

Lesson 8: Component Placement

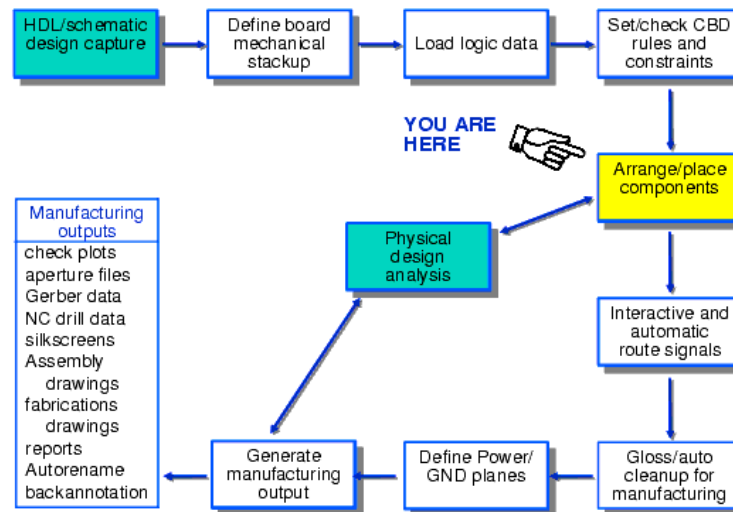
Learning Objectives

In this lesson you will:

- ◆ Use floorplanning to organize the placement of components with the same ROOM property.
- ◆ Assign reference designators to preplaced parts.
- ◆ Interactively place components using various commands.

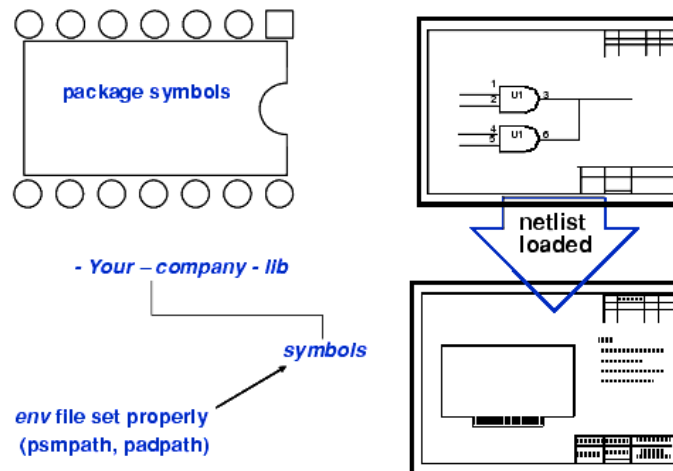
In this section you will place components on your board. You will learn how to create Rooms and assign components to rooms, how to assign reference designators to preplaced symbols, and how to quickly place components. You will also learn the interactive commands available when working with placement.

Design Layout Process



This design flow is used throughout the entire course. Each box in this flow represents a common step in the design of a printed circuit board. As indicated in the flow, the Arrange/place components box will now be discussed.

Prerequisites



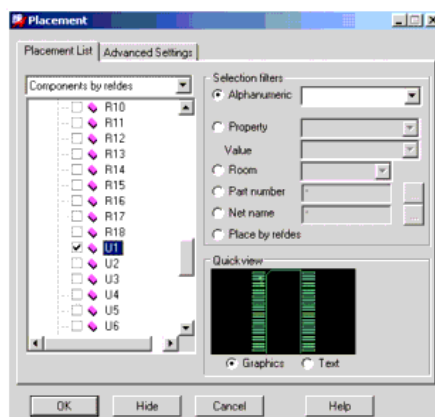
It is important to remember how PCB Editor determines where the footprints and padstacks are located on disk. The variables PSMPATH and PADPATH are used to determine the locations on disk of the footprints and padstacks, respectively. These variables are defined in the *env* file and can also be set and modified using the User Preferences Editor.

The prerequisites for manual placement are:

- **Symbols:** The package symbols and padstacks required for parts in the netlist must exist. Point to the location of the package symbols in the library search path. You can define these paths in the *env* file.
- **Netlist:** You must load a schematic database into a PCB Editor design file (*.brd*). See the lesson titled *Importing Logic Information into OrCAD PCB Editor* for more details.
- **Alternate Package Symbols:** If you plan to select alternative package symbols during manual placement, the alternate symbol definitions must be contained in the appropriate part definition files.
- **Floorplanning:** You can create a "block diagram" of the logical functions that need to be arranged on the board by using Rooms. Specify this part property within the DE HDL or DE CIS schematics, or you can add it to a third-party netlist before the database is read in.
- **Package Keepouts:** If your master design file did not contain package keepouts, add them before you begin placing components, by selecting **Setup - Areas - Package Keepout**.

Interactive Placement

Place - Manually

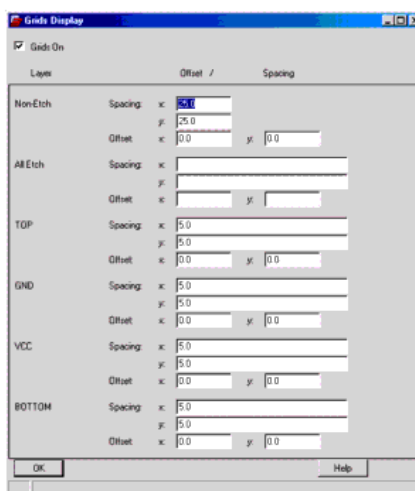


Manual placement can be used to place parts by reference designator, place all parts, place IC components, place IO components, and place discrettes, as well as other options. Components are defined as IC, IO or discrete, using the CLASS property as defined in the device file for third-party netlists, or in the chips.prt file for DE HDL. For DE CIS, you will need to manually add the CLASS property to the parts in your library.

You also use the **Place - Manually** command to place package symbols (spare footprints), mechanical symbols (board outline or board mechanical) and format symbols (company formats).

Placement Grid

Setup - Grids



The placement grid is a Non-Etch grid (it is not the grid used for routing connections). The origin of the package symbol (defined during symbol creation) locks on to the Non-Etch grid.

Select **Setup - Grids** to set the spacing for manual placement on the Non-Etch grid. The origin of the placement grid is the origin of the PCB Editor design file (x 0, y 0). Use the Grid form to toggle the grid visibility ON or OFF.

You can use the **Grid Toggle** icon to turn the grid display ON or OFF.

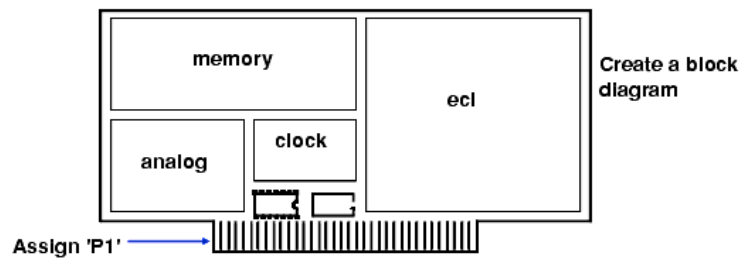


It is not unusual to attempt to keep all through-hole components on a 100- or 50-mil placement grid (this facilitates bareboard and in-circuit testing). For dense or surface-mount designs, a 25-mil placement grid (or less) is not uncommon.

Also consider the route grid for completing pin-to-pin connections. Keeping the placement grid compatible with the route grid will reduce the number of "off-grid" pins.

Strategy

- ◆ Create rooms for floorplanning.
- ◆ Assign reference designators to "preplaced" devices.
- ◆ Place I/O bound devices.
- ◆ Place critical logic functions.
- ◆ Evaluate and revise placement.
- ◆ Place bulk decoupling and bypass caps.
- ◆ Use reports to aid placement process.



1. Floorplanning: You can create a "block diagram" of the logical functions through the use of Rooms.
2. Assign fixed IO devices: Use the **Assign** command to correlate any connector package symbols (mechanically placed within the master or template file) to reference designators in the database (such as P1, J2). This process also applies to any mechanically constrained devices preplaced in the master design file (such as LEDs).
3. Place IO bound devices: Place any parts that send or receive nets from backplane connectors to minimize overall net length.
4. Place critical logic functions: Place clock circuits, memory arrays, buffers, controllers, and address buses. (See Floorplanning on the next page.)

circuits: P