

Tutorial of *SpectreRF* Simulations (no submission required)

1. In this tutorial, we will introduce *SpectreRF* circuit simulation using the LNA circuit shown below as an example. You will acquire the hands-on experiences of performing common RF simulations such as noise figure (NF), input-interception point (IIP_x) and S-parameters (S_{xx}) by following the step-by-step instructions in the manual file provided. The simulation examples in the manual include advanced topics such as mixer's conversion gain and VCO's phase noise and are left for reader's interest.

General guidelines are as follows:

- (a) The step-by-step instruction is well documented in the file *spectreRF.pdf*. You can download the file at <https://inst.eecs.berkeley.edu/~inst/pub/nis/?file=Cadence/spectreRF>, EECS instructional account required.
- (b) Like HSPICE, Spectre needs the user to point the path to a device model file. You can use the one at </share/instsw/cadence/tools.sun4v/dfII/samples/artist/models/spectre/rfModels.scs>. You can save a copy of *rfModels.scs* at your local directory and include the model path as instructed in p.407 of *spectreRF.pdf*
- (c) Create a temporary directory under your home directory, e.g. user/142Examples
- (d) Make sure Xwindow connection is established. Bring up Cadence by typing icfb2, go to Tools→Library Manager→Edit→Library Path, fill in new library name, e.g. EE142, and the path, e.g. user/142Examples (p.223)
- (e) In library manager, click rfExamples→click lnaSimple→right click schematic (view)→copy and choose to copy from rfExamples into EE142
- (f) Perform LNA gain simulation following the instructions in p.401-415 in *spectreRF.pdf*, S-parameters simulation in p.422-429, noise simulation in p.450-462, linearity simulation in p.463-478. Try to plot your results in the result window and compare with those in the manual. Before you start the simulation by hitting *netlist and simulate* in the analog design environment, be sure that the *check and save* icon in the upper left of the schematic window is always clicked after a change to the schematic (e.g. wiring, device properties,...etc.) has been made since the last simulation. Otherwise the simulation won't get start.
- (g) repeat for cell ne600p (for mixer) and tline3oscRF (for VCO)
- (h) for interested readers, <https://inst.eecs.berkeley.edu/~inst/pub/nis/?file=Cadence/spectreRFTheory>.

