



IC TEST SYSTEM

Integrating the Simulation Models in PSpice/Capture

Content	Page
1 Supplying the Models	3
2 Integrating the Simulation Models in PSpice/Capture	4

1 Supplying the Models

To obtain the free PSpice/Capture simulation models for the Langer IC Test System, please contact our sales team with the following e-mail address: sales@langer-emv.de.

On our website under the section simulation you may choose from the selection of the simulation models: <https://www.langer-emv.de/de/category/simulation/76>

The following simulation models of the probes / sources are available:

- | | |
|---|-------------|
| • Probes for conducted pulse-injection | P200/300 |
| • Probes for E-field and magnetic field pulse-injection, ESD/ EFT | P1200/P1300 |
| • Probes for RF E- field and magnetic field injection | P1400/P1500 |
| • RF measurement of E- field and magnetic field emission | P1600/P1700 |

The models are used to simulate different EMC immunity and emissions issues. The sources emulate the typical trace of the interfering signals. The measuring systems emulate the typical load impedance sequence and the transfer function.

The underlying substitute circuits correspond to the real probes arising on ICs.

The models can be parametrized according to the original probes or sources.

Deviations between the curve trend of the model and the real devices may occur. No warranty claims can be derived from the deviations. Furthermore, changes in the sense of further development are reserved.

2 Integrating the Simulation Models in PSpice/Capture

The models can be used for simulations using the simulator PSpice / Capture.

You will get a .lib file and an .olb file.

.lib: library file where the network lists of the models are stored

.olb: Component library created for Orcad / Capture from the elements of the .lib file

2.1 Insert the .lib file for the simulation properties

In order to use the models, the library file must be stored in the simulation properties.

1. Open the simulation settings ("Simulation Settings")
2. In the "Configuration Files" tab, change to "Category" Library
3. Using the "Browse" function, locate the .lib file in the directory tree
4. Add "Global" to the .lib file as a global library file

The models are only usable with a restart of the software PSpice / Capture.

2.2 Insert the .olb file to the component libraries

We supply pre-defined switching symbols for all models. These are contained in the .olb file. To include the .olb file, proceed as follows:

1. Open the Part Menu using the Place / Part item in the menu bar
2. In the part menu, select "Add Library"
3. Locate the .olb files in the directory tree and add them to the build list
4. Now the elements from the list can be selected and inserted in the circuit diagram

Please note: The models have several connections

M is for Reference / GROUND. This should always correspond to the reference point of the simulated circuit.

The number and use of the connections depends on the type of probe / source.

The function of the probe, the meaning of the connections and the integration into the simulation can be found in the corresponding user manuals. The simplified replacement circuit and the functional principle of the probe are shown in the user manual.

The connection of the LTspice models with the IC replacement circuits is shown in the "Instructions for use".