p-cad® 2002

PROFESSIONAL TOOLS FOR BOARD LAYOUT SPECIALISTS ™

Parametric Constraint Solver

p-cad®

PCB layout system from Altium™

Copyrights

Altium Limited
www.altium.com

Table of Contents

chapter 8 Using PCS with InterPlace

chapter 9 File Commands

chapter 10 View Commands

chapter 11 Help Commands

Appendix A Keyboard Reference

Appendix B PCS References

Index

CHAPTER 1

Parametric Constraint Solver Introduction

Congratulations on your purchase of the P-CAD Parametric Constraint Solver. P-CAD's Parametric Constraint Solver (PCS) is a productivity-enhancing tool that enables true rules based design.

With PCS, designers can comprehensively view and manipulate Schematic and PCB design rules in rule-precedence order. Most importantly, design rules can be defined as constant values or mathematical expressions, and may also be defined in terms of other design rules, providing a parametric, precedence-based, rules-driven system.

The Parametric Constraint Solver provides a vehicle in which a design's constraint values can be defined in a variety of ways: as a constant, formula, vector, or as the result of a mathematical expression which can be based on one or more other design constraints.

The updates made to a design in PCS are saved to the PCB or Schematic design file. PCB routing, copper pour, online DRC, and placement tools honor PCS width and clearance restrictions. In addition, DRC validates the completed design against the constraints defined in PCS, including widths, clearance, and room constraints such as room inclusion and height restrictions.

With PCS, a circuit board designer has easy access to the PCB and Schematic design rules. You can view the hierarchical order of the design constraints from an individual component rule up to the global design rules and quickly determine the order of precedence in which these rules will be applied to and affect your design. Constraints are easily added to and removed from any level of the rules hierarchy and can also be added to and removed from multiple items with just one operation.

P-CAD Parametric Constraint Solver is an advanced constraint management tool for the Microsoft® Windows 95™, Windows 98™, Windows 2000™, or Windows NT™ operating systems. The Parametric Constraint Solver contains a combination of features designed to significantly reduce the number of design iterations needed to produce a successful board layout.

P-CAD PCS Features

This section highlights some of the important P-CAD Parametric Constraint Solver features:

- Manipulation of constraints. PCS provides easy access to the design rules in the Constraint Editor. All of the design's rules are displayed in their order of precedence from highest to

lowest: component, room, class-to-class, net, net class, layer and design. From this hierarchy you can easily add constraints to and remove constraints from any of the levels.

- Constraints as expressions. PCS provides the ability to use expressions to define your design rules. An expression can use constants, formulas or vectors and may include references to other constraints.
- Constraints based on other constraints. A constraint expression can be based on and/or reference other constraints from any precedence level in the design. When the value of one constraint is modified, the result is applied to any expressions that referenced it, and their values are changed as well.
- Math and Trig Functions. A comprehensive suite of mathematical and trigonometric functions are supported by the PCS Constraint Editor.
- Constraint validation. Constraints are evaluated at the time they are defined and whenever a design is loaded into PCS. Synchronizing the updates between the applications helps eliminate errors that may occur due to changes made in PCB or Schematic.
- Unit selection. You can choose a constraint's unit. For instance, a clearance constraint can be a length unit such as an inch, mil, millimeter or centimeter. User-defined constraints provide the opportunity to assign any of the available units.
- Assignment to multiple design objects at one time. With PCS it's easy to assign a constraint to many items at once, instead of many times to individual items.
- Constraint data saved in originating design. Updates made to the design in PCS are saved to the originating design file, eliminating any need to repeat the process in multiple applications.
- Organizing the design objects. You can use the Design Manager in PCS to organize the various objects in the design so that assigning constraints becomes quick and easy. The contents of the design data (i.e., components and nets) can be defined in terms of partitions, rooms, net classes and class-to-class rules.
- Application integration. P-CAD PCB, P-CAD Relay and P-CAD Schematic software is tightly integrated with P-CAD Parametric Constraint Solver assuring the synchronization of design file updates between the applications.

About this Guide

This manual provides information about the P-CAD Parametric Constraint Solver. It includes the following sections:

- *Getting Started*: Chapters 2 and 3 tell what you need to get started using the P-CAD Parametric Constraint Solver. It provides installation instructions and walks you through the basic capabilities of P-CAD PCS and its interaction with P-CAD PCB or Relay.
- *Using the Parametric Constraint Solver*: Chapters 4 through 7 provide information that you need to work with P-CAD PCS. They give you details on applying design constraints, assigning units and finding the best way to capture a constraint's value in the constraint editor.

- *Using InterPlace and PCS*: Chapter 8 provides information on the additional placement functionality available when both P-CAD Parametric Constraint Solver and P-CAD InterPlace are used together.
- *The Parametric Constraint Solver Reference*: Chapters 9 through 11 and Appendices A and B include an extensive reference on the PCS commands and covers all of the variables the Parametric Constraint Solver allows when assigning values to the constraints.

PCS within the P-CAD Product Suite

The interface of the Parametric Constraint Solver varies depending on the design file you have loaded. For instance, designs loaded from P-CAD Schematic may not use the Room precedence level in which to assign constraints since only PCB designs employ the Room as an organizational tool.

P-CAD PCB Icon

When a design is loaded from PCB, the full suite of Parametric Constraint Solver features is available. When a feature is unique only to a PCB design the PCB icon is displayed in the margin.

CHAPTER 2

Installation and Setup

This chapter lists the required hardware and software settings you need to install the P-CAD Suite.

System Requirements

Make sure that your PC and its software conform to the following P-CAD requirements and recommendations.

Recommended System

- Windows NT 4/2000 Professional
- PC with Pentium III Processor
- 128MB RAM (256MB for high component/net count)
- 400MB Hard Disk Space
- Desktop area 1024x768 pixels
- 32-bit Color Palette
- CD-ROM Drive
- Mouse or compatible pointing device.

Minimum System

- Windows 95/98/2000Me
- PC with Pentium 166MHz
- 64MB RAM
- 200MB Hard Disk Space (without ISO libraries)
- Desktop area 800x600 pixels
- 256 Color Palette
- CD-ROM Drive
- Mouse.

Installing P-CAD Products

For up-to-date installation information refer to the file `Readme.WRI`, located on the product CD. This file can also be found in the application program folder (`\Program Files\P-CAD 2002`) after installation. Note that the setup program on the Product CD can also be used to Repair or Remove an existing P-CAD Installation.

CHAPTER 3

Parametric Constraint Solver Basics

This chapter introduces many of the basic features you need to know when using the P-CAD Parametric Constraint Solver. It includes general information on such topics as:

- Using PCS from PCB, Relay, or Schematic
- PCS user interface.
- Loading, saving, and closing a design.
- Right mouse commands.

What is presented here is a high level look at Parametric Constraint Solver capabilities. These capabilities are detailed in later chapters.

To keep with the easy-to-use tradition of P-CAD applications, much of the basic functionality of P-CAD Parametric Constraint Solver is similar to P-CAD PCB. As a Windows user, you will also be familiar with the basic manipulations employed in managing constraints in PCS.

Using the Parametric Constraint Solver

The Parametric Constraint Solver is a versatile application that can be used to manage constraints on both a Schematic and a PCB design. Invoked from one of the P-CAD applications, a design's data is transported from its originating application into PCS and, when updates are complete, passed back to PCB or Schematic.

PCS provides a vehicle for adding constraints to your design at various enforcement levels using diverse organizational tools.

The ability to directly assign a constraint's value or define that value using expressions is an exciting feature of PCS. Employing its close integration with P-CAD PCB and P-CAD Schematic allows PCS to validate any changes to the design's constraint values whenever a design is loaded into PCS.

The role of the Parametric Constraint Solver in the P-CAD product suite is summarized in the sections below.

PCB and Relay Designs in PCS

The Parametric Constraint Solver can be activated from P-CAD PCB or P-CAD Relay to organize the components and nets in a design into physical and logical groups in preparation for applying design constraints. PCS is especially accommodating in providing easy constraint viewing and management in the Constraint Editor. Once design objects are properly grouped, rules can be added to or removed from any of the precedence levels of the design, including design, layer, net class, net, class-to-class, room and component. Constraint values may also be determined by employing constants, formulas, vectors or expressions.

Unless specified, all references to the Parametric Constraint Solver features, relevant when a PCB design is loaded, are also applicable when a Relay design is loaded.

When you access PCS from PCB you have access to the Design Manager and the Constraint Editor in which to organize your design objects and define the desired constraints. Each of these aspects of the Parametric Constraint Solver interface is discussed in About the User Interface below.

Schematic Designs in PCS

Parametric Constraint Solver can be activated from P-CAD Schematic to help organize a design's components and nets in logical groups (e.g., partitions and net classes, respectively), and define placement requirements to keep the PCB Designer informed of your design intentions.

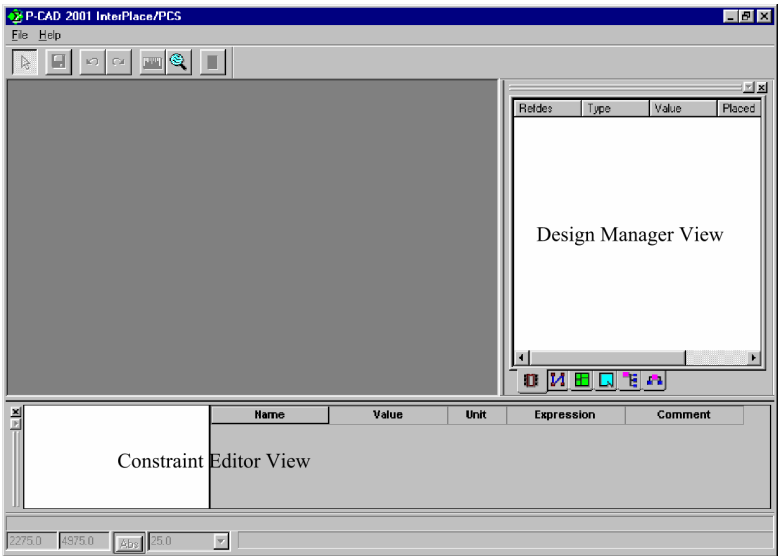
To achieve these functions, when you access PCS from Schematic you have access to the Design Manager and the Constraint Editor. The Parametric Constraint Solver interface is discussed in About the User Interface below.

About the User Interface

The P-CAD Parametric Constraint Solver interface follows the standard Windows format, with the addition of PCS specific controls. This section provides an overview of the user interface, including the menu bar, toolbars, and basic screen layout.

Parametric Constraint Solver Interface

When you first access the P-CAD Parametric Constraint Solver, the interface displays an empty area for each of the views. All menu functions except **File » Load PCB Design**, and **File » Load Schematic Design** are unavailable.



When you load a design file from a P-CAD application, the design information is displayed in the appropriate viewing area. The Parametric Constraint Solver workspace views include:

- The Constraint Editor View
- The Design Manager View

Designs loaded from P-CAD PCB, Relay or Schematic provide the design information as it pertains to the individual views. The Design Manager View shows the design objects in each of its pages, while the Constraint Editor View provides access to the constraints.

Each of the Parametric Constraint Solver views is summarized in the Parametric Constraint Solver Views section.

Menu Bar

The menu bar allows you easy access to Parametric Constraint Solver commands and functions.

Before you load a design, the PCS menu bar contains only the **File** and **Help** menu items. From the **File** menu you can choose to load a PCB or Schematic design.

When you load a design from PCB or Schematic, the menu bar of PCS appears as follows:

File View Options Help

To activate a menu, click the menu title or press the **Alt** key in combination with the underlined letter of the menu title (e.g., **Alt**+**F** to open the **File** menu). When the menu appears, click a menu item, or press the underlined key, to enable a command.

- A command followed by three dots (e.g., Open...) opens a dialog when you choose it.

- The menu bar wraps if you reduce the width of the window. In contrast, other areas of the screen truncate with screen reduction.

Parametric Constraint Solver Toolbar

Toolbars provide shortcuts to commonly used commands and functions. Toolbars appear when you activate the appropriate View command. Once a toolbar is visible on your screen, you can use your mouse to drag it to a new position. It can be docked along the edges of your screen, or left floating in the middle of your display. A floating toolbar can be resized.

P-CAD Parametric Constraint Solver has one basic toolbar: the command toolbar.

Command Toolbar

The Command Toolbar provides shortcuts to menu commands and other system functions.

The Command Toolbar attached to the Parametric Constraint Solver contains only one shortcut command and appears as follows:

Update PCB/Schematic

Parametric Constraint Solver Views

Each view in the Parametric Constraint Solver specializes in a unique design function. The Design Manager View is superb at organizing design objects and providing many different aspects of the design's data. The Constraint Editor View provides the means to add to and remove constraints from any of the precedence levels, and determine how the constraint values are established.

The Design Manager View

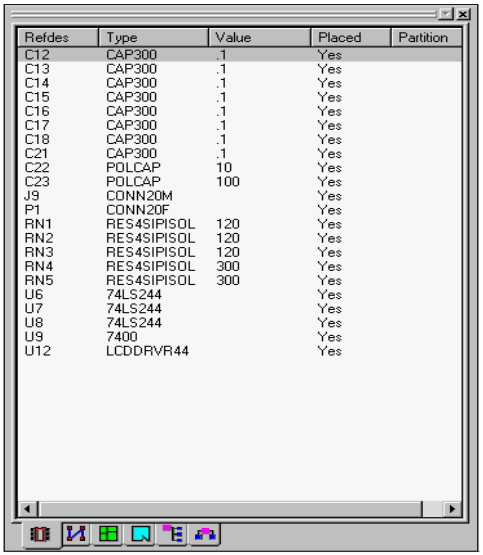
The Design Manager View provides a simple interface for arranging design components into physical and logical divisions, and affords easy access to information about the nets, net classes and class-to-class rules in your design.

With the Parametric Constraint Solver, you can create a logical division in the design, called a partition, and assign the components within these divisions. You can also create, modify and delete net classes and move components from one net class to another, and assign net classes to a class-to-class.

You can also manage physical regions on the PCB board layout, called rooms, where the components and partitions can be assigned.

The Design Manager View may be enabled or disabled using the View Design Manager command. When visible, the view can be disabled by clicking the appropriate button at the top right corner. The Design Manager can be resized by positioning the cursor on the edge of the line you want to move, then dragging it to the right or left and releasing the button at the desired location. Additionally, if you double-click on the gray lines at the top of the view, the view is undocked and floats in the middle of your display. You can then use your mouse to drag it to a new position.

When enabled, the Design Manager View appears in your workspace as shown below:



Refdes	Type	Value	Placed	Partition
C12	CAP300	.1	Yes	
C13	CAP300	.1	Yes	
C14	CAP300	.1	Yes	
C15	CAP300	.1	Yes	
C16	CAP300	.1	Yes	
C17	CAP300	.1	Yes	
C18	CAP300	.1	Yes	
C21	CAP300	.1	Yes	
C22	POLCAP	10	Yes	
C23	POLCAP	100	Yes	
J9	CONN20M		Yes	
P1	CONN20F		Yes	
RN1	RES4SIPISOL	120	Yes	
RN2	RES4SIPISOL	120	Yes	
RN3	RES4SIPISOL	120	Yes	
RN4	RES4SIPISOL	300	Yes	
RN5	RES4SIPISOL	300	Yes	
U6	74LS244		Yes	
U7	74LS244		Yes	
U8	74LS244		Yes	
U9	7400		Yes	
U12	LCDDRVR44		Yes	

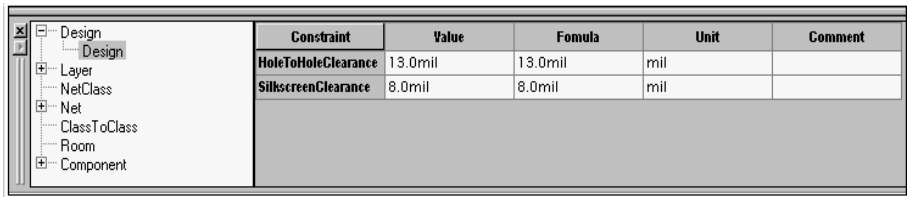
For more information about using the Design Manager, refer to *The Design Manager View, (page 15)*.

The Constraint Editor

The Constraint Editor displays all of the design constraints in an easy-to-use Graphical User Interface (GUI). It is organized in a tree structure that shows the relationship between these constraints and the design. The Constraint Editor provides easy access to all of this information for viewing or modifying.

The Constraint Editor View may be enabled or disabled using the **View » Constraint Editor** command or by clicking the appropriate button at the top right side of the view. The view can also be resized by positioning the cursor on the edge of the line you want to move, then dragging the cursor to the right or left and releasing the button at the desired location.

When enabled, the view appears in your workspace as shown below:



The Constraint Editor spreadsheet provides the ability to establish constraint values using simple or complex methods. Formulas can be as simple as a mathematical operation or as complex as referencing other constraint values to define relationships between constraints.

For more information about using the Constraint Editor, refer to *The Constraint Editor View, (page 31)*.

Loading, Saving and Exiting a Design

This section details basic design loading and updating functions as well as how to exit the Parametric Constraint Solver application.

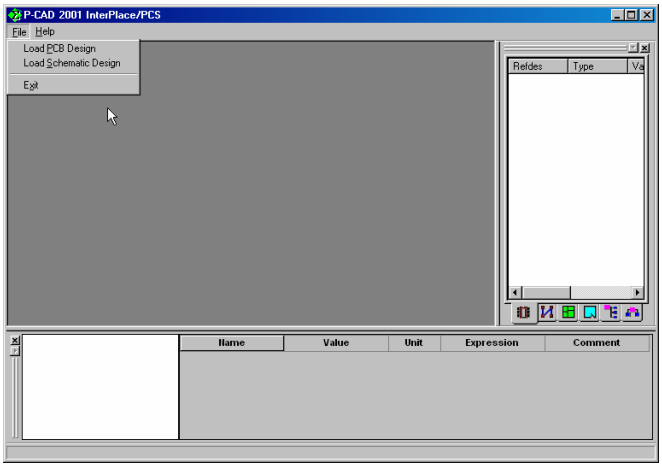
Loading a Design

To load a design into the P-CAD Parametric Constraint Solver, the design must first be opened in P-CAD PCB, P-CAD Relay, or P-CAD Schematic.

Once the design is open and active in the originating P-CAD application, choose the **Utils » P-CAD Design Constraint Manager** command to launch the Parametric Constraint Solver. You can also launch PCS from your desktop.

The **Design Constraint Manager** command in the PCB and Schematic Utils menus provides access to both PCS and P-CAD InterPlace. P-CAD searches your license files and, depending on which license(s) are found, launches the appropriate application(s).

From the **Parametric Constraint Solver File** menu choose the **Load PCB Design** or the **Load Schematic Design** command to begin the design loading process.



The design is loaded automatically and the Parametric Constraint Solver workspace displays the Design Manager and Constraint Editor Views.

While a design is loaded, the originating P-CAD application is locked so that conflicting changes to the design or the design constraints cannot be made. Refer to *P-CAD Application Lock* for details.

P-CAD Application Lock

When the P-CAD Parametric Constraint Solver is running, the P-CAD application from which the design originated is locked. The lock prevents conflicting changes to the PCB or Schematic design. The following warning is displayed over the PCB or Schematic workspace while the design is loaded into the Parametric Constraint Solver.

Cancel terminates the DBX (P-CAD Database Exchange programmer's interface) connection between the applications. If this connection is broken any changes made in PCS are not saved to the design file.

Design information is communicated between the originating application and the Parametric Constraint Solver via DBX. If you wish to update the design with changes made in the Parametric Constraint Solver, do not click **Cancel**. To update the design, use either the **File » Update PCB** or **File » Update Schematic** command to save the changes to the original design. Then, close the design or exit the Parametric Constraint Solver.

See *Working with the Constraint Editor View, (page 41),* for more information on maintaining the update link between PCS and the originating application.

Saving the Design

If you want to save the changes you have made to the design in PCS, and would like to continue working on the current design, use the **File » Update PCB or File » Update Schematic** commands or click the **File » Save Design** button to save the changes to the PCB or Schematic design file. With any of these commands, the file remains open in PCS so you can continue working on it.

If you have finished working with the current design in PCS, choose **File » Close**. This command updates the design when you confirm that you want to save the changes and closes the Parametric Constraint Solver views. The design is unlocked in the originating P-CAD application, permitting changes again in the current PCB or Schematic design editor. When you save the design or exit the originating application, all changes to the design are applied.

When you exit PCS and return to the originating application, make sure you do a **File » Save** in PCB or Schematic to apply the PCS updates to the design. Updates made in PCS are saved in the memory of the originating application only until you choose the File Save command or confirm that you want the updates saved on exiting the originating application.

Exiting the Parametric Constraint Solver

Choose the **File » Exit** command to exit the P-CAD Parametric Constraint Solver.

If an open design or its constraints have been modified since the last save, you are prompted whether you want to save the changes.

The design files are unlocked in the originating P-CAD application, permitting changes again in the PCB or Schematic design editor.

The program writes information to the `.ini` file when you choose **Exit**. This information, which will apply to subsequent Parametric Constraint Solver sessions, consists of parameters and settings such as the size of the views, their positions in the workspace, etc.

Right Mouse Commands

When you select an object and click the **right mouse button**, a pop-up menu appears providing shortcuts to common commands performed on selected objects. This menu changes depending on the object you select and the view from which they are selected.

The following section summarizes the commands, which appear on the pop-up menu:

- **Add Constraint:** This command in the Constraint Editor View displays the Add Constraint dialog where you can define a new constraint.
- **Add To:** This command is used to add selected components to a Room.
- **Delete:** This command deletes a selected partition or class-to-class.
- **Delete Net Class:** This command deletes the selected net class.
- **Move To:** This command moves selected components to a partition, or selected nets to a Net Class.
- **New Class-to-Class:** This command is used to add a new class-to-class to the design.
- **New Net Class:** This command is used to add a new net class to the design.
- **New Partition:** This command creates a new partition.
- **Remove Components:** This command removes selected components from a partition or Room.
- **Remove Constraint:** This command in the Constraint Editor deletes the selected constraint.
- **Remove Nets:** This command removes nets from a net class.
- **Rename:** This command is used to rename a partition.
- **Rename Net Class:** This command allows you to rename a selected net class.

CHAPTER 4

The Design Manager View

The P-CAD Parametric Constraint Solver provides the Design Manager View to help in organizing the design components into logical partitions and physical regions called rooms. The Design Manager View also gives access to information about the nets, net classes and class-to-classes in your design in an easy-to-use interface.

In this chapter, each method of organizing components, nets and net classes is discussed, as well as how you can use them to expedite the design and production of an efficient and cost effective circuit board. You will also learn how organizing design objects can facilitate the assignment of constraints.

Working with the Design Manager View

The Design Manager View of the P-CAD Parametric Constraint Solver displays the design information in either a tree-like structure or a simple list. You can view design data from a number of perspectives: components, nets, partitions, rooms, a net class and a class-to-class, each of which is available in a Design Manager View page.

In the Components page you can rearrange the display of the details shown there by sorting the list of components using any of the column headings. You can create partitions in the partitions page, and add components to or remove components from the partitions. If you created a room in the PCB design, you can add or remove components in those rooms. From the Nets page you can view the list of nets in the design, and the nets can be organized into net classes and class-to-class groupings in their respective pages.

In the next sections you will learn how to access the Design Manager View, move around in each of the pages, and employ its powerful features to expedite your design efforts.

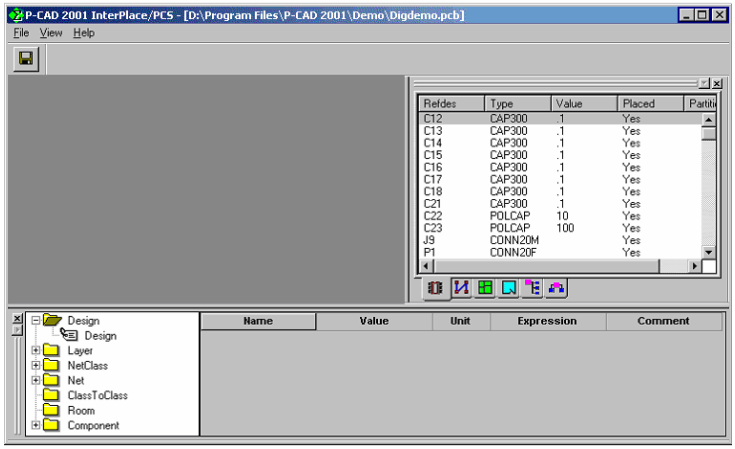
Accessing the Design Manager View

When you launch the Parametric Constraint Solver, the Design Manager View appears automatically in the workspace. The position and size of the Design Manager View from the last session in which you loaded a design into PCS is stored in the `.ini` file and reconstructed in the workspace for the new session.

Before a design is loaded, the Design Manager View is empty. Once a design is loaded using the **File » Load PCB** or **Schematic Design** commands, the Design Manager View displays the design data in the relevant pages.

You can choose to view or hide the Design Manager View using the View Design Manager command. The view can also be resized and rearranged in the workspace. See *Parametric Constraint Solver Basics, (page 7)*, for details on changing the way the workspace is organized.

The Design Manager View is displayed in the right half of the Parametric Constraint Solver workspace when PCS is launched for the first time, as shown below:



Accessing Design Manager Pages

To access any of the pages in the Design Manager View, simply click the appropriate tab at the bottom of the view. The data presented in the six different views of the design data are shown below with their associated icon:

 Components Page

 Nets Page

 Partitions Page

 Rooms Page (for PCB designs only)

 Net Class Page

 Class-to-Class Page

On each page, except the Components Page, is a tree structure or hierarchy, containing all of the information pertinent to that page. You can view the page's contents at various levels of detail by expanding or collapsing the branches of the tree.

Groupings containing collapsed levels are shown with a ⊞ sign. To expand the grouping, simply click the ⊞. Expanded groupings are shown with a ⊟ sign. To collapse the grouping, simply click the ⊟ sign. You can also use the ← and → keys on a selected group to collapse or expand groupings, respectively.

Selecting Items in Design Manager

In order to perform functions on the data items in your design, you must first select one or more items in a page. For instance, you may want to select components to add to a room or partition. You might also want to create a new net class or class-to-class from selected nets and net classes.

Items can be selected using any of these methods:

- **To Select a Single Item:** Position the cursor on the item and click the **left mouse button**.
- **To Select a Contiguous Block of Items:** Position the cursor on the first item in the block and click the **left mouse button**, press the **Shift** key and click on the last item in the block. You can also select an item, then press the **Shift** + **End** keys to select the items that follow to the end of the list.
- **To Select Multiple Individual Items:** Press the **Ctrl** key while clicking the **left mouse button** on each item you want to select.

You can right mouse click on a selected item(s) in the tree to access its pop-up command menu. Refer to the section, *Design Manager Right Mouse Commands, (page 23),* for the complete list of right mouse commands.

Using the Components Page

The Components Page lists all the components in the design, along with some pertinent information about each of them, including the Type, Value, Placed indicator and Partition name.

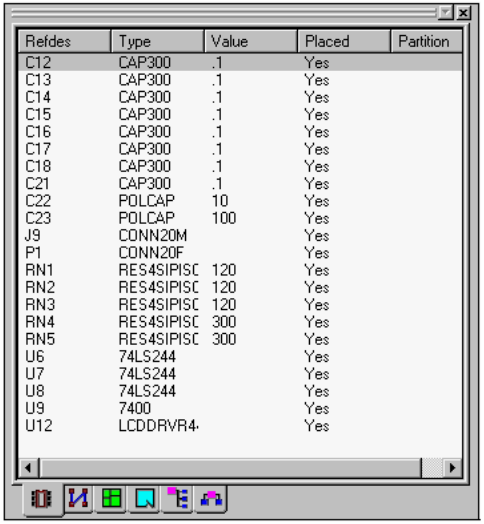
You can sort the components by clicking any of the column headers.

Component Information

For each component in the design, the following information is displayed:

- Reference Designator or component identifier.
- Type of component.
- Value assigned to the component.
- Placed indicates whether the component is or isn't located within the board outline.
- Partition displays the name of the partition the component is assigned to, if any.

The Components Page appears as follows:



Refdes	Type	Value	Placed	Partition
C12	CAP300	.1	Yes	
C13	CAP300	.1	Yes	
C14	CAP300	.1	Yes	
C15	CAP300	.1	Yes	
C16	CAP300	.1	Yes	
C17	CAP300	.1	Yes	
C18	CAP300	.1	Yes	
C21	CAP300	.1	Yes	
C22	POLCAP	10	Yes	
C23	POLCAP	100	Yes	
J9	CONN20M		Yes	
P1	CONN20F		Yes	
RN1	RES4SIPISC	120	Yes	
RN2	RES4SIPISC	120	Yes	
RN3	RES4SIPISC	120	Yes	
RN4	RES4SIPISC	300	Yes	
RN5	RES4SIPISC	300	Yes	
U6	74LS244		Yes	
U7	74LS244		Yes	
U8	74LS244		Yes	
U9	7400		Yes	
U12	LCDDRVR4·		Yes	

Sorting the Components

You can sort the list of components for easier viewing by simply clicking on one of the column headings. For instance, if you want to see which components have been placed on the board, or which ones remain to be placed, click the **Placed** column heading and the unplaced components float to the top of the list.

Using the Nets Page

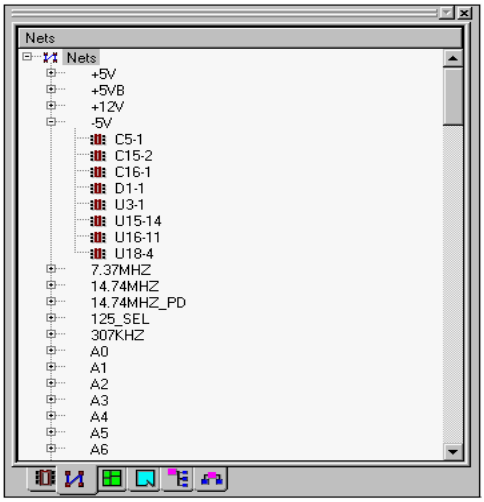
The Nets Page of the Design Manager View displays a list of the nets in the design, and their associated pins. If you have purchased P-CAD InterPlace, from the Nets Page you can highlight a net, show or hide the connections, jump to a particular node or cluster the components in a net into a common work area.

Net Information

The Nets page displays a list of the nets resident in the loaded design. Nodes associated with each net are listed for designs loaded from PCB.

> A Schematic design loaded into Parametric Constraint Solver displays only the list of nets in the design.

The Nets Page appears as follows:

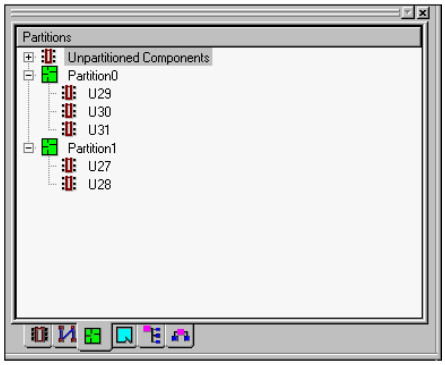


Using the Partitions Page

The Design Manager View provides a way to assemble components logically in a group referred to as a partition. Using a partition to group components is a great way to start organizing your design's components. For instance, you can establish partitions with like properties, such as the analog or digital components, or partitions that contain components destined for a particular sheet or layer.

Partition names can be used to describe the way the components are organized for easy identification. For instance, you may have a partition containing the analog components and name it AnalogPart, or for components located on a Schematic Sheet number one you can create a partition called Sheet1Part.

The Partitions Page is displayed in the workspace as follows:



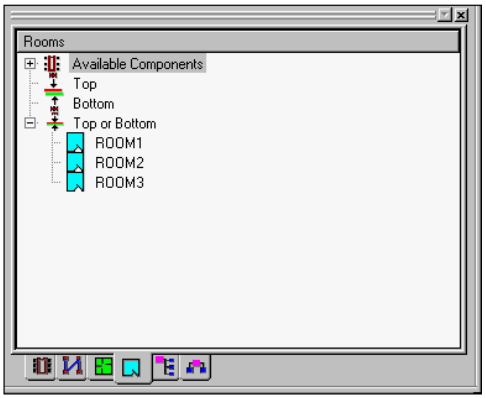
Organizing Components, (page 25), has more information on using partitions to help organize designs.

Using the Rooms Page

The Parametric Constraint Solver allows you to group components into a physical area on the board known as a Room. As an example, you may want the analog components in your design to be physically located at the front of the board and the digital components in the back. The Rooms page provides an easy way to organize these components in the PCB design, simply by adding or removing components from a room's contents.

Rooms must be placed on the board in PCB before the design is loaded into the Parametric Constraint Solver. A Room cannot be created in the Parametric Constraint Solver. See your *PCB User's Guide* for instructions on creating rooms.

The Rooms Page is displayed in the Design Manager View as follows:



Refer to *Organizing Components, (page 25),* for more information on the use of Rooms.

Using the Net Class Page

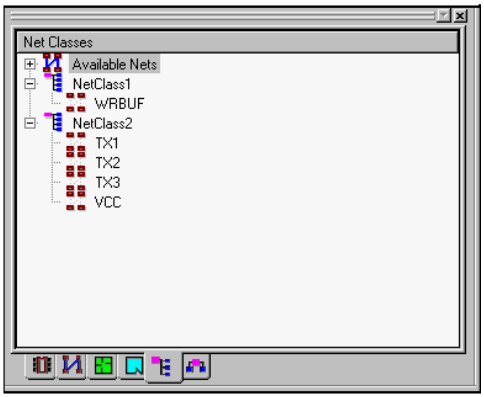
Nets in a design loaded into the Parametric Constraint Solver can be organized into Net Classes using the Net Class page by creating new net classes or modifying the nets that are part of a net class. Grouping nets into a Net Class saves time and effort when you want to apply specific design constraints to a number of nets. With the nets organized in a Net Class, constraint(s) can be applied once to the Net Class, instead of repeating the constraint for each of the individual nets.

Once a Net Class is established, you can delete and rename it. You can move the nets from one Net Class to another using the drag and drop method or the Move To command.

Net Class Information

The Net Class page of the Design Manager View displays the design's net classes and nets. When a net has not been assigned to a Net Class, it is listed in the `Available Nets` folder. The `Available Nets` and any `Net Class` folders can be expanded for viewing the included nets by clicking the ⊞ button or pressing the ← key. To collapse the lists, click the ⊟ button or press the → key.

The Net Class Page, showing the expanded list for each of the existing Net Classes, appears as follows:



Creating a Net Class

A new Net Class is created by choosing the **New Net Class** command from the right mouse menu in the Net Class page.

To Create a Net Class

1. Select the `Available Nets` folder, an existing net class or simply click in the workspace.
2. Choose the **New Net Class** command from the right mouse menu. The net class is given a default name of NetClassX, where X is a number incremented with each added net class.

Deleting a Net Class

Any Net Class in the design can be deleted.

To Delete A Net Class

1. Select an existing net class.
2. From the right mouse menu, choose **Delete Net Class** or press the **Delete** key. If the net class has assigned nets, the nets are placed back in the Available Nets list and the net class is deleted.

Renaming a Net Class

A Net Class can be renamed.

To Rename a Net Class

1. Select an existing net class.
2. From the right mouse menu choose the **Rename Net Class** command. You may also double click on the net class name to put the field in edit mode.
3. Enter the new net class name and press **Enter**.

Adding Nets to a Net Class

Any net in the design is available to be added to a Net Class. A net can belong to only one Net Class at a time. To add a selected net to a Net Class, use the drag and drop method or the **Move To** command in the *right mouse* menu.

Removing Nets from a Net Class

Nets removed from a Net Class are placed back into the Available Nets list. Selected nets are removed from a Net Class by dragging them from the Net Class into the Available Nets list or by using the **Remove Nets** command in the right mouse menu.

Using the Class-to-Class Page

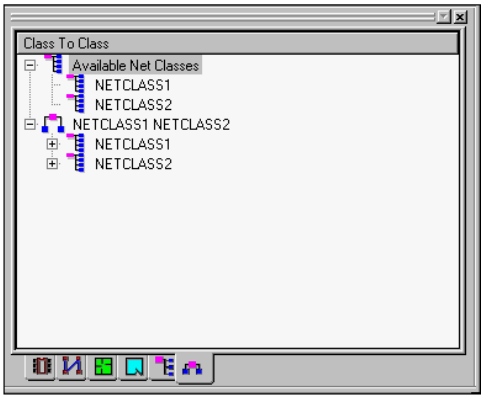
When you have established Net Classes in your design, you can define constraints that apply to only those unique Net Classes. You can create, delete and modify Class-to-Classes in this page as well.

For instance, a single net in your design requires a clearance greater than any other net clearance. You can assign this one net to a net class, and all the rest in another net class. The clearance constraint between the two net classes essentially keeps the single net in the first net class a unique distance from the other nets in the second net class.

Class-to-Class Information

The Class-to-Class page of the Design Manager View groups the design's net classes in the list of *Available Net Classes* and does not display them until you click the button.

The Class-to-Class Page appears as follows:



Creating a Class-to-Class

A new Class-to-Class is created by selecting one or two net classes and then choosing the New Class-to-Class command from the right mouse menu.

To Create a Class-to-Class

1. Expand the `Available Net Classes` folder, if it is not expanded, to display the available net classes, by clicking the ⊞ button or pressing the ➔ key.
2. Select one or two of the Net Classes in the list of available net classes. The new Class-to-Class name is the name of the first net class, followed by a space, then the name of the second net class appended to it. If you only select one net class, the new Class-to-Class name is the net class name repeated.

Deleting a Class-to-Class

You can delete any Class-to-Class you have created.

To Delete A Class-to-Class

1. Select an existing class-to-class.
2. From the right mouse menu, choose **Delete** or press the **Delete** key.

Design Manager Right Mouse Commands

When you select an object and click the right mouse button, a pop-up menu appears providing access to the commands you can perform in the Design Manager View.

The following table summarizes the commands, which appear on the pop-up menus, and the page where the command is available.

The column headings indicate the page as follows: **C**=Components; **N**=Nets; **P**=Partition; **R**=Rooms; **NC**=Net Class; **CC**=Class-to-Class.

C	N	P	R	NC	CC	Command
		X				**Add Selected Components:** This command adds the selected components to a partition.
			X			**Add To:** This command is used to add selected components to a room.
		X			X	**Delete:** This command deletes a selected partition or class-to-class.
				X		**Delete Net Class:** This command deletes a selected net class.
		X		X		**Move To:** This command moves selected components to a partition, or selected nets to a Net Class.
					X	**New Class-to-Class:** This command is used to add a new class-to-class to the design.
				X		**New Net Class**: This command is used to add a new net class to the design.
		X				**New Partition:** This command creates a new partition.
		X	X			**Remove Components:** This command removes selected components from a partition.
				X		**Remove Nets:** This command removes nets from a net class.
		X				**Rename:** This command is used to rename a partition.
				X		**Rename Net Class:** This command allows you to rename a selected net class.

CHAPTER 5

Organizing Components

Whether you are working on a PCB or a Schematic design, the P-CAD Parametric Constraint Solver will help you organize the many objects in your design. The Parametric Constraint Solver specializes in two of the basic areas of board design: object organization and constraint assignment. A well organized board is essential to minimizing constraint violations resulting in more efficiently designed and manufactured circuit boards.

The Design Manager View in the Parametric Constraint Solver provides easy access to information on the components and nets in your design. In the Design Manager View you can define logical blocks or partitions and assign components to them. For a PCB design, you can even group components into a room, providing an additional organizational tool to help place them in a specific physical location on the board.

This chapter explains in detail how to use the organizational tools in the Partitions and Rooms pages for your PCB or Schematic design objects. The other pages of the Design Manager View are described in *The Design Manager View (page 15)*.

Once the design objects are organized, you will be better prepared to assign constraints to individual objects or the groups of objects you have created.

Organizing Design Objects

When a PCB or Schematic design is loaded into the Parametric Constraint Solver, the components and nets are divided in the Design Manager View and displayed in one of these individual pages: Components, Nets, Partitions, Rooms, Net Class and Class-to-Class.

This portion of the chapter contains an explanation of the Parametric Constraint Solver organizational processes and step-by-step instructions on how to use them. The two pages designed to provide exceptional help in organizing components are the Partitions and Rooms pages, and they are described in great detail in this chapter.

The Design Manager View, (page 15), contains an overview of all functions available in the Design Manager View, along with detailed instructions on functions unrelated to organizing design data.

The Design Manager View

Each of the design data sections uses a tree like structure in which lists of objects can be expanded or contracted depending on your desired view. Refer to *The Design Manager View, (page 15),* for detailed information.

The Components Page

The Components Page is an ideal way to view the complete list of components in your design. For each component in the list the following essential information is displayed:

- **Reference Designator:** The reference designator of the component.
- **Type:** The component type.
- **Value:** The value assigned to the component.
- **Placed:** If any part of the component touches the board edge, or is located inside of the board outline, the Placed indicator displays Yes.
- **Partition:** If the component has been assigned to a partition the name of the partition is shown here.

Using the sorting capability of the Components Page you can quickly determine, for example, which components have been placed and which are assigned to a partition. Refer to *The Design Manager View, (page 15),* for more detailed information.

The Nets Page

The next informational page is the Nets Page. All nets in the loaded design are present in the list of nets for both PCB and Schematic designs. Refer to *The Design Manager View, (page 15),* for more detailed information.

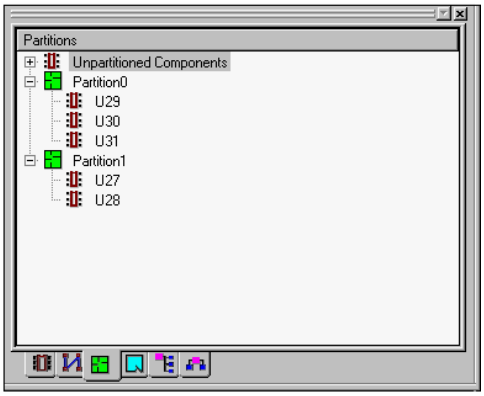
The Partitions Page

The Partitions page of the Design Manager View displays the list of components not assigned to a partition, provides the ability to create and delete partitions, and add or remove components from the partitions.

Components can belong to only one partition at any time.

When you first enter the Partitions page of the Design Manager View, and no partitions have been created, the components are grouped in the Unpartitioned Components list and are not visible. Click the ⊞ button next to Unpartitioned Components to display the list of components.

The Partitions Page is shown below:



Creating Partitions

Components in a design with no partitions are, by default, unpartitioned. A new partition is created using the **New Partition** command in the right mouse menu. The partition name can be changed at any time.

To Create a Partition

1. Select the `Unpartitioned Components` folder or an existing partition.
2. Choose the **New Partition** command from the right mouse menu. The partition is given a default name of PartitionX, where X is a number incremented with each added partition.

Deleting Partitions

Any partition you create can be deleted. If a deleted partition has components assigned to it, those components are placed back into the Unpartitioned Components list and the associated Partition name is deleted from the Components Page.

To Delete A Partition

1. Select an existing partition.
2. From the right mouse menu, choose **Delete** or press the **Delete** key. If the partition has assigned components, the components are placed back in the Unpartitioned Components list and the partition is deleted.

The `Unpartitioned Components` folder cannot be modified or deleted.

Renaming Partitions

The partition can be renamed at any time. Partition name changes are also reflected in the Partition column of the Components Page.

To Rename a Partition

1. Select an existing partition.

2. From the right mouse menu choose the **Rename** command. You may also double click on the partition name to put the field in edit mode.
3. Enter the new partition name and press **Enter**.

Adding Components to a Partition

When a design is loaded into the Parametric Constraint Manager, the components not assigned to a partition are part of the Unpartitioned Components list. Selected components can be moved to a newly established partition using the **Move To** command or by dragging them from the Unpartitioned list into the target partition.

To Add Components to a Partition

1. Display the list of Unpartitioned Components by clicking the button.
2. Select the desired components from the list.
3. Choose the **Move To** command from the right mouse menu, and then select the target partition from the right pull down list. You can also press the left mouse button and drag the selected components into the target partition.

Removing Components from a Partition

When you remove components from a partition they are placed back into the Unpartitioned Components list. Selected components are removed by dragging them from the target partition into the Unpartitioned Components list or by using the **Remove Components** command in the right mouse menu.

To Remove Components from a Partition

To remove all components from a partition and delete the partition:

1. Select the partition.
2. From the right mouse menu, choose **Delete** or press the **Delete** key. If the partition has assigned components, the components are placed back in the Unpartitioned Components list and the partition is deleted.

To remove a portion of the assigned components:

1. Select the desired components.
2. Choose the **Remove Components** command from the right mouse menu, or drag them back into the `Unpartitioned Components` folder.

The Rooms Page

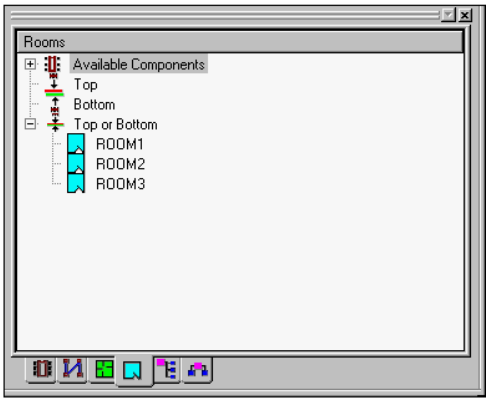
When a room has been placed in a PCB design, the Parametric Constraint Solver provides a vehicle with which to organize a set of components destined for placement in the room. You may want to position the analog components in a room located in the back of the board, for instance, and add a height constraint to that group of components.

When you enter the Rooms page of the Design Manager View, all of the design components are listed in the Available Components list and are not visible until you click the button.

Components can be assigned to more than one Room, therefore, the list of *Available Components* does not change even after a component has been assigned to a room. All components remain available for room assignment at all times.

In addition to the `Available Components` folder, there is a Top, Bottom and Top or Bottom folder. Existing Rooms are displayed in their appropriate folder, depending on the layer placement.

The Rooms Page appears as follows:



Creating a Room

Physically drawing the boundaries of a room must be completed in PCB before the design is loaded into Parametric Constraint Solver. The location of the room, Top, Bottom or Top or Bottom, is determined at the time the room is placed in the design. See your *PCB User's Guide* for more information on placing a room.

Adding Components to a Room

When a design is loaded into the Parametric Constraint Manager, the components available to be assigned to rooms are listed in the Available Components list. Selected components can be added to a room using the **Add To** command or by dragging them from the Available Components list into the target room.

Components can be included in more than one room, therefore the list of Available Components does not change when components are added to or deleted from a room. Instead of being moved, component information is copied from the Available Components list into the target Room.

To Add Components to a Room

1. Select the desired components in the list of Available Components.
2. Choose the **Add To** command from the right mouse menu and the specific room from the pull down list, or press the left mouse button and drag the selected components to their room destination.

Removing Components from a Room

Selected components can be removed from a room only by using the **Remove Components** command in the right mouse menu.

To Remove Components from a Room

1. Select the desired components in the room.
2. Choose the **Remove Components** command from the right mouse menu. The components are removed from the room.

A component can belong to any number of rooms at the same time.

The Net Class Page

Organizing nets into one class is, again, an easy way to apply constraints once instead of individually to each net. The Net Class Page is described in detail and instructions provided in *The Design Manager View, (page 15)*.

The Class-to-Class Page

Once you have established Net Classes in your design, you can define constraints that will apply to only those unique Net Classes in a Class-to-Class definition.

For example, you may have a single net in your design, which requires a clearance greater than any other net clearance. You can assign this one net to a net class, and all the rest to another net class. These two net classes can then be associated using a class-to-class.

The clearance constraint applied to the class-to-class essentially keeps the single net in the first net class a unique distance from the other nets in the second net class.

The Class-to-Class Page is described in detail, and instructions provided in *The Design Manager View, (page 15)*.

CHAPTER 6

The Constraint Editor View

Design constraints are an integral part of P-CAD PCS. The Parametric Constraint Solver provides an easy-to-use interface in which you can organize, prioritize and define constraints or design rules. In one convenient location you can access all the constraints for a design and easily manipulate those constraints to satisfy your design requirements. You can even define constraint formulas, which are based on other constraints values.

Design constraints and any constraint expressions are automatically loaded from an open design in P-CAD PCB or Schematic to the Parametric Constraint Solver. The constraints are consolidated in the Constraint Editor where you can view, add, copy, modify, or remove constraint definitions. Constraints defined or modified in PCS can be saved back to the original PCB or Schematic design.

This chapter summarizes the constraint management power of the Constraint Editor. You will learn what a constraint is and how constraints are applied to printed circuit board and Schematic designs. The Constraint Editor's functions are detailed along with instructions on building formulas based on other constraints.

Working with the Constraint Editor

The Constraint Editor in the P-CAD Parametric Constraint Solver displays the design constraints in an easy-to-use Graphical User Interface (GUI). It is organized in a tree-like structure that shows the relationship between these constraints and the design. The Constraint Editor provides easy access to all of this information for viewing or modifying.

In order to apply the power of the constraint management tools to your design, you must first understand the basics of the Constraint Editor.

In this section you will learn how the design's constraints are organized and where they appear in the workspace. A constraint definition is included, along with how constraints modified in the PCS work with P-CAD PCB's Design Rules Checking.

What is a Constraint?

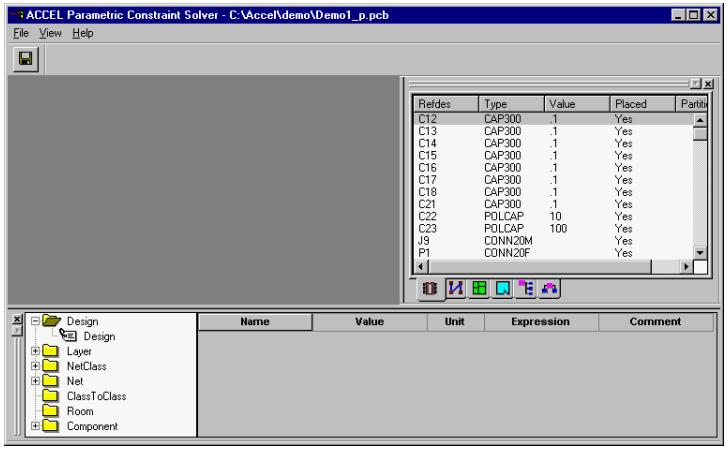
A design constraint may be more familiar to you from P-CAD PCB or P-CAD Schematic as a design rule or attribute. Rules or attributes specify requirements that must be met to successfully design, route and manufacture a printed circuit board.

For example, with PCS you can assign components to a physical, predefined area of the board called a room, and then, using the Constraint Editor View, apply a maximum height constraint to all the components located in that area.

In the above example, PCB Design Rules Checking can automatically verify that components placed in a room satisfy design rules or constraints.

Viewing the Constraint Editor

The Parametric Constraint Solver displays the Design Manager View and the Constraint Editor View as shown below.

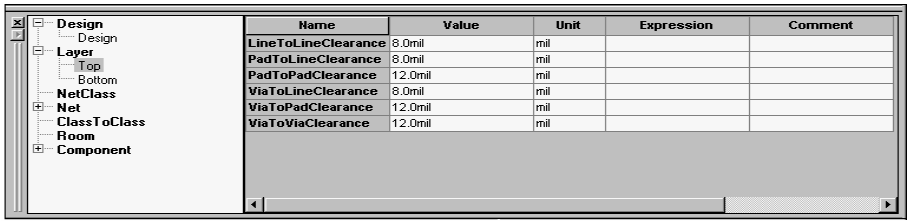


The Constraint Editor View may be enabled or disabled using the **View » Constraint Editor** command. When visible, the view can be disabled by clicking the appropriate button at the top right corner. The Constraint Editor can also be resized by positioning the cursor on the edge of the line you want to move, then dragging it to the right or left and releasing the button at the desired location. Additionally, if you double-click on the gray lines at the left of the view, the view is undocked and floats in the middle of your display. You can then use your mouse to drag it to a new position.

The windows can be rearranged and, along with the columns, resized. PCS stores the visibility, position and size of the Constraint Editor View from your last session in the `.ini` file and reconstructs those settings in the workspace for the new session.

Accessing the Constraint Editor

To access the Constraint Editor, if it is not visible when you launch PCS, choose the **View » Constraint Editor** command after the design file is loaded. The Constraint Editor View, with a design file loaded, is shown below:



The Constraint Editor View has two components:

- **The Precedence View:** Located on the left side of the workspace the Precedence View lists the constraint categories in their hierarchical order of precedence.
- **The Constraint View:** Located on the right side of the workspace, the Constraint View displays the constraints assigned to a selected item(s) in a category.

The Precedence View

Objects in the design are organized into predefined categories, which are displayed in the Precedence View. These categories are arranged hierarchically according to their application precedence and appear in a tree-like structure.

The primary access point for the Constraint Editor is the Precedence View. Depending on the objects selected, you can view your design constraints in a number of ways. For instance, you can see which components, nets and class-to-classes have no constraints assigned. You can also select a number of items in a category, and display the constraints they have in common.

For more information on how to use the Precedence View, see *The Precedence View (page 33)*.

The Constraint View

When you select an item in the Precedence View, such as the Top layer in the Layer category, the constraints that have been assigned to the item appear in the Constraint View. The Constraint View displays the name, value, unit, expression and comments for each constraint.

In the Constraint View you can add a new constraint, remove a constraint, or modify an existing constraint's value. You can also create an expression which, when evaluated, supplies a constraint's value based on other constraints. The unit used to display the constraint's value can be selected based on the constraint's type.

For more information on how to use the Constraint View, see *Using the Constraint View, (page 36)*. Expressions are explained in detail in Evaluating Expressions.

Constraints and Rules Checking

Certain design constraints added, deleted or modified in PCS and saved in the design file, are used by P-CAD PCB Design Rules Checking to verify your PCB board or Schematic design.

For more information on how constraint rules are processed during design, refer to the *Utils Commands* chapter in your *PCB User's Guide*.

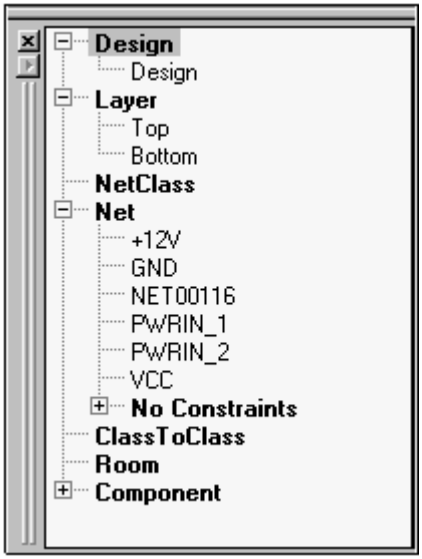
Using the Precedence View

The predefined categories in the Precedence View specify the hierarchy of the design constraints. Their order indicates which value takes precedence if the same constraint is defined in multiple categories for a single design object. Design constraints are further explained in Design Constraint Categories below.

The Precedence View is the main access point for adding, deleting and modifying constraints assigned to the items in your design. Items such as net classes and class-to-classes cannot be added directly to the categories in the Precedence View, they must be added using the Design Manager View. Once added, however, they are displayed in the Precedence View and constraints can be assigned to them.

Viewing the Precedence View

The tree structure of the Precedence view displays all design objects in their categorical precedence, as shown below:



Within each constraint category in the Precedence View tree are constraint items. For example, the Layer category will have the item Top, which contains all of the constraints defined for the Top layer of the design. When a category has a large number of items, such as a Net or a Component, the items in the category are divided into two lists. The first list displays the items that have constraints, while the remaining items are grouped into the No Constraints folder.

Constraint Categories

There are seven constraint categories organized hierarchically, whose order precedence is applied to the design. For example, a Design Width constraint can be overridden for a particular net by specifying a Net Width constraint.

The constraint categories of PCS are listed, and displayed in the Constraint Editor View, in their precedence order as shown below:

- **Design:** Design constraints are constraints that apply to any object when no other constraint has been set.
- **Layer:** Layer constraints are constraints that apply to any object on a particular layer.
- **Net Class:** Net Class constraints apply to all nets that have been assigned to the specified net class.
- **Net:** Net constraints apply to a specific net.
- **Class-to-Class:** Class-to-Class constraints specify constraints between two net classes.
- **Room:** Room constraints apply to the components included in a room. Note that the Placement Side Attribute on the Rooms dialog in PCB is used for Design Rules Checking. If the attribute is added in PCS, it will not override the value on the dialog.
- **Component:** Component constraints apply to a specific component.

Constraint categories are used when evaluating and enforcing design rules.

> Certain constraints assigned to items in the Layer category are predefined in the PCB design and they cannot be deleted. Those constraints are: PadToPad, PadToLine, PadToVia, LineToVia, LineToLine and ViaToVia. The display units for these predefined constraints are fixed and cannot be changed, but the constraint values can be modified.

Accessing Items in a Category

You can view all design constraint categories and their contents at various levels of detail by expanding or collapsing the branches of the tree.

Groupings containing collapsed levels are shown with a ⊞ sign. To expand the grouping, simply click the ⊞. Expanded groupings are shown with a ⊟ sign. To collapse the grouping, simply click the ⊟ sign. You can also use the ← and → keys on a selected group to collapse or expand groupings, respectively.

Precedence View Right Mouse Commands

When one or more items are selected in the Precedence View, and you click the right mouse button, a menu appears providing commands that can be performed on the selected items.

The following list summarizes the commands, which appear on the pop-up menu in the Precedence View.

Add Constraint: This command is used to add a constraint to the selected item(s) and can be accessed from both the Precedence View and the Constraint View.

Remove Constraints: This command is used to remove all constraints from the selected item(s) in the Precedence View or individually selected constraints in the Constraint View.

Using the Constraint View

The Constraint View displays pertinent information about each constraint assigned to a selected item or items in the Precedence View tree. When you select an item in the Precedence View tree, such as Top in the Layer category, its contents appear to the right side in the Constraint View.

In the Constraint View you can add constraints, and choose the unit in which the constraint value is displayed. Existing constraints can be removed. A constraint's value can be directly modified, or indirectly updated by creating an expression based on other constraints values. In addition, comments can be added to each constraint.

Viewing the Constraint View

When the Constraint Editor is visible in the PCS workspace, and an item has been selected in the Precedence View, the Constraint View appears as follows:

Name	Value	Unit	Expression	Comment
LineToLineClearance	8.0mil	mil		
PadToLineClearance	8.0mil	mil		
PadToPadClearance	12.0mil	mil		
ViaToLineClearance	8.0mil	mil		
ViaToPadClearance	12.0mil	mil		
ViaToViaClearance	12.0mil	mil		

The Constraint View displays the constraints within the selected item and includes the information in these columns:

- **Name:** This is the name of the constraint or attribute, for example, LineToLineClearance or PadToLineClearance.
- **Value:** The value assigned to the constraint. The value displayed here can be entered directly into the field, or will automatically appear based on the evaluation of an expression in the Expression field.
- **Unit:** The unit is the method of presentation for the quantity entered in the Expression field. For instance, a width measurement can be presented as a unit of inch, millimeter, mil, etc. The unit is assigned when the constraint is added.

  If you add a user-defined constraint and do not choose a unit, you will be required to enter the unit along with the value each time you change the constraint's value. For instance, if you have

a constraint whose value should always be mil, by assigning that unit to the constraint you need not enter "mil" each time the numeric measurement changes, you will only need to enter the appropriate figure.

- **Expression:** This field displays a formula, which can use numeric values, expressions, functions, or other constraint values to mathematically calculate the value applied to a constraint.
- **Comment:** Notes about the constraint.

The columns in the Constraint View can be resized by clicking the **left mouse button** on the dividing line and dragging it to the right or left, then releasing the button at the desired location.

For more information about constraint types, see *Constraint Data Types* below. Additional information on building expressions can be found in *Evaluating Expressions, (page 44)*.

Constraint Data Types

Each design constraint has a data type, which specifies the required format of the constraint value. The data type is important when building a formula based on other constraints.

Design constraints can be one of many types. Each constraint type has a unique set of units that can be applied to the selected type. The complete list of data types for each constraint can be found in *Appendix B, PCS References, (page 69)*.

The constraint types are listed below:

- **Length**: A constraint with type Length can be specified in a variety of different measurement units such as inches, mils, etc.
- **Angle**: A constraint of type Angle can be specified using degree or radian.
- **Resistance**: A constraint of type Resistance can be specified as ohm.
- **Conductance**: A constraint of type Conductance can be specified as mho.
- **Voltage**: A constraint of type Voltage has a base unit of volt.
- **Current**: A constraint of type Current has a base unit of ampere.
- **Inductance**: A constraint of type Inductance has a base unit of henry.
- **Capacitance**: A constraint of type Capacitance has a base unit of farad.
- **Time**: A constraint of type Time has a base unit of second.
- **Frequency**: A constraint of type Frequency has a base unit of hertz.
- **Power**: A constraint of type Power has a base unit of watt.
- **Boolean**: A constraint of type boolean indicates a True/False value.
- **Quantity**: A constraint of type Quantity is a whole positive number.

- **String**: A constraint of type String includes a series of alphanumeric characters. For example, the constraint PartNumber may have the String value 76-8BEH98-42.
- **Layer**: A constraint of type Layer indicates the layer on which the constraint applies (e.g., Top, Bottom, etc.).
- **ViaStyle**: The ViaStyle type is specific to the ViaStyle constraint. It indicates the style of the via for a particular design, net class or net.

Constraint types described as having a base unit all provide additional units of milli-, micro-, nano-, and pico- in addition to the base unit. The one exception is the Frequency type where the additional units are kilo- and mega- hertz.

Constraint types are particularly important when defining a constraint expression based on other constraints. See *Evaluating Expressions, (page 44),* for more information

Expression Syntax

When building an expression to determine a constraint's value, the syntax, or structure, of the expression must follow the format supported by the Constraint Editor. PCS grammar defines where spaces, quotes, brackets, periods and other special characters are allowed and the format required to successfully utilize expressions and operators in your formulas.

In its simplest form, using the expression field, you can derive the constraint value from a sequence of numbers, constraint, vector or an expression where an expression is a combination of constraints, operators and integers.

A very simple example is the one shown below which defines a RiseTime constraint by using a numeric value (number) in the Expression field.

Name	Value	Unit	Expression
RiseTime	1.5ns	nanosecond	1.5ns

The next example uses an expression referencing the RiseTime constraint shown above to compute the MaxNetLength.

Name	Value	Unit	Expression
MaxNetLength	8700.0mil	mil	RiseTime*5800[mil/ns]
RiseTime	1.5ns	nanosecond	1.5ns

Moving on to a sample expression that is a bit more complicated, the next example uses two other constraints and a trigonometric function in its expression to determine the ComponentSpacing value. Note that one of the referenced constraints, ComponentHeight, resides in the same precedence level while the InspectionAngle constraint comes from the Design level.

Name	Value	Unit	Expression
ComponentHeight	51.0mil	mil	
ComponentSpacing	4.0mil	mil	int(ComponentHeight*tan(Design.InspectionAngle))

In the last example, an expression is used to produce the value of the Capacitance constraint.

Name	Value	Unit	Expression
BoardThickness	9.0mil	mil	9
Capacitance	1.37264uF	microfarad	.67uF * (diel_const + 1.41) / ln(5.98 * BoardThickness / (.8 * Cu_width + cu_thickness))
Cu_thickness	0.8mil	mil	.8
Cu_width	10.0mil	mil	10
Diel_Const	2.3		2.3

This expression includes the Diel_Const, BoardThickness, Cu_width and cu_thickness constraints and incorporates their values into a formula to determine capacitance. Whenever the value of any of the referenced constraints changes, the value of the capacitance constraint is re-evaluated and changed automatically.

For more information on expressions, see *Working With Constraints, (page 41)*. Additional information on valid units, data types, function, operators and other PCS definitions and grammar can be found in *Appendix B, PCS References, (page 69)*.

CHAPTER 7

Working with the Constraint Editor View

Knowing how to organize and assign constraints to the various objects in your design is just the beginning of putting the power of the Parametric Constraint Solver to work for you. Building an expression to represent a constraint value is an advanced, intelligent way to produce the desired value when one constraint's change affects others down the line.

In this chapter you will learn how to add the constraints you need to the items in the precedence levels of the design. You will also put the Constraint View to use in providing the proper values in the constraints used in your design.

There is also a section, which explains how the updates you make in the Parametric Constraint Solver affect the design file in PCB and Schematic.

Working With Constraints

All constraints present in a PCB or Schematic design are displayed in the Constraint Editor when the file is loaded into PCS. New constraints can be added, or once established, deleted from the Constraint View. Constraint values can be modified, expressions built and comments noted as well.

The Constraint View also provides an easy method of adding constraints to any number of items in one easy step, instead of adding the same constraint many times to individual items.

Assigning Units to Constraints

When adding a constraint to a design item, a unit may be designated. The unit determines how the constraint value is displayed, and is only used when an expression is evaluated.

After the category and type of constraint have been chosen in the *Add Constraint* dialog, the desired unit is assigned by selecting from the list of allowed units for that constraint type. For instance, a clearance constraint unit is chosen from the list of Length units, such as inch or mil.

For constraints requiring a data type of Length, use units of mil, mm, cm or inch in order to correspond with the units used in a PCB design.

If the constraint being added is user-defined, the unit can be chosen from the complete list of allowable units. If you do not specify a unit for a user-defined constraint, PCS makes the unit Unitless.

How Units are Displayed

The unit that appears in the Value field is dependent on how the value is determined. The value can be entered directly into the Value field, the Expression field, or can be the result of an expression formula. Each type of entry produces a different unit display, as described below.

Value Field Entry: When you modify a constraint's value by typing a new value directly into the Value field, that entry remains exactly what you have entered. PCS does not append the assigned unit to the entered quantifier in the value field nor does it try to convert the value to the value assigned to the unit.

Expression Field Entry: An alphanumeric value can be entered directly into the Expression field and the Constraint Editor will convert the entry into the constraint's assigned unit.

Expression Formula: A constraint's value can be the result of an expression based on other constraint values. When PCS displays the outcome of the expression, its unit becomes the assigned unit for the target constraint.

If there is a mismatch between the expression's evaluated unit and the assigned unit, an error message appears in the Value field. See *Appendix B, PCS References, (page 69),* for an explanation of the PCS error messages.

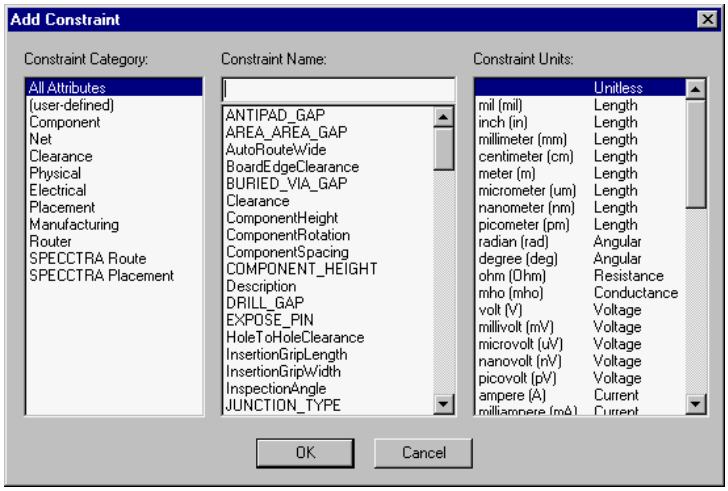
Evaluating Expressions describes in detail how PCS handles unit conversions and display.

Adding Constraints

A new constraint may be added to one or more selected items in the Precedence View using the **Add Constraint** command in the right mouse menu. For instance, in a PCB design with a room, you can organize the components that reside in the room in the Design Manager View's Rooms Page, and then add constraints to that entire group of components in the Precedence View.

A unit is assigned to every predefined constraint in the design. Constraints that are automatically assigned to the design (such as the clearance constraints in the Layer level) are given the unit chosen as your default in the *Options Configure* dialog of the originating application.

When you choose the **Add Constraint** command, the *Add Constraint* dialog appears as follows:



For each constraint category, you can select the constraint and its unit. Units are unique to the type of constraint chosen. For example, when adding a clearance type constraint the unit selection is Length, while for a component rotation constraint type the unit is Angular.

Constraint Categories and Types

The *Add Constraint* dialog lists all constraint categories and, within each category, the types of constraints available in a design. Any of these constraints can be added to the loaded design. When saved, additions and changes made to design constraints are committed to the design file.

Removing Constraints

Any selected constraint that you have added to the design can be removed by using the **Remove Constraints** command in the right mouse menu.

> The predefined Layer constraints: PadToPad, PadToLine, PadToVia, LineToVia, LineToLine and ViaToVia cannot be removed.

If you have selected one or more items in the Precedence View, and then choose the **Remove Constraints** command, all the constraints for that item are removed.

In the Constraint View, only the constraints attached to the selected items are removed with the **Remove Constraints** command.

Constraint Reports

The PCS provides reports that sort your design constraints in a variety of ways: by constraint domain, precedence level, net class, class-to-class and partitions.

File Commands, (page 57), has instructions on producing the reports, along with an explanation of the information each contains.

Evaluating Expressions

With the Constraint Editor you can define a constraint's value in a number of ways:

- By entering a value in the Value field.
- By entering a value in the Expression field.
- By entering a formula in the Expression field that uses constant values, functions or references to other constraints' values. For example, you can assign a single clearance constraint at the design level. Then, a constraint added to a different precedence level (e.g., a single component) can reference the design level constraint in the expression field. Whenever the design level constraint's value is changed, the component's constraint is evaluated and reflects the resulting value.

The Constraint Editor checks each constraint's expression for proper syntax and correct usage of assigned units. It also converts units and displays error messages where necessary.

Using Data Types in Expressions

Every constraint in a design uses a specific data type. For constraints like clearances and widths, the data type is Length. Other data types include such types as string, boolean, current and many more.

A correctly assigned data type is important when the Constraint Editor is evaluating expressions and converting units to produce a constraint value. You cannot mix data types in an expression. For instance, if your expression is a simple addition of a data type of Length, both sides of the operand must be numeric and contain one of the measurements in the data type such as mil and inch (e.g., 10mil + 0.5in). PCS displays an error if, for instance, you attempt to add a Length data type and a Resistance data type.

The complete list of data types, the units supported in each, and their unit conversions can be found in *Appendix B, PCS References, (page 69)*.

Unit Conversions

When there is a difference between the assigned unit and a unit resulting from an expression, the Constraint Editor attempts to convert the expression's resulting unit to an equivalent value in the assigned unit. For example, you may have a clearance constraint whose assigned unit is inch. If you type a value of 2mm in the expression field and press Enter, the value field displays the equivalent measurement of 0.0787402inch.

Appendix B, PCS References, (page 69), contains a table of unit conversions provided in the Constraint Editor.

Unitless Expressions

You can assign a unit of Unitless to a user-defined constraint. With a Unitless constraint it is possible to allocate a simple number without a unit in the value field, and it will not be subject to a conversion.

Strings and Evaluations

A unit type of String applies to such constraints as description and part number. The String unit allows a combination of alphanumeric characters to be entered verbatim in the Value field.

PCS will, when evaluating an expression, attempt to resolve the expression to a known value that can be applied as the constraint value. Any alphanumeric string entered as part of an expression is subject to this evaluation, and resulting conversion errors, if any.

You can use string constraints in other string constraint expressions. For instance, you may define a component string constraint called `refdes2` and enter "`refdes`" in its expression field. When evaluated, the value of the refdes2 constraint is assigned to the reference designator of the component.

Expression Errors

PCS checks expression formulas for correct application of functions, operators, units and constraint references. When the Constraint Editor evaluates an expression and encounters an illegal situation, an error message appears in the Value field.

The errors you may encounter include:

- Syntax Error.
- Divide by Zero.
- Operands type mismatch.
- <Operator> units mismatch
- Function <function> math error.
- Not a number.
- Unknown identifier <name>.
- Undefined identifier <name>.
- Array index <index> out of bounds.

The complete description of each of the PCS error messages can be found in *Appendix B, PCS References, (page 69)*, along with examples and possible solutions.

Constraint Validation and Design Updates

Whenever you are working with your design, whether in PCS, P-CAD PCB or P-CAD Schematic, the changes you make, when saved, are applied to a single design file.

If a design has expression formulas, the expressions are evaluated for mismatched values at the time the design is loaded into PCS.

If you have turned on the ECO recorder, the design updates become available for subsequent transfer between PCB and Schematic as a P-CAD `.eco` file.

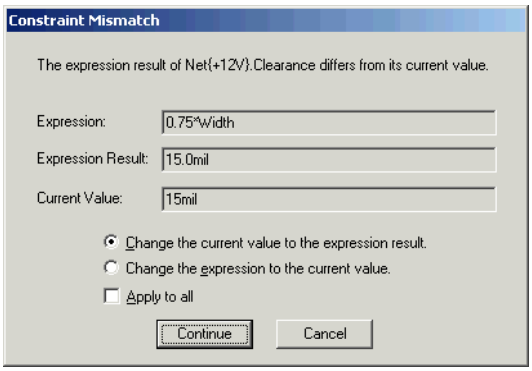
Expression Validation at Design Load

At the time a design is loaded into PCS existing expressions are evaluated. The values produced by those evaluations are compared with the attribute values for the same constraints in the PCB design. If the two values do not match, the *Constraint Mismatch Error* dialog provides the chance to choose the correct value.

For example, the following scenario shows how a mismatch occurs during a series of updates:

1. The last time your design was loaded into PCS, a Design Level Clearance constraint was added. The Clearance constraint has an expression of 0.75 * Width. The Design Level Width constraint value is 20mil. When evaluated, the Clearance constraint is 15mil (0.75 * 20mil).
2. Later, the Design Level Width constraint value attribute is changed in P-CAD PCB to 15mil.
3. The design is loaded into PCS. At the time PCS loads the file the expression for the Clearance constraint is again evaluated. The expression 0.75 * Width hasn't changed, but since the value of the Width constraint is now 15mil, the expression (0.75 * (Width(15mil)) evaluates to a value of 11.25mil.

The *Constraint Mismatch* dialog, with the described error, appears as follows:



The message at the top of the dialog includes the name of the design constraint in conflict, in this case the Design Level Clearance constraint. The following information is displayed as well:

- **Formula**: This is the expression defined for the constraint.
- **Formula Result**: This is the result of the expression (i.e., Value field entry) in the Constraint Editor.
- **Current Value**: This is the value currently assigned to the constraint in PCB.

  Based on the information supplied in the *Constraint Mismatch* dialog, you may choose one of the following actions:

- **Change the current value to the formula result**: This option changes the value of the constraint in PCB to match the value of the evaluated constraint in PCS.

- **Change the formula to the current value.** This option assigns the value of the constraint in PCB to the expression and value in the PCS constraint, and drops the original expression.

4. Once you have chosen the desired change option, you can apply that option to all of the detected mismatches by enabling the **Apply to all** box. If this option is not checked, a *Constraint Mismatch* dialog is presented for each error and you can choose an individual, appropriate action for each mismatch.

Saving Design Updates

Design updates made in the originating PCB or Schematic application are dynamically applied to the design file. To save the changes you have made to the design in PCS use the **File » Update PCB** or **File » Update Schematic** commands or click the **File » Save Design** button to save the changes to the PCB or Schematic design file memory. With any of these commands, the file remains open in PCS so you can continue working on it until you exit PCS.

If the DBX link has been broken between the originating application and the Parametric Constraint Solver, when you try to update the design, you will see a message saying that the DBX connection has been broken and you must confirm that you want the connection reestablished. PCS will validate that the design name and RefDes' in both applications are the same.

Additionally, PCS checks the values of the predefined layer clearance constraints. If any value is less than zero, an error message is displayed and you must choose to continue the update or cancel it. If you choose to continue the update the illegal value will be ignored, leaving the value with its last valid entry.

If you have finished working with the current design in PCS, choose **File » Close**. This command updates the design when you confirm that you want to save the changes and closes the Parametric Constraint Solver views. The design is unlocked in the originating P-CAD application, permitting changes again in the current PCB or Schematic design editor. When you save the design or exit the originating application, all changes to the design are applied.

When you exit PCS and return to the originating application, make sure you do a **File » Save** in PCB or Schematic to apply the PCS updates to the design. Updates made in PCS are saved in the memory of the originating application only until you choose the **File » Save** command or confirm that you want the updates saved on exiting the originating application.

Design update options are described in *Parametric Constraint Solver Basics, (page 7)*.

How PCS Updates Affect PCB and Schematic

Updates made to a design in PCS are saved to the original PCB or Schematic design file when a file save occurs. These changes, once committed to the design file, can become part of an Engineering Change Order if the ECO recorder is turned on when the updates are committed to the file, and subsequently applied to a matching design in PCB or Schematic.

Features unique to PCS, such as formulas and comments, are included in the design file and preserved by PCB and Schematic. They cannot, however, be viewed in PCB or Schematic.

Recording Changes with ECOs

If you have turned on the ECO recorder in your originating application before making changes to the design in PCS, the changes will be recorded in the `.eco` file when the design is saved. Those of you familiar with the `.was` file should note that since only RefDes changes are recorded here, and you cannot change the RefDes in PCS, the Parametric Constraint Solver will make no updates to this file.

Once you exit PCS, you can import or export from PCB or Schematic the changes recorded in the ECO file, just as you would have done if those changes had been made in PCB or Schematic.

Make sure to turn on the ECO recorder before launching PCS and loading the design.

If you should forget to turn on the ECO recorder, you can still record the changes by following these steps:

1. Return to the originating application.
2. Terminate the link between the applications by **Cancel** button on the *P-CAD DBX* dialog, shown below:

3. Turn on the ECO recorder using the **Utils** » **Record ECOs** command or by clicking the  button.
4. Return to PCS and update the design using the **File** » **Update PCB/Schematic Design** command.
5. The error message dialog informs you that the connection has been terminated. Click the **OK** button to continue.

6. In the next *DBX Error* dialog you must choose whether to re-establish the connection and continue the update process or cancel it.

7. If you wish to continue the update, click the **OK** button. P-CAD DBX is automatically re-activated and the link between the applications re-established. The changes saved to the file in PCS are also saved to the ECO recorder and can then be exported to PCB or Schematic.

If you chose either the **File » Close** or **File » Exit** commands in step 4, and you click the **Cancel** button in the *DBX Error* dialog, your design is erased from the PCS workspace and the changes you made are not recorded in the design file.

Using a Design Technology Parameters File

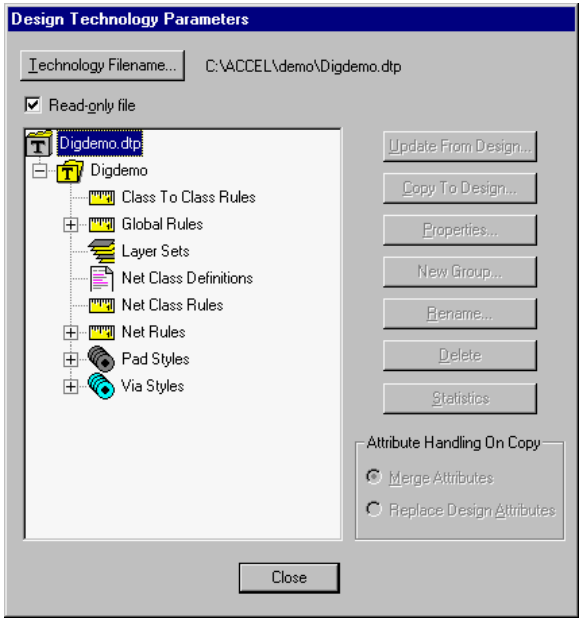
When a design is loaded into the Parametric Constraint Solver the existing constraints are included and displayed in the workspace. If you have a Design Technology Parameters file containing constraints that have proven successful in previous designs, you can load that file into the Parametric Constraint Solver and copy pertinent attributes into the current design.

This section describes which of the features of the Design Technology Parameters file are available to the design while in the Parametric Constraint Solver.

For more complete information on the Design Technology Parameters file, see your *PCB User's Guide* or *Schematic User's Guide*.

Accessing a DTP

To access a Design Technology Parameters file, choose the **File » Design Technology Parameters** command. The *Design Technology Parameters* dialog, shown on the next page, appears:



DTP Operations

The following list shows which operations are available in PCS from the *Design Technology Parameters* dialog:

- Update From Design
- Copy To Design
- Properties
- New Section or Item
- Rename
- Delete
- Statistics

For each of the categories in the file, a particular set of operations is allowed. In all cases, you can **Update From Design** and **Copy To Design**, as well as **Delete** items in each category. In addition to those basic functions, the following additional operations are available in each of the categories:

- **Class-to-Class Rules**: In this category you can add a New Section or item.
- **Global Rules** (Layers): For the entire category you can add a New Section or item. For an individual layer in this category you can access the Properties and Statistics, as well as Rename an item.

- **Net Class Definitions**: In this category a New Section or item can be added.
- **Net Class Rules**: In this category a New Section or item can be added.
- **Net Rules**: A New Section or item can be added to the category. The Properties and Statistics are accessible for individual nets, which can also be Renamed.

The Attribute Handling On Copy section of the *Design Technology Parameters* dialog is enabled for the Copy to Design function for selected Class-to-Class Rules, Net Class Rules, Net Rules or individual nets. You may choose to Merge Attributes in the design with those in the DTP file, or Replace Design Attributes with those in the DTP file by clicking the appropriate radio button.

Refer to the *File Commands* chapter of the *PCB User's Guide* for more information on how the Merge Attributes function behaves in each of the relevant categories.

Viewing Parametric Constraint Solver Features in PCB and Schematic

Some of the unique Parametric Constraint Solver features become part of the design file, and are viewable in the originating application as described below:

- **Values**: The value given to a constraint in the Parametric Constraint Solver can be the result of an entry in the Value field, an entry in the Expression field, or the evaluation of an expression. Whatever the entry method, the subsequent content of the Value field becomes the value in all applications for the constraint.
- **Formulas**: Parametric Constraint Solver provides a method of determining the value for a constraint by using a formula. When a formula entered in the Expression field of the Constraint Editor is evaluated, the result becomes the entry in the Value field. If the evaluation results in an error, the error is entered in the Value field of the constraint in PCS, and is also displayed in the same field in the originating application for that constraint.

The P-CAD Parametric Constraint Solver permits you to attach comments to the constraints in your design and assign expressions to determine constraint values. PCB and Schematic do not display the comments or expressions, but they are easily accessible in the Parametric Constraint Solver Reports. For more information on reports, see *File Commands, (page 57)*.

CHAPTER 8

Using PCS with InterPlace

While you can use the P-CAD Parametric Constraint Solver as a stand alone application, its power is dramatically increased when the P-CAD InterPlace (IPL) application is running along side it. In addition to being able to organize the design objects and constraints in PCS, you can use the placement features in IPL to increase the effectiveness of the design.

The Layout View in P-CAD InterPlace provides the graphic representation of the circuit board for a PCB design. Manipulation of the design objects, including controlling design layers, using rooms, moving components, and viewing connections are just a small part of the productivity enhancing tools that are available with InterPlace. InterPlace also provides a Visible Placement Area tool which shows an area on the board, based on constraints, where a component can be placed without incurring errors.

If you have purchased InterPlace, refer to the *InterPlace User's Guide* to learn about its basic functionality, which includes all of the functions described in this chapter.

Highlighting Objects

The Layout View provides the means to highlight a number of objects using the right mouse commands from the Parametric Constraint Solver's Precedence View.

When net connections are visible, you can highlight individual nets, the nets assigned to a net class or class-to-class, or the nets attached to selected components. You can also highlight a room and its components, when a room exists in the design.

Highlighting options are described in the following sections.

Net Highlighting

There are several methods you can use to highlight the nets in your design. Each highlighting method is described below:

> You must enable the visibility of the net connections before they can be highlighted. See *The Layout View* in the *InterPlace User's Guide* for more information on showing net connections.

To highlight a single net: Select a net item in the Net category of the precedence tree. Choose the **Highlight** command in the right mouse menu.

To highlight the nets in a net class: Select the desired net class in the net class category of the precedence tree. Choose the **Highlight** command in the right mouse menu.

To highlight the nets attached to components: Select the desired components from the Component category list in the precedence tree. Choose the **Highlight Attached Nets** command in the right mouse menu.

To highlight the net classes in a class-to-class: Select the desired class-to-class from the Class-to-Class category in the precedence tree. Choose the **Highlight** command in the right mouse menu.

To highlight a group of nets: Select the desired Nets, Net Class or Class-to-Class items in their respective pages and choose **Highlight** from the right mouse menu. For instance, in the Net Class page you can select the Net Class instead of having to select the nets in the Net Class and then highlight them.

To remove highlights: To remove highlighting from any highlighted object, select the object and from the right mouse menu choose the appropriate **Unhighlight** command.

Room Highlighting

When rooms exist in a PCB design, and components have been assigned to and placed in them, you can highlight the room and its components.

To highlight rooms and assigned components: Select the desired room(s) in the Rooms Page of the Design Manager View and choose the **Highlight Room and Assigned Components** command from the right mouse menu.

To remove highlights: To remove highlighting from any highlighted object, select the object and from the right mouse menu choose the appropriate **Unhighlight** command.

Component Highlighting

Components and their attached nets can be highlighted in the Layout View from the Parametric Constraint Solver's Precedence View.

To highlight one or more components: Select the desired components in the PCS Precedence View component category. From the right mouse menu choose the **Highlight Component** command.

To highlight the nets attached to a component: First make sure that the display of the net connections is enabled, then select the desired components in the PCS Precedence View component category. From the right mouse menu, choose the **Highlight Attached Nets** command.

To remove highlights: To remove highlighting from any highlighted object, select the object and from the right mouse menu choose the appropriate **Unhighlight** command.

Component Selection

In addition to being able to highlight components and nets, the components in the design can be selected from the Precedence View in PCS. Once selected, you can perform any of the functions available to selected objects such as moving, fixing, rotating, clustering, etc. All of these functions are explained in detail in *The Layout View*, in your *InterPlace User's Guide*.

To place component(s) in select mode: Select the desired component(s) from the list of components in the Precedence View tree and choose the **Select Component** command from the right mouse menu.

To place components in a net in select mode: Select the desired net from the list of Nets in the Precedence View tree and choose the **Select Net Components** command from the right mouse menu.

Clustering Components

Locating one or more specific components in a design with a large quantity of components can be almost effortless when you use the **Cluster By** command. The Cluster tool finds designated components and lets you place them in a chosen location arranged by the property of your choice, such as type, pin count, package size, etc. Once clustered they can easily be placed on the board.

The **Cluster By** command is available on every menu in the Layout View and the Design Manager View. In addition, the **Cluster By** command appears in the right mouse menu of the PCS Precedence View when one or more items in the Net, Net Class, Room or Component categories are selected.

The *InterPlace User's Guide* provides detailed information on how to use the clustering tool in *Organizing and Placing Components*.

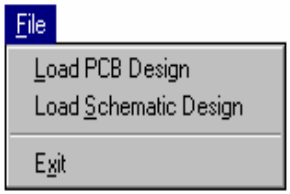
Jumping to a Component's Location

If you need to find a component and do not know its exact location, InterPlace can locate it for you.

To jump to a component: Select a single component from the component category in the PCS Precedence View tree and from the right mouse menu choose the Jump To Component command. The selected component is highlighted in the Layout View and the cursor is positioned over the selected component. If the data tips are enabled, the component's reference designator, type and value are displayed as well.

CHAPTER 9

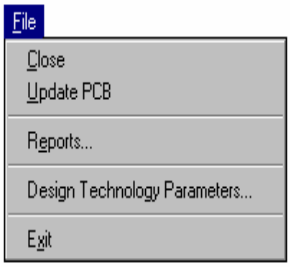
File Commands



The File commands allow you to load, close, and update designs with their design constraints into P-CAD Parametric Constraint Solver. From the File menu, you can also exit the application.

The **File** menu displayed in the margin, containing the **File » Load PCB Design** and **File » Load Schematic Design** commands, is available when a design is yet to be loaded. An expanded **File** menu is available once the PCB or Schematic design has been loaded into the Parametric Constraint Solver. See the following pages for details.

File Load PCB Design



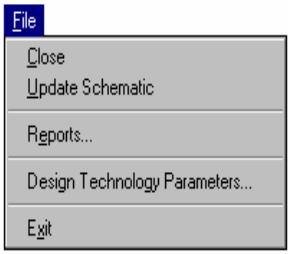
The **File » Load PCB Design** command loads an existing PCB design file into the P-CAD Parametric Constraint Solver.

Before a design has been loaded, the **File** menu appears as shown in the chapter introduction. When a design is loaded, the **File** menu changes, as shown in the margin, to include the additional commands: Close, Update PCB, Reports, Design Technology Parameters and Exit.

To load the design into Parametric Constraint Solver, the design must be currently open in P-CAD PCB or P-CAD Relay. When you choose **Load PCB Design**, the design that is active in the P-CAD application is automatically loaded into the Parametric Constraint Solver workspace. The design will be locked in the originating application while accessed through Parametric Constraint Solver to prevent conflicting modifications.

After loading, the Design Manager View organization tools become available. Refer to *The Design Manager View, (page 15)*, for more information. In addition, the Constraint Editor View, detailed in *The Constraint Editor View, (page 31)*, is accessible as well.

File Load Schematic Design



The **File » Load Schematic Design** command loads an existing Schematic design file into P-CAD Parametric Constraint Solver. If a design is already loaded, this command is unavailable.

Before a design has been loaded, the File menu appears as shown in the chapter introduction. When a design is loaded, the *File* menu changes, as shown in the margin, to include the additional commands: Close, Update Schematic, Reports, Design Technology Parameters and Exit.

To load the design into the Parametric Constraint Solver, the design must be currently open in P-CAD Schematic. When you choose **Load Schematic Design**, the design that is active in the P-CAD application is automatically loaded into the Parametric Constraint Solver workspace. The design will be locked in the originating application while accessed through Parametric Constraint Solver to prevent conflicting modifications.

After loading, the Design Manager View organization tools become available. Refer to *The Design Manager View, (page 15),* for more information. In addition, the Constraint Editor View, detailed in *The Constraint Editor View, (page 31),* is accessible as well.

File Close

Closes all views for the design in the Parametric Constraint Solver. The design is unlocked in the originating application, permitting changes again in the PCB or Schematic design editor.

If the design has been changed but not yet saved, you are asked whether or not you want to save your changes before closing.

File Update PCB

Saves the changes to the PCB or Relay design. When you select **File » Update PCB**, the file remains open so you can continue working on it in the Parametric Constraint Solver.

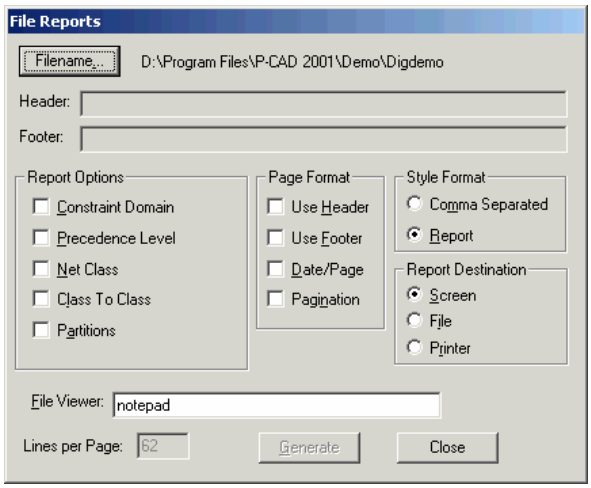
File Update Schematic

Saves the changes to the Schematic design file. When you select **File » Update Schematic**, the file remains open so you can continue working on it in the Parametric Constraint Solver.

File Reports

Allows you to generate reports with specific output options. The selected Style Format dictates the options available for each of the styles: Comma Separated and Report. With the Report style, in addition to the Report Options, Lines per Page and Report Destination, you may select the page format and define the Header and Footer. These options are saved when you exit the program.

The dialog appears as follows:



Filename

You may specify the report or reports you wish to generate from the list of report types listed in the Report Options area. Each report type has its own, unique file extension; the filename defaults to the name of the current design.

Report File Extensions

The extensions are set as listed below and cannot be changed. The files are all saved as text files.

- constraint_domain.txt
- precedence_level.txt
- net_classes.txt
- class_to_class.txt
- partitions.txt.

Report Options

The reports are described below:

Constraint Domain: The Constraint Domain report lists the three domains pertinent to placement constraint management: Physical, Electrical, and Room. Within each domain, in their precedence order, are each item's constraint names, values, expressions and comments.

Precedence Level: The Precedence Level report shows, for each precedence level and item, the constraint names, values, expressions and comments.

Net Class: The Net Class report shows the nets that are assigned to each net class in the design.

Class-to-Class: The Class-to-Class report shows the net classes assigned to each class-to-class and the class-to-class constraints.

Partitions: The Partitions report lists the components assigned to each Partition.

Page Format

These Page Format options are enabled when the Report format is selected:

- Use Header and Use Footer include the information you specified in the header and footer dialog fields.
- Date/Page includes the current date and the page number.
- Pagination allows you to create your own pagination (lines per page). When this option is enabled print from the **DOS Print** command, not the Notepad.

Style Format

Comma Separated puts all data in comma-separated format. This format can be imported into other spreadsheet and database programs.

Report produces a report format with columns and spaces, etc.

Lines per Page

Lines per Page allows you to specify the number of lines per page in your report.

Report Destination

Screen sends the output to a file and opens the file using the Notepad Utility.

File sends the output to a file.

Printer sends output directly to the printer without creating files.

Generate

Generates all selected reports using the options selected.

File Design Technology Parameters

Captures design data, including design constraints, and stores this data in a design technology parameters file. This data can be used in any P-CAD design. Thus this file becomes a template from which you can selectively use the captured design data in any design.

A design technology parameters file is like a storage box. You can reorganize it, add or remove items, alter some of its contents, and use what is stored inside. For a project or series of projects, you may have several items that you will have to use repeatedly for creating PCB or Schematic designs. The design technology parameters file is a perfect storage box for the following design items: class-to-class rules, global rules, layer sets, net class definitions, net class rules, net rules and pad and via styles.

Design constraints, or rules, can be transferred using a design technology parameter file between the Parametric Constraint Solver and any PCB or Schematic design.

In the P-CAD Parametric Constraint Solver, Design Technology Parameters can be used in a variety of ways, including:

- Applying company-standard design rules to your board layout.
- Storing design rules created or organized in Parametric Constraint Solver for later use in other PCB or Schematic designs. You can create or change design rules pertaining to Class-to-Class Rules, Layer Rules, Net Class Definitions, Net Class Rules and Net Rules in Parametric Constraint Solver.

For more information, see *Using a Design Technology Parameters File, (page 49)*.

File Exit

Exits the Parametric Constraint Solver program.

If the design has been modified since the last save, you are prompted whether you want to save the changes.

The design file is unlocked in the originating application, permitting changes again in the PCB or Schematic design editor.

The program writes information to the `.ini` file when you choose **Exit**. This information, which will apply to subsequent PCS sessions, consists of parameters and settings such as values set in Options Configure, etc.

CHAPTER 10

View Commands

View commands allow you to temporarily alter your view of the workspace. You can enable or disable the display of the Design Manager View or the Constraint Editor View.

The *View* menu of Parametric Constraint Solver, for a design loaded from either PCB or Schematic, is shown in the left margin.

View Constraint Editor

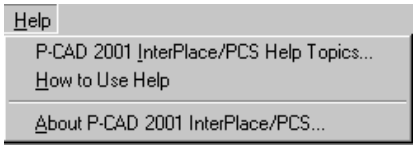
Displays the Constraint Editor View. In the Constraint Editor, you can define and organize all design constraints. For more information, see *The Constraint Editor View, (page 31)*.

View Design Manager

Displays the Design Manager View. In the Design Manager, you can organize design components into logical partitions or physical rooms. These rooms and partitions can be used for grouping unplaced components, as well as placing them in a common board region. For more information, see *The Design Manager View, (page 15)*.

CHAPTER 11

Help Commands



The P-CAD Parametric Constraint Solver provides extensive online help. The online help contains reference information, but differs from the manual in its format and accessibility.

The main advantages to online help are the availability of hypertext links between related subjects (an electronic cross-reference utility), and the keyword search function (an electronic index).

Help P-CAD Parametric Constraint Solver Help Topics

Displays the P-CAD Parametric Constraint Solver online help including the **Contents** tab, which is structured to match the order of commands as they appear in the product, and the **Index** tab, which lets you look up a specific concept or keyword.

How to Use Help

Connects you to the Windows help system, where instructions on how to use the help system are provided.

About P-CAD Parametric Constraint Solver

Displays a dialog that contains information such as the product version number, release date, memory used, memory available, and license number.

APPENDIX A

Keyboard Reference

This chapter is a reference of commands and functions accessed through P-CAD shortcut keys and standard Windows accelerators.

Standard Windows key combinations are functional for all of the menu commands; use the normal combination **Alt**,**x**,**y**, where x equals the underlined menu character, and y equals the underlined command character.

The Parametric Constraint Solver Keyboard Reference section includes general keystrokes and those specific to P-CAD Parametric Constraint Solver.

Parametric Constraint Solver Keyboard Reference

Alt+**F4** (File Exit)	Shortcut for **File Exit**, which exits the P-CAD program. If the current design has been modified since the last save, you will be prompted (YES or NO) as to whether you want to save the changes to the file. The program will write information to the `.ini` file when you exit.
arrow keys	A directional arrow key moves the cursor in the Constraint Editor to the constraint above or below, or to the field to the left or right of the starting position. In the Design Manager View only the up and down arrows are operational.
Ctrl+**S** (File Update PCB)	Saves changes to the current design without closing it.
F1 (Help)	Displays context-sensitive help. If you put focus on a command or dialog and press **F1**, the Help window appears containing information specific to the focus item.
Shift + **End**	Selects all items in a list on the Design Manager pages from the first one selected to the end of the list.

APPENDIX B

PCS References

This appendix contains references pertaining to the Parametric Constraint Solver field entries, primarily the Expression field.

Included are tables:

- Grammar
- Data Types
- Postfix mode
- Functions
- Operators
- Units
- Constants
- Constraint Referencing
- Sample formulas
- Error Messages
- Limitations.

The Constraint Editor View, (page 31), provides detailed information on how these references are utilized in the Constraint View.

PCS Grammar

The Parametric Constraint Solver understands specific use of variables, operators, special characters and phrases when evaluating an expression. The notations shown below will help you understand the PCS grammatical rules.

Notations

The following notations are used to describe the PCS grammar.

Notations	Definition
‘+’	Items in single quotes are literals.
[ ] ?	Zero or one occurrences of item in brackets.
[ ] +	One or more occurrences of item in brackets.
[ ] *	Zero or more occurrences of item in brackets.
|	Or
<Name>	A string of characters (e.g., Width)
<Integer>	An integer number (e.g., 123)
<Float>	A floating point number (e.g., 1.23e-3)
<Unit>	A unit abbreviation (e.g., pF)

Grammar

The following table shows the list of allowable syntax usage. Any entry in the second column can be used.

Term	Allowable syntax
expression	expr booleanExpr vector
expr	‘postfix’ exprpf ‘-’ expr expr ‘+’ expr expr ‘*’ expr expr ‘/’ expr expr ‘%’ expr expr ‘!’ ‘(’ expr ‘)’ number

	constraint constraint ‘[‘ expr ‘]’ ‘size’ ‘(‘ constraint ‘)’ ‘if’ booleanExpr ‘then’ expr ‘else’ expr ‘abs’ ‘(‘ expr ‘)’ ‘acos’ ‘(‘ expr ‘)’ ‘asin’ ‘(‘ expr ‘)’ ‘atan’ ‘(‘ expr ‘)’ ‘cos’ ‘(‘ expr ‘)’ ‘cosh’ ‘(‘ expr ‘)’ ‘exp’ ‘(‘ expr ‘)’ ‘int’ ‘(‘ expr ‘)’ ‘in’ ‘(‘ expr ‘)’ ‘log’ ‘(‘ expr ‘)’ ‘max’ ‘(‘ expr[,expr]* ‘)’ ‘min’ ‘(‘ expr[,expr]* ‘)’ ‘sin’ ‘(‘ expr ‘)’ ‘sinh’ ‘(‘ expr ‘)’ ‘sqrt’ ‘(‘ expr ‘)’ ‘sum’ ‘(‘ xpr[,expr]* ‘)’ ‘tan’ ‘(‘ expr ‘)’ ‘tanh’ ‘(‘ expr ‘)’
booleanExpr	‘yes’ | ‘no’ | ‘true’ | ‘false’ expr ‘and’ expr expr ‘or’ expr ‘not’ expr booleanExpr ‘and’ booleanExpr booleanExpr ‘or’ booleanExpr

	‘not’ booleanExpr ‘(‘ booleanExpr ‘)’ expr ‘<’ expr expr ‘>’ expr expr ‘<=’ expr expr ‘>=’ expr expr ‘=’ expr expr ‘=’ booleanExpr expr ‘!=’ expr expr ‘!=’ booleanExpr
constraint	name precedenceMember ‘.’ name
precedenceMember	‘Design’ precedenceLevel ‘{‘ member ‘}’
precedenceLevel	‘Layer’ ‘Net’ ‘NetClass’ ‘Room’ ‘Component’ ‘ClassToClass’
member	name
vector	[–] ?number[,[–]number]+
exprfp	number constraint exprpf exprpf ‘+’ exprpf exprpf ‘–’ exprpf exprpf ‘*’ exprpf exprpf ‘/’ exprpf exprpf ‘%’

	exprpf exprpf '^' exprpf '!' exprpf 'neg' exprpf 'abs' exprpf 'acos' exprpf 'asin' exprpf 'atan' exprpf 'cos' exprpf 'cosh' exprpf 'exp' exprpf 'int' exprpf 'in' exprpf 'log' exprpf 'sin' exprpf 'sinh' exprpf 'sqrt' exprpf 'tan' exprpf 'tanh'
number	<Integer>[unitSpec]? <Float>[unitSpec]? 'e' 'pi'
name	<Name> ' " ' <Name>
unitSpec	<Unit> '[' unitExpr ']'
unitExpr	<Unit> ['*' <Unit>] + ['/' <Unit> ['*' <Unit>] +]?
unit	<Unit> ['^' [–] ?Integer]

Data Types

The following data types are supported as a field entry or when used in an expression:

Type	Definition
Number	Any number with one of the following formats: 1000, 10.234, 10e2, 10e-2, 1.234e2 or 1.234e-2
Number with Units	Any number with a unit suffix, for example 1000mil, 3pW, 5[mA/s}
Boolean	True, False, Yes, No, 0 or 1
Vector	A series of comma-separated numbers, for example 1,2,3,4,5
String	Any text string

Data Types by Constraint

Every predefined constraint has allowable data types. The following table shows which data types are allowed for the constraint.

Constraint	Data Type
AutoRouteWide	Boolean
BoardEdgeClearance	Length
Clearance	Length
ComponentHeight	Length
ComponentRotation	Angular
ComponentSpacing	Length
Description	String
HoleToHoleClearance	Length
InsertionGripLength	Length
InsertionGripWidth	Length
InspectionAngle	Angular
LineToLineClearance	Length

Constraint	Data Type
Link	String
MaxComponentHeight	Length
MaxNetLength	Length
MinNetLength	Length
MaxVias	Quantity
NoAutoRoute	Boolean
NoSwap	String
Optimize	Boolean
PadToLineClearance	Length
PadToPadClearance	Length
PartNumber	String
PlacementSide	Layer
Ripup	Boolean
RoutingChannels	Quantity
Silkscreen Clearance	Length
SwapEquivalence	String
TieNet	String
ViaStyle	ViaStyle
ViaToLineClearance	Length
ViaToPadClearance	Length
ViaToViaClearance	Length
Width	Length

Postfix Mode

The keyword postfix starts a postfix expression. The postfix expression stack runs left to right (e.g., "postfix 2 3 + 5 /" is the same as "(2 + 3) / 5". A negative number must be noted using the reserved keyword neg. Do not use the minus sign. In the above example, if the number 3 is negative, the expression would read postfix 2 3neg + 5 / which is the same as "(2 + 3neg) / 5".

Functions

The following table describes which functions are supported in the Parametric Constraint Solver, along with the legal arguments and the expected results. Each function's legal argument contains an expression, which is any number, constant, function result, or known identifier. Unless otherwise stated, and expression may include any unit.

Function	Legal Arguments	Result	Postfix Mode
abs (x)	Expression	Absolute value of x.	Yes
acos (x)	Unitless expression ($-1 \le x \ge 1$)	Arccosine of x.	Yes
asin (x)	Unitless expression ($-1 \le x \ge 1$).	Arcsine of x.	Yes
atan (x)	Unitless expression ($-1 \le x \ge 1$).	Arctangent of x.	Yes
cos (x)	Rad, deg or unitless expression	Cosine of x.	Yes
cosh (x)	Unitless expression	Hyperbolic cosine of x.	Yes
In (x)	Unitless expression where $x > 0$	Natural logarithm of x.	Yes
log (x)	Unitless expression where $x > 0$	Log base 10 of x.	Yes
max (x1, x2, x3,…xn)	A series of expressions, which have the same units.	Maximum value in the series.	
min (x1, x2, x3,…xn)	A series of expressions, which have the same units.	Minimum value in the series.	
sin (x)	Rad, deg or unitless expression	Sine of x.	Yes
sinh (x)	Unitless expression	Hyperbolic sine of x.	Yes
size (v)	A vector name	Number of elements in the vector v.	
sqrt (x)	Expression ($x \ge 0$).	Square root of x.	Yes
sum (x1, x2, x3,…xn)	A series of expressions with all the same units.	Summation of the series.	
tan (x)	rad, deg or unitless expression.	Tangent of x.	Yes
tanh (x)	Unitless expression	Hyperbolic tangent of x.	Yes

Function	Legal Arguments	Result	Postfix Mode
if x then r1 else r2	x is Boolean expression, r1 and r2 are expressions.	If x is true then return r1. If x is false return r2.	

Operators

The following table of operators are those supported in PCS. Binary means the operator takes both a left and a right hand side. Unary means the operator only has a left hand side. Unit restrictions list how the units on the left and right hand side must be used. Unit category is the "type" of unit, such as length, current, time, power, etc. typically, linear combinations of units must have the same category.

Operator	Explanation	Unit Restrictions	Postfix Mode
x + y	Binary addition.	Units must be same category.	Yes
x – y	Binary subtraction.	Units must be same category.	Yes
– y	Unary negation.		Yes
x / y	Binary division.		Yes
x * y	Binary multiplication.		Yes
x % y	Binary modulo. Remainder of division ($x \neq 0$).	y must be unitless.	Yes
x!	x Factorial.	x must be unitless and $0 > x < 20$.	Yes
x ^ y	Binary power ($y \geq 0$).	y must be unitless.	Yes
x < y	Binary less than.		
x > y	Binary greater than.		
x <= y	Binary less than or equal to.		
x >= y	Binary greater than or equal to.		
x = y	Binary equal to.		
x != y	Binary not equal to.		

Operator	Explanation	Unit Restrictions	Postfix Mode
[x]	Vector index. x is a unitless expression.	x must be unitless.	
x and y	Boolean AND.	x and y must be Boolean.	
x or y	Boolean OR.	x and y must be Boolean.	
not x	Boolean NOT.	x and y must be Boolean.	

Supported Units

The following table shows the units supported in PCS along with applicable conversions.

Unit	Abbreviation	Conversion From	Data Type
mil	mil		Length
inch	in		Length
millimeter	mm		Length
centimeter	cm		Length
meter	m		Length
micrometer	um		Length
nanometer	nm		Length
picometer	pm		Length
radian	rad	degree	Angular
degree	deg	radian	Angular
Ohm	Ohm	Volt / Ampere	Resistance
Mho	mho		Conductance
Volt	V	Ampere * Ohm	Voltage
millivolt	mV	Ampere * Ohm	Voltage
microvolt	uV	Ampere * Ohm	Voltage
nanovolt	nV	Ampere * Ohm	Voltage

Unit	Abbreviation	Conversion From	Data Type
picovolt	pV	Ampere * Ohm	Voltage
Ampere	A	Volt / Ohm	Current
milliampere	mA	Volt / Ohm	Current
microampere	uA	Volt / Ohm	Current
nanoampere	nA	Volt / Ohm	Current
picoampere	pA	Volt / Ohm	Current
Henry	H		Inductance
millihenry	mH		Inductance
microhenry	uH		Inductance
nanohenry	nH		Inductance
picohenry	pH		Inductance
Farad	F		Capacitance
millifarad	mF		Capacitance
microfarad	uF		Capacitance
nanofarad	nF		Capacitance
picofarad	pF		Capacitance
second	s		Time
millisecond	ms		Time
microsecond	us		Time
nanosecond	ns		Time
picosecond	ps		Time
Hertz	Hz		Frequency
kilohertz	KHz		Frequency
megahertz	MHz		Frequency
Watt	W		Power
milliwatt	mW	Volt * Ampere	Power
microwatt	uW	Volt * Ampere	Power
nanowatt	nW	Volt * Ampere	Power

Unit	Abbreviation	Conversion From	Data Type
picowatt	pW.	Volt * Ampere	Power
Boolean	bool		Boolean
string	string		string
layername	layer		layer
viastyle	viastyle		viastyle

Constants

The following are predefined constants:

Constant	Unit
e	Unitless
pi	Unitless
True	Boolean
False	Boolean
Yes	Boolean
No	Boolean

Constraint Referencing

Referencing other constraints is done in two ways: 1) if the constraint is in the same precedence member (e.g. a net), then use the name of the constraint (e.g., “Width”), 2) if the constraint is in another precedence level, use one of the fully qualified constraint name formulas:

- Design.constraint-name
- Layer{layer-name}.constraint-name
- NetClass{net-class-name}.constraint-name
- Net{net-name}.constraint-name
- ClassToClass{“net-class-name net-class-name”}.constraint name
- Room{room-name}.constraint-name
- Component{component-name}.constraint-name

Spaces are optional except for ClassToClass where quotes around and a space between the net class names are required. Constraint referencing formulas are not case sensitive.

Reserved Keywords

The following keywords are reserved and cannot be used as constraint names:

abs		e	neg	sinh
acos	else	Net	size	
and		false	NetClass	sqrt
asin	for	no	sum	
atan	if	not	tan	
ClassToClass	ln	or	tanh	
Component	Layer	pi	then	
Cos		log	postfix	true
Cosh	max	Room	while	
Default	min	sin	yes	
Design				

Limits

The limits supported by PCS are:

Integer (64 bit internal storage) -9223372936854775807 to +9223372036854775807

Integer (32 bit internal storage and only when the unit is quantity) 2147483647

Floating Point (IEEE standard)

Minimum value: 2.2250738585072014E – 308

Maximum value: 1.7976931348623158E + 308

Syntax Examples

The following table shows sample expressions for different types of constraints and the expected results. Notice that, for each section of the table, there are constraint expressions based on the values of other constraints, some at different levels of the precedence hierarchy.

For each example the targeted constraint value is displayed in bold letters. Those constraints that are not bold are those that supply values to the target constraint's expression.

EXAMPLE #1

In this example the value of the constraint Dist is the sum of the values of the thickness constraints from the Top Layer and the INT1 Layer.

Constraint	Precedence Level	Expression	Results
Dist	Design	Layer(Top).thickness + Layer{INT1}.thickness	160.0mil
thickness	Top Layer	.08inch	80.0mil
thickness	INT1 Layer	.08inch	80.0mil

EXAMPLE #2

In this example the value of the constraint width in the –5v net is the value of the width constraint in the mux1 net multiplied by 2.

Constraint	Precedence Level	Expression	Results
width	-5v Net	Net{mux1}.width*2	30.0mil
width	mux1 Net	15	15mil

EXAMPLE #3

In this example the value of the constraint width in the Top Layer is determined by using the Dist and Cu_Height constraints in the Top Layer and the B constraint in the Design level. In addition, as you can see, each expression references another constraint at a different level in the precedence hierarchy.

Constraint	Precedence Level	Expression	Results
width	Top Layer	int(((5.98*Dist)/Design.B)–(Cu_Height)/8)	55.0mil
Dist	Top Layer	Layer{Top}.thickness+ Layer{INT1}.thickness	160.0mil
thickness	Top Layer	.08inch	80.0mil
thickness	INT1 Layer	.08inch	80.0mil
B	Design	(Impedance*(sqrt(1.41+ Diel_Const))/87Ohm)^10	17.1076
Impedance	Design	60	60Ohm

Constraint	Precedence Level	Expression	Results
Diel_Const	Design	1.9	1.9
Cu_Height	Top Layer	2mil	2.0mil

EXAMPLE #4

In this example the value of the constraint MaxNetLength for the A80 net is determined using the RiseTime constraint's value from the Design level of the hierarchy.

Constraint	Precedence Level	Expression	Results
MaxNetLength	A80 Net	Design.RiseTime*(5800[mil/ns])	8700.0mil
RiseTime	Design	1.5	1.5ns

EXAMPLE #5

In this example the value for the VDdiff constraint is obtained using various constraints from the design level.

Constraint	Precedence Level	Expression	Results
VDiff	Design	2*Resistance*(1–(.48*e^(.96*Spacing/Cu_width)))	125.368Ohm
Resistance	Design	870hm/(sqrt(diel_const+1.41))*(ln(5.98*BoardThickness/(.8*Cu_width+ Cu_thickness)))	81.7947Ohm
BoardThickness	Design	9.0	9.0mil
Cu_thickness	Design	0.8	0.8mil
Cu_width	Design	10.0	10.0mil
Diel_Const	Design	1.9	1.9
Spacing	Design	Cu_Width*.75	7.5mil

EXAMPLE #6

In this example, to determine the ComponentSpacing value at the component level, the expressions use values of constraints at the design level and the component level.

Constraint	Precedence Level	Expression	Results
ComponentSpacing	Component	If CompSpaceCriterion=True then CompSpaceNumber else design.ComponentSpacing	10.0mil
CompSpaceCriterion	Component	Design.ComponentSpacing< CompSpaceNumber	False
ComponentSpacing	Design	10	10.0mil
CompSpaceNumber	Component	Int(ComponentHeight*tan (design.InspectionAngle))	0.0mil
InspectionAngle	Design	5	5.0deg
ComponentHeight	Component	0.450	0.450in

Illegal Cases (Error Messages)

Syntax Error

Cause: The parser could not understand the formula syntax.

Solution: Check the syntax and matching parenthesis. Make sure there are no spaces between a number and its unit.

Divide By 0

Cause: An attempt was made to divide by zero.

Solution: Evaluate the expression for the existence of a divisor with a value of zero.

Operands type mismatch

Cause: An attempt was made to use “size” on a non vector or to combine non numeric values (Boolean, vector) with numeric (integer, floating point quantity) operators (+, -, *, /, sin, sqrt, etc.). Example: 3 + true.

Solution: Make sure that the value types on either side of the operand are corresponding types.

<operator> units mismatch

Cause: An attempt was made to combine incompatible units in an expression. Example: 1mil + 2W; sqrt(1pF)

Solution: Check the units for compatibility.

Function <function> math error

Cause: An internal math or domain error was detected (e.g., square root or log of negative number, or arcsine of number > 1.

Solution: Check the mathematical functions for correct usage.

Not a Number

Cause: An internal floating point conversion was unsuccessful or an integer was too large.

Solution: Check the expression for evaluations, which may result in an integer outside the limits.

Unknown identifier <name>

Cause: A reference to a nonexistent constraint was made.

Solution: Verify that the constraint name was spelled correctly or that the constraint exists.

Undefined identifier <name>

Cause: A reference to a constraint with no value was made.

Solution: Enter a value for the referenced constraint or correct any errors in the referenced constraint.

Array index <index> out of bounds

Cause: The value of a vector index was less than 0 or greater than or equal to the size of the vector (e.g., vector v=1,2,3;v[4])

Solution: Correct the entry to be greater than zero and less than or equal to the size of the vector.

Index

-D-

-E-

-F-

-H-

-I-

-J-

-K-

-L-

-M-

-N-

-O-

-P-

-R-

-S-

-T-

-U-

-V-

-W-