



SPLSim Channel Sim Flow

V2019.06



Topics:

- Background
- Modules used
- Flow overview
- Flow details
 - A: Project folder structure
 - B: Prepare channel component models
 - C: Assemble channel and simulate
 - D: Parameter sweep
- Demo



Background:

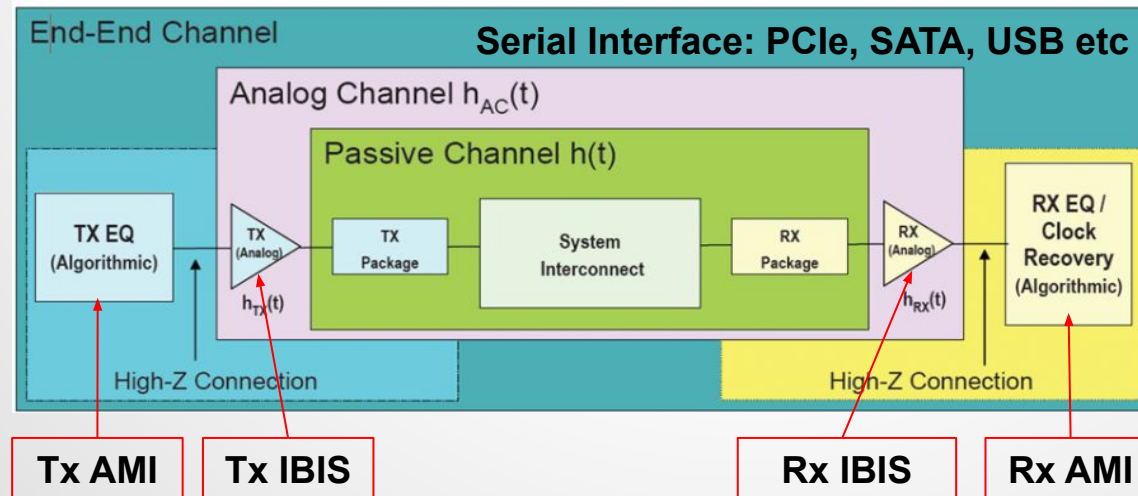
- Channel simulation:
 - Off-chip focused (thus does NOT use transistor)
 - Use system level components:
 - IBIS, w/ or w/o AMI for Tx and Rx
 - Passive interconnects may include:
 - Transmission line (RLGC tabular model)
 - S-parameter (package, connectors etc)
 - Spice subcircuit (Via etc)
- SPISim's ChannelSim supports:
 - Time domain (TD) channel simulation
 - Frequency domain (FD) .snp extraction (NOT field solving)
 - **StatEye**: Statistical and bit-by-bit IBIS-AMI analysis.



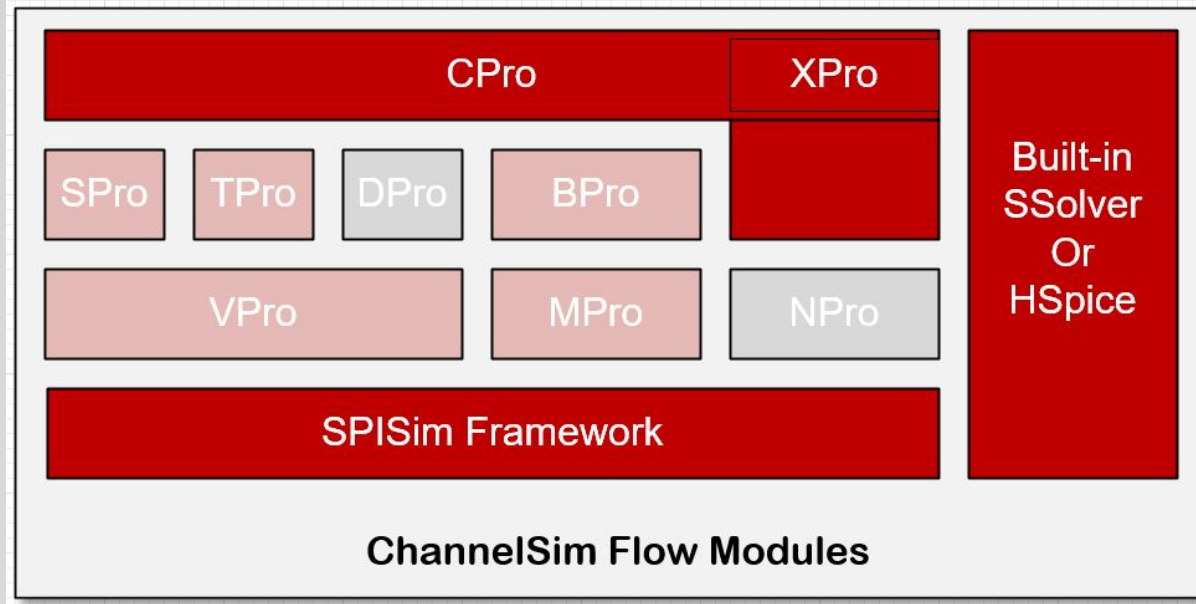
Background:

- Channel simulator:

- Has built-in engine, can also use user licensed HSpice
- Time domain and frequency domain:
 - Spice-like derived from ngSpice (new BSD licensed)
 - Added system level components models from SPISim
- StatEye (IBIS-AMI) domain: tool built-in.
 - IBIS-AMI modeling capabilities can be added.



Modules Used:



Full S/T/B/V/M flow etc are available as add-ons for complete modeling support.

Fully included:

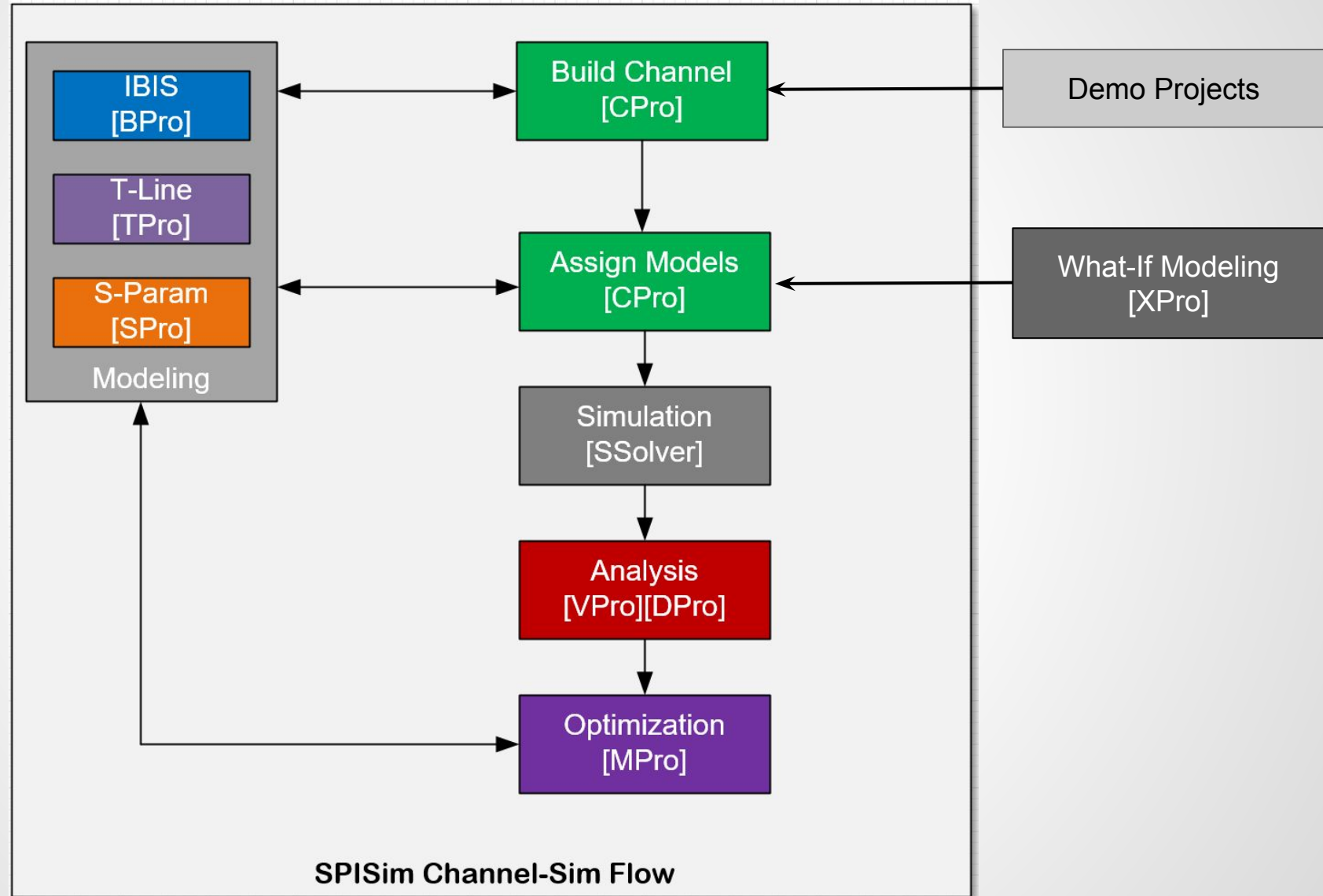
- CPro: Channel builder
- XPro: What-if channel modeling
- SSolver: Channel simulator

Partially included:

- V/S-Pro: Simulation result viewer
- TPro: Transmission line model
- BPro: IBIS inspection/tuning
- MPro: for param. sweep

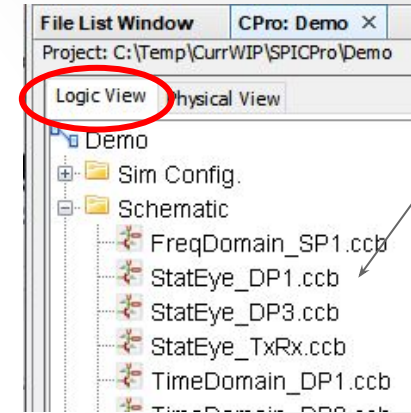
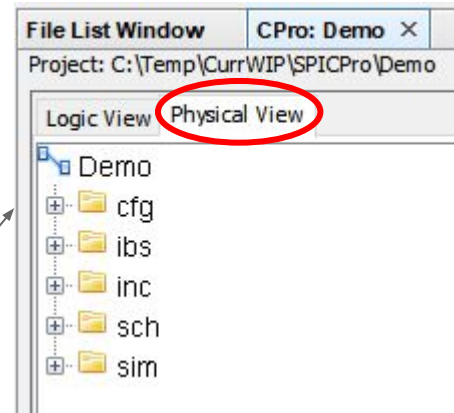
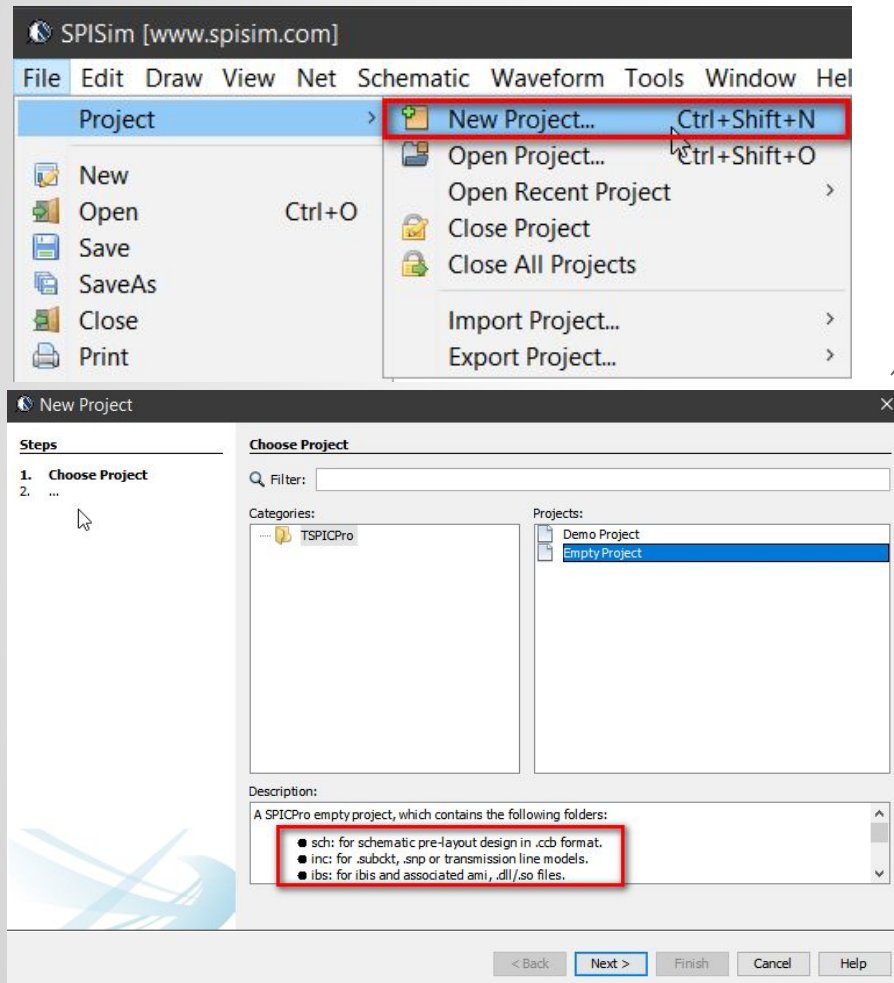


Flow Overview:



Project folder structure layout

- Create an empty or demo project:
 - In the working folder:

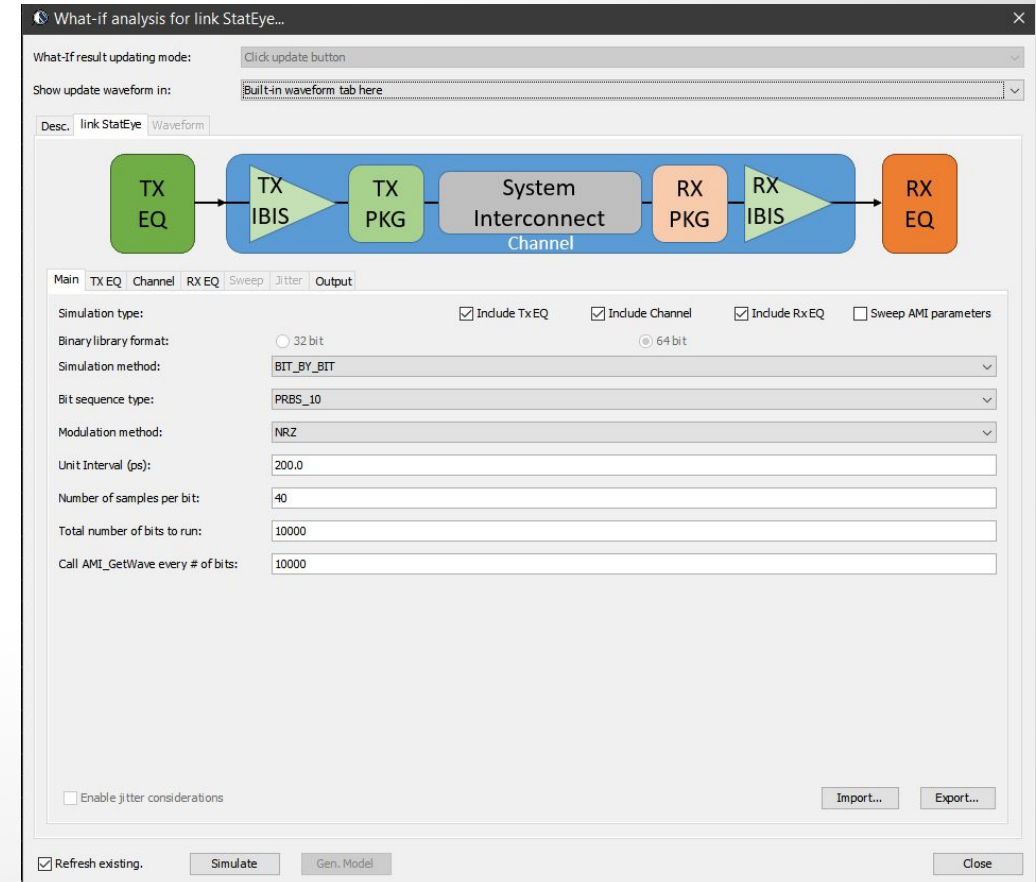
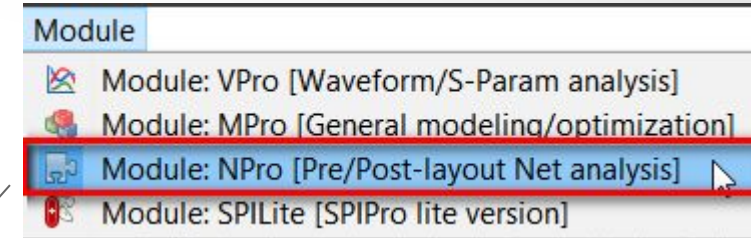
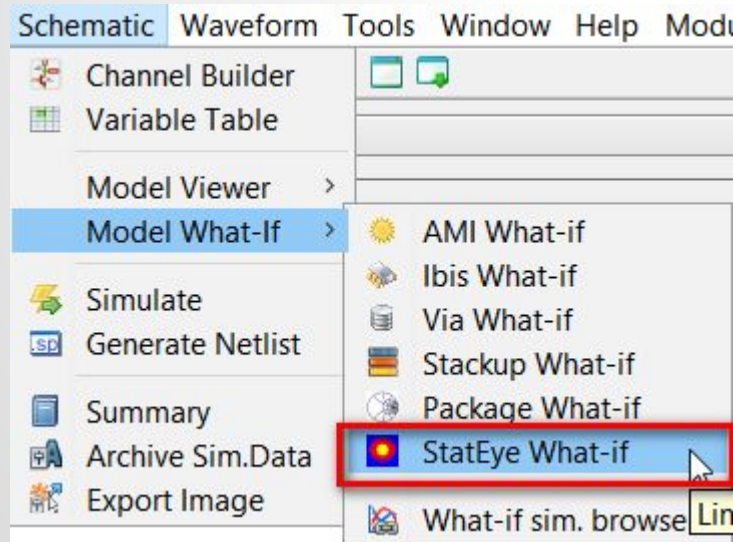


double-click to open

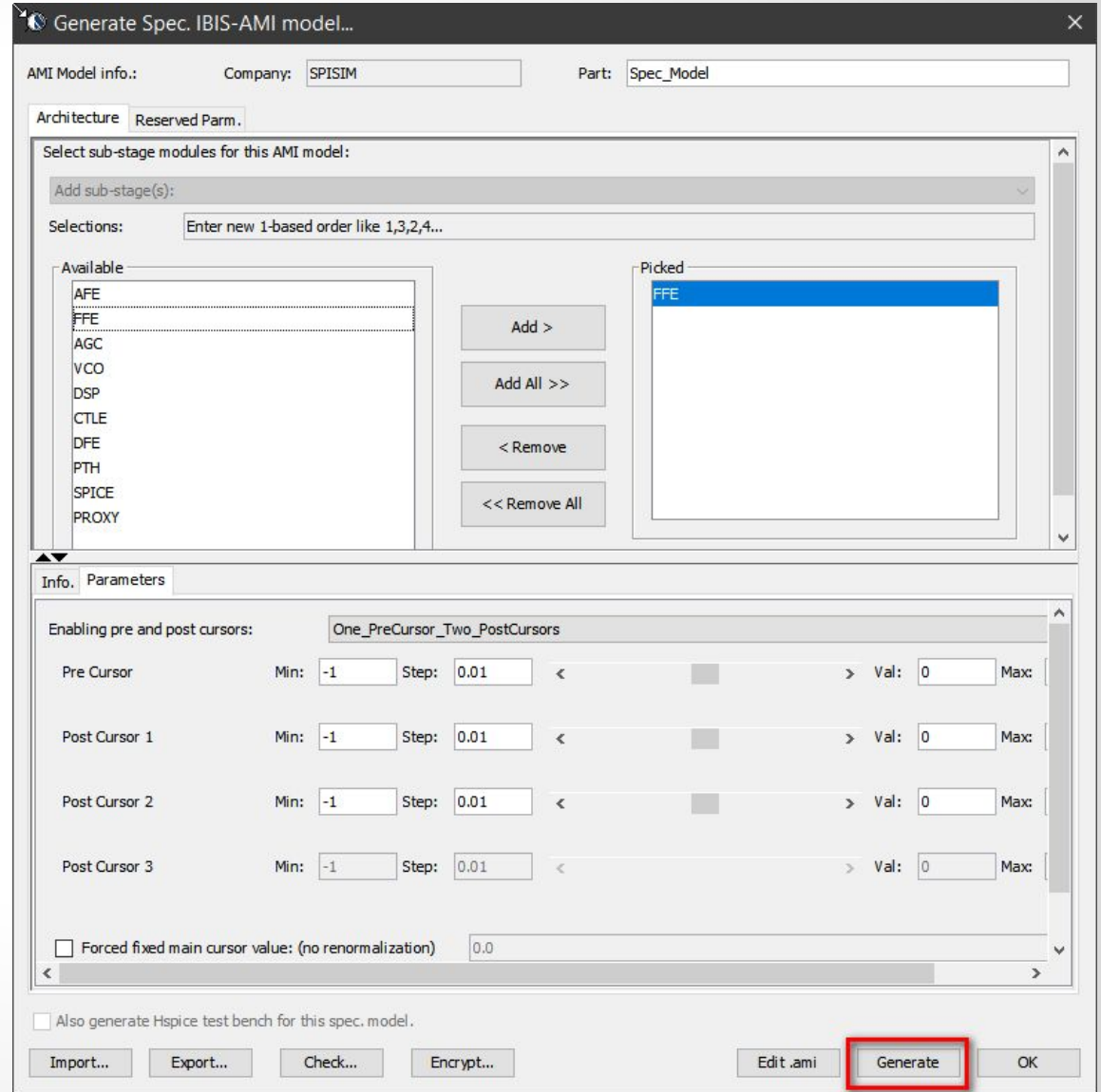
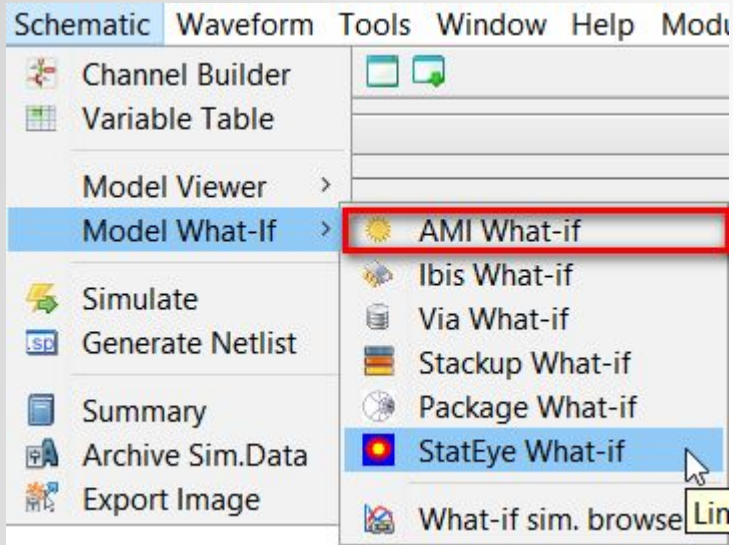
- Save schematic design built from channel builder as “.ccb” files put under “Sch”
- Put IBIS (.ibs, .ami, .dll/.so) model files under “Ibs”
- Put transmission lines RLGC models (“.tab”), S-parameter (“.snp”) files or spice sub-circuit (“.sp”) files under “Inc”
- Tool will save config. settings as “.cfg” into “Cfg” folder
- Tool will generate files and run simulation “Sim” folder
 - Waveform format is Berkeley Spice “.raw”
 - Sim folder will be cleaned-up before simulation begin
- Tool will archive simulation results under “Bak” folder

Model preparation: 1/2 “What-If”

- Switch to NPro:
 - For Pre-layout net analysis
- Don't have any models yet?
 - Try “StatEye What-if for quick analysis.

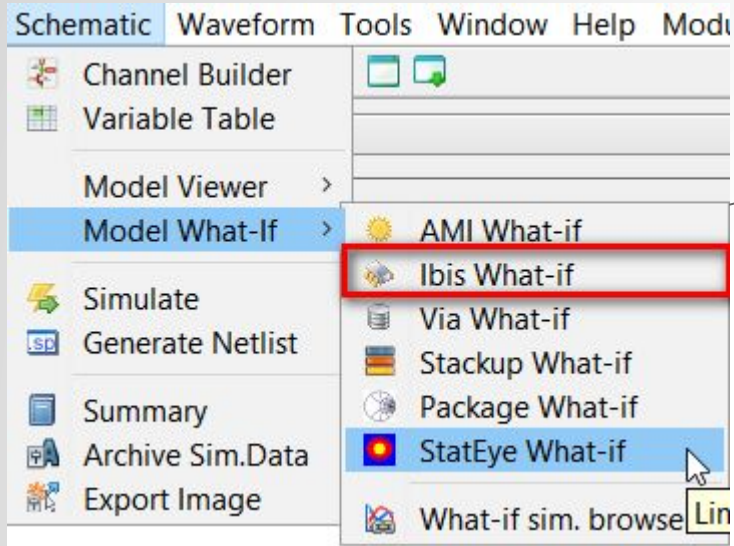


Generate “What-If” AMI models:

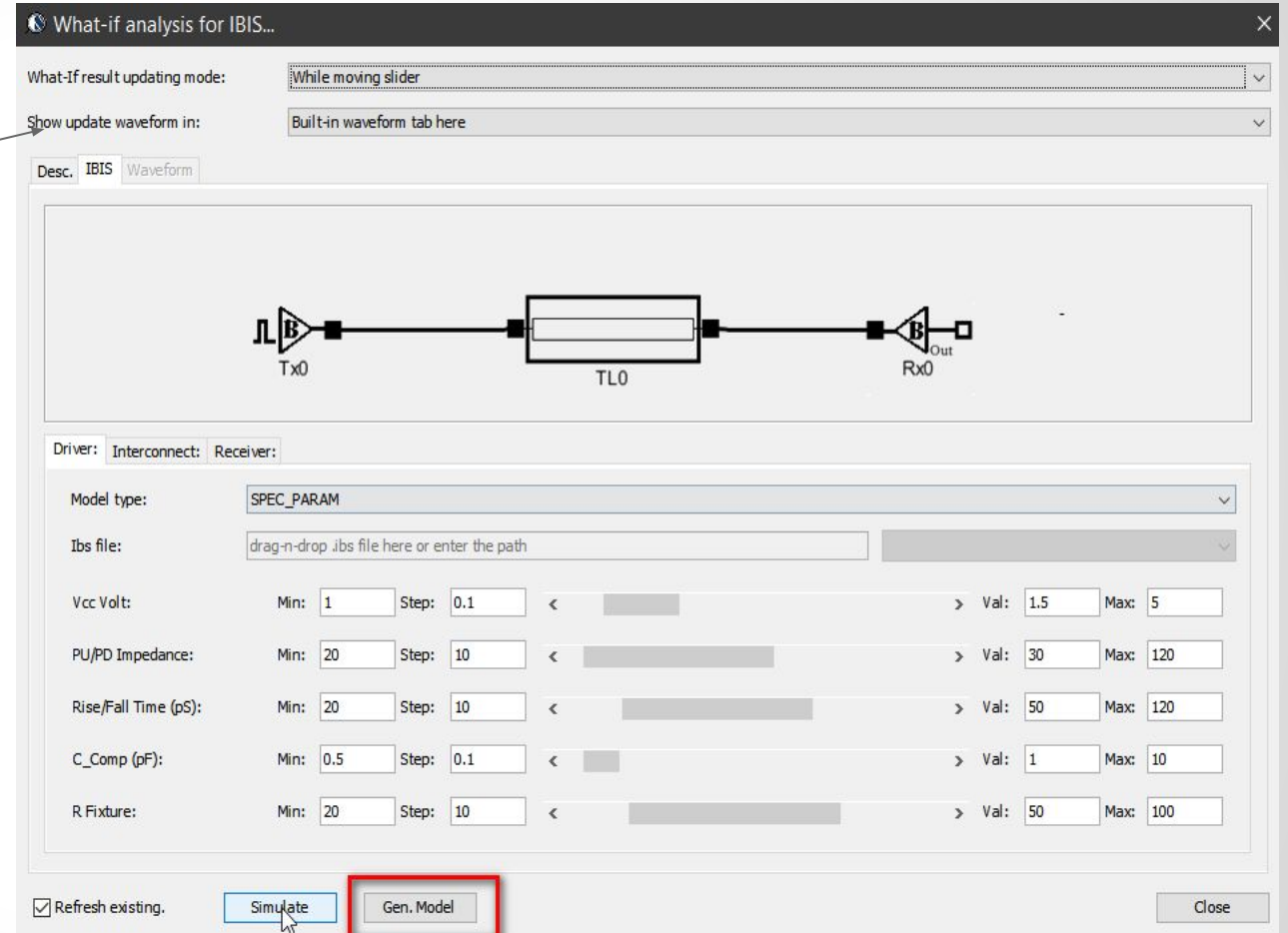


- Generated AMI model will expire in two weeks.
- Can be fully unlocked by BPro w/ AMI.

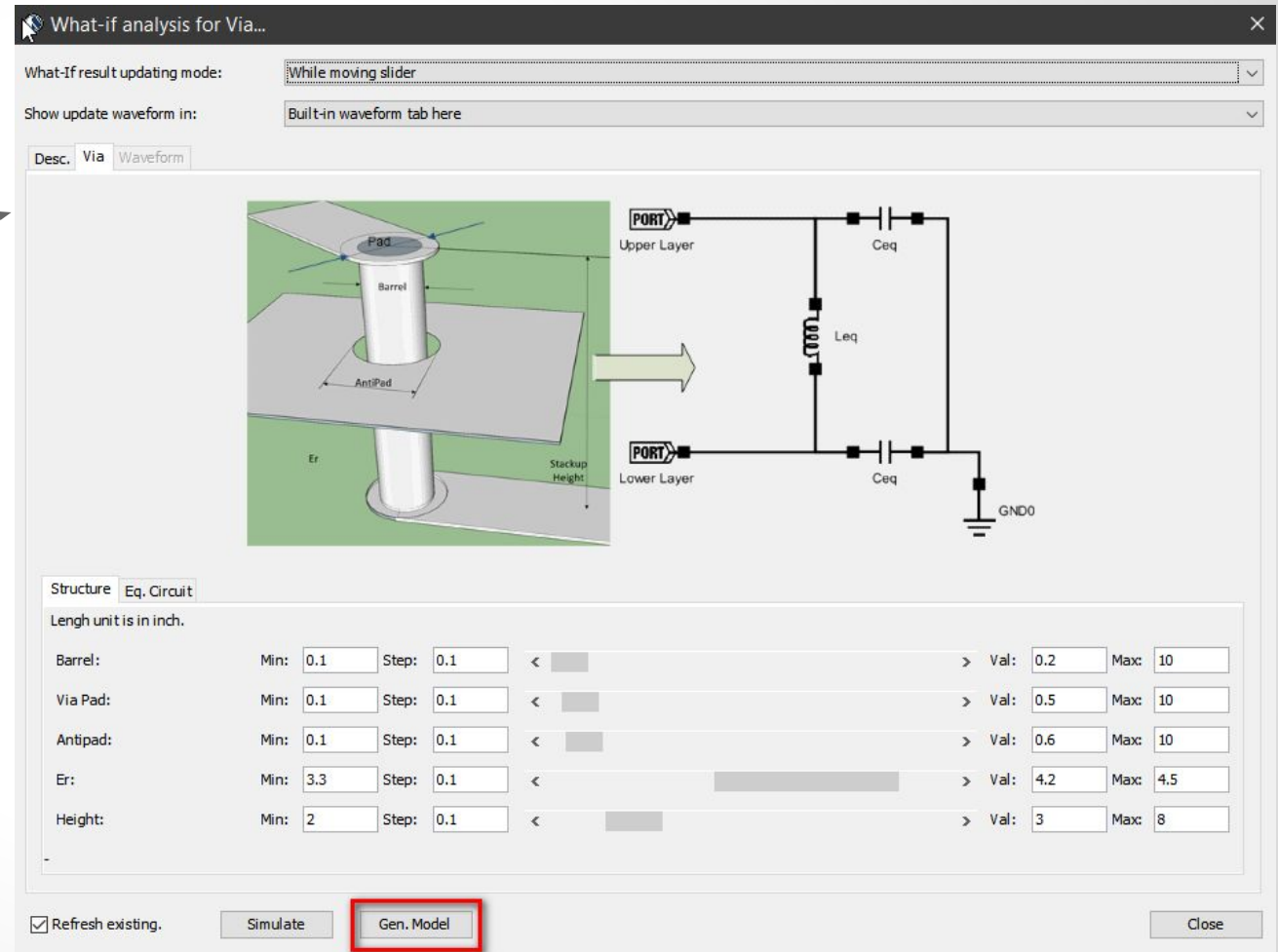
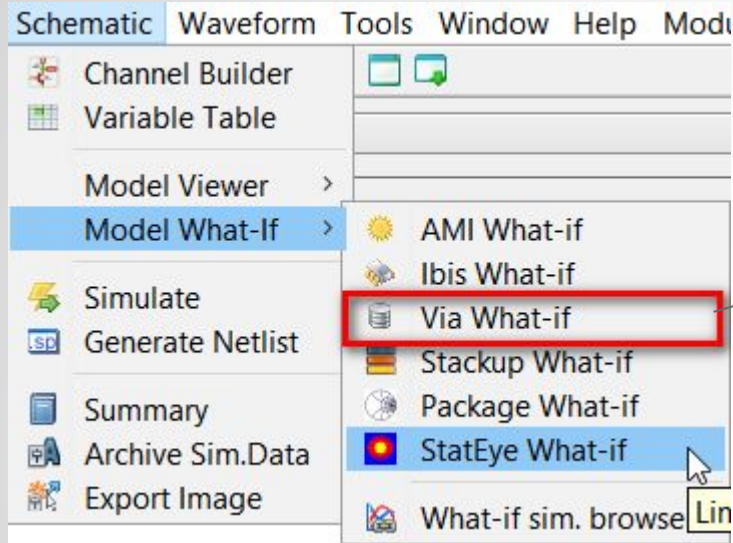
Generate “What-If” IBIS models:



- Generated IBIS model based on performance.
- IBIS from silicon etc supported by BPro.



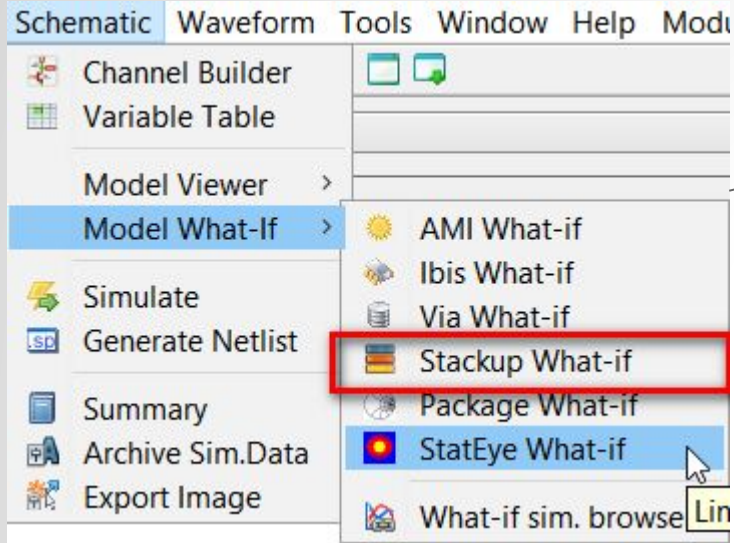
Generate “What-If” Via models:



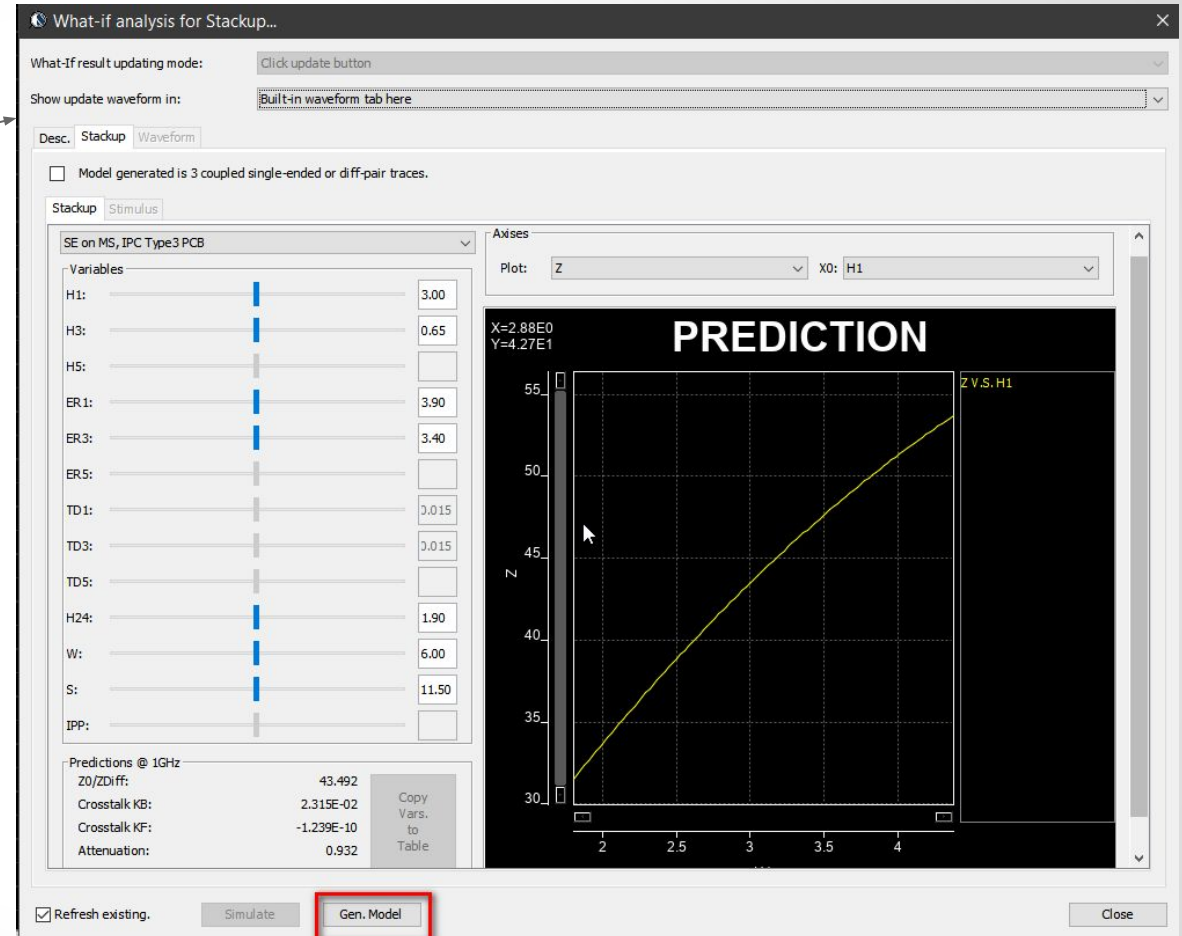
- Generated .s4p for equivalent model.
- No crosstalk coupling.
- Use 3rd party tool like HFSS for full extraction.



Generate “What-If” T-Line RLGC models:

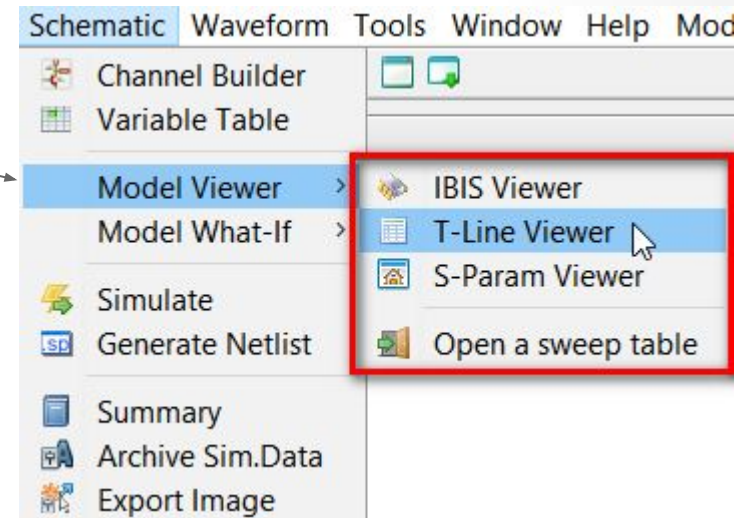
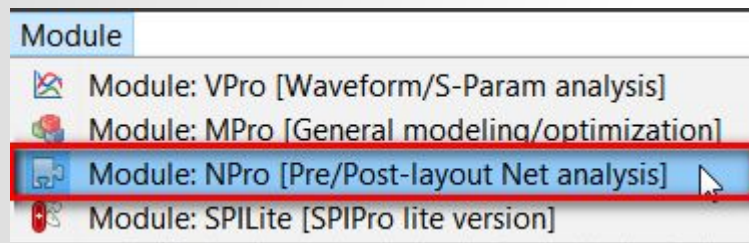


- Generated frequency dependent tabular RLGC models (.tab).
- Full stack-up modeling supported by TPro.



Model preparation: 2/2 Inspect Quality

- For existing or generated component models:
 - Check/tune their quality against the spec.
 - These “viewers” are subset of other SPISim modules



Inspect IBIS models

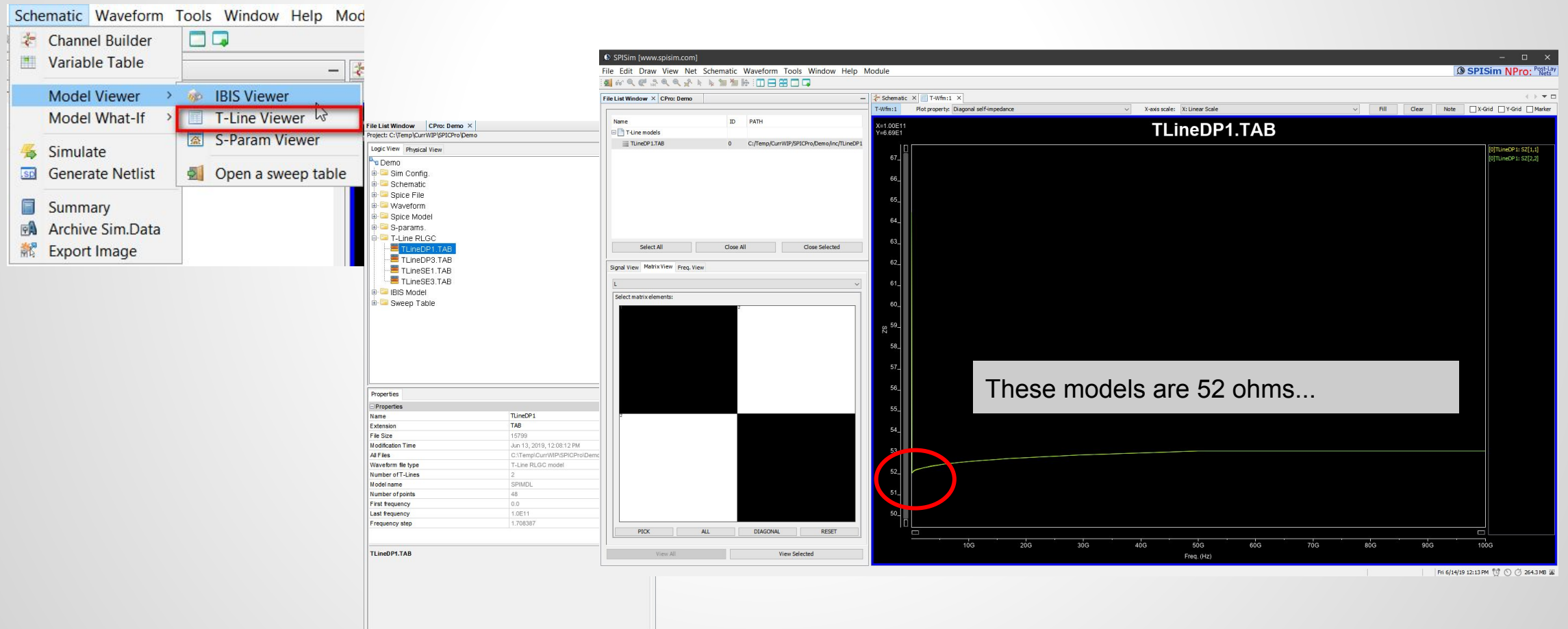
- Check IBIS model's impedance/slew etc, tune if needed with built-in model editor.
 - Double click ".ibs" file in project navigator or open via "IBIS viewer"

The image illustrates the workflow for inspecting an IBIS model in SPISim:

- Model Viewer Menu:** The 'Model Viewer' menu is open, showing options like 'IBIS Viewer', 'T-Line Viewer', 'S-Param Viewer', and 'Open a sweep table'. The 'IBIS Viewer' option is highlighted with a red box.
- Open File Dialog:** An 'Open' dialog box is shown with 'Lab1' as the location. The file 'v69adq.ibs' is selected. A large grey box with the text 'Select IBIS file' is overlaid on the dialog.
- IBIS Data Library:** The 'IBIS Data Library' window shows the project structure. The 'v69adq.ibs' file is selected in the 'Physical View'.
- Rising Waveform Plot:** The 'Rising Waveform' plot is displayed, showing the voltage (V) versus time (s) for the selected IBIS model. The plot shows three curves: TYP (red), MIN (blue), and MAX (green).

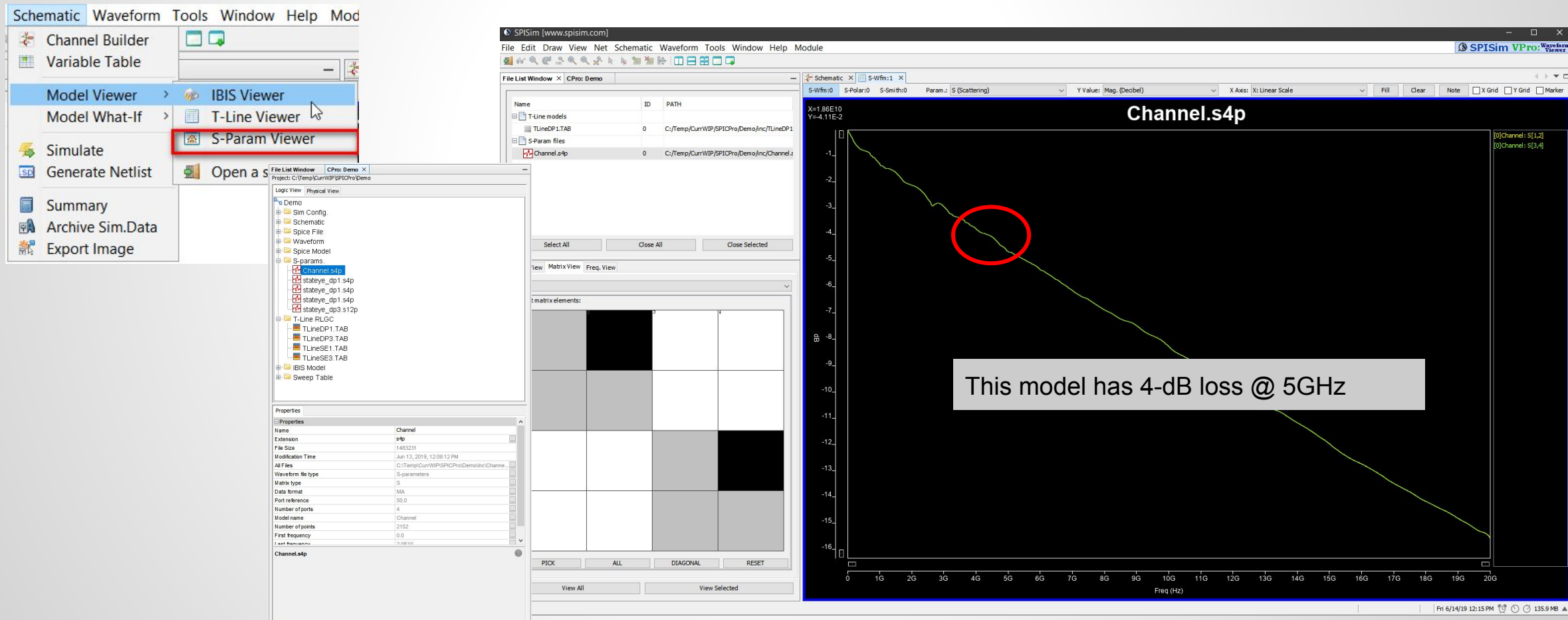
Inspect transmission line models

- Check Transmission line model's impedance, attenuation etc
 - Double click “.TAB” file in project navigator or drag-N-drop into “FileList window”, then view with “T-Line viewer”



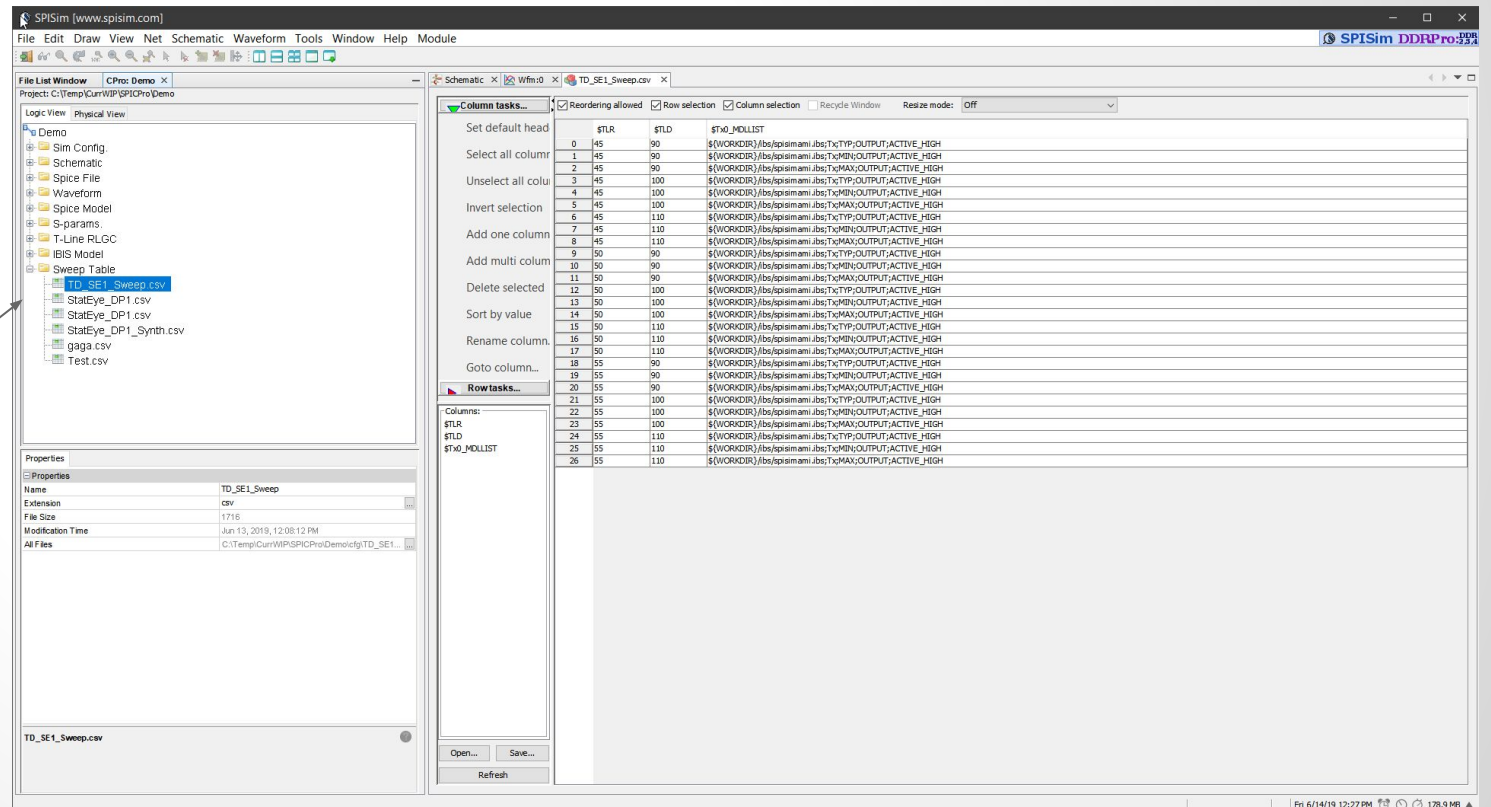
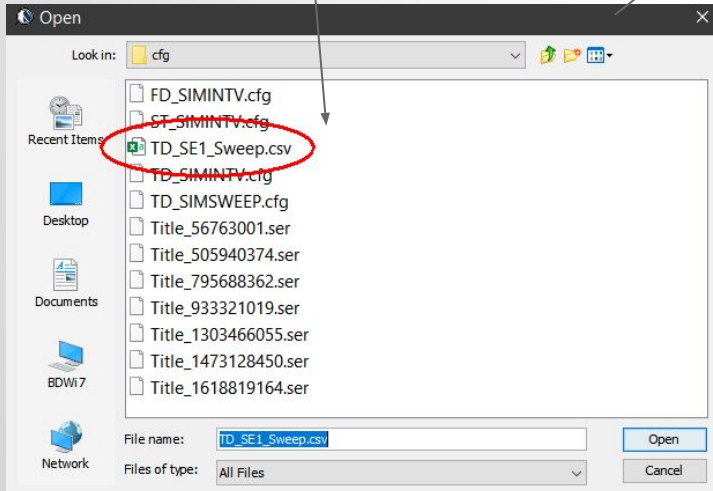
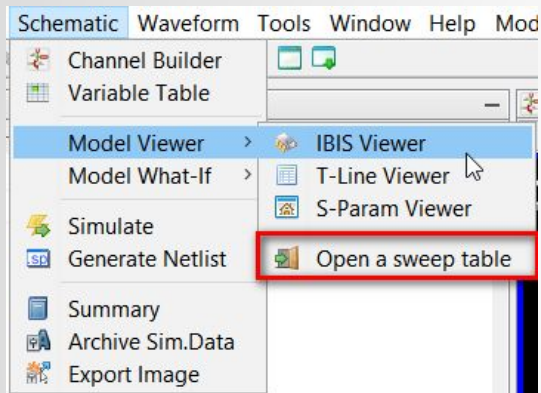
Inspect S-parameter models

- Check model losses, TDR/TDT etc
 - Double click “.Snp” file in project navigator or drag-N-drop into “FileList window”, then view with “S-param viewer”



Parameter sweep table

- Double click “.csv” file in project navigator or via menu “S-param viewer”
- Plain text “.csv” files can be generated by SPISim’s MPro or using tool like Excel
 - MPro has DOE, Full factorial etc sampling algorithm



Flow Details: Channel builder

The screenshot displays the SPISim software interface with several annotations explaining the workflow:

- Remember to save first before simulating...**: A callout pointing to the Save icon in the toolbar.
- Open the schematic window from main menu...**: A callout pointing to the 'Channel Builder' option in the 'Schematic' menu.
- Double click any ".ccb" file will open its schematic.**: A callout pointing to a '.ccb' file in the File List Window.
- Drag-N-drop from palette...**: A callout pointing to a component in the right-hand palette.

The main workspace shows a schematic diagram titled 'StatEye Channel Sim' with the following components and connections:

- Tx0**: Transmitter component with properties: IBIS=spisimami.jbs, MDL=Tx, CORNER=TYP, AMI=spisimbcami, LIB=SPISimAMI_WIN64.dll, and model parameters: Snp File: Channel.s4p, Z=50.
- S0**: S-parameter component labeled 'S_PARAM'.
- Rx0**: Receiver component with properties: IBIS=spisimami.jbs, MDL=Rx, CORNER=TYP, AMI=spisimrxami, and model parameters: Snp File: Channel.s4p, Z=50.

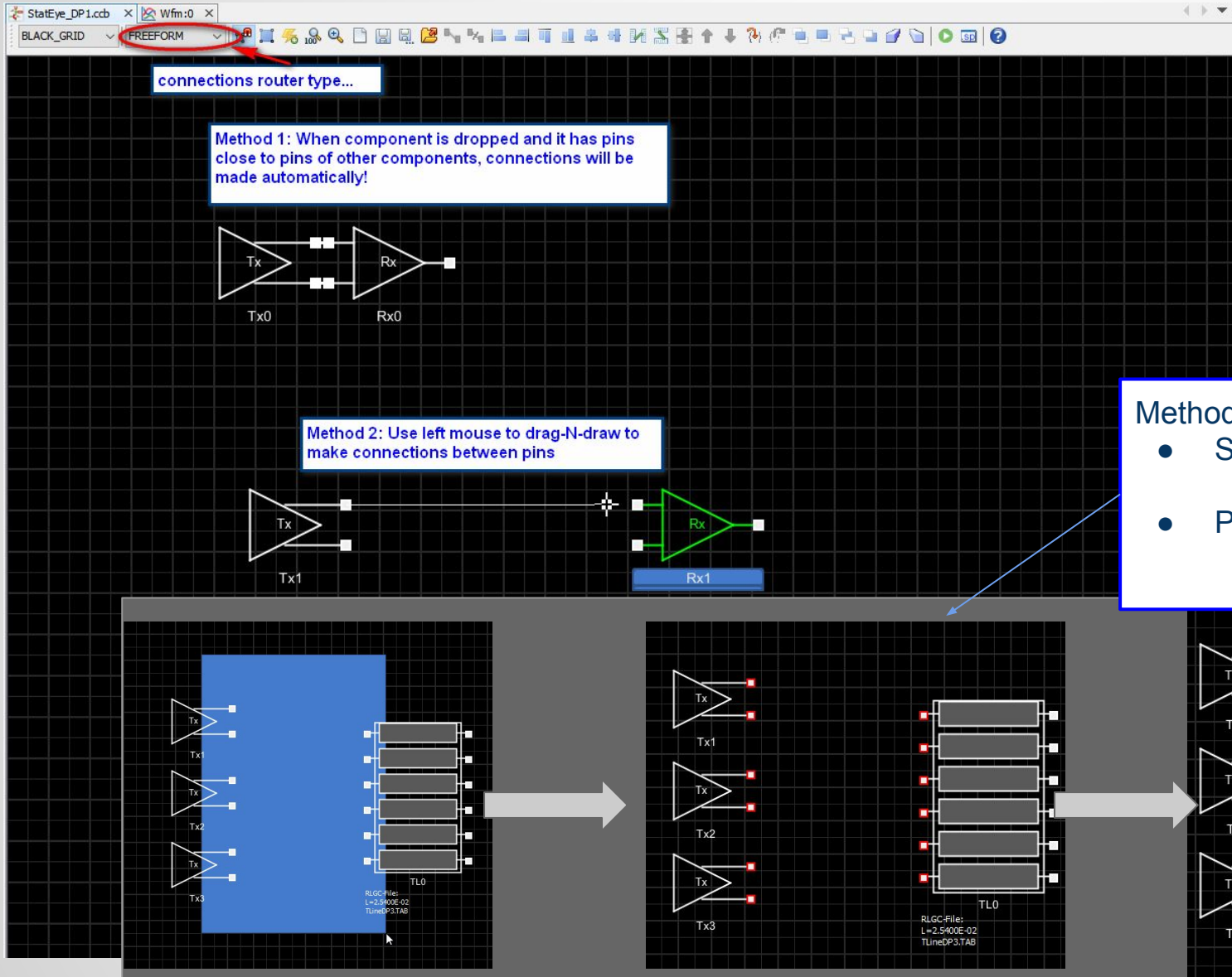
The components are connected in a series: Tx0 → S0 → Rx0.

The right-hand palette contains various components for circuit simulation, including:

- StatEye Sim.
- StatEye Swe
- TD Sim.
- TD Sweep
- FD Sim.
- FD Sweep
- Plot [Drag H
- IBIS Diff. Dr** (highlighted with a red circle)
- IBIS Drv
- IBIS Diff. Rc
- IBIS Rcv
- Repeater/Re
- S-parameter
- Sub-circuit

The bottom status bar shows the date and time: Fri 6/14/19 3:20 PM, and the file size: 219.4 MB.

Channel builder: Making connections



Method 3:

- Select group of unconnected pins
 - No need to be aligned
- Press "C" key to connect
 - Left pins will connect to right pins

Channel builder: Edit properties

StatEye_DP3.ccb

Wfm:0

BLACK_GRID

FREEFORM

StatEye Interactive

C:/Temp/CurrWIP/SPICPro/Demo/sim
SimMode: BIT_BY_BIT
200p
1E5

StatEye Channel Sim With Crosstalk

Setup ST simulation...

Time Domain | Freq. Domain | **Stat-Eye**

Setup | Probe | Driver | DCD/Jitter | HSpice Options

Simulation method: STATISTICS

Bit sequence type: PRBS_15

Modulation method: NRZ

☒ Bit rate (Hz): 5.0E9

☐ Unit interval (Sec): 200p

Number of samples per bit: 32

Number of bits to run: 1E5

Call AMI_GetWave every # of bits: 10000

Ignore first number of bits: 0

☐ Generate un-wrapped waveform: from time (sec.): 10 to time (sec.): 50

[To display waveform in external window, launch Window: VPro module from main menu first!]

Display waveform at: Built-in waveform widget

Sweep simulation plan:

Automatical probe(s)/ports:

☒ Driver/Receiver pins ☐ RC Terminator

☒ Driver stimulus ☐ IO ports

☐ Via

☐ Coupled transmission line

OK Cancel

Must have a simulation set-up (Time-domain, Freq.-domain or StatEye) in order to simulate...

Double click or right-click and select "Edit" to edit device's properties.

Right-click pop-up...

- Delete node
- Edit properties**
- Map to layout
- Instantiate as array
- Descend in hierarchy
- Smash this subckt
- Define pin location



Flow Details: Interactive simulation

TimeDomain_SE1.ccb **Wfm:0**

BLACK_GRID FREEFORM

TD Sim Interactive

WORKDIR: C:/Temp/CurrWIP/SPICPro/Demo/sim
SimStep: 10p
SimTime: 50n
Standard: SPICEM

Analog IBIS TD Simulation 1 Single-Ended TLine

Setup TD simulation...

Time Domain Freq. Domain Stat-Eye

Simulator specified in SPICPro's option tab will be used.

Simulation step: (ps) 10

Simulation duration: (ns) 10

Waveform plot on channel-builder is optional. Tool can also use waveform plot window to display waveform (better)

[To display waveform in external window, launch Window: VPro module from main menu first!]

Display waveform at: Built-in waveform widget

Sweep simulation plan: Built-in waveform tab

Automatic probe(s)/ports:

☒ Driver/Receiver pins ☐ RC Terminator

☒ Driver stimulus ☐ IO ports

Trace: <-> Clear ☒ X-Grid ☒ Y-Grid ?

Waveform

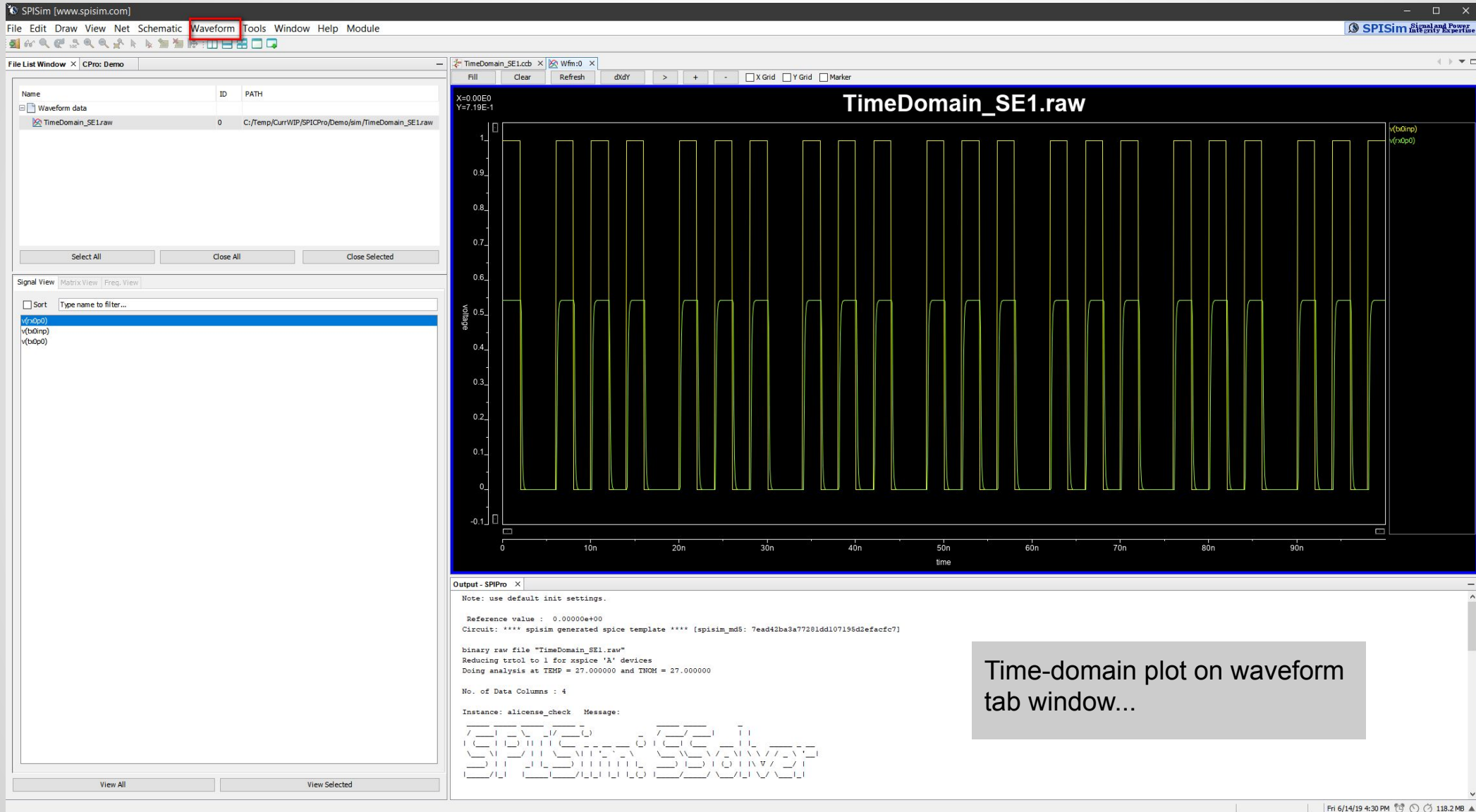
Volt

Time(ns)

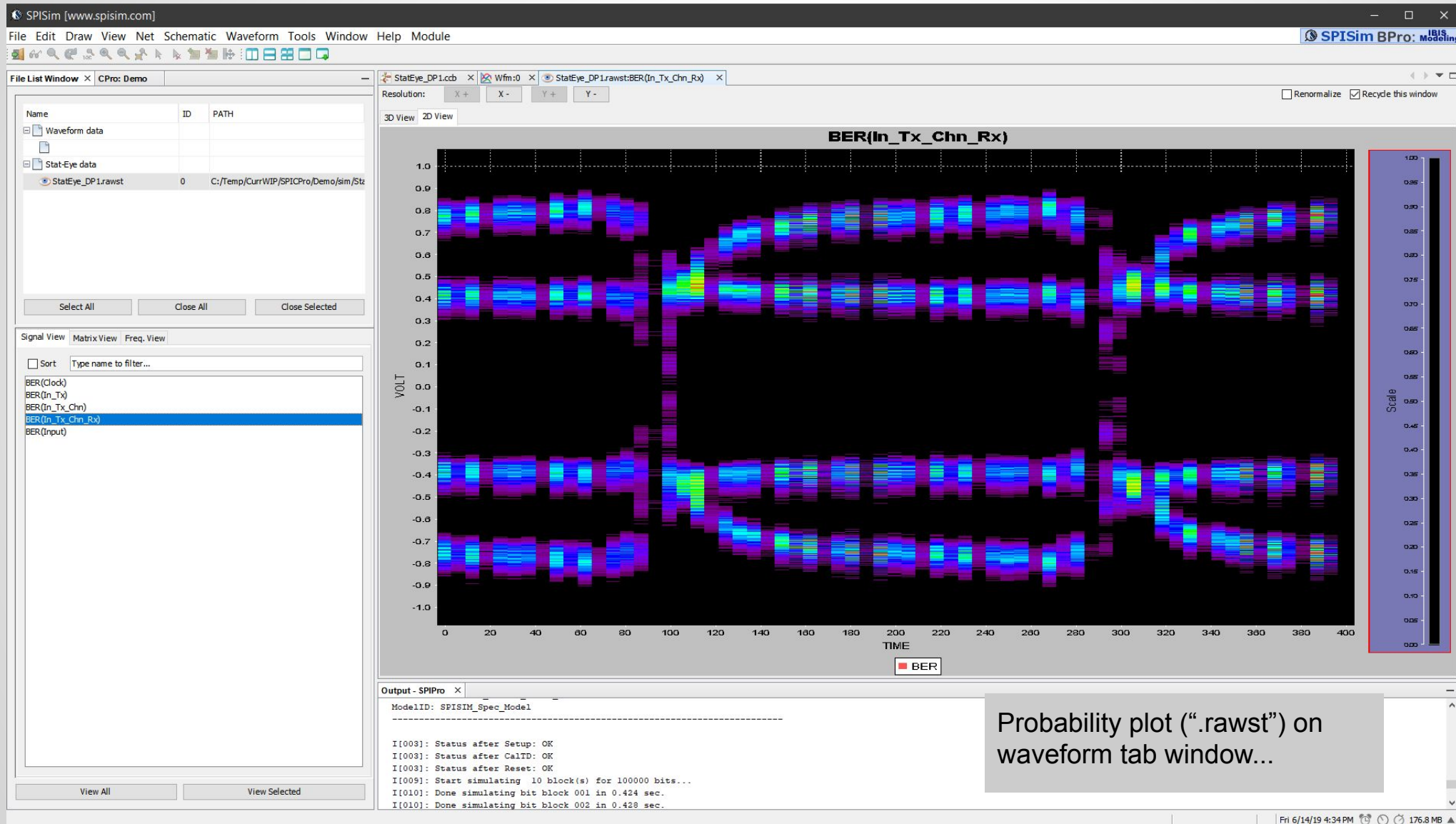
Plot [Drag Here]

Need to drag here to move around... NOT the plot itself!

Flow Details: Interactive simulation



Flow Details: Interactive simulation



Flow Details: Prepare Parameter sweep

- Choose desired variable pattern
- Define variable(s) in one or more device nodes
 - e.g. multiple IBIS corners or model combinations

The screenshot displays the SPICE Pro interface for setting up a parameter sweep. On the left, the 'Options' dialog is open, with the 'CPro' tab selected. The 'Variable pattern' is set to '\$VarName'. In the center, the 'TD Sim Sweep' window shows a schematic diagram of a transmission line simulation. The schematic includes a voltage source 'Tx0' connected to a transmission line block 'TL0'. The 'TL0' block is highlighted with a green box, and a red arrow points from it to the 'Define T-Line...' dialog on the right. The 'Define T-Line...' dialog shows the 'T-Element' properties, with the 'Characteristic impedance (ohms)' and 'Propagation delay (ps)' fields set to '\$TLR' and '\$TLD' respectively, both circled in red. The background of the TD Sim Sweep window shows the text 'Analog IBIS TD Simulation 1 Single-Ended TLine'.

Options

General Editor Fonts & Colors Keymap Appearance Miscellaneous SPISim

BPro CPro Main MPro NPro SPro TPro TDR/TDT VPro

Info. Path etc. Spice netlisting

Default working root: C:/Temp/CurrWIP/SPICPro/Demo/

Variable pattern: \$VarName

Connection router: FREEFORM

IBIS2Spice library type: COEF

Schematic theme: BLACK_GRID

Field solve trace model when extracting nets from post-layout to schematic.

Remove non-functional pad(s) when exporting post-layout nets to CPro's channel builder.

Export... Import... OK Apply

TD Sim Sweep

Simulation: C:/Temp/CurrWIP/SPICPro/Demo/sim
SimStep: 10p
SimTime: 10n
Simulation: C:\TEMP\...

Analog IBIS TD Simulation
1 Single-Ended TLine

Define T-Line...

Transmission line block: Name: TL0

Info. Model

Transmission line properties: ☒ T-Element (lossless) ☐ W-Element tabular

Line Length:

T-Element Prop. W-Element Models Stackup

T-Element (uncoupled simple transmission line) properties:

Characteristic impedance (ohms): \$TLR

Propagation delay (ps): \$TLD

Tx0
IBIS=spisimami.ibs...
MDL=Tx
CORNER=TYP
AMI=spisimbcami
LIB=SPISimAMI_WIN64.dll
PRBS_7 FREQ=5E8

TL0
T-Element:
L=2.5400E-02
Z=50.0 T=0.0

Flow Details: Prepare Parameter sweep

- Generate simulation plan

The screenshot illustrates the process of generating a simulation plan in a software interface. The process involves several steps:

- Channel Builder:** The 'Variable Table' is highlighted in the Channel Builder menu.
- Variable Table:** A table showing the hierarchy of variables to be swept.
- Sweep sim...:** A dialog box where the sweeping plan is configured. The 'Full Factorial' option is selected.
- Config. settings for full factorial:** A dialog box where the input variables and their values are specified. The 'Value format' is set to 'A, B, C,...,D (comma delimited for all possible values) or: BeginValue : StepValue : EndValue (for numerical sweeping values)'.
- TD_SE1_Sweep.csv:** The resulting simulation plan file, which is a CSV table with columns for the variables and their values.

Variable Table:

HIER	INST	NAME	VALUE
1	TOP	TX0	\$TX0_MDLIST
2	TOP	TL0	\$TLR
3	TOP	TL0	\$TLD

Sweeping plan:

☒ Full Factorial

☐ Space filling

☐ Load existing table

☐ Tune w/ sliders

Config. settings for full factorial:

Number of input variables: 3

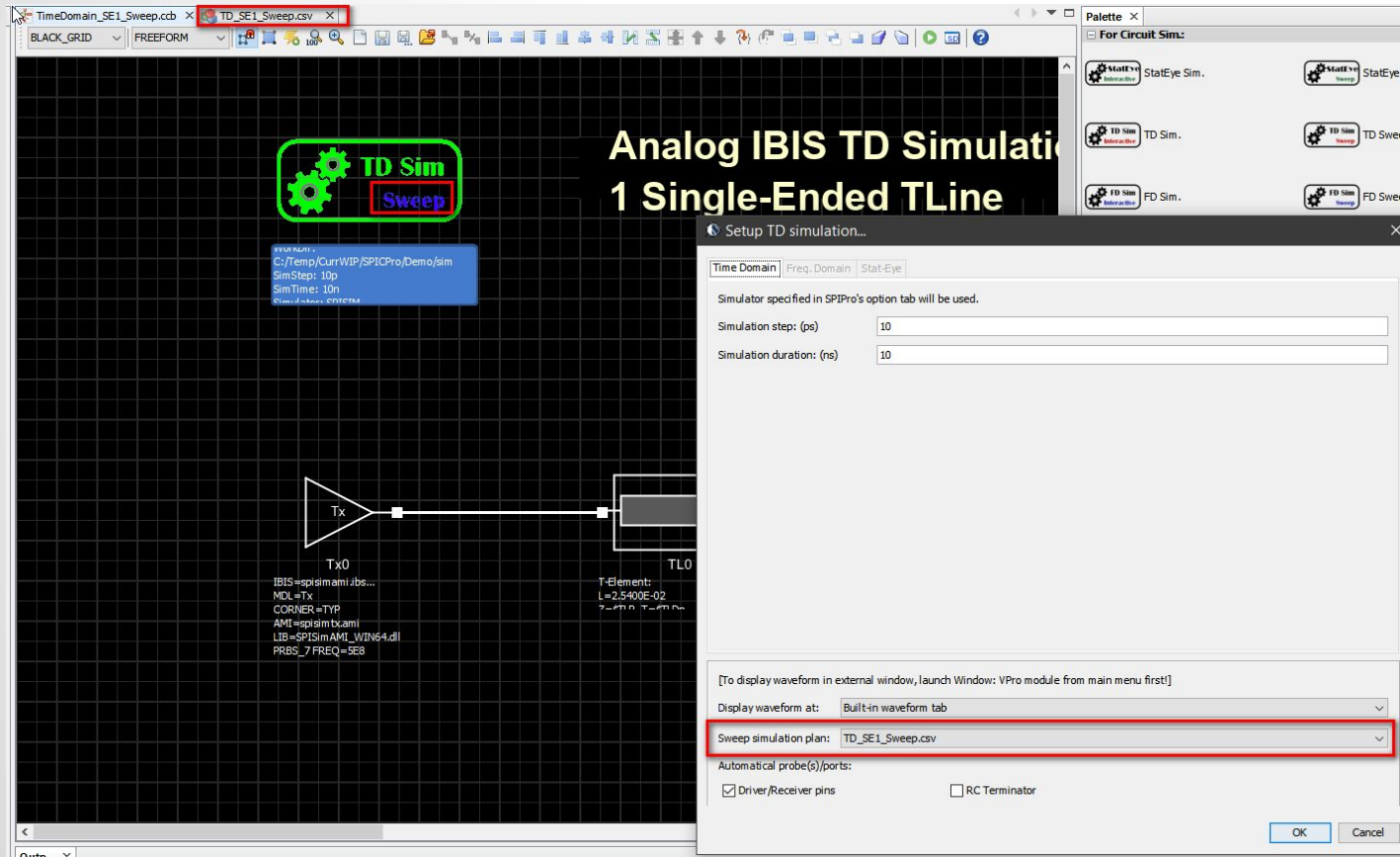
Name	Values
\$TLR	45
\$TLD	90
\$TX0_MDLIST	0, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19

TD_SE1_Sweep.csv:

	\$TLR	\$TLD	\$TX0_MDLIST
0	45	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
1	45	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
2	45	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
3	45	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
4	45	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
5	45	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
6	45	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
7	45	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
8	45	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
9	50	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
10	50	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
11	50	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
12	50	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
13	50	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
14	50	100	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
15	50	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
16	50	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH
17	50	110	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MAX;OUTPUT;ACTIVE_HIGH
18	55	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;TYP;OUTPUT;ACTIVE_HIGH
19	55	90	\${WORKDIR}/ibs/spisim ami .ibs;Tx;MIN;OUTPUT;ACTIVE_HIGH

Flow Details: Prepare Parameter sweep

- Specify in the simulation set-up
 - A sweep table needs to be opened or specified in the simulation setup for parameter sweep.





SPISim is an InSync member.

