



CST PCB STUDIO TECHNICAL SPECIFICATIONS

Frontend Module

- ***For functionality and CAD/EDA import filter, see technical specifications of the CST STUDIO SUITE(tm)***

CST PCB Studio is embedded into the CST Studio Suite, which is referred to in the *CST Studio Suite Getting Started* manual. The following explanations assume that you have already installed the software and familiarized yourself with the basic concepts of the user interface.

What is CST PCB Studio?

CST PCB Studio is an electromagnetic simulation tool specially designed for the fast and accurate simulation of real-world PCBs and can be used for pre-layout and post layout analysis. It allows the simulation of effects like resonances, reflections or crosstalk on any kind of PCBs from single-layer up to multilayered high-speed PCBs.

CST PCB Studio can be easily integrated into any existing design flow by importing PCB designs directly from many popular EDA layout tools and provides a powerful tool for the automated layout checking and correction of geometric errors.

CST PCB Studio has an intuitive user interface that makes it easy to define a design from scratch for pre-layout analyses. There are advanced functions to navigate through a design and to select, hide or visualize any objects like traces or areas.

CST PCB Studio incorporates three different solver techniques to account for all kinds of PCBs. Single- or two layer PCBs are normally designed without any special ground reference layers and therefore dedicated to the lower- or medium frequency range. The method best suited to this kind of application is the quasi-static Partial Element Equivalent Circuit method (PEEC). The program generates equivalent circuits out of any selected combination of conductors. Skin effect and dielectric loss are modeled in both the frequency and the time domain.

CST PCB Studio uses CST Design Studio to define passive and active devices on the modeled PCB layout with the help of an easy-to-use schematic editor. The powerful built-in network simulator inside CST Design Studio enables the simulation of the whole system consisting of the equivalent circuit of the PCB and its termination in both frequency and time domain. Broadband equivalent circuits can be exported in several SPICE formats for a further usage in other commercial network simulators.

CST PCB Studio offers a modeling technique dedicated to the analysis of power distribution networks (PDN) in multi-layer PCBs. Given a set of PDN nets to be characterized, the full-wave three-dimensional Finite Element Frequency Domain solver, hereafter referred to as 3D (FE FD), is able to compute PDN impedances directly. The results can be used to check whether the design margins imposed by the IC component are met. Using the CST PCB Studio component library, this modeling option enables the assessment of different decoupling capacitor strategies, taking into account the full-wave electromagnetic effects in the PDN.

Applications

- SI analysis on single/multilayer PCBs and packages
- PI analysis on single/ multilayer PCBs and packages
- IR-Drop analysis on single/ multilayer PCBs and packages

CST PCB Studio Key Features

An overview of the main features of CST PCB Studio is provided in the following list. Please note that not all options may be available to you due to license restrictions. Please contact your local sales office for details.

For the circuit simulator only some selected key features are listed below. A full list can be found in the *CST Design Studio Workflow* manual.

General

- Native graphical user interface based on Windows operating systems.
- Tight interface to CST Design Studio.
- PEEC method specializing in the simulation of single- and two-layer boards.
- Transmission line modeling method for SI analysis of high-speed multi-layer PCBs and packages.
- Specialized FEM method for PI analysis of high-speed multi-layer PCBs and packages.
- IR-Drop analysis to simulate DC power/ground behavior of a PCB and package.

PCB Structure Modeling

- Import of PCB designs from Cadence Allegro/ADP/SiP.
- Import of PCB designs from Mentor Graphics Expedition.
- Import of PCB designs from Mentor Graphics Hyperlynx.
- Import of PCB designs from Mentor Graphics PADS.
- Import of PCB designs from Zuken C-R5000/8000 ASCII.
- Import of PCB designs from ODB++.
- PCB layout checker with automatic correction.
- Interactive PCB editing tools.
- Advanced navigation through the PCB.
- Hiding/visualizing selections.

PCB Electric Modeling

- Automatic meshing and extraction of 3D PEEC models.
- Automatic meshing and extraction of 2D transmission line models.
- Automatic meshing and extraction of 3D (FE FD) models and PDN impedances.
- Consideration of skin-effect and dielectric loss in time and frequency domain.
- Export of equivalent SPICE circuits.
- Export of current distribution and near fields for radiation analysis.
- Advanced export of PCB sub structures to CST Microwave Suite.

Circuit Simulator

- Schematic editor enables the easy definition of passive and active devices.
- Fast circuit simulation in time and frequency domain.
- Support of IBIS models.
- Import and Export of S-Parameter data via TOUCHSTONE file format.
- Parameterization of termination circuitry and parameter sweep.



About This Manual

This manual is primarily designed to enable a quick start to the modeling capabilities of CST PCB Studio. It is not intended as a reference guide of all available features, but rather as an overview of the key concepts. Understanding these concepts will allow you to learn working with the software efficiently with additional help from the online documentation.

To learn more about the circuit simulator please refer to the *CST Design Studio Workflow* manual.

The next chapter *Overview* is dedicated to explaining the underlying concepts of CST PCB Studio and to showing the most important objects and related dialog boxes. The chapter *Examples* will guide you through the three important analysis types. We strongly recommend studying both chapters carefully.

Document Conventions

- Buttons that should be pressed within dialog boxes are always written in italics, e.g. *OK*.
- Key combinations are always joined with a plus (+) sign. *Ctrl+S* means that you should hold down the *Ctrl* key while pressing the *S* key.
- The program's features can be accessed through a Ribbon command bar at the top of the main window. The commands are organized in a series of tabs within the Ribbon. In this document a command is printed as follows: *Tab name: Group name ⇒ Button name ⇒ Command name*. This means that you should activate the proper tab first and then press the button *Command name*, which belongs to the group *Group name*. If a keyboard shortcut exists, it is shown in brackets after the command. Example: *View: Change View ⇒ Reset View (Space)*
- The project data is accessible through the navigation tree on the left side of the application's main window. An item of the navigation tree is referenced in the following way: *NT: Tree folder ⇒ Sub folder ⇒ Tree item*.
- Example: *NT: Components ⇒ IC100 ⇒ IC100-A1*