NAME

putdevmod - put device model(s) into the circuit database

SYNOPSIS

putdevmod [-v] file [...]

OPTIONS

-v Verbose mode.

DESCRIPTION

Putdevmod places a device model description into the circuit database. One or more filenames may be given as argument, each containing exactly one device model description.

For new projects with xcontrol use the program *xcontrol* to set and list the cell device status. In that case, the program *putdevmod* only adds the "devmod" file and does not set the device status.

A device model description contains a description of the name of the device, a description of its terminals and (optionally) a simulation model for the device (currently only SPICE and SPECTRE models are supported). The device model description is used by other programs to detect which cells are network primitives (e.g. by the layout-to-circuit extractor *space(1ICD)* and by programs that retrieve circuit descriptions from the database) and/or to collect simulation input.

To remove a device model for a cell, use *device(1ICD)* with the option **-u**.

The syntax of an input file for *putdevmod* is as follows: On the first line the keyword "device" is specified followed by the name of the device. On a next line the keywords "begin" and "spicemod" are specified to denote the beginning of the SPICE device information. The next lines are considered to be SPICE input that can directly be appended to a SPICE network description in order to form the model description of the device. The end of the SPICE model description is denoted by a line that contains the keyword "end".

In addition, a SPECTRE model may be specified with keywords "begin" and "spectremod" and must be ended with a line that contains the keyword "end". Note that the terminals must be specified in the SPICE model.

In the header part of the input file that is used as SPICE input, the *terminal* pins, a possible *bulk* and a *prefix* for instance names for the device, may be included as SPICE comments. For MOS transistors also a *dw* and *dl* can be specified. This is done by specifying them on lines that start with the comment character "*".

To specify *terminals*, use the keyword "terminals" followed (on the same line) by the names of the terminals. The order in which the terminals are specified must agree with the order required in the SPICE circuit description (see the SPICE User's Guide). The following terminal names are used for the standard devices: "d", "g", "s" and "b" for the drain, gate, source and bulk of MOSFET transistors; "c", "b", "e" and "s" for the collector, base, emitter and (optional) bulk of BJT transistors; and "p" and "n" for capacitors, diodes and resistors.

To specify a *bulk* (if appropriate) and voltage, use the keyword "bulk" followed (optional) by a floating point number for the voltage (default 0). It specifies also that a bulk *terminal* must be added.

To specify a *prefix*, use the keyword "prefix" followed by one of more characters. The first character must be conform the SPICE syntax. If no prefix is specified the SPICE default is "x". The prefix is not only used for instance names of the device, but also for device-model *type* recognition. For MOSFETs you can specify the type using "mn" or "mp". For BJTs you can specify the type using "qn" or "qp". If missing, default the n-type is assumed. However, when the *spicemod* contains a ".model" statement, the *type* of the .model statement is used. When the *spicemod* contains a ".subckt" statement, then this statement needs first to be specified, else an incorrect type can be set.
Note that the *type* of a predefined device can also be found by specifying a *model* statement in the control file for the *xspice* netlist program.

To specify a *dw* and/or *dl*, use respectively the keyword "dw" or "dl" followed by a floating point number. This can be used to correct respectively the "w" or "l" attribute of MOS transistors. Note that a *positive* value gives a *smaller* attribute value. For example specify "1e-7" for a correction of 0.1 micrometer.

Note that also the keyword "rename" may be specified with two node name arguments. For example used, to connect the "n" node of a p-diffusion capacitor always to the "pbulk" node. Note that this capacitor must be specified with another name in the technology file.

By using *putdevmod* the user can easily perform extraction of devices that can not be recognized with the built-in device extraction algorithms of *space(1ICD).* A requirement for this is that the layout description of the device is stored in separate cell in the layout view. Using *putdevmod* a device model is then stored into the database that has a same name as the name of the layout cell and a list of terminals that is similar as, or a subset of, the list of terminals of the corresponding layout cell. The layout to circuit extractor *space* will consider these devices to be leaf-cells whose contents should not be extracted, but that should be included as primitives in the network description instead.

EXAMPLE

An example of an input file for *putdevmod* is:

```
device nenh
begin spicemod
* terminals     d g s
* bulk          0
* prefix        m
* parameters for n_enhancement NMOS  w/l = 4/4
.model nenh 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
begin spectremod
model nenh mos2 type=n \
        vto=0.6838      kp=32u          gamma=0.2787 \
        phi=0.55        pb=0.84         cgso=440p \
        ...
end
```

SEE ALSO

device(1ICD), macro(1ICD), space(1ICD), xspice(1ICD),
SPICE2 / SPICE3 User's Guide.