Useful FPAA Tools / Remote Tool comments

General Tool Setup Issues

The VM will run slowly if you have not allocated enough space for the virtual machine in the virtual box. If so, go to the system settings, increase memory space reserved for the VM. Do install the extension pack for USB 2.0 or 3.0 depending on what your USB is.

While updating/resetting the tools from the blue GUI, make sure you have an internet connection and wait for the confirmation message to appear and do not close the tools.

General Scilab / Xcos Tool items

Define your input myVariable and all the other variables you have used in the Xcos design, as a row vector, 1 x n matrix. If the number of blocks are vectorised, say, there are 'm' blocks, define the input as a m x n array. This is the primary cause for the "invalid index" errors. Make sure you don't have any spaces in your design file or directory names.

For load capacitances, you don't need to explicitly define a capacitance block at the node, in your Xcos design; the parasitic/routing capacitances at the node exist by default, due to the routing of the circuits. You can say "view routing" from the blue GUI, in case you wish to see how your design is routing through the blocks.

The "join" block allows connecting one output to multiple inputs, while the "in2in_x1" block helps to connect multiple outputs into one input. These can be found under Utility blocks.

Xcos Simulation System Issues

During simulation, you can adjust the resolution by adjusting the time period of the clock block. The final integration time under simulation setup controls the time duration for which you want to simulate your design. The different solvers like RK-45, Sundials etc. can be chosen too from the solver kind drop-down in simulation setup.

The refresh period in the scope block determines the time for which the figures are plotted in the figure window. I would recommend to setup the refresh period to be same as the final integration time you have specified under simulation setup, to observe the figures over the entire time interval of simulation.

The "To Workspace" blocks allow you to save your variables and the variable.values and variable.time can be saved to .csv files, using csvWrite commands. If you are not able to see anything in the graphic window which plots the result, it maybe because the range of the result set may not have been in the right region. The Ymax, Ymin vector in the plot are set through the scope block and their ranges can be edited also from the axes properties in the graphic window itself.

In the simulation schematic, if some connections are missing in the design, you may get the error: "the number of equations does not match the number of variables", so make sure you haven't left any floating nodes in the design schematic. Make sure to connect the ammeter in

series or the voltmeter in parallel, otherwise you may get a structurally singular jacobian error.

The explicit blocks can be connected to the implicit blocks, those which are used for the level=2 simulation, through connectors like CVS. For example, arbitrary function generator is a non-electrical block and since mosfet is an electrical implicit block, the arbitrary generator can be connected to the transistor terminal through the CVS- controllable voltage source block (found in electrical blocks palette).

Circuit Design Issues

You can program the design from 40pA to 10uA.

The decision to use OTA/ FG-OTA depends on what you wish to achieve. An OTA gives a higher gain, while the FG-OTA gives a better linearity and also helps to set the DC offsets, by modulating the p bias and n bias.

Remote System Issues

For using the remote system, make sure you have turned the setting to allow for access for “less secure apps” ON in your email account settings. Otherwise, you may encounter authentication issues. After entering your details in the GUI, make sure you press the enter key, before saying send email.

After downloading the remote system results, save them as it is (do not rename the zip file or do not unzip it) in the directory same as the one you compiled the design. Note that the results are stored in “rm_results” variable which can be accessed from the scilab workspace. While using the boards locally, if you are using measure voltage / ramp adc, the data is stored in MITE_ADC_Output_voltage.csv or RAMP_ADC_Output_voltage.csv files in the same directory where you compiled your design.