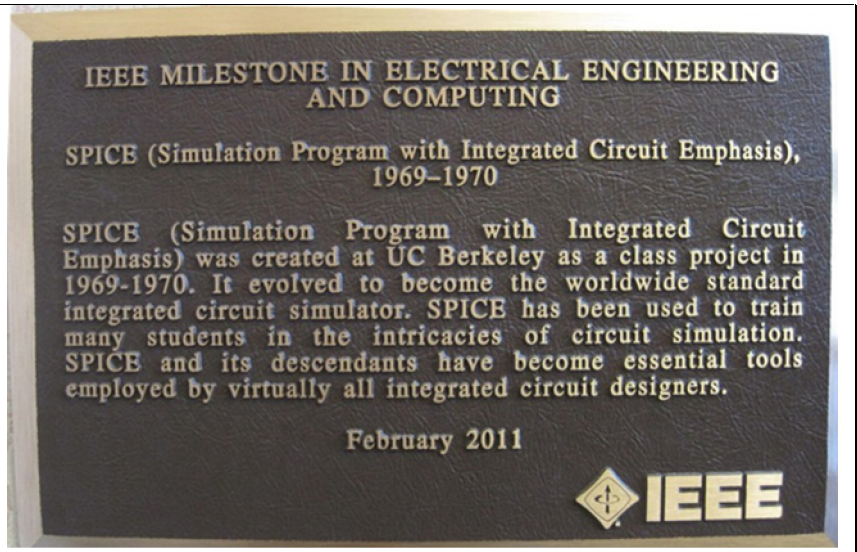
What is a Spice Model Card ?

SPICE



Sept. 2015
MOS-AK, Graz:

Laurence Nagel
Omega Enterprises Consulting
510.558.0842
lwn@omega-enterprises.net
251 Stanford Avenue Kensington, CA 94708
www.omega-enterprises.net

SisConsult

-2-

Dr.-Ing. Franz Sischka

Consulting Services
for Electronic Device
Measurements,
Data Verification
and Modeling

www.SisConsult.de

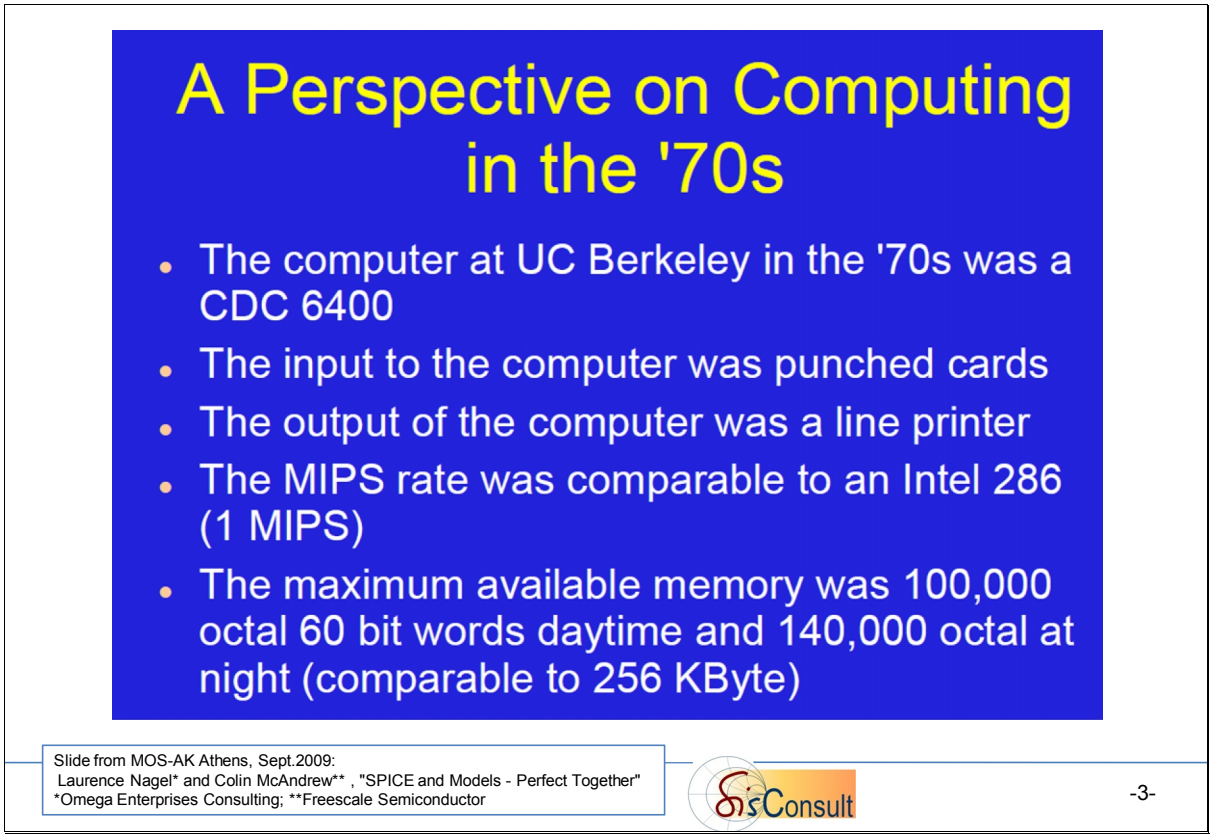
SisConsult
Ingenieurbüro - Engineering Office

27. March 2016, updated 31.March 2023

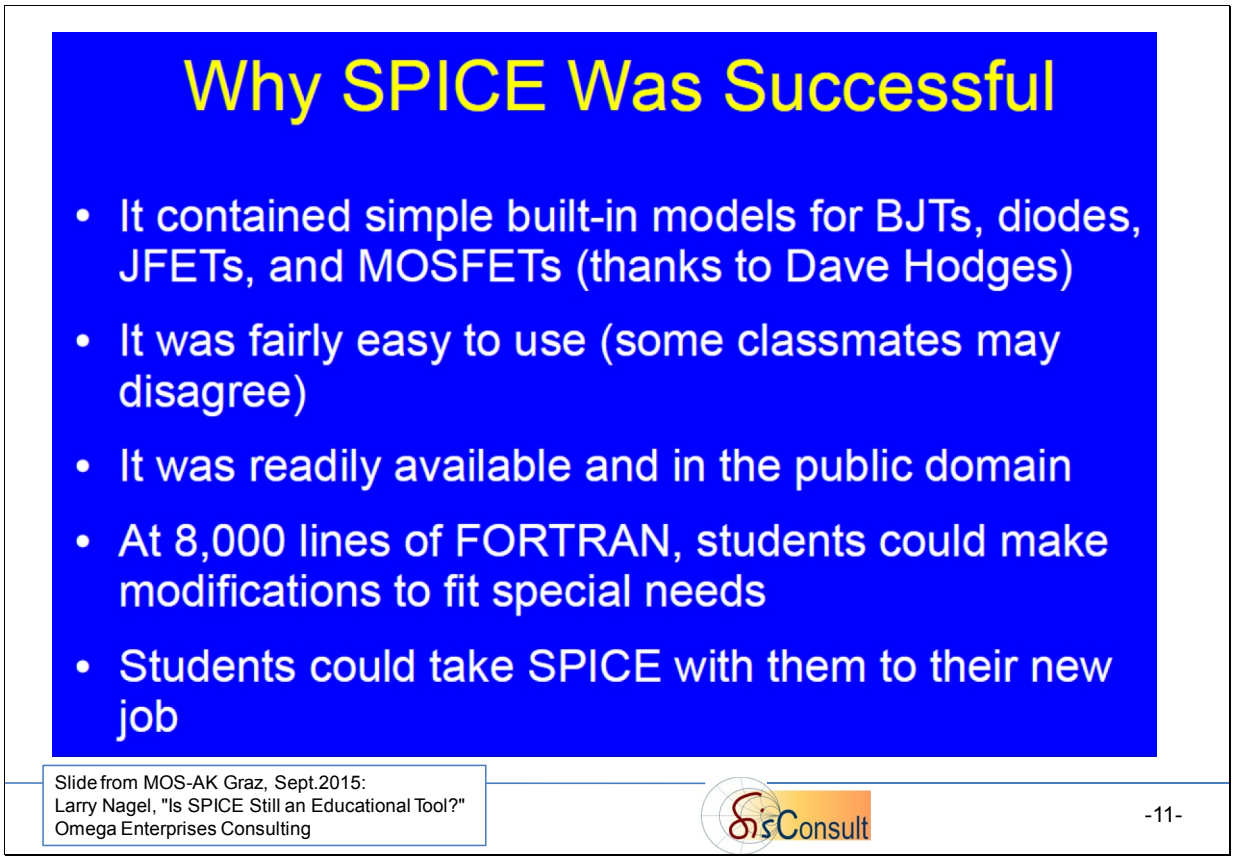
SPICE (Simulation Program with Integrated Circuit Emphasis) is a standard tool for circuit designers to simulate and predict the behavior of integrated circuits, before they are manufactured. Once manufactured, the circuits are measured and compared to the previous simulations.
A prerequisite for this flow is to have accurate and valid device models available, from which the designer composes his circuit.
The term 'with Integrated Circuit Emphasis' in the naming of Spice is an indication to the smart concept of the model implementation for integrated circuits (ICs):

- many transistors or diodes based on the same production process, but possibly different in size (for different current flow requirements)
- these many transistors/diodes with their individual connection nodes and geometry information, each link to the same model description (formulas) with the basic model parameters, which are valid for the geometrically normalized device size.

SPICE has been developed in the early 1970's at UC Berkeley/California by Laurence Nagel, to assist with teaching electronic circuit design to students . At that time, it was about 6000 lines of FORTRAN program code.

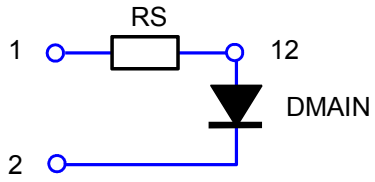


It became popular because of its simple and efficient netlist syntax, the implemented models (Diode, Gummel-Poon bipolar model) and its availability as a public domain software.



Let us discuss a simple netlist, consisting of a diode in series with a resistor, as shown below:

Diode Spice Netlist




```
.SUBCKT myDiode 1 2
RS  1  12  2.5
DMAIN  12  2  MAIN
.MODEL  MAIN  D  IS=1E-18  N=1  CJO=1p  VJ=0.6  M=.5  FC=.5  TT=1f  BV=100  IBV=1m  KF=0  AF=1  XTI=3  EG=1.11
.ENDS
```

SiConsult

-4-

The beauty of a SPICE netlist is its clear and simple structure:
for every electronic device, a line needs to be defined, beginning with a key character for the component (e.g. R for a resistor, C a capacitor, L an inductor, D a diode, Q a transistor), followed by the user-defined component name. Then come the nodes, and finally the device parameter value.
Example for a 2.5 Ohm resistor between nodes 1 and 12:
Rmyname 1 12 2.5

Diodes and transistors are usually described by two components:
- first the so-called instance, i.e. the key-character and the user-name, the nodes, and finally a link keyword to a 'model card'.
- the other line of code, the 'model card', starts with the keyword '.MODEL', followed by the previous mentioned keyword of the device instance, the link to the SPICE-internal model equations ('D' for diode model), and the model parameters (the specific parameter values for the SPICE model equations).

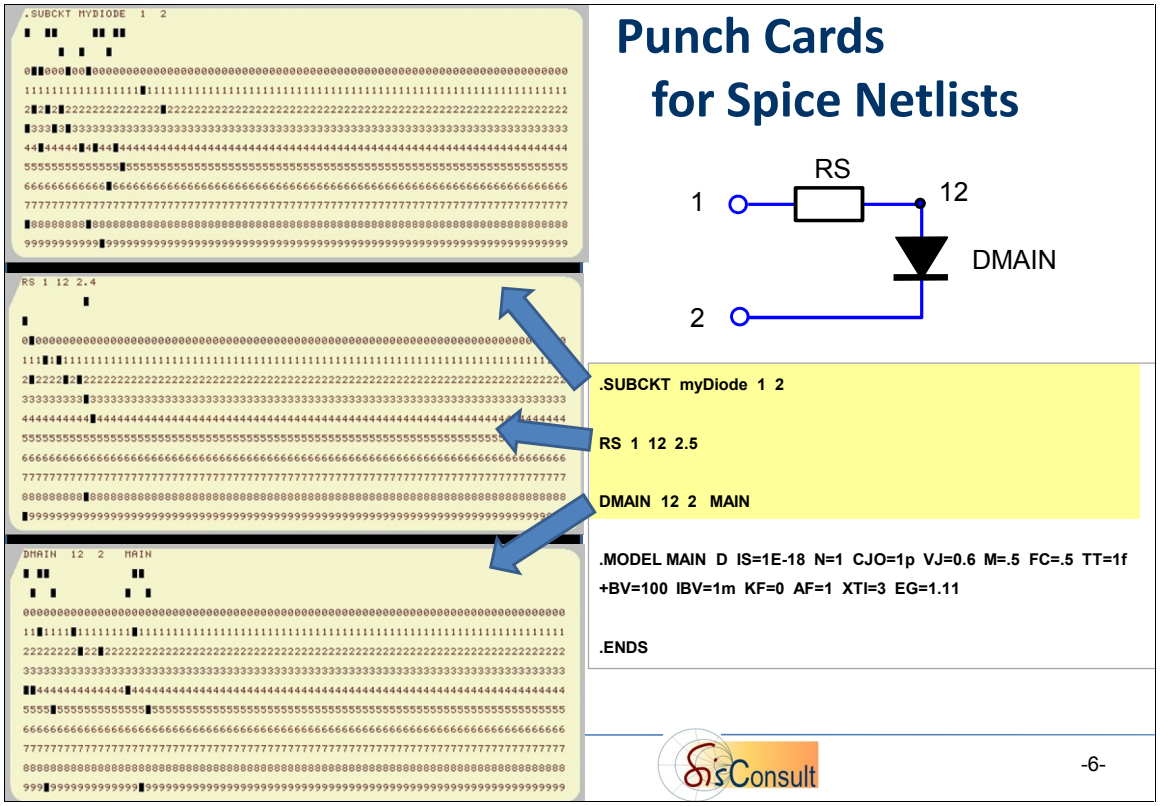
This idea of an instance line (connections) and a model line (description of the electrical performance) is a key usability factor for the design of integrated circuits, where many identical components are used at different locations in the circuit netlist.

The 1970's were the time of programming computers using punch cards.
Every line of code was represented by an individual punch card.

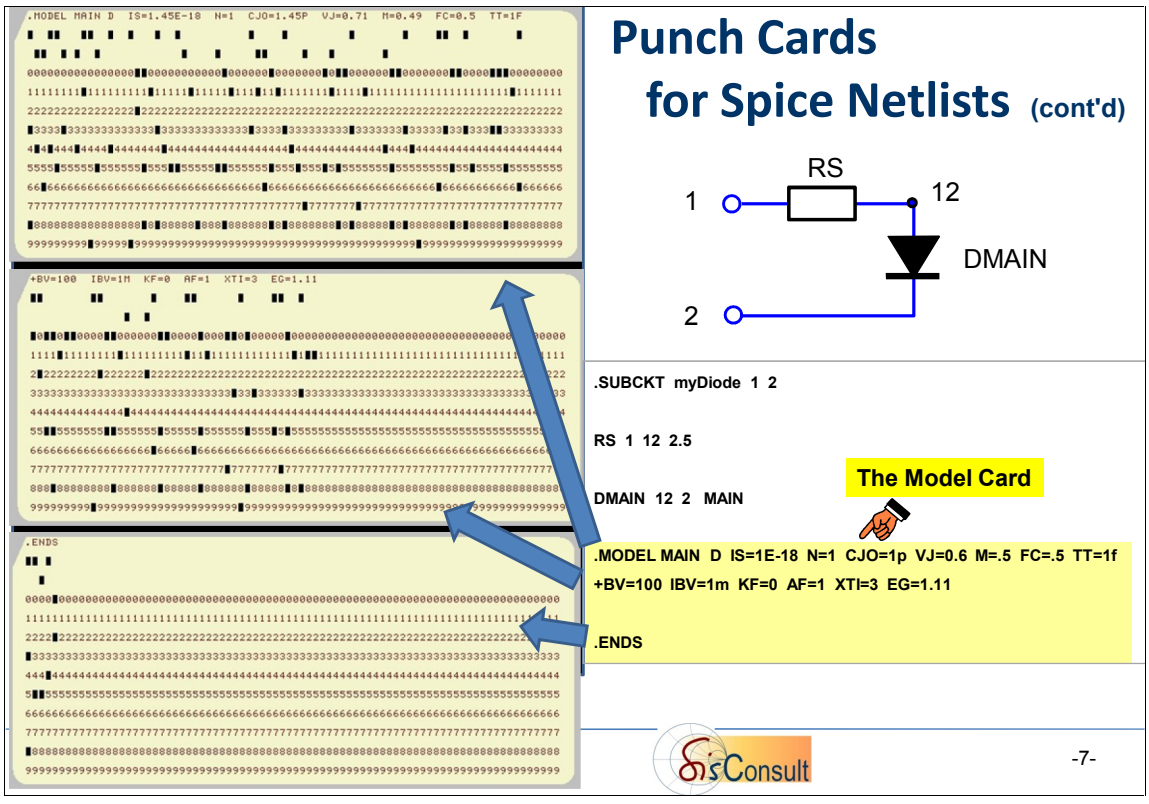
Punch Card Writer

SisConsult

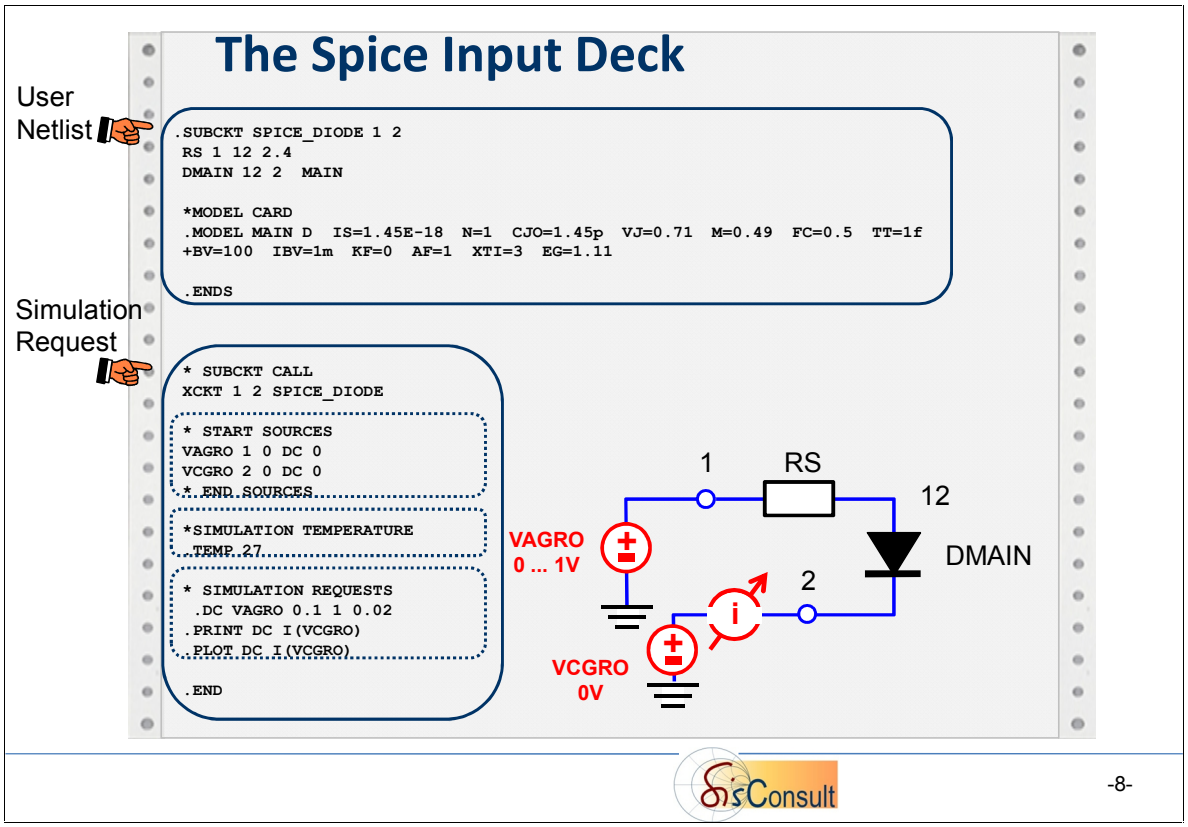
Below, the punch cards for the first 3 lines of our SPICE netlist example:



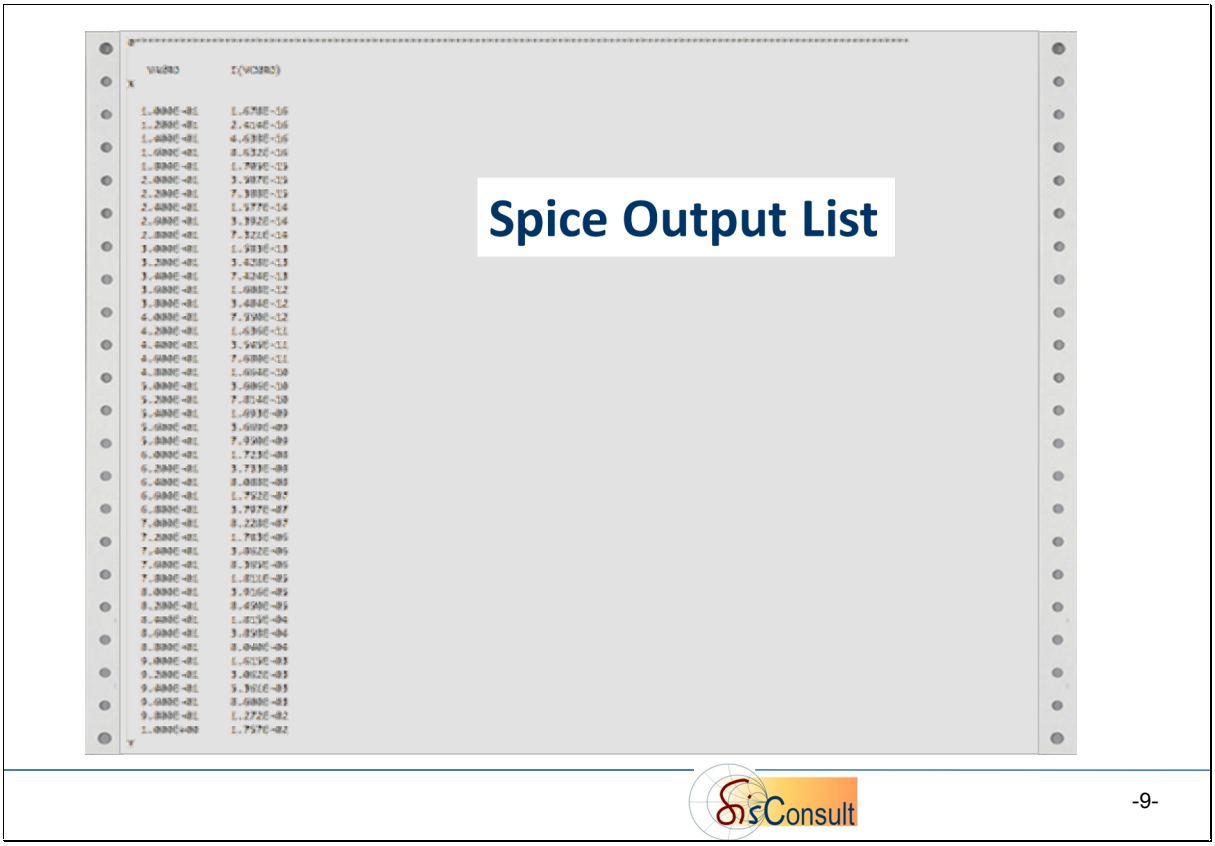
Consequently, the next line, hosting the .MODEL description was also represented by a punch card: **the so-called Model Card**.
If the .MODEL line was longer than what fits on a punch card, a 2nd card was applied, starting with the key character '+', as in our example:



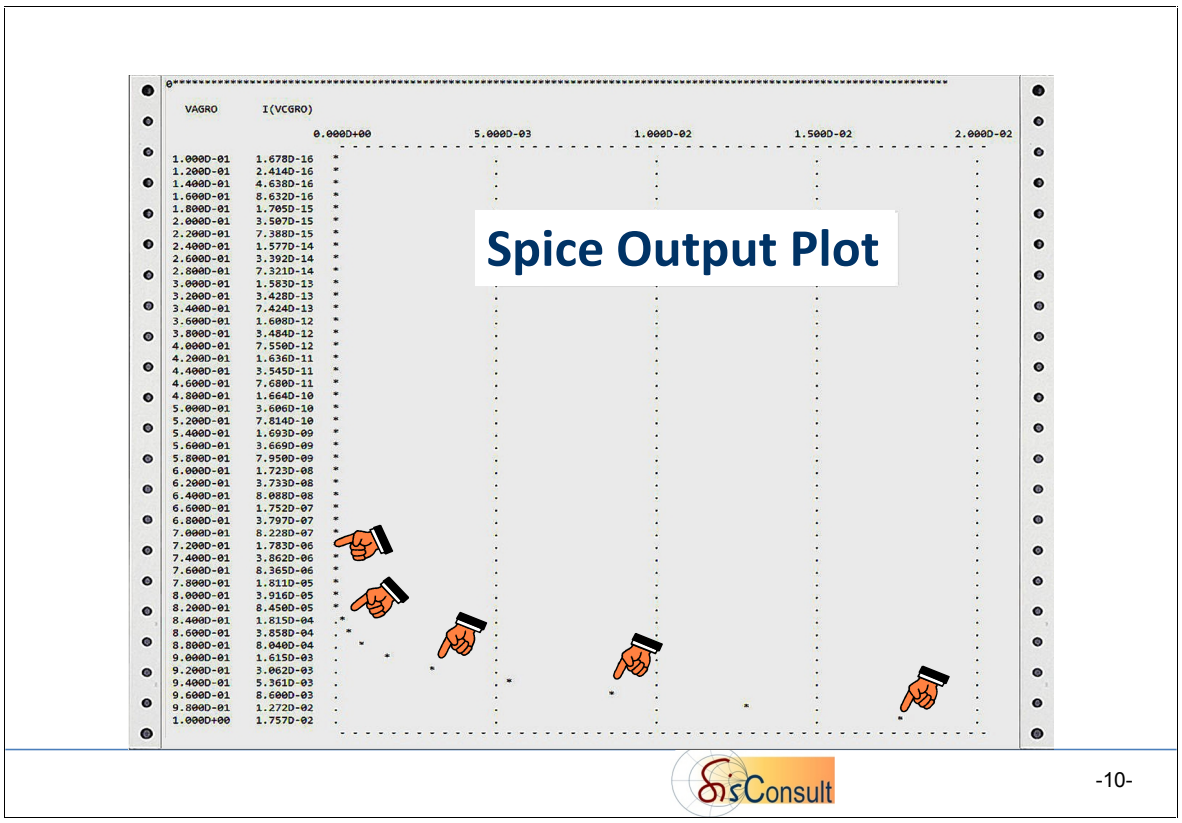
The next slide depicts the simulation request for a DC current characteristics of a diode, with the anode voltage from 0V -> 1V, and cathode grounded.



And the resulting Spice output was then printed in data columns on endless computer paper like this:



Using a so-called 'chain printer', "plotting" was using ASCII characters for x-y charts.



Back to Top

Dr.-Ing. Franz Sischka

Consulting Services
for Electronic Device
Measurements,
Data Verification
and Modeling

SisConsult
Ingenieurbüro - Engineering Office
www.SisConsult.de

eMail: franz.sischka@SisConsult.de