



P-CAD to Altium Designer Terminology Guide

Summary

Guide
GU0121 (v1.1) October 31, 2006

This is a high-level roadmap to guide you in understanding how P-CAD terminology translates and can be found in Altium Designer. High-level points of interest or difference are noted where appropriate. Resources for additional reading in the Altium Designer Knowledge Center to further your understanding of this new design environment are also given.

The **Knowledge Center** panel (**Help » Knowledge Center** or **F1**) will assist you as you learn about designing your electronic product in Altium Designer. Reference topics automatically load in response to your actions in the environment. In the lower region of the panel you will find links to PDFs that will help you perform design tasks, or understand the concepts behind Altium Designer. You can also bring up a list of shortcuts at any time through the **Help » Shortcuts** panel near the lower right of the workspace.

P-CAD Terms

The following is a list of the more commonly-used P-CAD terms and their closest respective Altium Designer equivalents.

P-CAD Term	Altium Designer Term	Comments
Ascend/Descend	Up/Down Hierarchy	Used for moving the focus to the next level up or down in the design hierarchy, from the current document. Found in the Tools menu.
Board Outline	Board Shape	The Board Shape defines the physical edge of the board, and is used to calculate the automatic power plane pullback. By default the board is shown in black, bounded by the Board Shape. Commands to modify the board shape are in the Design » Board Shape submenu. During import of a P-CAD PCB the P-CAD board outline is mapped to an Altium Designer layer, which is then used to define the Board Shape. See Also: TU0110 Preparing the Board for Design Transfer

P-CAD to Altium Designer Terminology Reference

Polygon	Solid Region	Solid multi-sided polygonal object. Does not pour around obstacles.
Connections	Connections	Connections are not graphical objects that can be assigned through a pad, but instead display once a pad is assigned to a net. Pads can be edited through their Properties dialog to be assigned to a net provided that the net already exists. Nets are created through Design » Netlist » Edit Nets .
Copper Pour	Polygon Pour	Intelligent object that pours around existing copper objects, such as routing and component pads. Can be poured solid (uses solid region objects), or hatched (uses tracks and arcs). See Also: AP0101 Polygon Pours and Copper Regions
Copy Matrix ; Paste From File / Circuit	Snippets; Rubber Stamp	Sections of circuitry can be stored and re-used, using the Snippets system. After selecting the circuitry right click and choose from the Snippets commands. Snippets are placed and managed from the Snippets panel.
Design Manager	PCB Panel	Used to browse, locate and edit PCB: Components, Nets, Rules, From-Tos, Split Planes, Differential Pairs, Polygons.
Design Rules	Design Rules	Rules in Altium Designer are defined separately from the objects (Design » Rules). The objects that they apply to is determined by the Scope of the rule, which is expressed as a Query, such as InNet(GND). The Rule priority, in combination with the rule scope, determines which rule applies to each object. Right Click on any object and choose an Applicable Rule command to see which rules are being applied to it. Note that some rules specify design requirements or preferences, such as Solder Mask expansion or Routing Corner Style, these type of rules are not checked by the DRC command. See Also: AR0111 Specifying the PCB Design Rules and Resolving Violations and TR0116 Design Rules Reference .
ECO/Netlist	Design Transfer	Once a project is compiled, the equivalent of a netlist load of the design can be made using the Design » Update command. This will generate ECOs that transfer the schematic information to the target PCB. Before transferring the design from schematic to PCB, create and add the

		<p>PCB file to the .PrjPCB project. Required PCB footprints libraries must be available, install them via the Libraries panel.</p> <p>Netlists can be exported from Design » Netlist » Export Netlist From PCB</p>
ECO	Design Synchronization	<p>Keeping the schematic synchronized with the PCB is performed as a direct process in Altium Designer, there is no need to generate a netlist or ECO file. Design changes, such as annotation and component property updates are made in both directions using the Design » Update command.</p> <p>See Also: AR0106 Finding Differences and Synchronizing Designs</p>
DRC	DRC	<p>PCB Design Rule Checking can be performed on-line, or as a batch process. Found in the Tools menu.</p> <p>See Also: AR0111 Specifying the PCB Design Rules and Resolving Violations.</p>
Drill Table	.LEGEND	<p>Place » String command places a .LEGEND string that is converted into a symbol legend for mechanical drill plots at output time (Gerber or ODB++).</p>
Fixed	Locked	<p>Each PCB object can be locked by right-clicking on it and selecting appropriate Locked command from the context menu.</p>
Edit; Rename Net	Double-click, right-click Properties	<p>Double-click on any object to edit its properties in the relevant properties dialog. Alternatively, right click on the object and select Properties from the context menu. Nets are edited via the PCB panel, or the Design » Netlist submenu.</p>
ERC	Error Reporting	<p>Electrical Rules Checking is performed during compilation of a project according to the error check settings in Projects » Project Options » Error Reporting. Any violations detected will display in the Messages panel, double-click on an entry to trace an error.</p> <p>See Also: AR0136 Verifying Your Design in Altium Designer</p>
Export	Save As	<p>As well as different file types, the Save As and Save Copy As command are used to export to other formats. Note that Save As will modify the project file to reference the new name given in the Save As dialog. To not affect the project use the Save Copy As command.</p>

P-CAD to Altium Designer Terminology Reference

Field	String	
		Used to display a special string in the workspace (ahead of printout/output stage, consider using the Convert Special Strings option in Tools » Preferences » PCB Editor . Many display options can also be set from here.
File	Design document (sch, PCB, etc), Project, Design Workspace.	As well as individual schematic (*.SchDoc), PCB (*.PcbDoc) and other types of design files, Altium Designer also has the Project file (*.PrjPcb), which links together the entire set of files used create the PCB, and the Design Workspace (*.DsnWrk) which is a user-defined set of related Projects. See Also: AP0129 Project Essentials
Find Errors	Browse Violations	PCB Rules can be browsed in the PCB panel, set to Rules mode. Violations are listed for each rule type, or select [All Rules] to list all violations. See Also: TR0104 Altium Designer Panels Reference
Force Update	Update from Libraries (sch)	Used to update components in the schematic design. Full control over the level of updating, from full component refresh, to selective symbol/model/parameter updates. See Also: AP0144 Keeping Components Up-To-Date
Force Update	Update from PCB Libraries	Used to update all footprints on the board with information from corresponding footprints in PCB Footprint libraries (*.PcbLib). The update involves a detailed physical comparison library-to-board footprint primitives, and can be applied to all footprints on the board. See Also: AP0144 Keeping Components Up-To-Date
Hierarchy (Simple)	Hierarchical Design	Altium Designer supports true hierarchal design, of any complexity. Sub-sheets are represented by sheet symbols (Modules in P-CAD), connectivity is established between sheet entries to ports on the sheet below. See Also: AR0123 Connectivity and Multi-Sheet Design
Hierarchy (Complex)	Multi-channel	Multi-channel design is the ability to reference a child sheet multiple times, either by referencing the same sub-sheet from multiple sheet symbols, or

		<p>by placing a single sheet-symbol and using the Repeat statement to generate an array of sub-sheets.</p> <p>Multi-channel design, like complex hierarchy, is built on the architecture of multiple instances. The fundamental difference is that there is no need to 'flatten' the design, Altium Designer's expands the design during compilation, mapping the single logical symbol to the many physical PCB components.</p> <p>See Also: AR0112 Multi-Channel Design Concepts</p>
Options Preferences	Preferences	<p>In Altium Designer the term preferences is used to describe environment settings (as distinct from file settings), such as selection colors, mouse wheel behavior, and so on. Preference settings for all editor can be accessed via the DXP » Preferences menu, or Tools » Preferences.</p> <p>Preferences are Altium Design environment settings that are stored as part of the environment and used for all applicable documents being edited in that environment – settings persist from one design to the next. Example settings include menu customizations, the position of panels, mouse click behavior, and how project folders present when a project is opened.</p> <p>Found in both the DXP and the Tools menu.</p>
Options (Display, Grid, Board)	Options (Document / Project)	<p>Options for documents are settings that are specific to and are stored with that document (file-specific settings). Changing a document option affects the current document only. Examples include settings for board layers and colors, grid settings and measurement units. Document options are found in the Design menu.</p> <p>Options for projects are settings that are specific to and are stored in only the project file. This includes error checking settings, design synchronization options, project parameters and print settings. Project options are found in the Project menu.</p>
Padstack / Pad Styles	Pad	<p>Layer stackup (padstack) can be defined and edited only on a placed pad, padstacks cannot be predefined or stored. Padstyle definitions are limited to simple shapes – complex pad shapes can be created by placing other primitives in the footprint library that extend the pad shape as required.</p>

Pattern	Footprint	They are the same – both define where the component sits on the board.
Pattern Graphic	Multiple Footprints	Multiple footprints can be attached to the schematic component, switching from one to another can only be done on the schematic.
Power Ports	Power Ports	Global net identifier that automatically connects all nets of the same name throughout the design. Bus Power Ports are available, these are used in FPGA designs.
Reports (BOM)	Reports » Bill of Materials or Simple BOM	Options exist for a simple ASCII BOM, or a customizable BOM report, supporting various output formats, including Microsoft Excel using Excel templates. BOMs can also be generated from an Output Job file, which is added to the project like any other project document. See Also: TU0104 Generating a Custom Bill of Materials and TR0127 OutputJob Editor Reference
Reports (Fabrication/Assembly only)	File » Fabrication Outputs and Assembly Outputs	CAM outputs can be generated via the File » Assembly Outputs and File » Fabrication Outputs . CAM outputs can also be generated from an Output Job file, which is added to the project like any other project document. Details on each output type are available in the Knowledge Center panel, Documentation Library > Generating Manufacturing Outputs > Output Generation .
Stack-up diagram	Place Stackup Legend	Found in the Design » Layer Stack Manager , command places a legend on the board. Right-clicking while in this dialog also copies the image to clipboard.
Sub-object	Primitive object	The set of basic objects used to create group objects (such as components), or used in building up the design.
Templates for designs	Templates	The templates used when a File » New command is selected are defined in the Tools » Preferences System » New Document Defaults . Alternatively, use any template by clicking on the New from Template options in the Files panel. A new Schematic template can be applied to an existing schematic using

		<p>the Design » Template submenu.</p> <p>PCB templates can only be used when creating a new PCB, they cannot be applied to an existing one.</p>
Tie Net	Net Tie Component Type	<p>Different nets can be connected to each other via a special Net Tie component type (a setting in the Component properties dialog). A net tie footprint includes shorting-routing between the component pads, creating the connection between the nets. When placing components of this type, use the Verify Shorting Copper option in DRC to verify that the short exists. A net tie component is defined on both the schematic and PCB, and be excluded or included in the BOM.</p>
Trace	Track	<p>They are the same – both define where the nets are physically laid or routed on the board.</p>
Trace Clean-up	Clean All Nets	<p>Used to clean individually routed nets for undesired duplicate (stacked) track segments. A selection of tools for reworking tracks can be found in Edit » Move. Found in the Design » Netlist.</p>
Undo	Undo	<p>Undo last action. by default will undo groups of the last actions performed. Set the size of the undo stack in Tools » Preferences » PCB/SCH Editor » General. Can also control if undo actions apply to all in group (eg, all wires just placed), or each (the last wire only) using the Group Undo option.</p>
Variants	Assembly Variants	<p>Assembly Variants allow alternate population options of the same board. Support is included for variant BOMs and pick and place files. Variants are defined in the Projects menu.</p> <p>See Also: AP0128 Managing Design Variation with Variants</p>

Revision History

Date	Version No.	Revision
20-Jun-2006	1.0	New release
31-Oct-2006	1.1	Updates made for additional terminology.

Software, hardware, documentation and related materials:

Copyright © 2006 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, Board Insight, CAMtastic, CircuitStudio, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, Nexar, nVisage, P-CAD, Protel, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.