KING FAHD UNIVERSITY OF PETROLEUM AND MINERALS
DEPARTMENT OF ELECTRICAL ENGINEERING
Electronic Circuits II – EE303

Tutorial # 2
TRANSISTOR MODELS FOR SPICE

An electronic circuit does not have to be very complex or contain many elements before the hand calculation effort becomes unwieldy. In fact, for many of the examples worked in the lectures we were forced to use a number of device-model approximations and circuit simplifications. These circuit solutions are usually quite adequate for a first look; however, for a more detailed design and analysis approach, Computer Aided Design (CAD) and Computer Aided Engineering (CAE) tools are used.

Currently, one of the more widely used general purpose circuit simulation program for industrial and academic computer systems is SPICE. As you know SPICE can be used to simulate circuits containing resistors, capacitors, inductors, mutual inductors, independent and dependent voltage and current sources, and basic semiconductor devices. Separate SPICE models are used for the BJT, JFET, MOSFET and diode. These models are generally complex. For example, the BJT model can include the ohmic base resistance r_b and the current dependent collector resistance r_o. The MOSFET models can include the effects of charge controlled capacitances, short channel effects to the degree they are understood, and the channel length variations as a function of terminal voltages. All these information, and many others, can be included in the model statement used for describing semiconductor devices. SPICE allows varying degrees of circuit element model complexity. In this tutorial we intend to provide basic default model descriptions and more complex model descriptions. Examples will be used to illustrate the differences between the results obtained using hand calculations, default device models and complex device models. By the end of this tutorial we expect that the student will get an appreciation of the advantages of using SPICE complex models.

Diode Models

The diode element model is given in Figure 1. The element statement format is given by
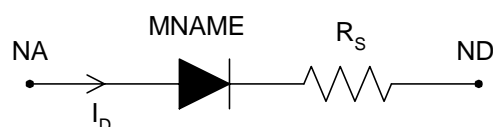
DXXX NA NC MNAME [AREA]

Figure 1

The associated model statement is

.MODEL MNAME TYPE [PNAME1=PVAL1 PNAME2=PVAL2…]

The anode of the diode is connected to NA; the cathode to NC. MNAME is an alphanumeric model designation for the device. The default value for the junction cross-sectional area is 1. Detailed model parameters are provided in Table 1.

Junction Field Effect Transistor

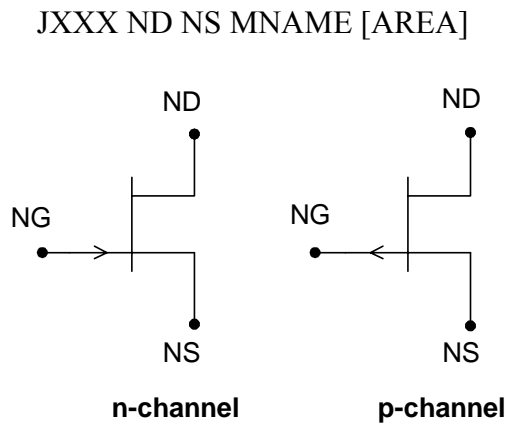The n-channel and p-channel JFET element model are given in Figure 2. The element statement format is

<p align="center">JXXX ND NS MNAME [AREA]</p>

<p align="center">Figure 2</p>

The default value for the cross-sectional AREA is 1. Detailed model parameters for JFET are provided in Table 2.

Bipolar Junction Transistor

The npn and pnp transistor element models are shown in Figure 3. The element statement format is
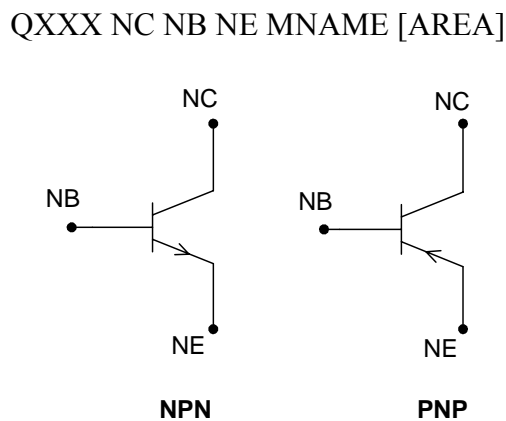
<p align="center">QXXX NC NB NE MNAME [AREA]</p>

<p align="center">Figure 3</p>

The default value of the base cross-sectional AREA is 1. Detailed model parameters are provided in Table 3.

MOS Field Effect Transistor

The n-channel and p-channel MOSFET element models are given in Figure 4. The element statement format is

MXXX ND NG NS MNAME [W=VALW][L=VALL][AD=VALD][AS=VALS]
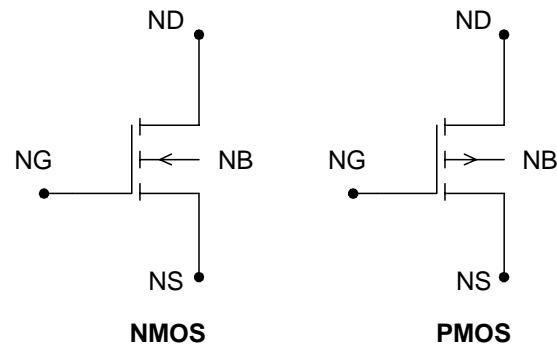

Figure 4

The default values for the gate length, VALL, and the gate width, VALW, are 1cm. Obviously these are not realistic values; however, the model uses the ratio of VALL and VALW rather than the individual values in its calculations. The drain and source areas are VALD and VALS respectively, and the default values are 10^{-6}cm^2. Detailed model parameters are provided in Table 4.

9