NAME

spicemod - model file for SPICE

DESCRIPTION

The file *spicemod* contains model information for the SPICE simulator. This file is used by *xspice* to prepare simulation input. However, the *spicemod* file can also be used by *xspectre* for the Spectre simulator.

The *spicemod* model file has a very simple format:

The lines above the '%' line contain a model index. These lines are used to specify device cell names. Only models specified in this index can possibly be included in the output. The lines can also be used to specify alternate output names and (if appropriate) correction values for device "w" and "l" attributes. The first argument on a line specify an alternate output name. The second and third arguments the "dw" and "dl" values. Note that the output name must be used in one of the specified models below the '%' line, else no model type is known or it must be a predefined model. And also no model lines can be included in the output.

The lines below the '%' line contain the models and possibly lines to specify "nbulk" and "pbulk" voltages and resistors. Only the used *model* lines after the '%' line are copied to the *xspice* output file. The comment lines and all other lines are not copied. The keywords *vnbulk* and *vpbulk* can be used to set the "nbulk" and "pbulk" voltages. The keywords *rnbulk* and *rpbulk* can be used to set the "nbulk" and "pbulk" resistor values. It are *xspice* control lines, which are used but not outputted. Thus the specified node numbers or names are not used.

Some remarks:

Do not use other separator characters than space and tab. Place ".model", model_name, model_type and the "level=N" parameter all on the same line. Use a lowercase modeltype. When the *level* parameter is used, it must be the first ".model" parameter. The *xspectre* program uses this for doing model conversion for the Spectre simulator. If needed, the SPICE scale factors are also converted. But scale factors 'a' and 'mil' are not supported.

The example below:

The lines above the '%' line contain the *xspice* or *xspectre* control information. The '#' line is a comment line. The "nenh" device must get the output name "nenhs" and every "w" must be corrected with a "dw" of "0.88" micrometer. The "l" attribute must not be corrected ("dl" is 0). Note that a *positive* value gives a *smaller* device attribute. Note that "nenh" and "ndep" are predefined device names, thus the used modeltype of the models must be "nmos".

EXAMPLE

For example, this is the *spicemod* file of the NMOS process:

```
# rcsid = "spicemod 1.2 2003/05/02 07:56:32 simon"
nenh   nenhs   0.88e-6   0
ndep   deps    1.20e-6   0
%

* parameters for depletion NMOS w/l = 4/4
.model deps nmos level=2
+          vto=-3.971    kp=31.54u    gamma=0.3642
+          phi=0.578     pb=0.84      cgso=430p
+          cgdo=430p     cgbo=130p    rsh=25.3
+          cj=62u        mj=0.73      cjsw=340p
+          mjsw =0.20    js=6.2u      tox=0.07u
+          xj=0.35u      ld=0.605u    uo=640
+          ucrit=534000  uexp=-0.025  utra=0
+          xqc=0.5       fc=0.6       lambda=0.0258
+          vmax=3.5e4    nsub=9.8e14  delta=6.3
```

```
* end deps w/l = 4/4

* parameters for n_enhacement NMOS w/l = 4/4
.model nenhs nmos level=2
+          vto=0.6838    kp=32u       gamma=0.2787
+          phi=0.55      pb=0.84      cgso=440p
+          cgdo=440p     cgbo=100p    rsh=25.3
+          cj=62u        mj=0.73      cjsw=340p
+          mjsw =0.20    js=6.2u      tox=0.07u
+          xj=0.35u      ld=0.625u    uo=650
+          ucrit=143000  uexp=0.154   utra=0
+          xqc=0.5       fc=0.6       lambda=0.02269
+          vmax=6e4      nsub=5.71e14 delta=5.5
* end nenh w/l = 4/4

vnbulk 1 0 -2.5V
rnbulk 1 0 100meg

* vdd ? 0 5V
* rgnd ? 0 1p
* rvss ? 0 1p
```

AUTHOR

S. de Graaf

SEE ALSO

SPICE2 / SPICE3 User's Guide,
xspectre(1ICD), xspice(1ICD).