB	SrcFreq
EIPSAT	SrcPower
EMDS	SDR
EOIP2	SGAIN
EOIP3	SIP1DB
EOP1DB	SNF
EOPSAT	TNP
ESFDR	UDCP
GAIN	VDC
GIMCP	ZIN
GIMCP2	ZOUT
GIMCP3	

Examples of Variables included in Path Measurements

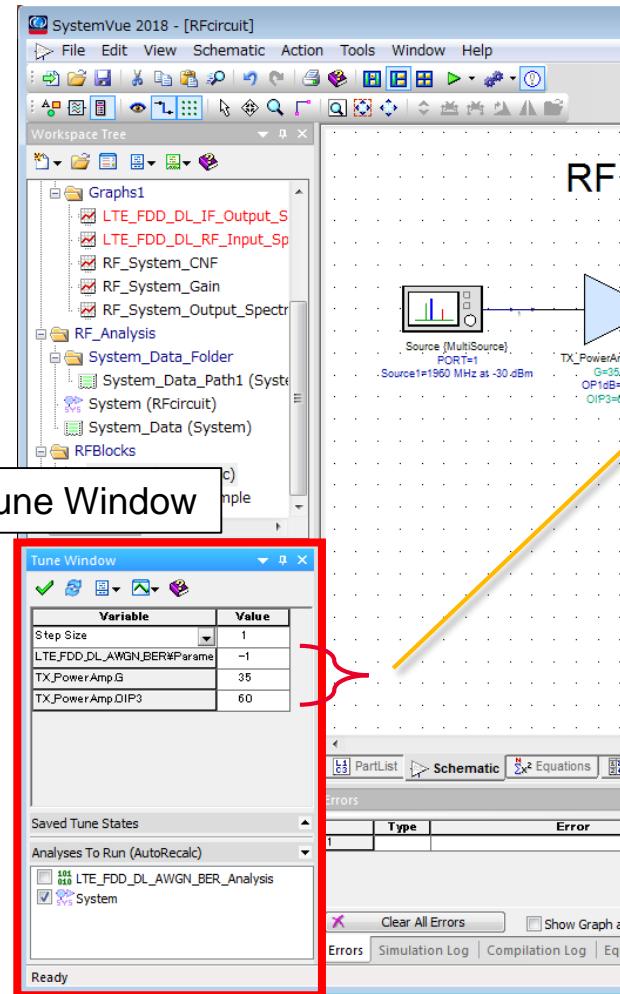
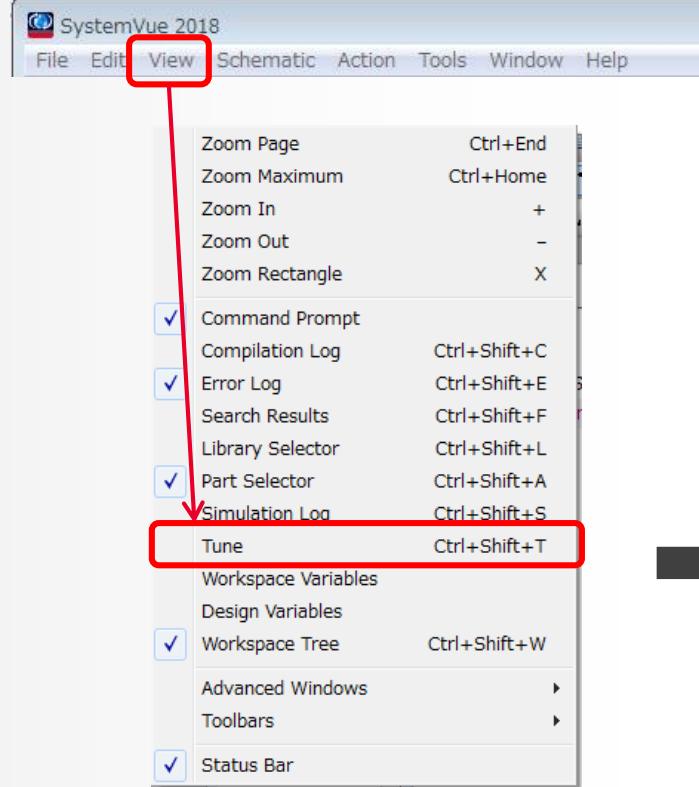
CGAIN	Cascaded Gain
CND	Channel Noise Density(1Hz当たりのCNP)
CNF	Cascaded Noise Figure
CNP	Channel Noise Power
CNR	Carrier to Noise Ratio
CP	Channel Power
DCP	Desired Channel Power
ECGAIN	Equation baseのCascaded Gain
ECNF	Equation baseのCascaded Noise Figure
PNCP	Phase Noise Channel Power
SDR	Stage Dynamic 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

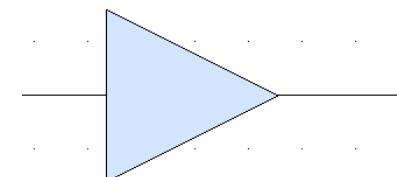


Tunable parameters in whole workspace are listed

SNR : N/A at this time (it is for Data Flow design)

G : TX_PowerAmp > Gain

OIP3 : TX_PowerAmp > Output Intercept Point



TX_PowerAmp {RFAMP}
G=35 dB
OP1dB=60 dB
OIP3=60 dB

Continue to the next page

Lab 3: Tune parameter

The screenshot shows the 'Tune Window' dialog box. At the top are several icons: a green checkmark, a blue floppy disk, a grey square with a downward arrow, a plot icon, and a purple gear icon. Below these are two buttons: 'Step Size of Tune' and 'Parameter name and its value'. A large red box highlights the main table area. The table has columns for 'Variable' and 'Value'. It contains the following data:

Variable	Value
Step Size	1
LTE_FDD_DL_AWGN_BER#Paramet	-1
TX_PowerAmp.G	35
TX_PowerAmp.DIP3	60

Below the table is a section titled 'Saved Tune States' with a dropdown menu. Under 'Analyses To Run (AutoRecalc)', there are two entries: 'LTE_FDD_DL_AWGN_BER_Analysis' (unchecked) and 'System' (checked). A red box highlights the checked 'System' entry.

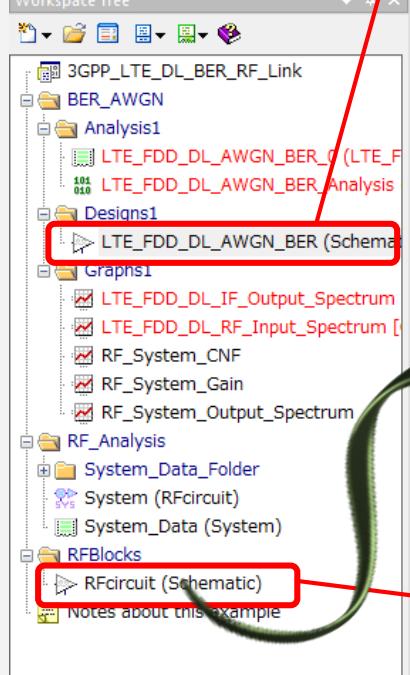
Callouts and Annotations:

- 'Apply tuned value to the plot': Points to the 'Parameter name and its value' button.
- 'Refresh tuned value': Points to the green checkmark icon.
- 'Config menu of Tune Window': Points to the blue floppy disk icon.
- 'Select plot to show Check Point': Points to the plot icon.
- 'Step size of Tune': Points to the 'Step Size of Tune' button.
- 'Parameter name and its value': Points to the table area.
- 'Check to run simulation automatically during Tune': Points to the 'System' entry in the 'Analyses To Run' list.
- 'Input the value directly': Points to the 'Value' column in the table.
- 'Click upper/lower icon (to change by Step Size)': Points to the up and down arrows in the 'Value' column of the table.
- 'During Tune, level diagram shows "before/after" lines.'': Points to the 'RF_System_Gain' and 'RF_System_CNF' plots below.

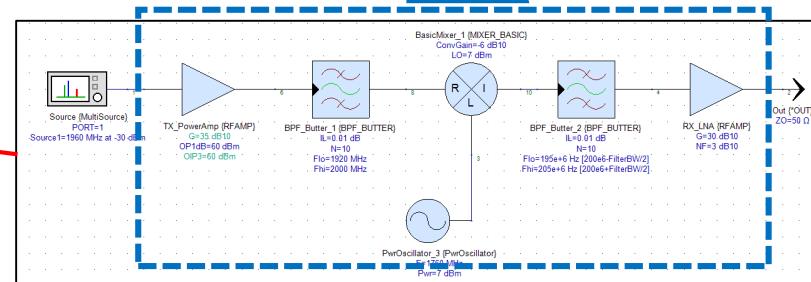
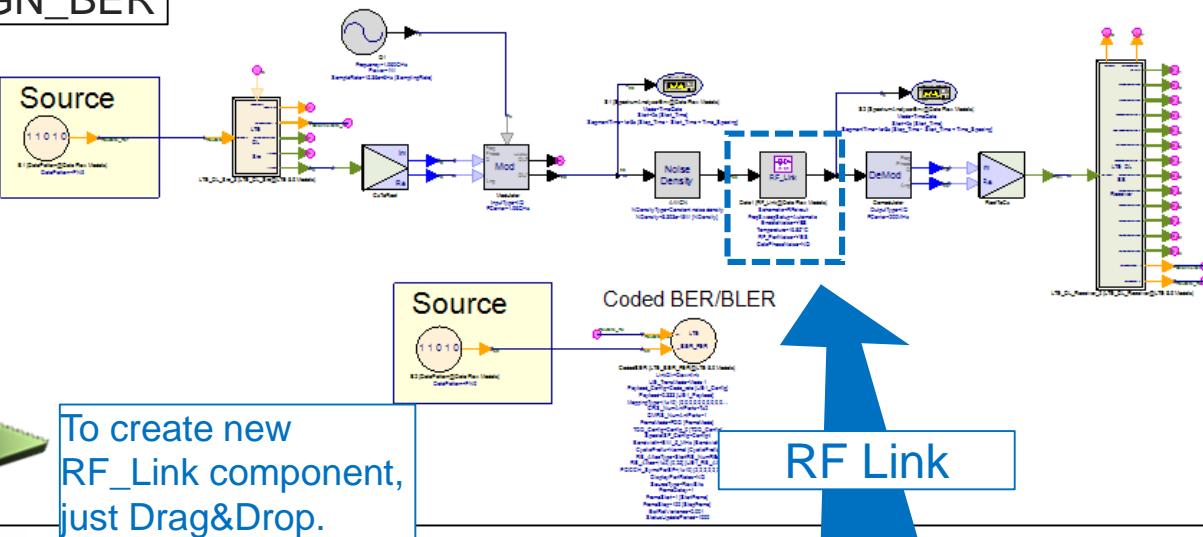
RF Link

Lab 3: How to use “RF_Link”

① Open LTE_FDD_DL_AWGN_BER



To create new
RF_Link component,
just Drag&Drop.



RFcircuit

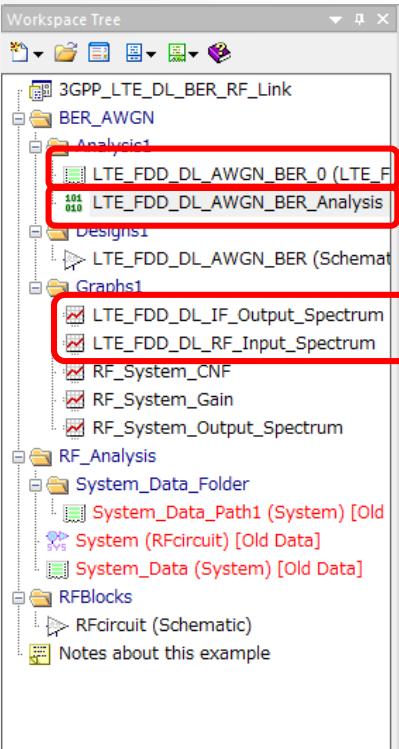
To consider RF front-end design in the system, user can use RF_Link.

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.

Lab 3: Run simulation and the result

② Right mouse click > Run (calculate now)

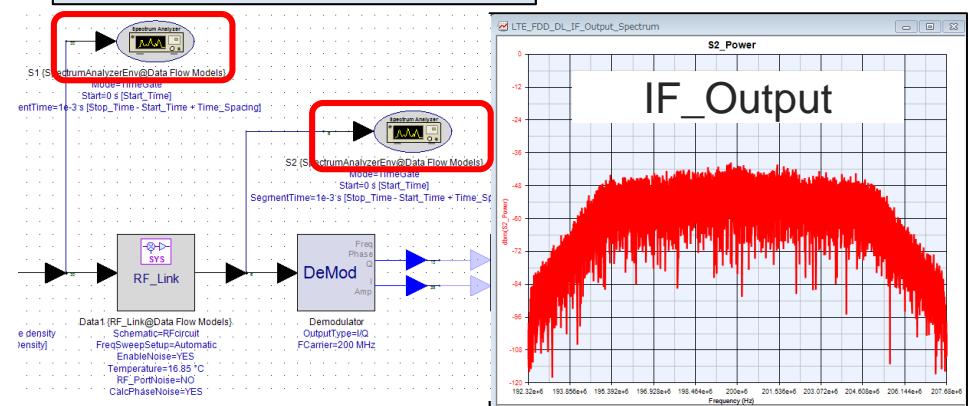
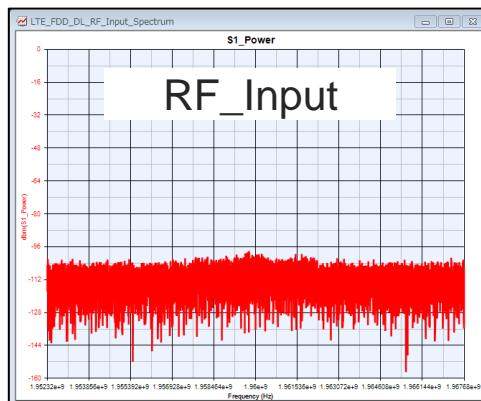


③ Open DataSet of
LTE_FDD_DL_AWGN_BER_0 and
check CodedBER_BER or
CodedBER_FER

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

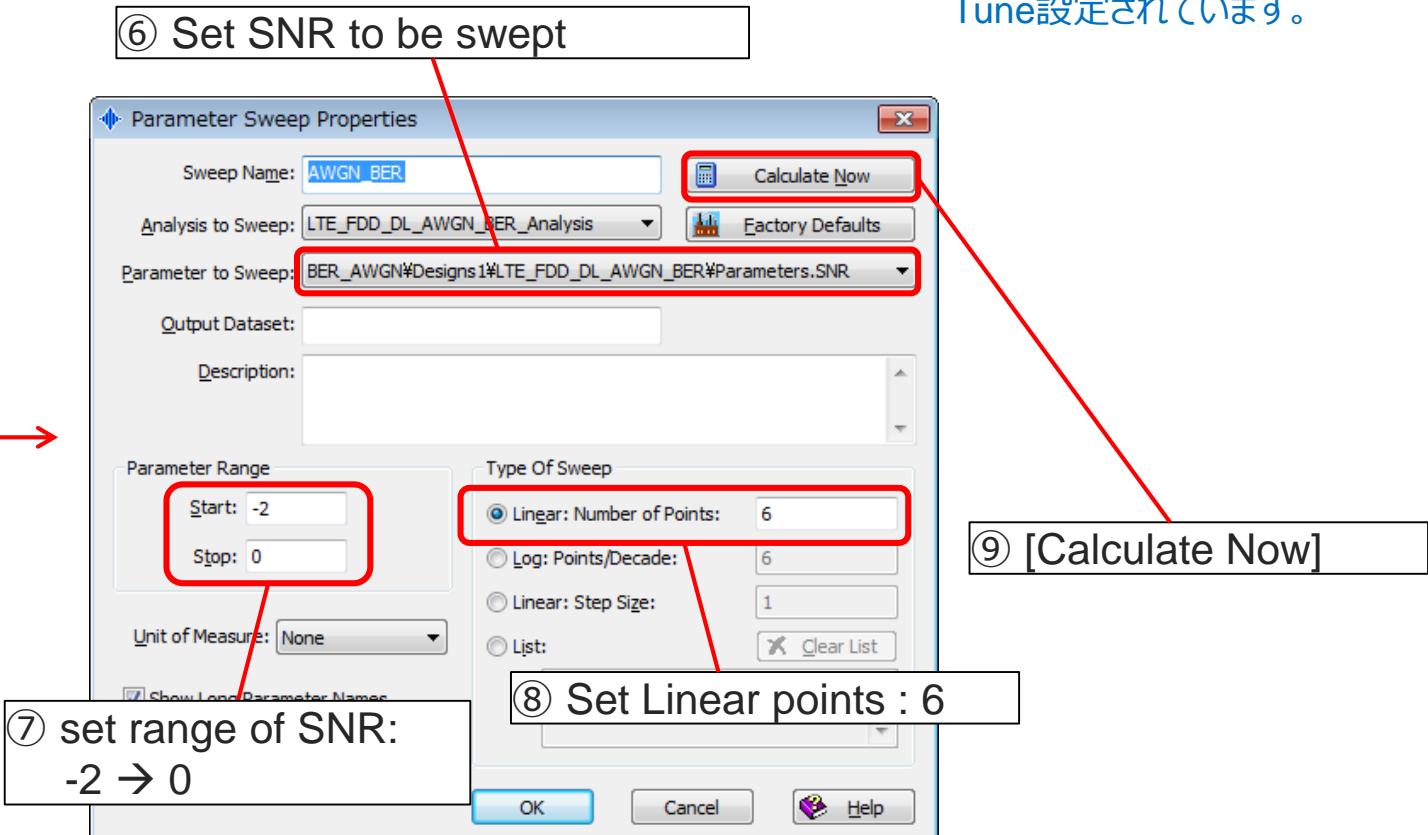
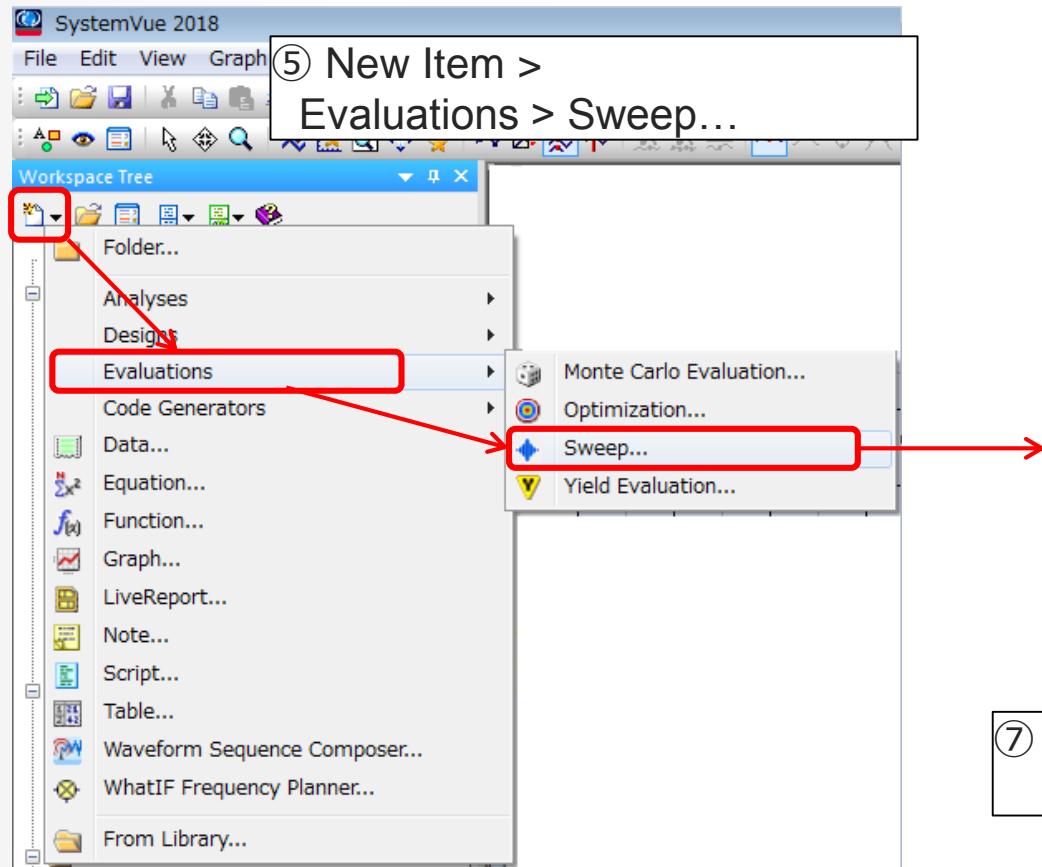
Variable	CodedBER_BER_Index	CodedBER_BER
CodedBER_BER	0	6.035106e-3

Variable	CodedBER_FER_Index	CodedBER_FER
CodedBER_FER	0	0.289474



Lab 3: Sweep SNR

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



It will take several minutes to complete

Lab 3: BER vs SNR

⑩ Open Sweep1_Data

The screenshot shows the Keysight SystemVue workspace. On the left is the 'Workspace Tree' containing various project components like '3GPP_LTE_DL_BER_RF_Link', 'BER_AWGN', 'Analysis1', 'Designs1', 'Graphs1', 'RF_Analysis', and 'RFBlocks'. A red box highlights 'Sweep1_Data (Sweep1)' under 'Analysis1'. The main area displays a table for the 'CodedBER_BER' variable:

Variable	Index	CodedBER_BER_Index	Parameters_SNR_Swp_CodedBER_BER_Index	CodedBER_BER
CodedBER_BER	1	0	-2	0.117545
CodedBER_BER_Index		0	-1.6	0.083137
CodedBER_FER		0	-1.2	0.022574
CodedBER_FER_Index		0	0.8	1.039367e-3
LogOutput=Sweep : S		0	-0.4	0
Parameters_SNR_Swp		0	0	0
Parameters_SNR_Swp		0	0	0

A right-click context menu is open over the first row of the table, specifically over the 'CodedBER_BER' entry. The menu items are:

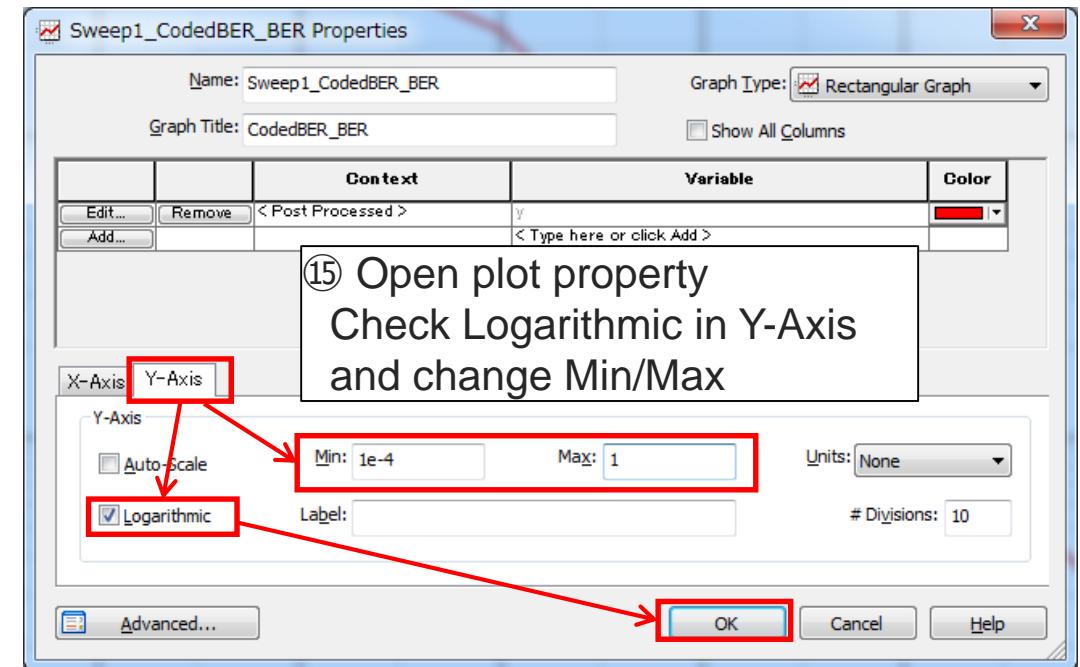
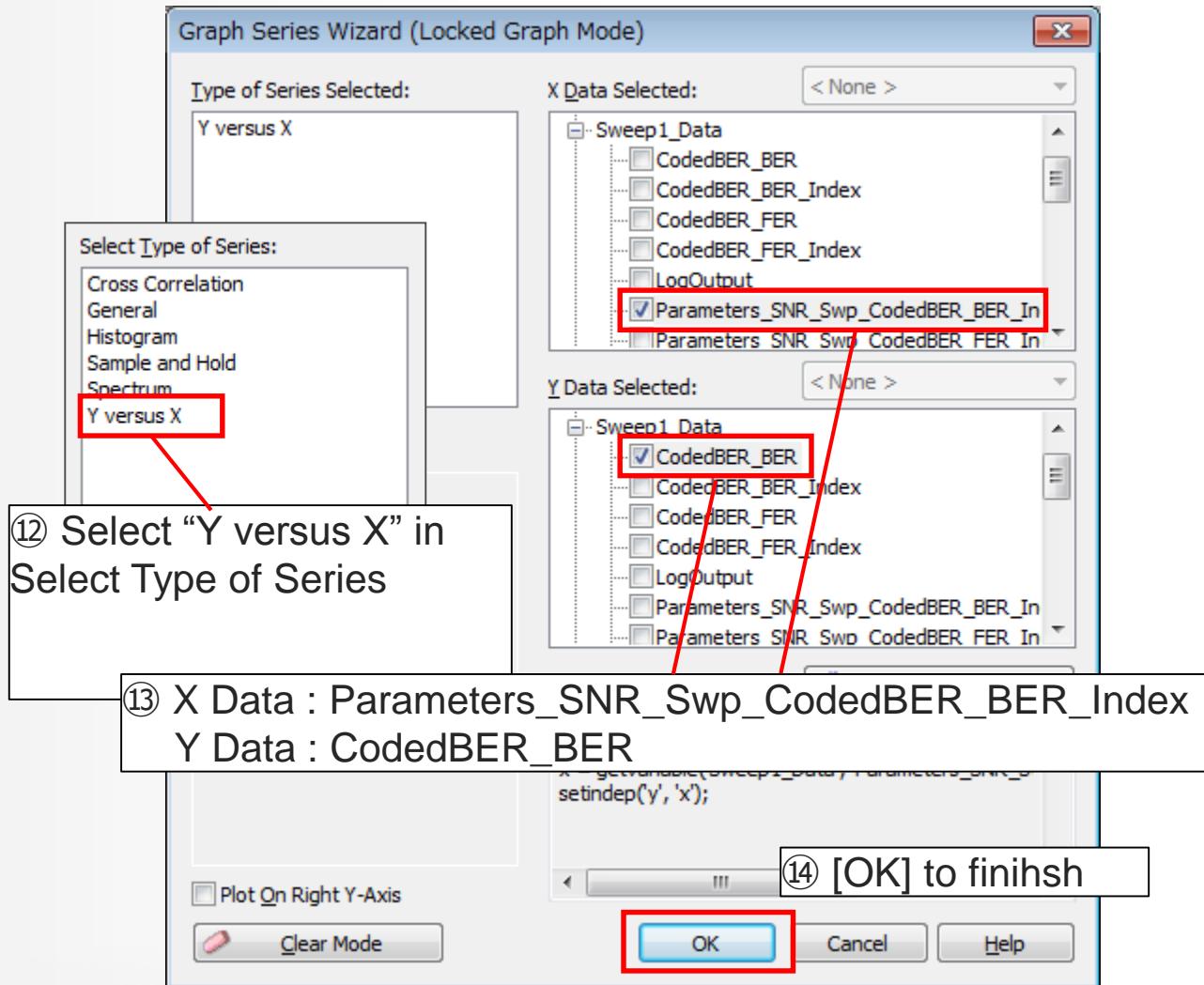
- Add New Variable...
- Add to Table
- Add to Graph** (highlighted with a red box)
- Snapshot...

An arrow points from the 'Add to Graph' option to another sub-menu window titled '⑪ Right mouse click on CodedBER_BER > Add to Graph > New Graph Series Wizard'. This sub-menu contains:

- Duplicate
- Export...
- Import Variable...
- New Graph** (highlighted with a red box)
- New Graph Series Wizard** (highlighted with a red box)
- Properties...
- Dataset Properties...

Continue to the next page

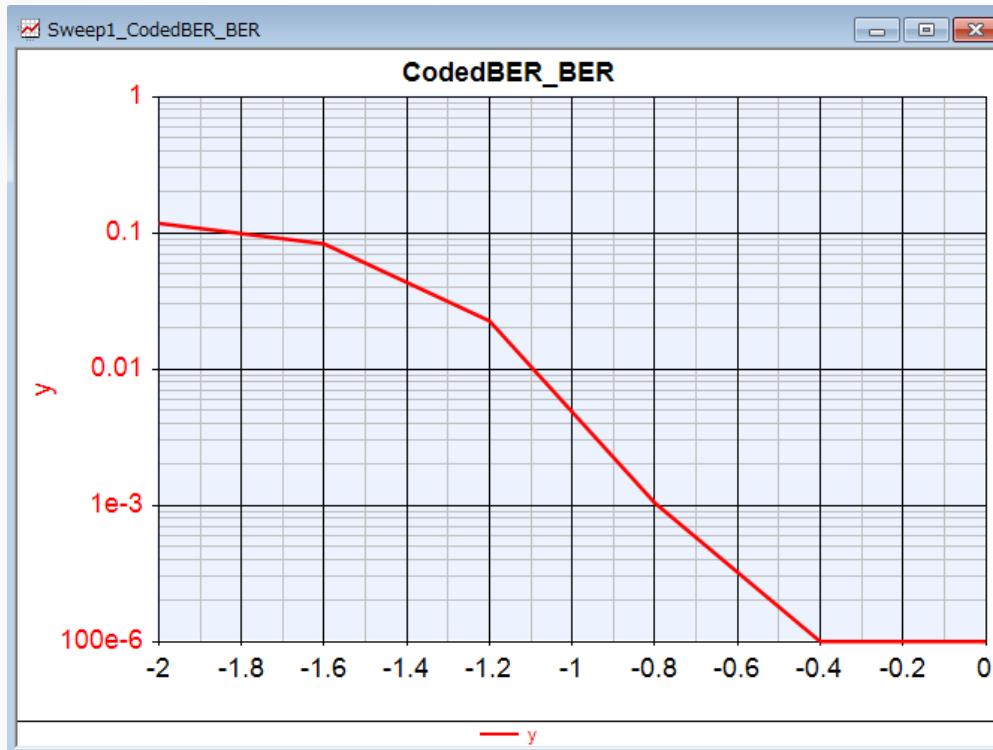
Lab 3: BER vs SNR (Cont'd)



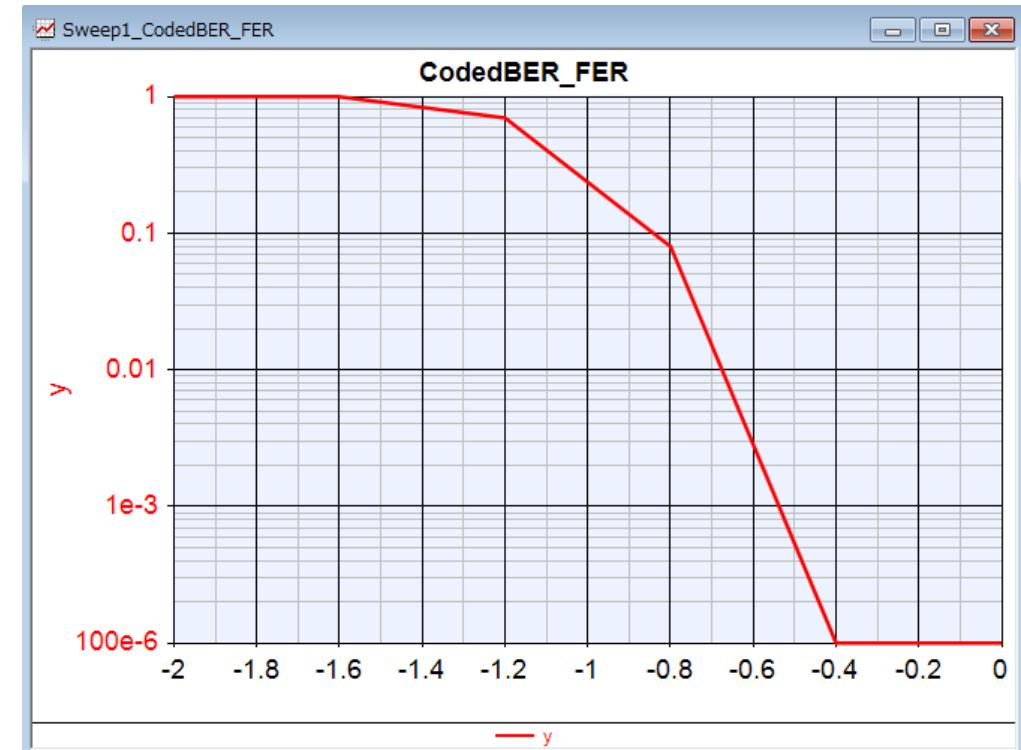
Lab 3: BER vs SNR (Cont'd)

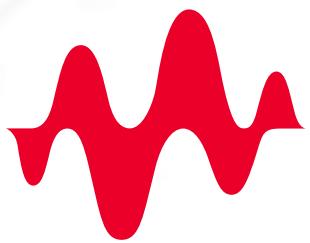
- Create FER vs SNR as well to be familiar with “Y versus X” plot
Just select variables for X axis and Y axis respectively

BER vs SNR



FER vs SNR





KEYSIGHT
TECHNOLOGIES