

Often I am only designing a few cells to fit into a customer's existing chip design.

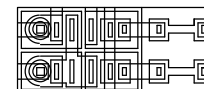
Rather than re-enter the customer's design into PSpice Schematics (with a potential for error), I simply include the customer's netlist(s).

To make this inclusion convenient I developed these PSpice parts (to the left).

To use, edit NETx to the appropriate node name, maintaining the same node order as in the customer's netlist, change "MODEL" to the netlist (subcircuit) name, then connect up with the additional circuit design.

Be sure to have either a .INCLUDE or .LIB in the schematic so that the subcircuit *definition* is established.

The Ux devices allow inclusion of 8, 14 and 16-pin discrete packaged parts.



ANALOG INNOVATIONS, Inc.
824 E. CATHEDRAL ROCK DRIVE
PHOENIX, AZ 85048-6300
(480) 460-2350 FAX: (480) 460-2142

Title: Import Netlists Into Your Schematic		
Size A	FileName: ...\SubcircuitImportByNetlist.sch	REV A
November 30, 2003, 9:27 AM		Sheet 1 of 1

