SPISim 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

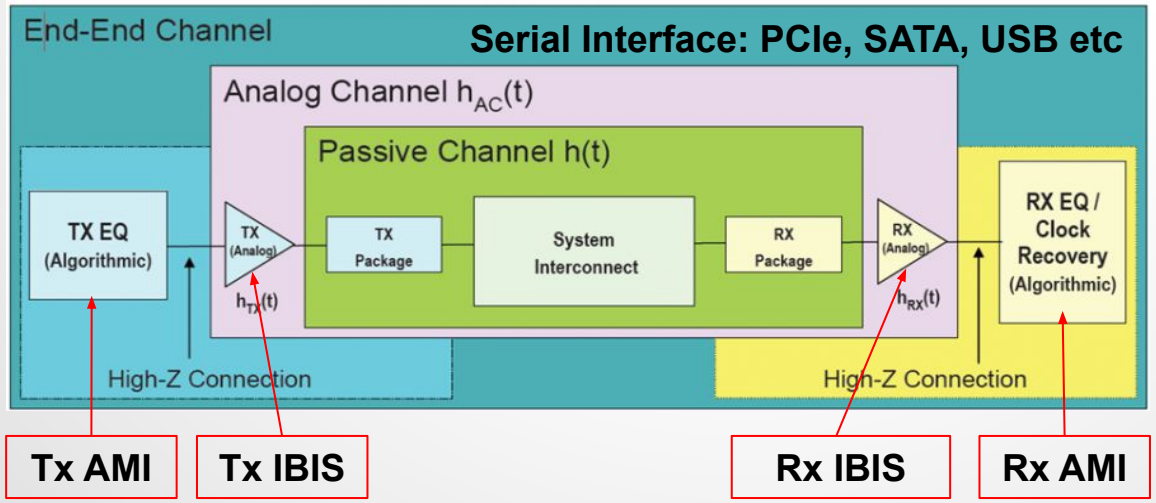
Background:

- 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)

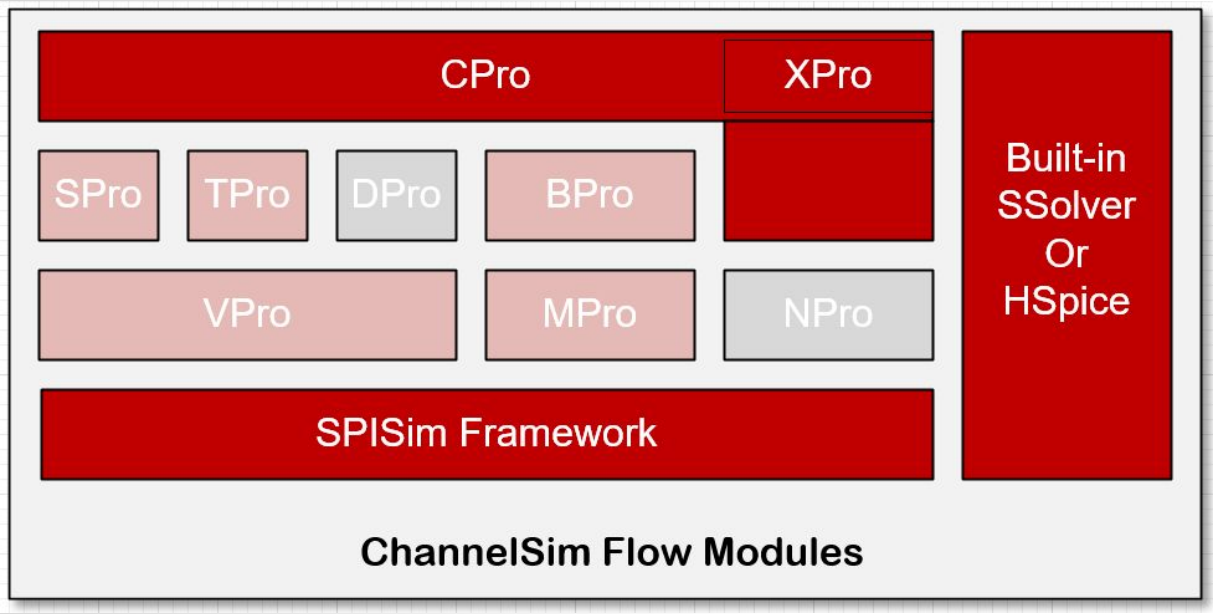
- SPISim's ChannelSim supports:
  - Time domain (TD) channel simulation
  - Frequency domain (FD) .snp extraction (NOT field solving)
  - **StatEye**: Statistical and bit-by-bit IBIS-AMI analysis.

Background:

- Channel simulator:
  - Has built-in engine, can also use <u>user licensed</u> 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:



Fully included:

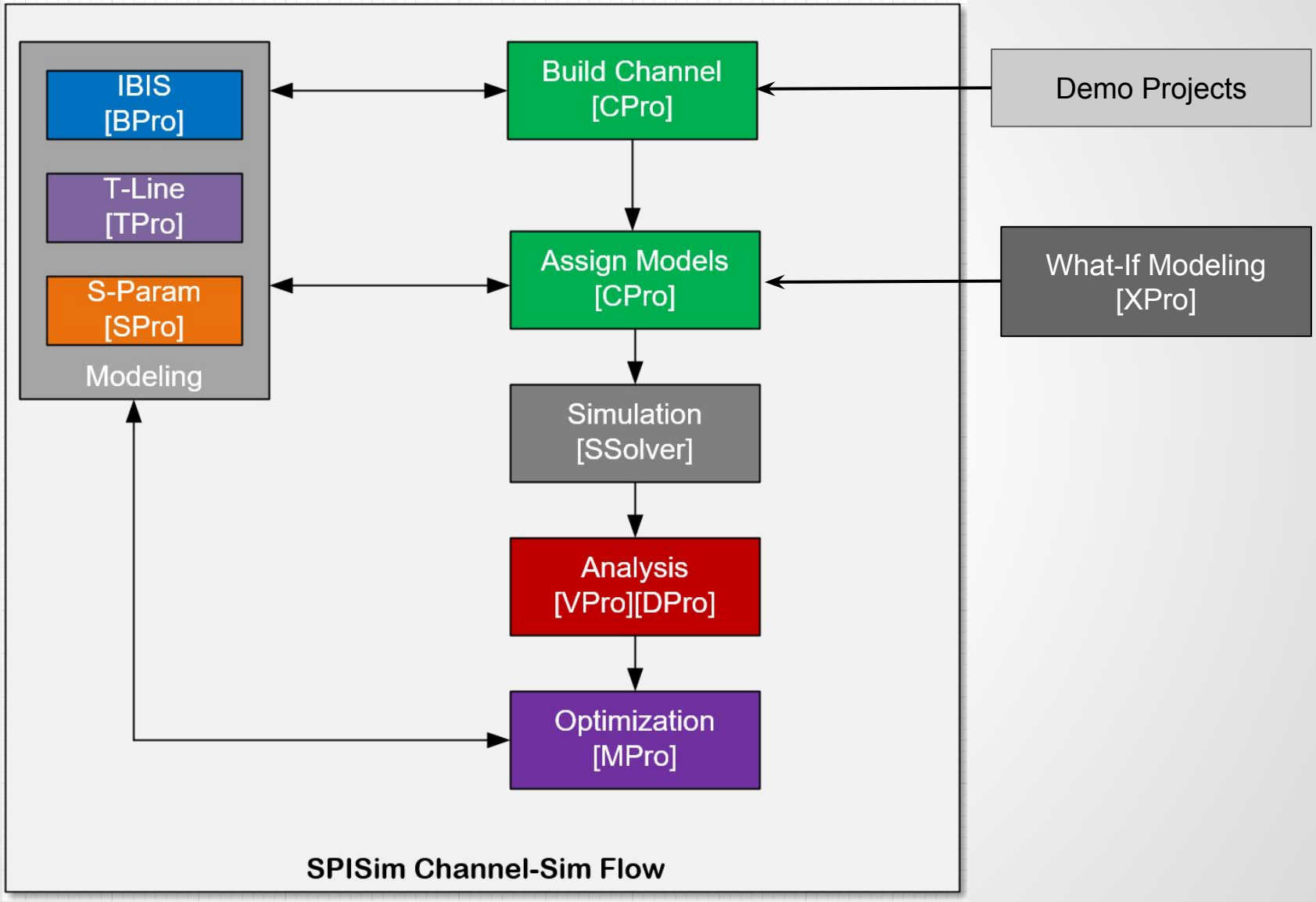
- CPro: Channel builder
- XPro: What-if channel modeling
- SSolver: Channel simulator

Partially included:

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

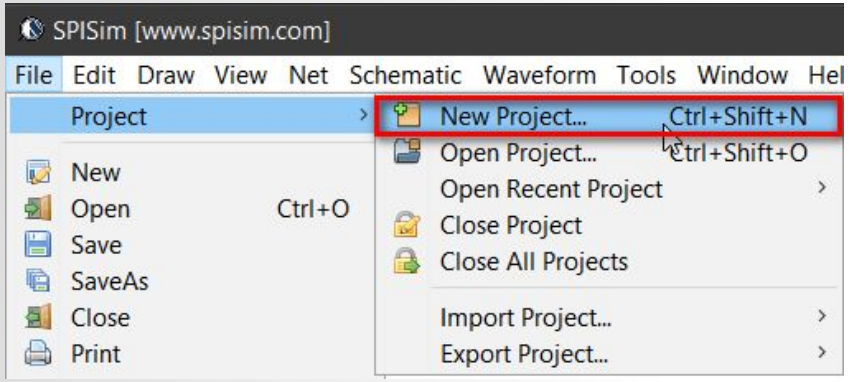
Full S/T/B/V/M flow etc are available as add-ons for complete modeling support.

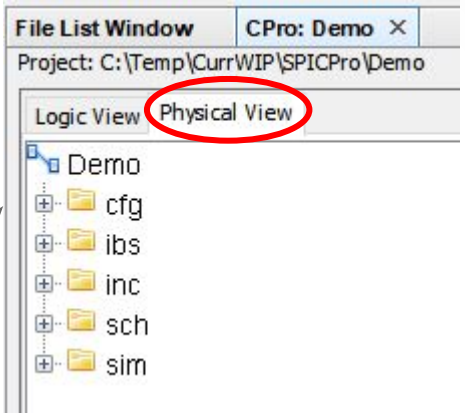
Flow Overview:

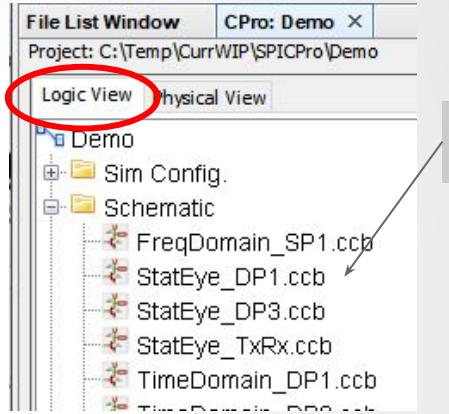


Project folder structure layout

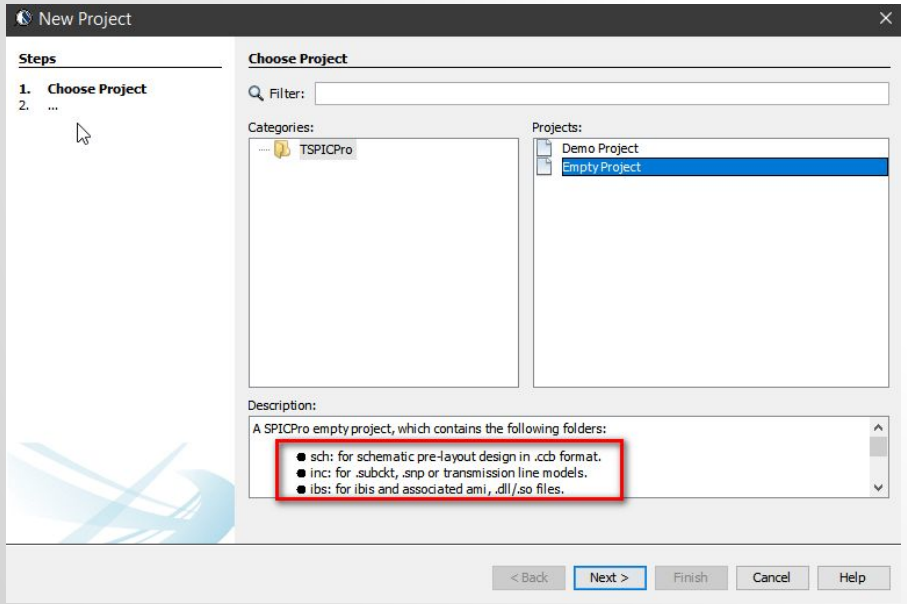
- Create an empty or demo project:
  - In the working folder:







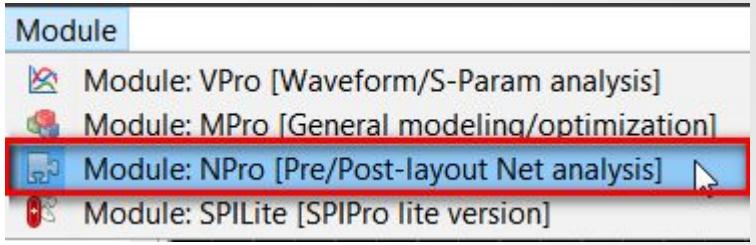
double-click to open

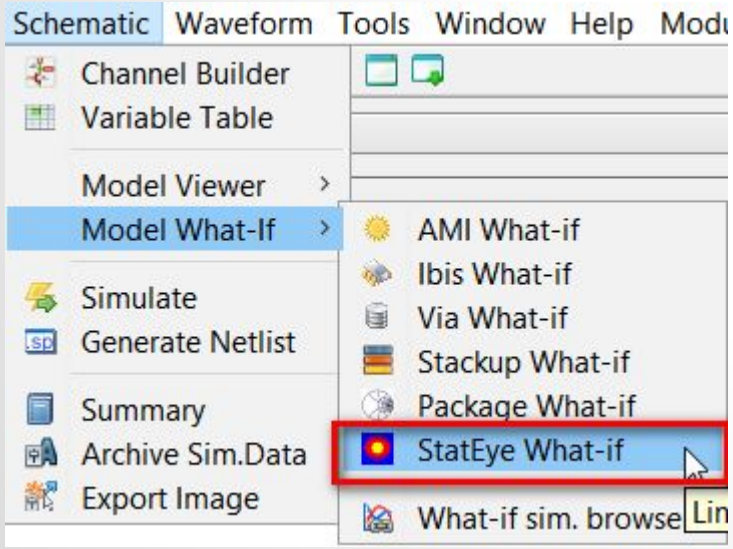


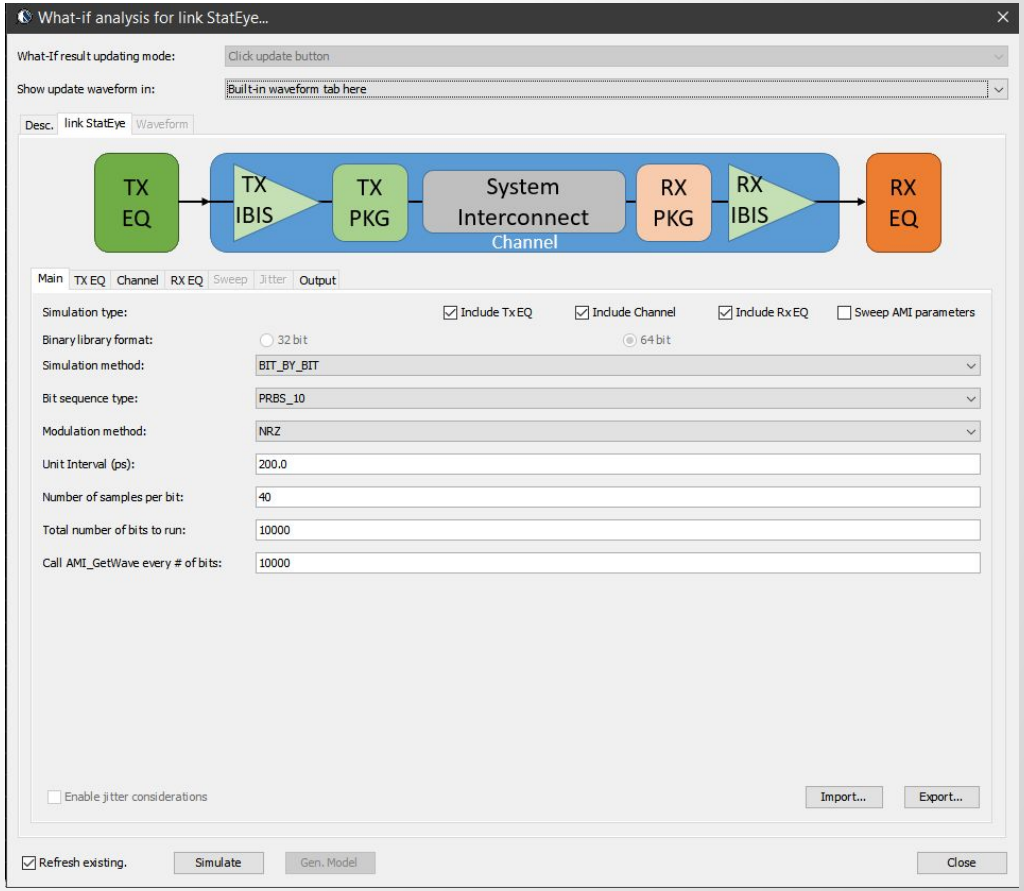
- Save schematic design built from channel builder as “.ccb” files put under “Sch”
- Put IBIS (.ibs, .ami, .dll/.so) model files under “Ibs”
- Put transmission lines RLGC models (“.tab”), S-parameter (“.snp”) files or spice sub-circuit (“.sp”) files under “Inc”
- Tool will save config. settings as “.cfg” 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”

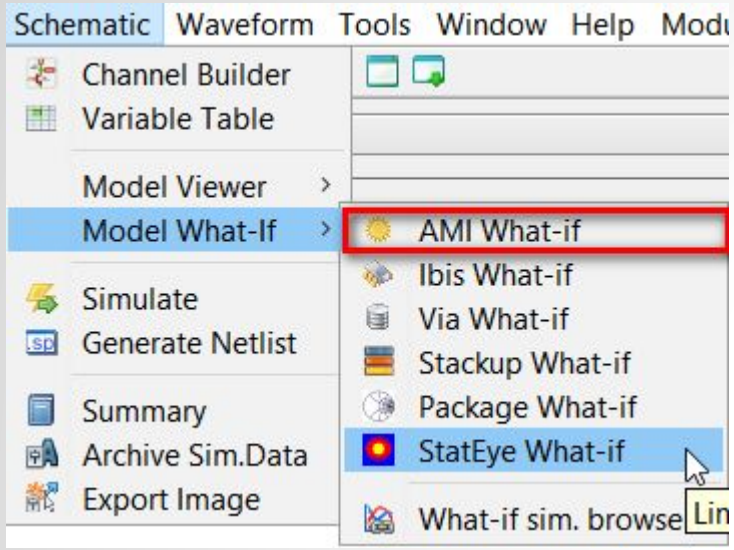
- Switch to NPro:
  - For Pre-layout net analysis
- Don’t have any models yet?
  - Try “StatEye What-if for quick analysis.

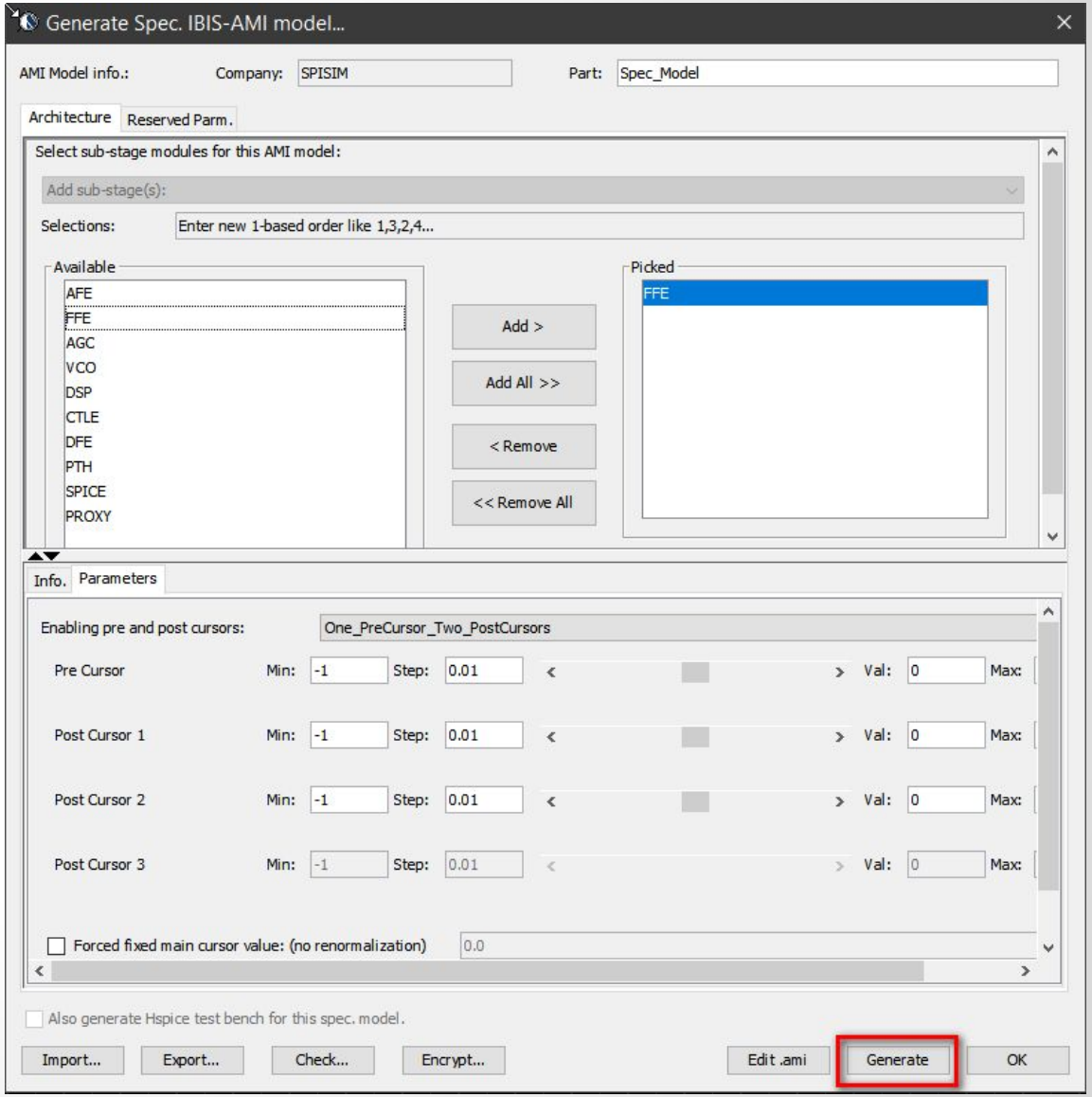






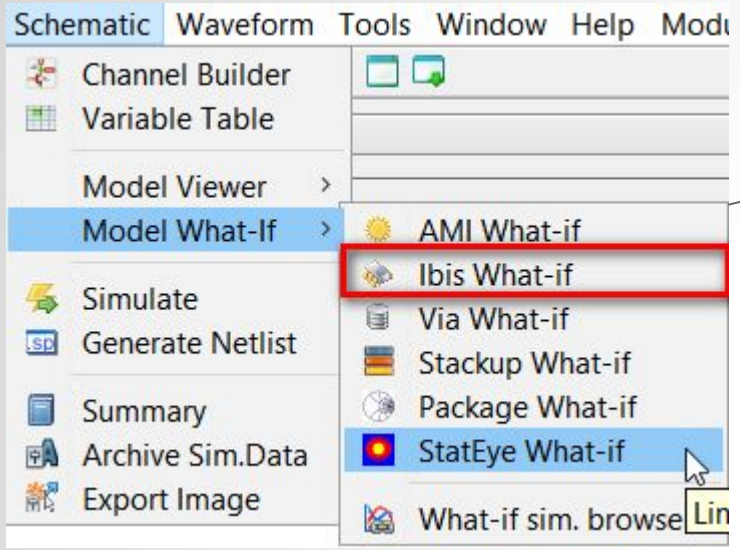
Generate “What-If” AMI models:

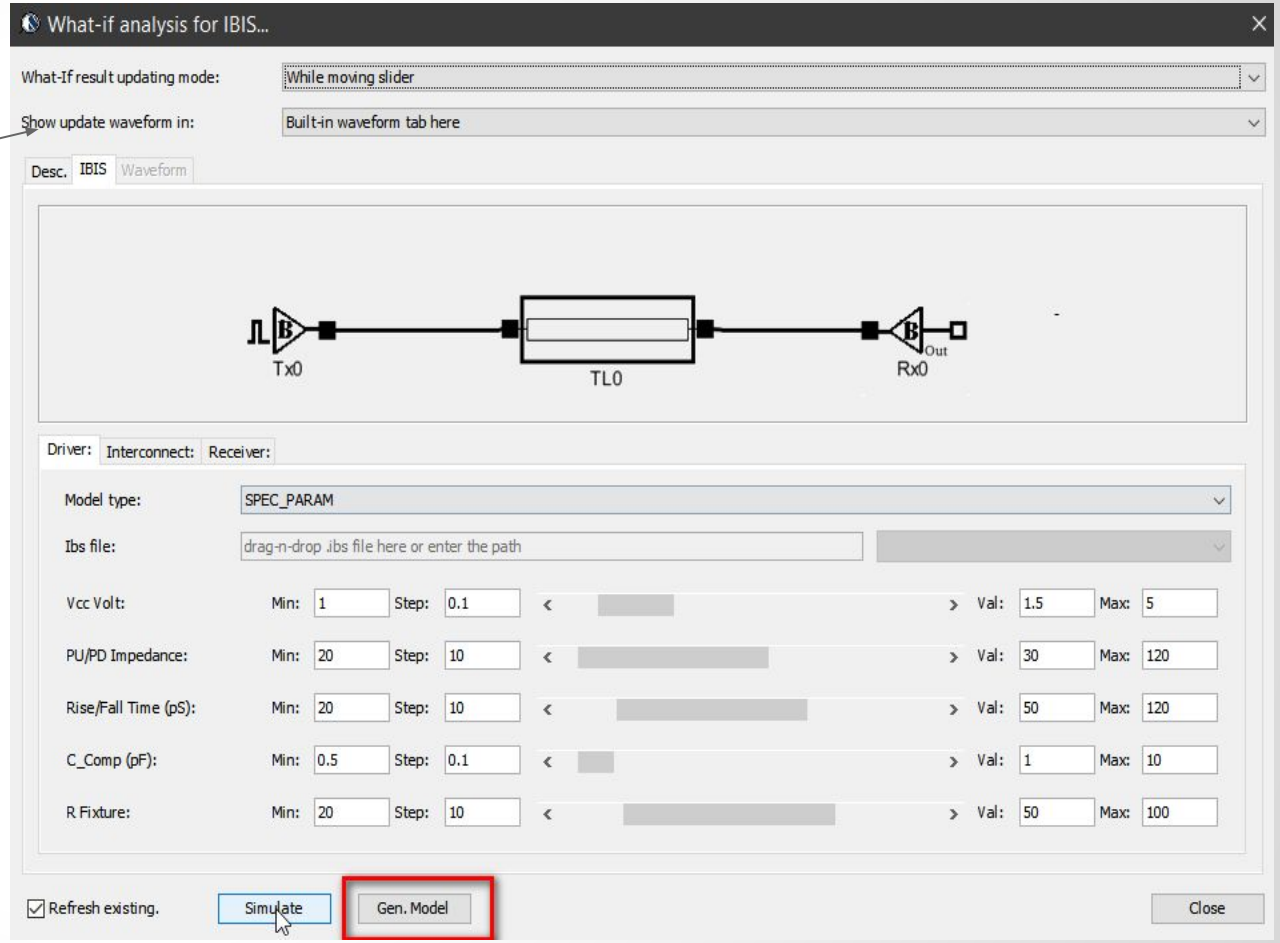




- Generated AMI model will expire in two weeks.
- Can be fully unlocked by BPro w/ AMI.

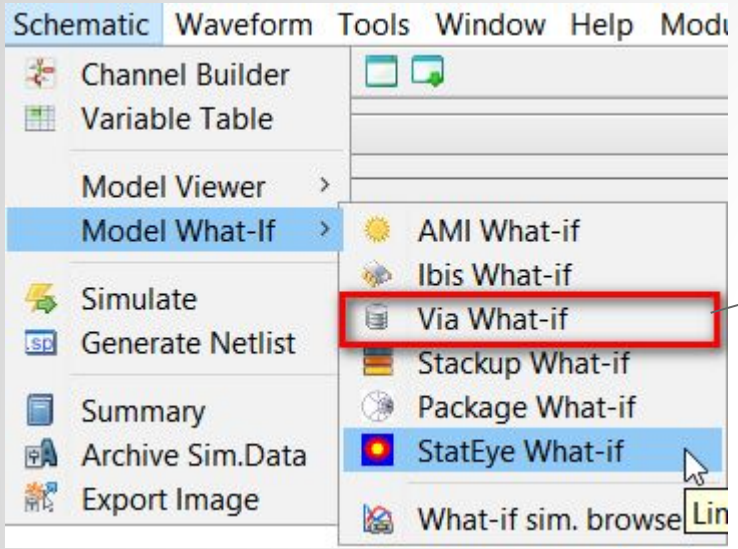
Generate “What-If” IBIS models:

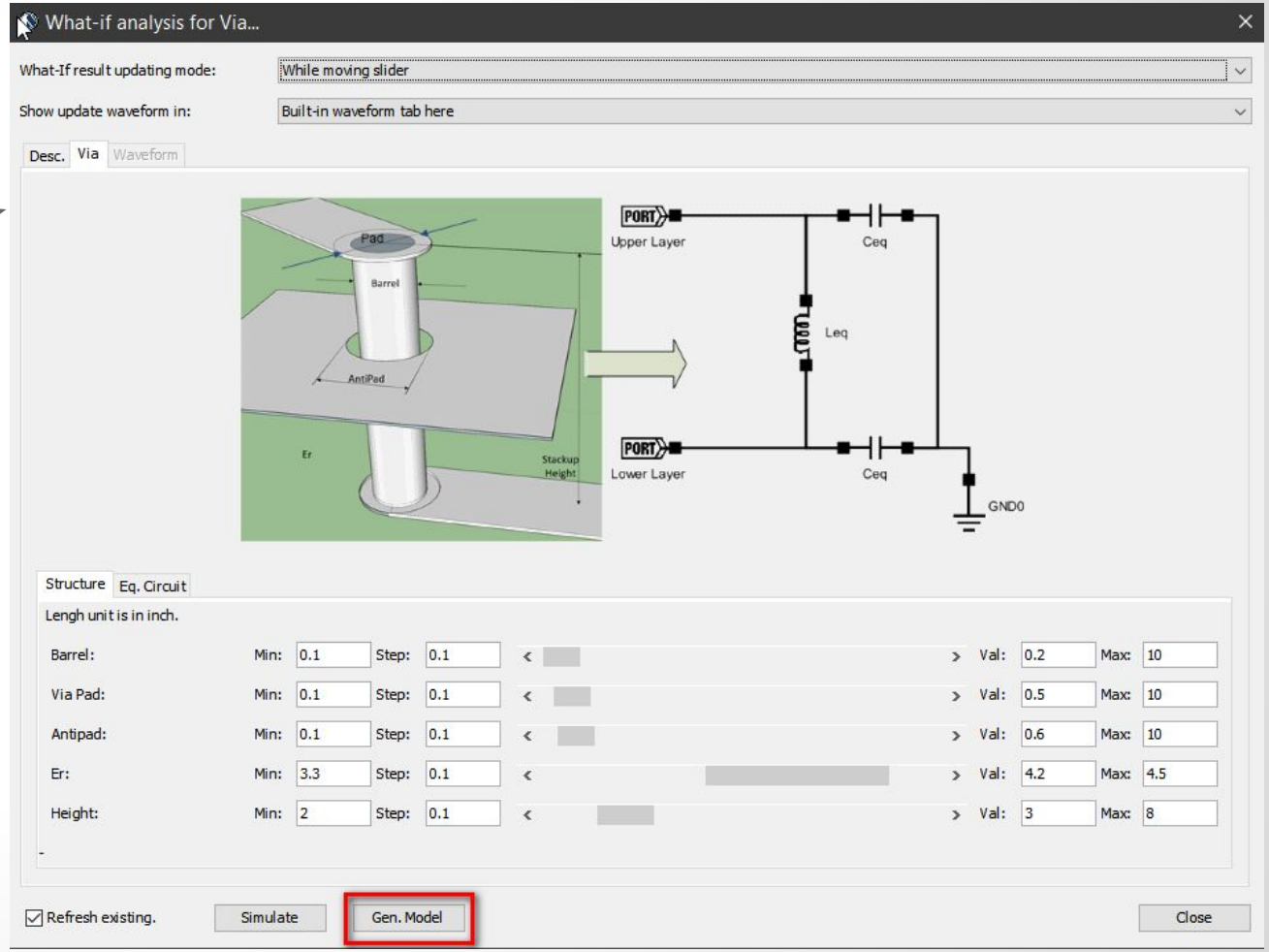




- Generated IBIS model based on performance.
- IBIS from silicon etc supported by BPro.

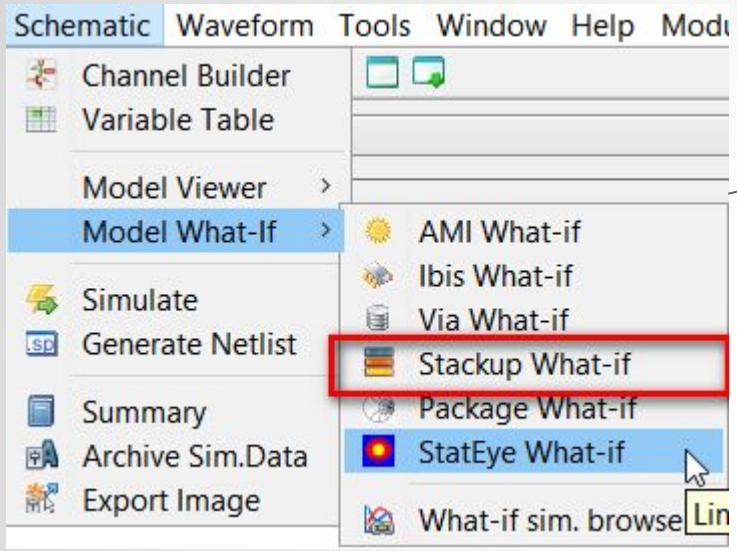
Generate “What-If” Via models:

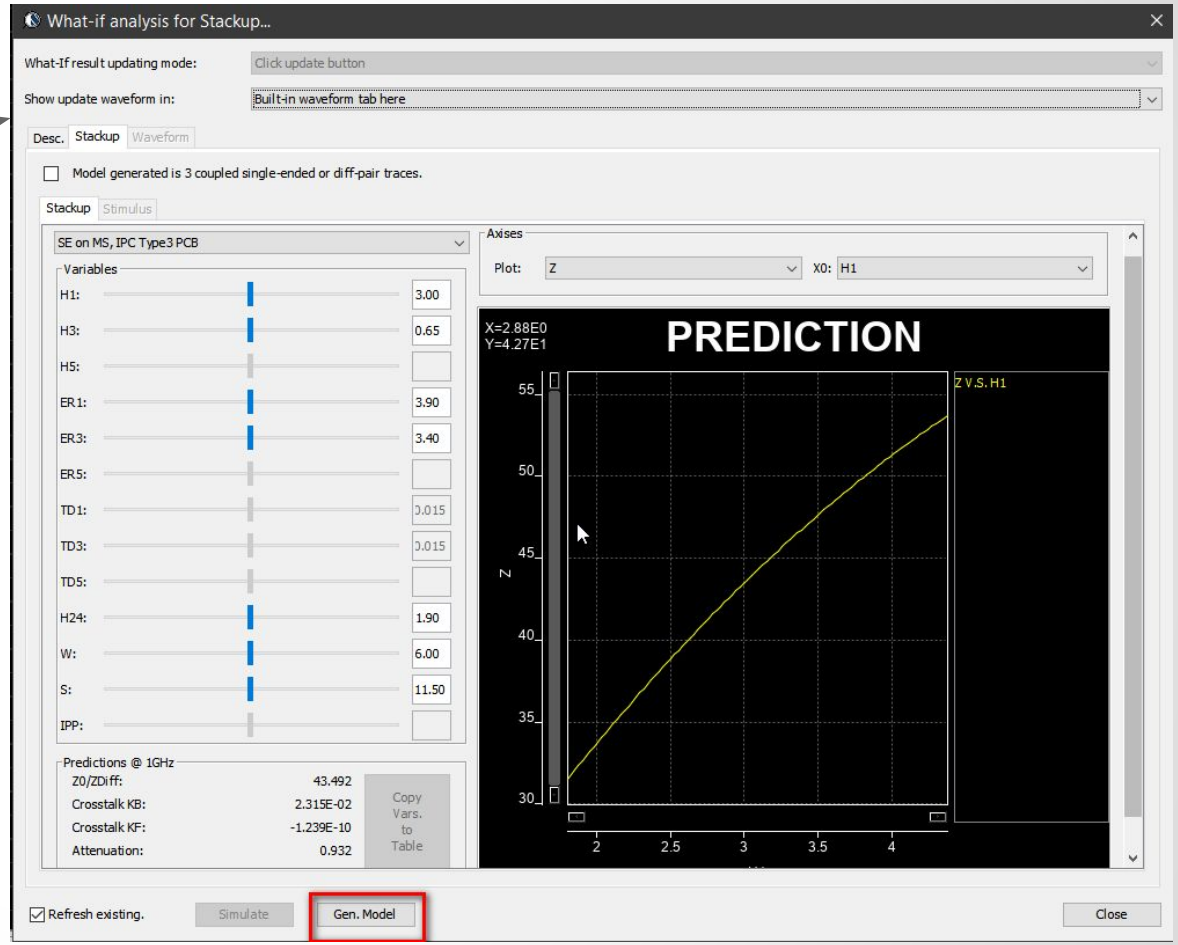




- Generated .s4p for equivalent model.
- No crosstalk coupling.
  - Use 3rd party tool like HFSS for full extraction.

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

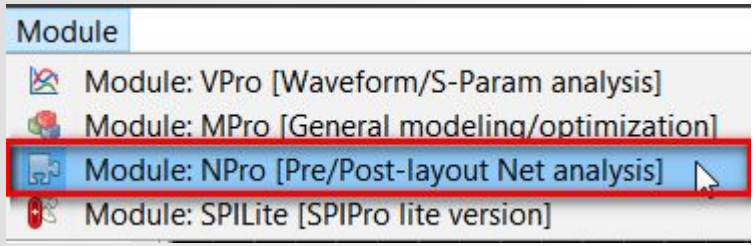


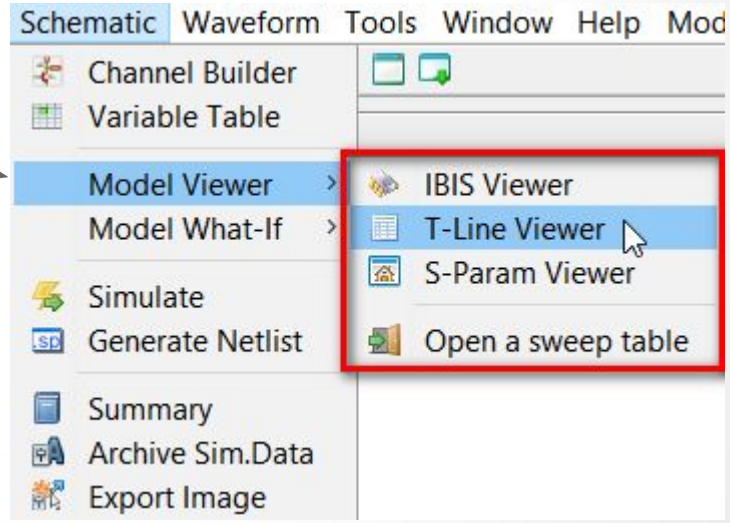


- Generated frequency dependent tabular RLGC models (.tab).
- Full stack-up modeling supported by TPro.

Model preparation: 2/2 Inspect Quality

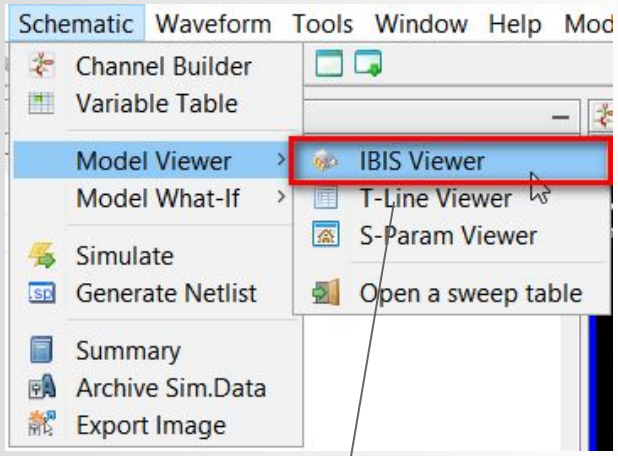
- For existing or generated component models:
  - Check/tune their quality against the spec.
  - These “viewers” are subset of other SPISim modules

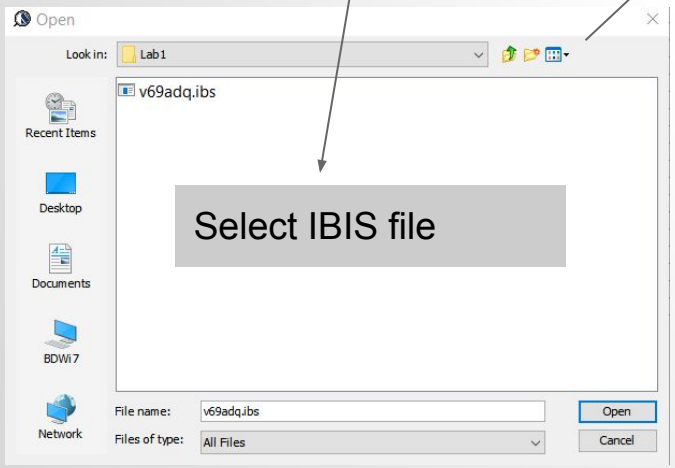


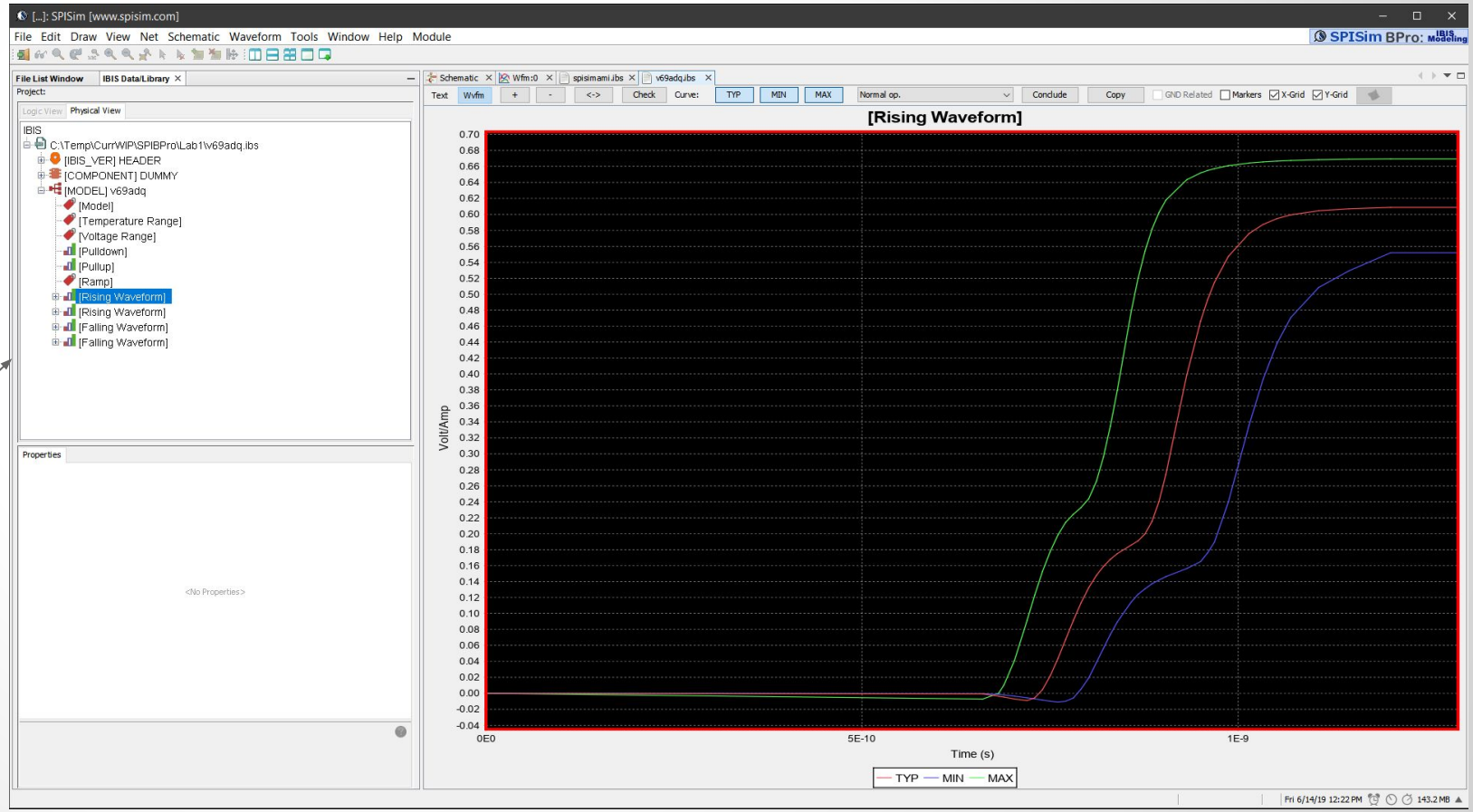


Inspect IBIS models

- 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"

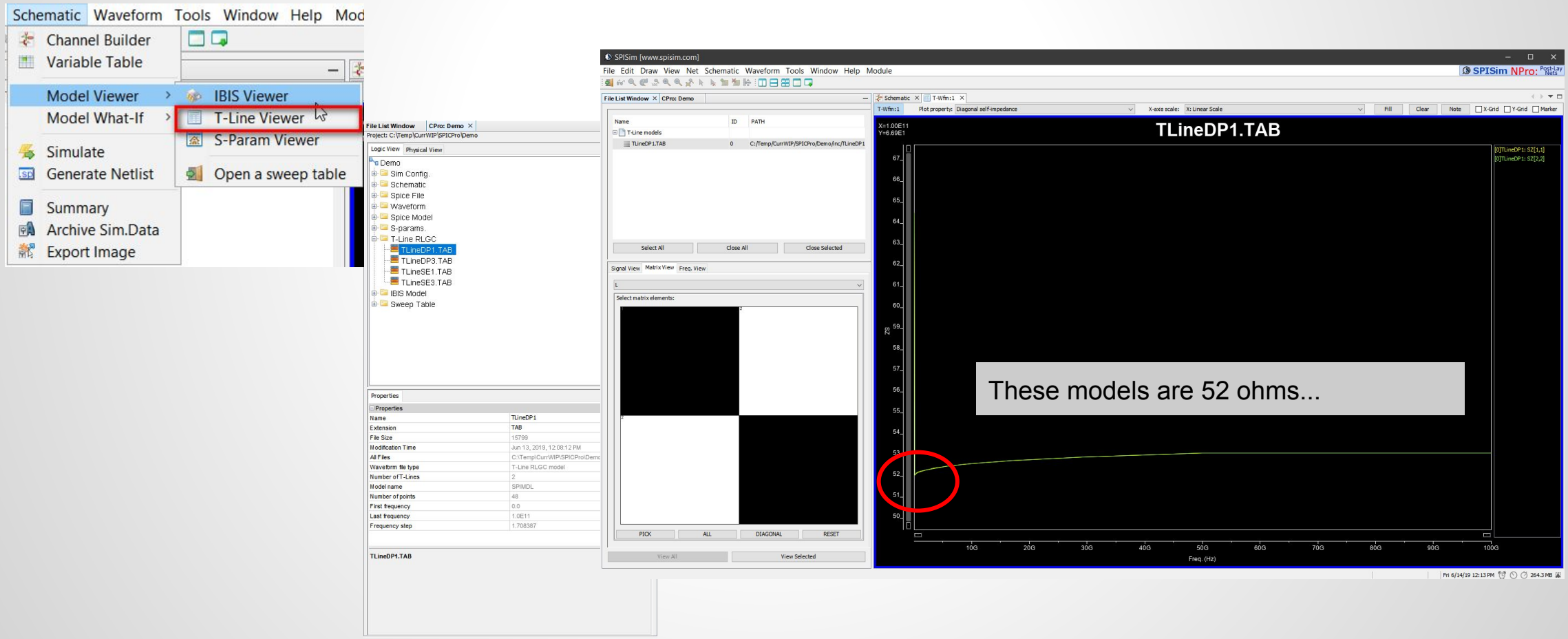






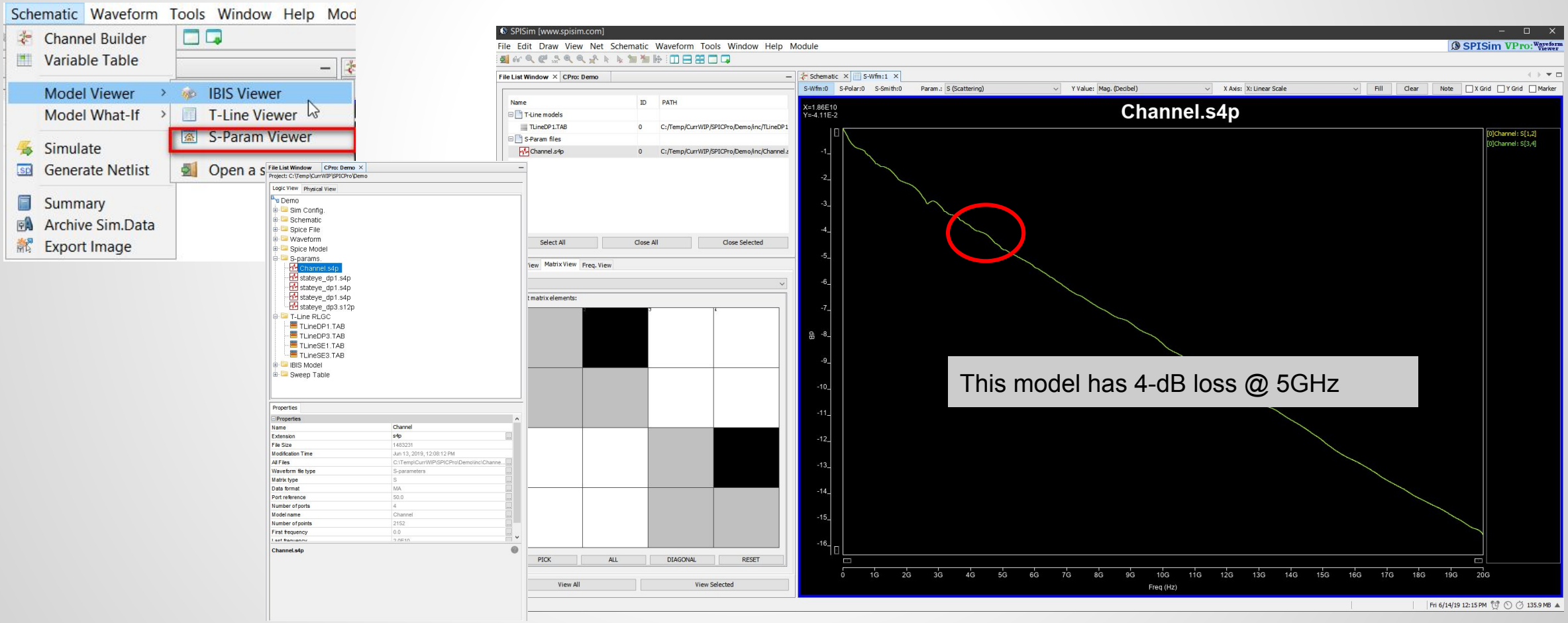
Inspect transmission line models

- 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"



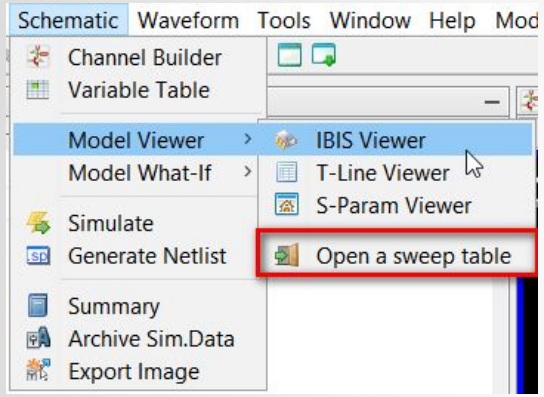
Inspect S-parameter models

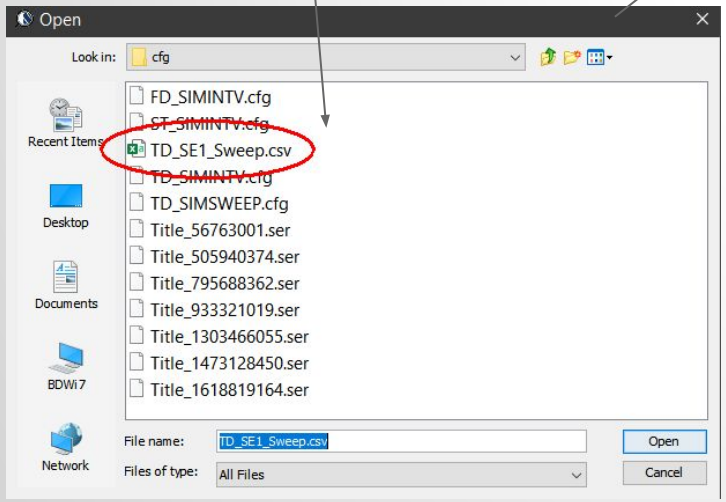
- Check model losses, TDR/TDT etc
  - Double click ".Snp" file in project navigator or drag-N-drop into "FileList window", then view with "S-param viewer"

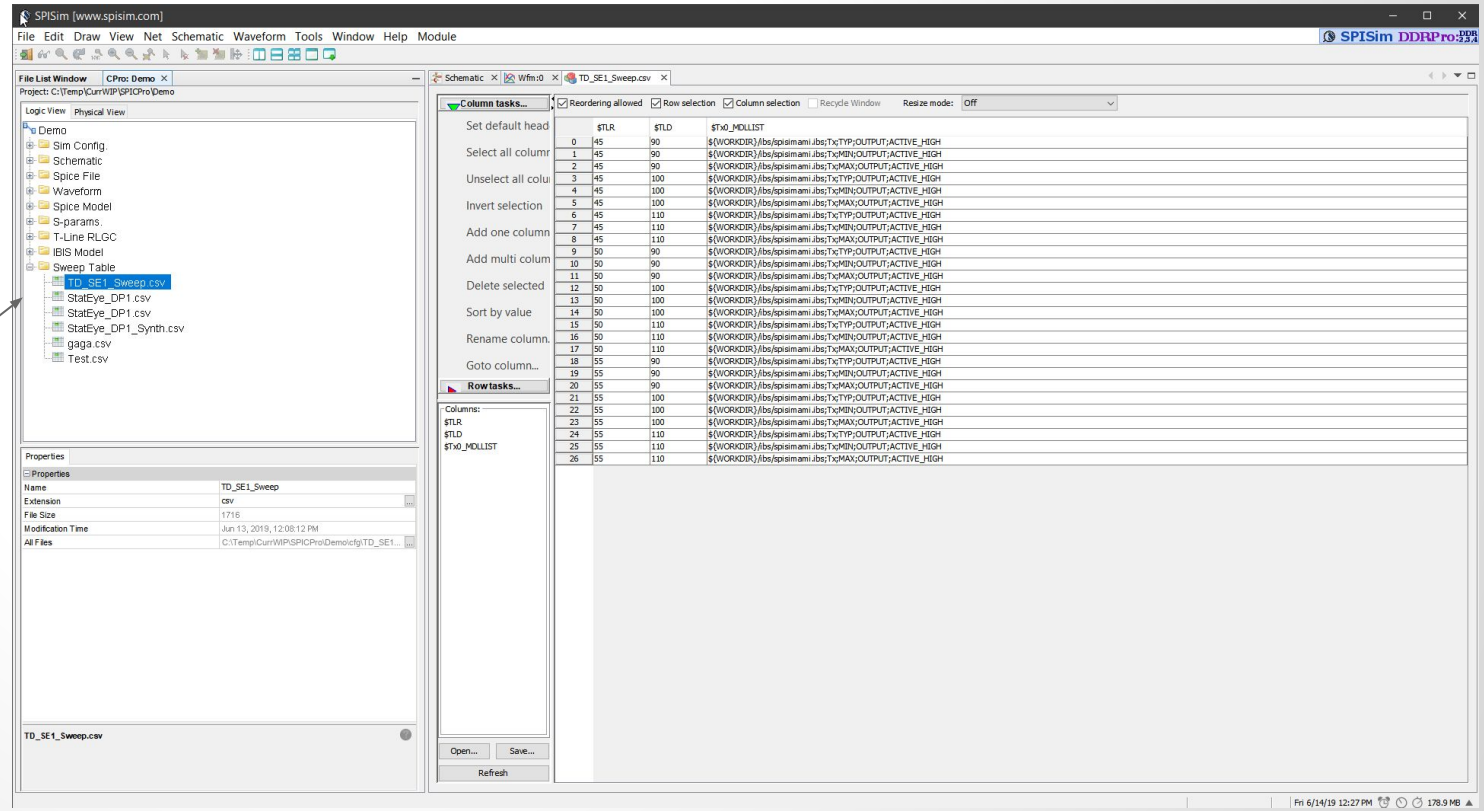


Parameter sweep table

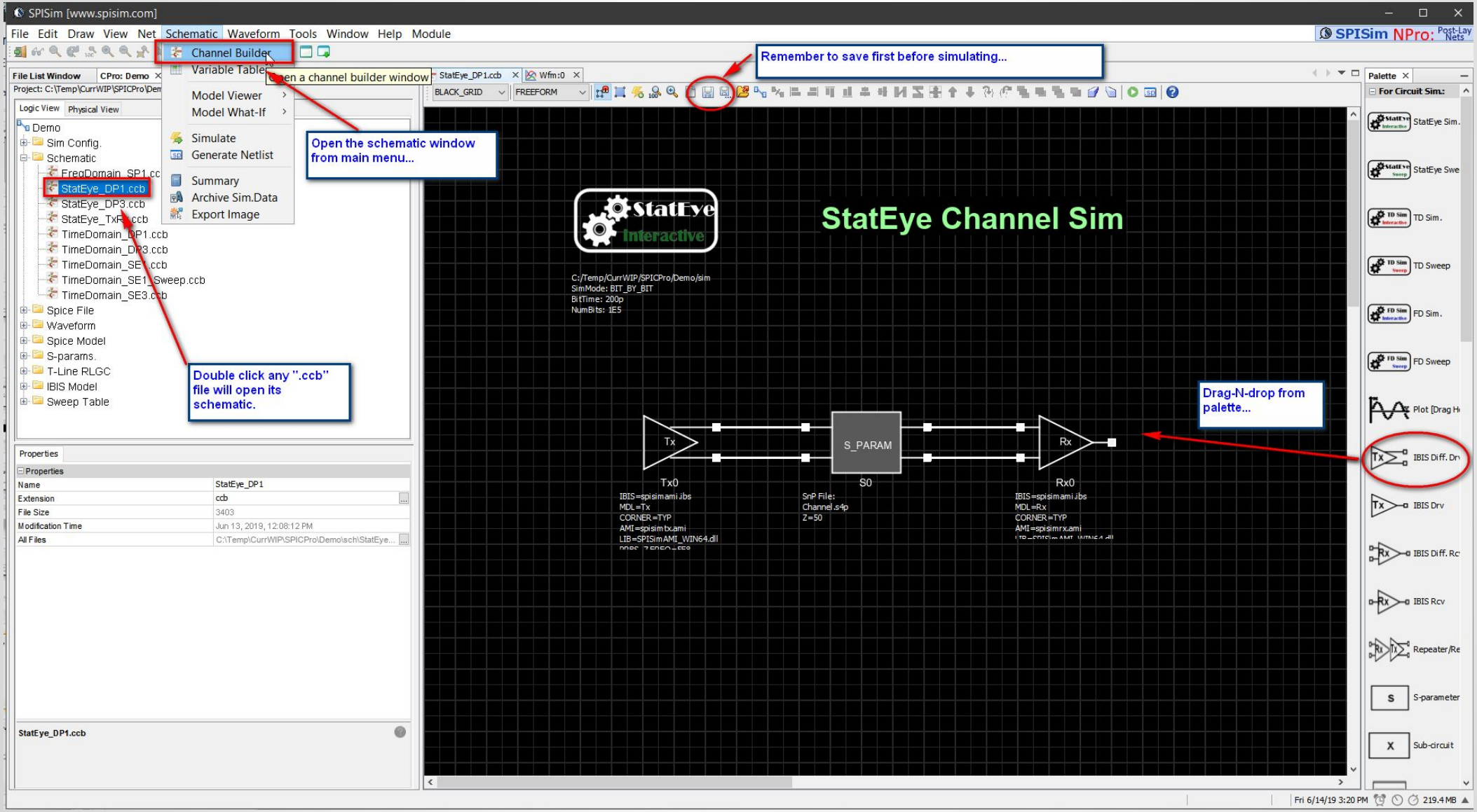
- Double click ".csv" file in project navigator or via menu "S-param viewer"
- Plain text ".csv" files can be generated by SPISim's MPro or using tool like Excel
  - MPro has DOE, Full factorial etc sampling algorithm



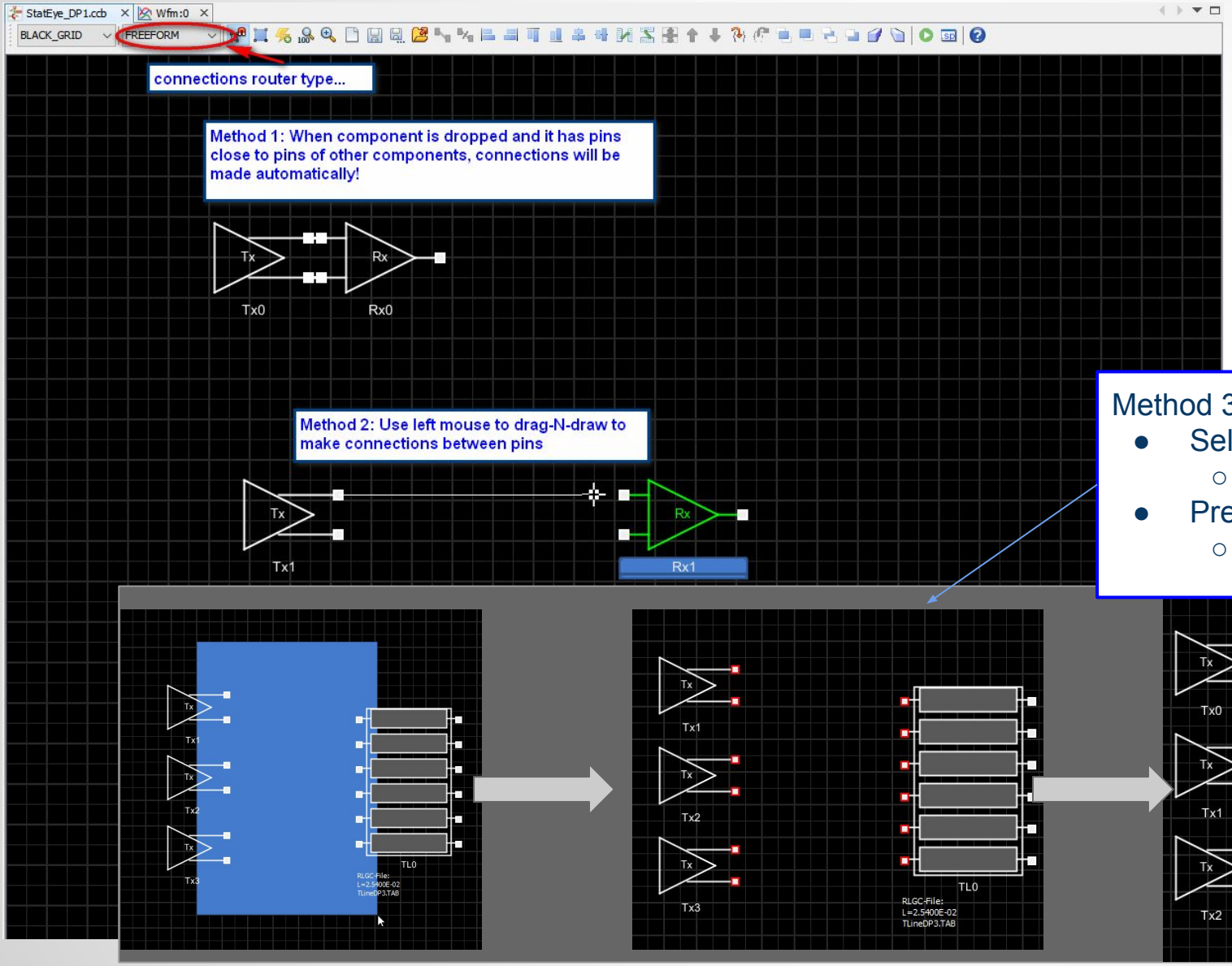




Flow Details: Channel builder

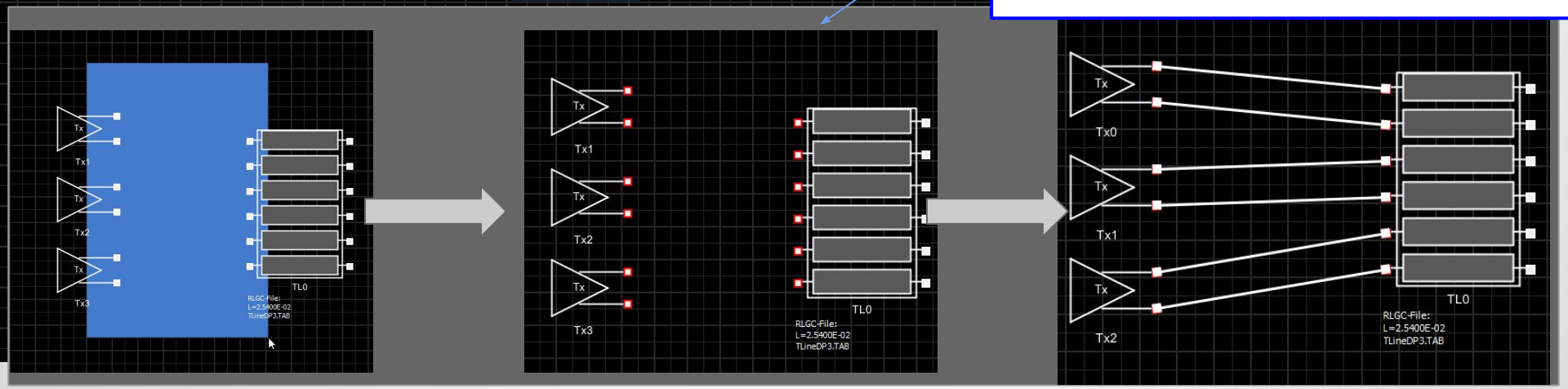


Channel builder: Making connections



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

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.

Tx0
IBIS=spisimami.ibs
MDL=Tx
CORNER=TYP
AMI=spisimtx.ami
LIB=SPISimAMI_WIN64.dll

Tx1
IBIS=spisimami.ibs
MDL=Tx
CORNER=TYP
AMI=spisimtx.ami
LIB=SPISimAMI_WIN64.dll
PRBS_7 FREQ=5E8

Tx2
IBIS=spisimami.ibs
MDL=Tx
CORNER=TYP
AMI=spisimtx.ami
LIB=SPISimAMI_WIN64.dll
PRBS_7 FREQ=5E8

EYE
EYE0

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

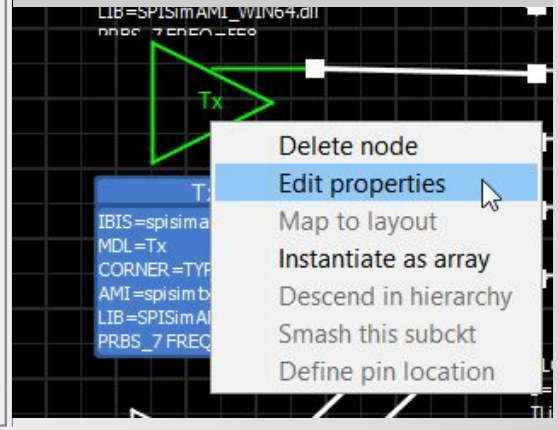
Sweep simulation plan:

Automatical probe(s)/ports:

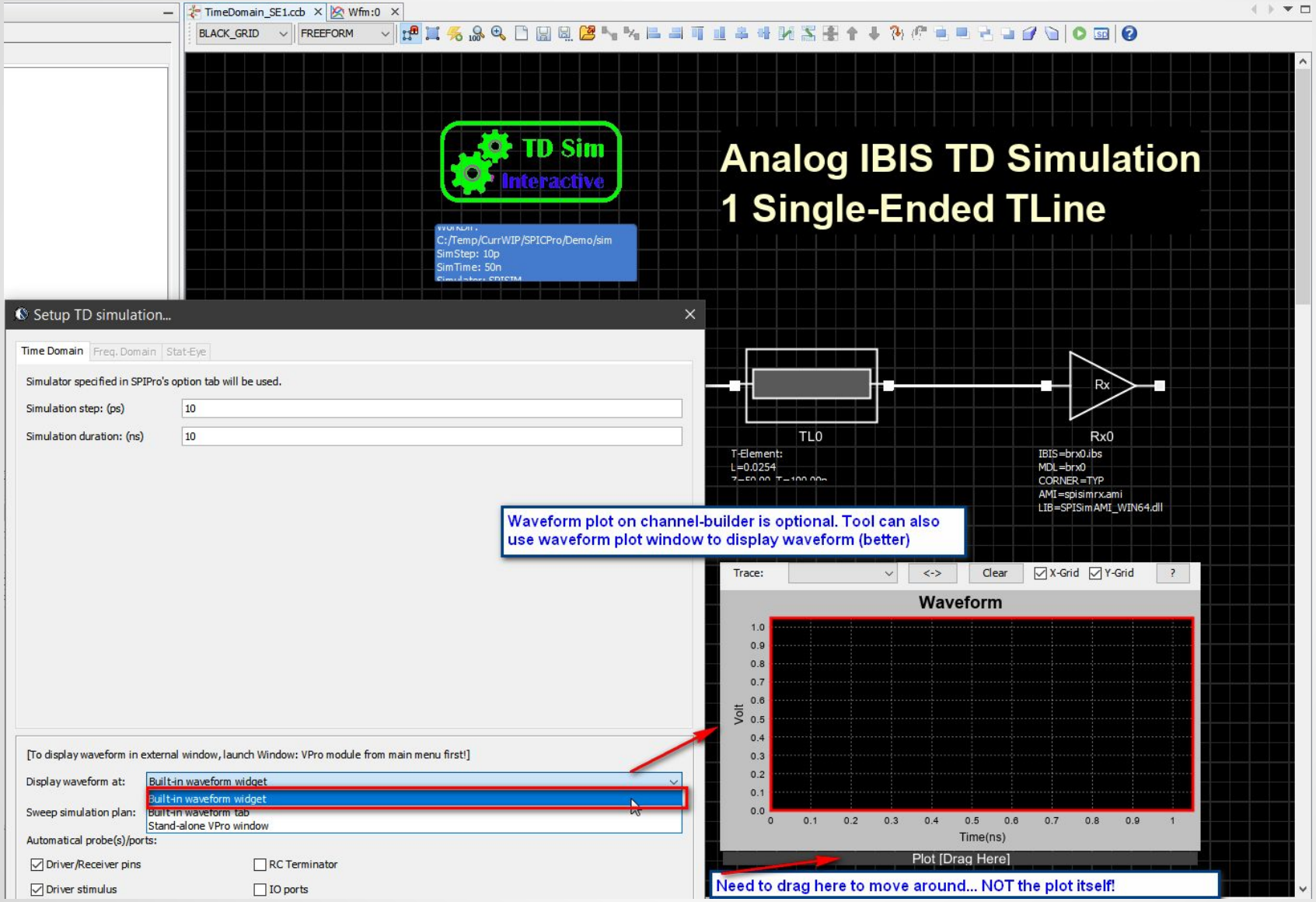
- [x] Driver/Receiver pins
- [ ] RC Terminator
- [x] Driver stimulus
- [ ] IO ports
- [ ] Via

OK | Cancel

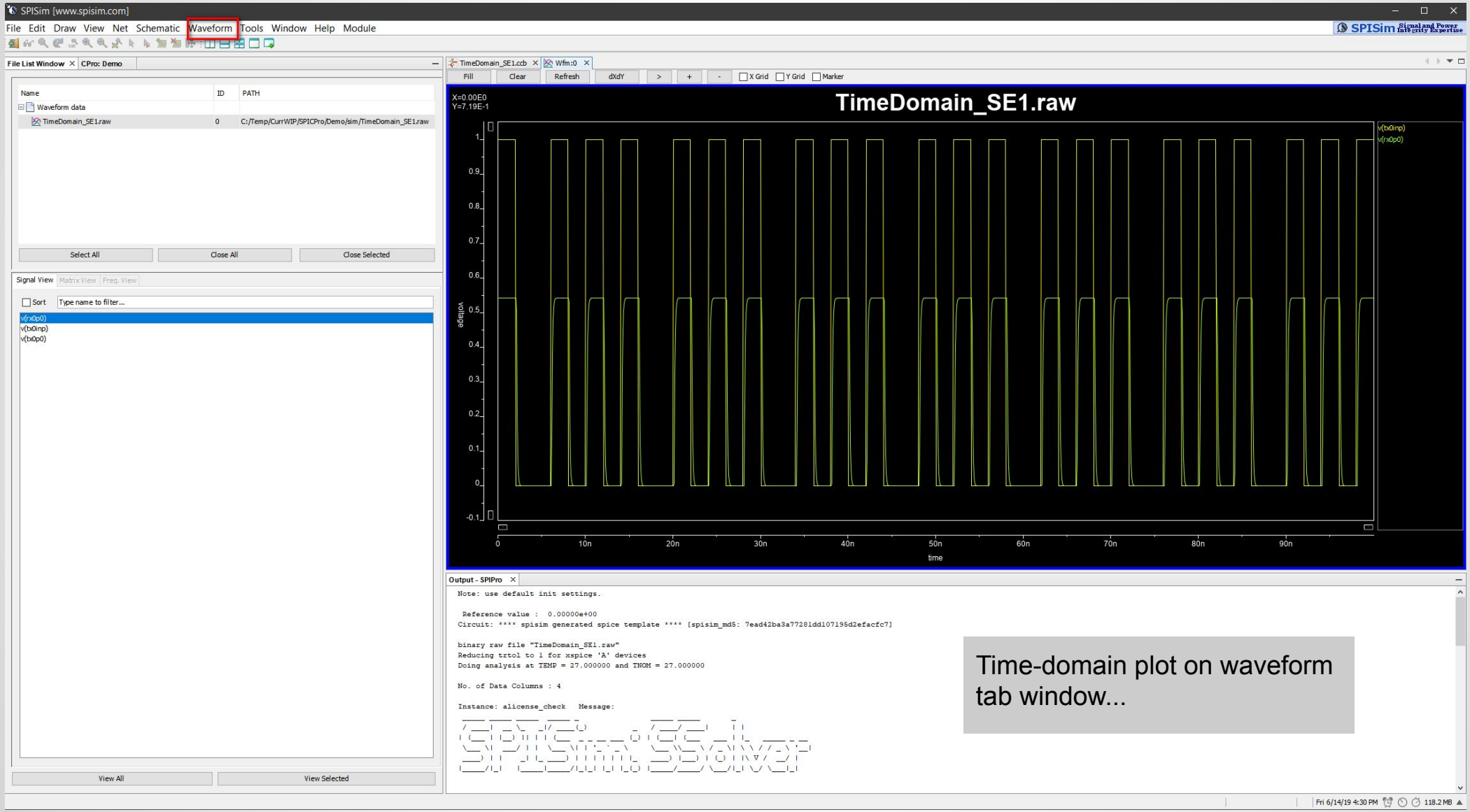
Right-click pop-up...



Flow Details: Interactive simulation

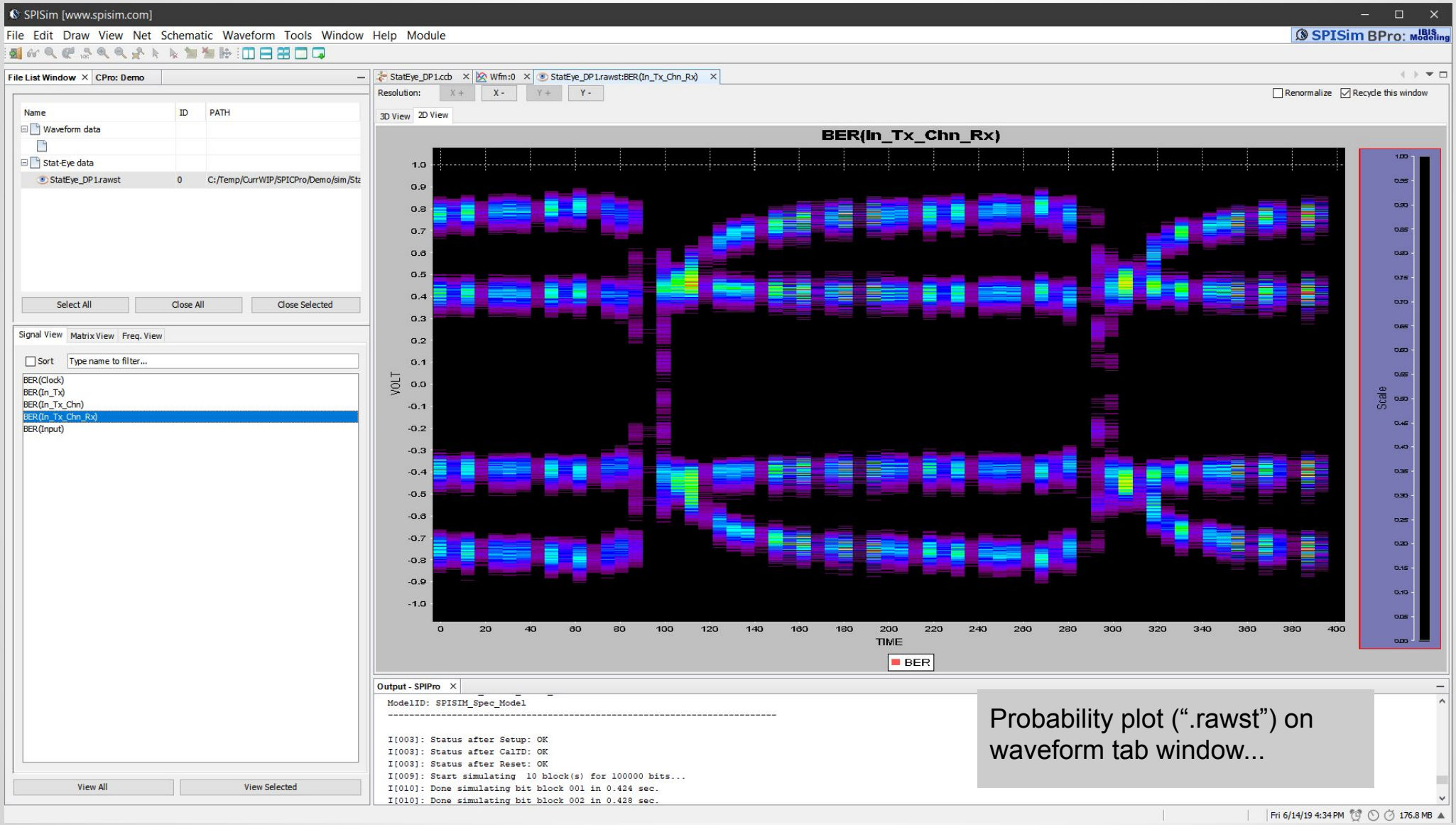


Flow Details: Interactive simulation



Time-domain plot on waveform tab window...

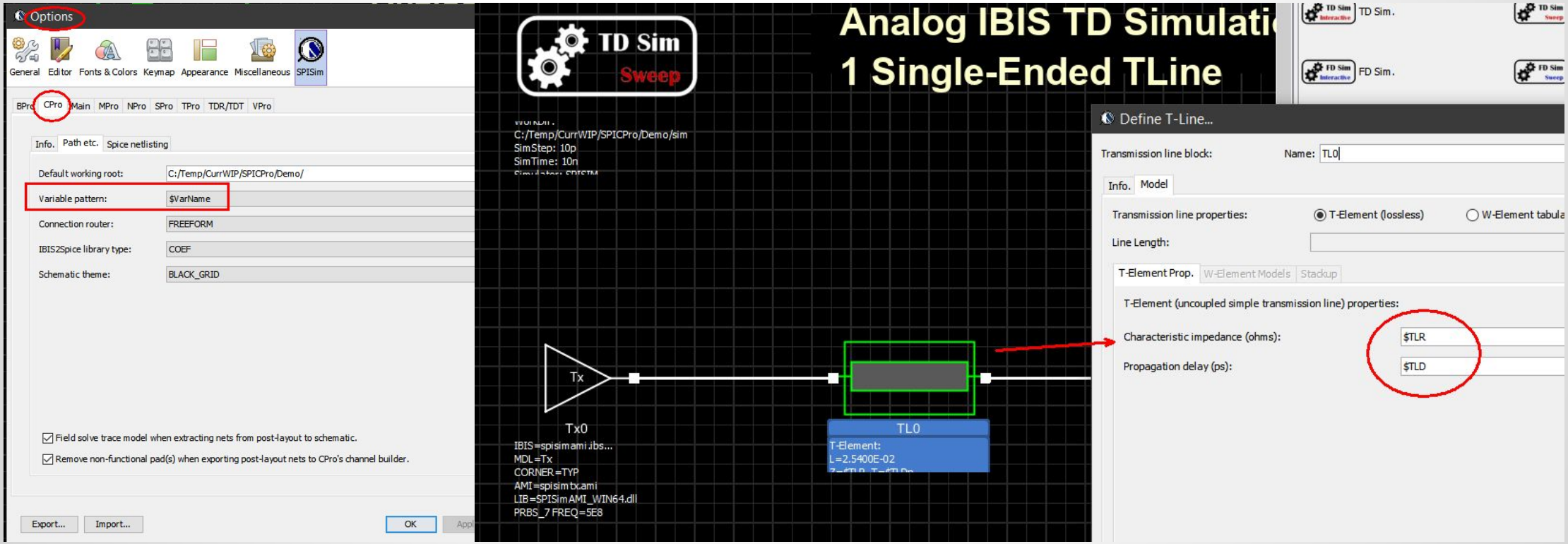
Flow Details: Interactive simulation



Probability plot (“.rawst”) on waveform tab window...

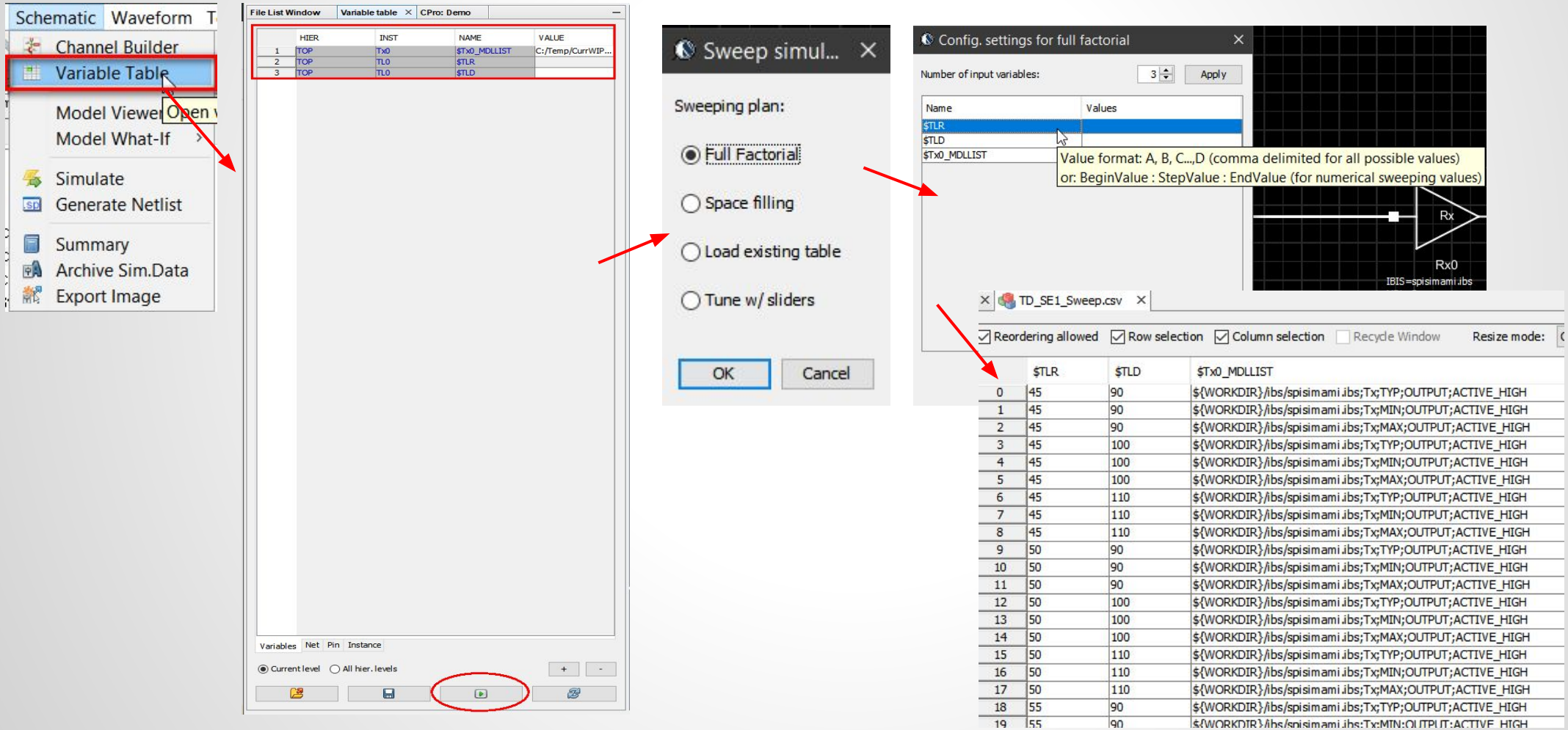
Flow Details: Prepare Parameter sweep

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



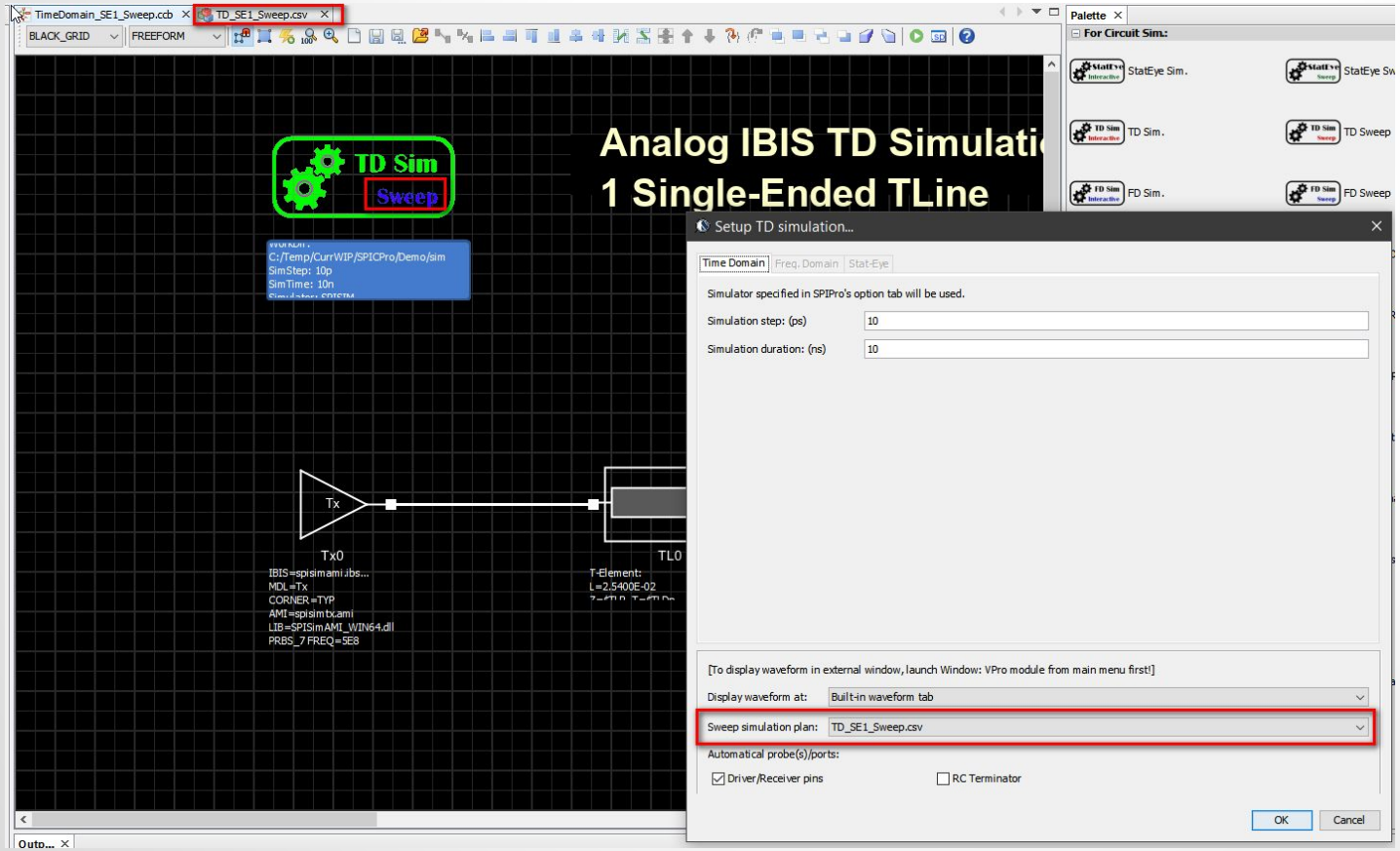
Flow Details: Prepare Parameter sweep

- Generate simulation plan



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

EDA Expertise in Signal, Power Integrity & Simulation

SPISim is an InSync member.

SYNOPSYS
Integrated with
HSPICE®
Learn More >>