

Simulation console

Ngspice started...

No compatibility mode selected!

Circuit: * qucs 0.0.23 c:/users/mbrin/.qucs-s/qucs-smanual_prj/rc1testngspicexyce.sch

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

No. of Data Rows : 169

ASCII raw file "RCLtestNgspicexyce_ac.txt"

Reset re-loads circuit * qucs 0.0.23 c:/users/mbrin/.qucs-s/qucs-smanual_prj/rc1testngspicexyce.sch

Circuit: * qucs 0.0.23 c:/users/mbrin/.qucs-s/qucs-smanual_prj/rc1testngspicexyce.sch

Doing analysis at TEMP = 27.000000 and TNOM = 27.000000

Initial Transient Solution

Node	Voltage
n2	0
n1	0
n3	0
l1#branch	0
v1#branch	0

No. of Data Rows : 1015

ASCII raw file "RCLtestNgspicexyce_tran.txt"

Reset re-loads circuit * qucs 0.0.23 c:/users/mbrin/.qucs-s/qucs-smanual_prj/rc1testngspicexyce.sch

Circuit: * qucs 0.0.23 c:/users/mbrin/.qucs-s/qucs-smanual_prj/rc1testngspicexyce.sch

ngspice-36 done

Simulation finished

100%

Simulate

Stop

Save netlist

Exit