LANGER
EMV-Technik

IC TEST SYSTEM

Integrating the Simulation Models in LTspice

Content   Page

1   Supplying the Models

To obtain the free LTspice 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 LTspice

The models can be used to perform simulations using the freeware tool LTspice.
In order to use the models, the model files must be copied into different subfolders of the LTspice installation.

        1. Copy the symbol files Pxxx.asy into the subfolder ... \ LTspiceIV \ lib \ sym into your LTspice installation
Note: For a better overview, you can also create a new folder. e.g. EMC; Langer EMV Probe

        2. Copy the Pxxx.cir files to the subfolder ... \ LTspiceIV \ lib \ sub
The models are only usable with a restart of the software LTspice.

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".