ACCEL PCB Contents

This table of contents lists available help information for PCB/Route.   Click on an underlined term to access information on that subject.

Series II Commands

The following list maps Tango PCB Series II commands/operations with their equivalent PCB commands, where equivalents exist.

The bolded terms on the left are the Series II commands, followed by their respective equivalent PCB commands/operations, which are non-bolded and indented.

For example:

Series II or PLUS command

Equivalent ACCEL PCB V2.00 Command/Operation

For more information on specific PCB commands, refer to their respective sections by way of the **Contents** or **Search** buttons in this on-line system. The Series II to PCB command/feature mapping is as follows:

Current Layer; or <L>

Options Layers; or <L>; or click on combo box on Status Line; or click scroll bar on Status Line

Current Line

Options Current Line

Current Pad

Options Pad Style

Current Text

Options Text Style

Current Via

Options Via Style

Delete Arc

Select arc, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Block

Select block, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Component

Select component, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Highlight

Select net using Edit Nets or select a segment of net <Right Mouse> Select Net, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Line

Select line, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Pad

Select pad, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Polygon

Select polygon, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Text

Select text, Edit Delete; or <Del>, or <RightMouse> Delete

Delete Via

Select via, Edit Delete; or <Del>, or <RightMouse> Delete

Edit Arc

Select arc, Edit Properties; or <RightMouse> Properties

Edit Component

Select component, Edit Properties; or <RightMouse> Mroperties

Edit Line

Select line, Edit Properties; or <RightMouse> Properties

Edit Pad

Select pad, Edit Properties; or <RightMouse> Properties

Edit Polygon

Select polygon, Edit Properties; or <RightMouse> Properties

Edit Text

Select text, Edit Properties; or <RightMouse> Properties

Edit Via

Select via, Edit Properties; or <RightMouse> Properties

File Clear

File New

File DOS

Select DOS prompt from Program Manager

File Load

File Open; see also File Gerber In; See also File DXF In

File Quit

File Exit

File Save

File Save; File Save As

File Save (ASCII)

File Save (ASCII); File Save As (ASCII)

Info Measure

Edit Measure; or click Toolbar Measure icon

Info Pad/Via

Select Component <RightMouse> Properties or Edit Properties Pattern Pads. For vias, select via <RightMouse> Properties or Edit Properties.

Info Status

File Design Info, click "Statistics"

Jump Component

Edit Components, Select Ref Des, Jump

Jump Location

View Jump Location

Jump Net

Edit Net, Select Net, Select Net Node, Jump to Node

Jump Pad_Center

Not applicable in ACCEL PCB

Jump Text

View Jump Text

Keys, <K> to Record/Stop, <E> to execute

Macro Record/Stop; Macro Delete; Macro Rename; Macro Run; Macro Assign To Key, <M> to Record/Stop; or click "M" on Status Line, <E> to execute

Library Add

Select pattern entities, Library Pattern Save As

Library Browse

Accessed in Place Component dialog box

Library Delete

Library Delete

Library Merge

Library Copy

Library Rename

Library Rename

Move Arc

Select arc, Drag; Move to Layer to move between layers

Move Block

Select block, Drag

Move Component

Select component, Drag

Move Endpoint

Select segment, Drag handle to re-size

Move Line

Select line, Drag; Move to Layer to move between layers

Move Pad

Select pad, Drag

Move Polygon

Select polygon, Drag; Move to Layer to move between layers

Move Reroute

Route Manual

Move Text

Select text, Drag; Move to Layer to move between layers

Move Via

Select via, Drag

Nets Clear

Not applicable in ACCEL PCB

Nets Display

Edit Nets, Show Conns, Hide Conns

Nets Generate

Utils Generate Netlist

Nets Highlight

Select net item, <RightMouse>, Select Net; or Edit Nets Select Net, click "Select"

Nets Identify

Select net, <RightMouse>, Net Info; or look on status line; or View Zoom In to see; or Edit Nets, Select Net click "Info"

Nets Load

Utils Load Netlist

Nets Optimize

Utils Optimize Nets, Utils Load Netlist, Choose Optimize Nets checkbox

Nets Reconnect

Utils Renumber, Utils Load Netlist, Choose Reconnect Copper checkbox

Nets Route

Route Manual

Nets Verify

Utils DRC, Netlist Compare option

Output Apertures

File Gerber Out, click "Apertures"

Output CAM

File N/C Drill; and File Gerber Out

Output Plot/Print

File Print; and File Printer Setup; For DXF Output, use File DXF Out

Output Reports

File Reports

Place Arc

Place Arc; or click Toolbar Arc icon

Place Block Copy

Select region or item, Edit Copy, Edit Paste; Edit Copy Matrix; or <RightMouse> Copy Matrix; or <Ctrl-LeftMouse>, drag and drop copy; see also Options Block Selection

Place Component

Place Component; or click Toolbar Component icon

Place Designator

Utils Renumber

Place Line

Place Line; or click Toolbar Line icon

Place Pad

Place Pad; or click Toolbar Pad icon

Place Polygon

Place Polygon; or Place Copper Pour; or Place Keepout; or Place Cutout; or click Toolbar icons

Place Text

Place Text; or click Toolbar Text icon

Place Via

Place Via; or click Toolbar Via icon

Setup Communications

Windows Program Manager, Main, Control Panel, Ports

Setup Display

Options Display

Setup DRC

Utils DRC

Setup Grids

Options Grids

Setup Libraries

Library Setup

Setup Options

Options Configure

Setup Palette

Not applicable in ACCEL PCB

Undo, or <U> to Undo

Edit Undo; or <U> to Undo; or click Toolbar Undo icon

Zoom All

View All

Zoom Board

View Extent

Zoom Center; or <C>

View Center; or <C>

Zoom In

View Zoom In; or <+>

Zoom Last

View Last

Zoom Out

View Zoom Out; or <->

Zoom Redraw

View Redraw

Zoom Window; or <Z>

View Zoom Window; or <Z>; or click Toolbar Zoom icon

Other Keyboard Shortcuts:

<E> to play default macro

<E> to play default macro

<F> to flip items

<F> to flip items

<R> to rotate items

<R> and <Shift-R> to rotate items

<O> toggle ortho modes

<O> to toggle ortho modes, <F> to flip

<U> unwinds placed routes

<Backspace> unwinds placed routes

<Arrow> to move cursor

<Arrow> to move cursor

<Shift-Arrow> moves 10X

<Shift-Arrow> moves cursor 10X

<Esc> exits commands

<Esc> exits commands

<RightMouse> cancels dragging

<RightMouse> cancels dragging

<?> on status line for help

<F1> for help, Help

<K> to start/stop Macro Recorder

<M> or Macro Record/Stop

<M>, or <SpaceBar>, or <DoubleMouse> to access menu

<Alt> or <F10> to access menu

For more information on specific commands, refer to their respective sections in this help system. Use the **Contents** or **Search** help buttons for easy access.

Shortcut and Special Use Keys

Alt+F4 **(File Exit)**

Shortcut for File Exit, which exits the ACCEL 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 PCB.INI file when you exit.

Alt **+ mouse click**

For any *click-and-drag* or *drag-and-drop* operations, you can hold down the *Alt* key, click the left mouse button, then move or drag the object wherever you want without having to keep the mouse button depressed. Without the *Alt* key, you would normally have to click and drag with the button depressed while you are dragging.

arrow keys

A directional arrow key moves the cursor to the next grid point. If *Ctrl+arrow*, then the cursor is moved 10 grid points, which can be used to pan the screen.

Ctrl+C **(Edit Copy)**

A shortcut for Edit Copy. Copies objects from your design to the clipboard.

Ctrl **+ Mouse Click (drag and drop copy)**

You can *copy-and-drag* an object by first selecting the object, holding down *Ctrl* and clicking the left mouse button in the selected object region, and dragging a copy of the object to a location and releasing to paste it.

Ctrl+N **(File New)**

A shortcut for File New. This command opens a window containing a new, untitled window

Ctrl+O **(File Open)**

Displays the File Open dialog, from where you can choose a file.

Ctrl+P **(File Print)**

Displays the File Print dialog, where you can set options for your output.

Ctrl+S **(File Save)**

Saves changes to the current design without closing it. To save the design to a different file, use File Save As. To clear the workspace, use File New.

Ctrl+V **(Edit Paste)**

A shortcut for Edit Paste. You can paste objects from the clipboard to the your design.

Ctrl+X **(Edit Cut)**

A shortcut for Edit Cut. Cuts objects from the design to the clipboard.

Ctrl+Z **(Edit Undo)**

A shortcut for Edit Undo, which reverses a *completed* action. If you have not completed your action (e.g., you are in Place Line mode and have not *finished* a series of segments), Undo will not reverse the action. Use the backspace key to unwind the unfinished actions when you are in a placement mode.

Ctrl+F4

Closes the active window.

Ctrl+F6 and Ctrl+Tab

Switches focus to the next window. Use *Ctrl+Shift+F6* or (*Ctrl+Shift+Tab*)to switch focus to the previous window.

Shift+T **(Move to Layer)**

Move to Layer shortcut.

Shift+F4 **(Window Tile)**

Shortcut for the Windows Tile command. Windows are resized and arranged side-by-side so that all windows are visible and none overlap.

Shift+F5 **(Window Cascade)**

Shortcut for the Windows Cascade command. All windows overlap, starting in the upper-left corner of your Workspace. You can see each window's title, making it easy to switch between windows.

Del **(Delete)**

Deletes all *selected* objects.

F1 **(Help)**

Displays the Help Contents window, from which you can access help information on ACCEL commands and tutorials.

Page Down

Scrolls one page down in the workspace. *Ctrl+Page Do*wn scrolls one page right.

Page Up

Scrolls one page up in the workspace. *Ctrl+Page Up* scrolls one page left.

Spacebar

The spacebar can be used in place of the left mouse button; but the action is different. To simulate a typical *click and release* of the mouse button, you need to press and release the spacebar <u>twice</u>. Therefore, to simulate the *click and hold* mouse action, you press and release the spacebar <u>once</u>.

As the left mouse button is used in such a variety of ways throughout the ACCEL program, this spacebar keystroke can become a regular part of your work.

Backspace (Unwind)

Used as unwind command while placing objects with multiple segments (e.g., lines, polygons, route manual). Each backspace stroke unwinds the previously placed item.

Esc **(Escape)**

Terminates placement of objects with multiple segments; it also cancels a redraw in progress. It is often equivalent to the right mouse button. *Esc* also exits from dialogs (equaling the **Close** or **Cancel** button), which is a common Windows feature.

+ Plus Key (Zoom In)

Shortcut for View Zoom In command. The keypad plus key also works. The plus key causes a *zoom in* to occur at the cursor location. Refer to the View Zoom In command documentation for more information. The plus key does not change the cursor to a zoom cursor (as do the zoom commands from the View menu).

- Minus Key (Zoom Out)

Shortcut for View Zoom Out command. The keypad minus key also works. The minus key causes a zoom out to occur at the cursor location. The minus key does not change the cursor to a zoom cursor (as do the zoom commands from the View menu).

A key (Grid Toggle)

Toggles between absolute and relative grid settings.

C Key (View Center)

Shortcut for View Center command. This command allows you to center your cursor location. Place the cursor in the area of your design that you want centered and press *C*. Repeated uses of the C key can be used to pan across the workspace.

D Key (Increment RefDes)

Increments the RefDes if while the Place Component tool is enabled. use *Shift+D* to decrement the RefDes.

E Key (Play Macro)

Plays back (executes) the *temporary* macro (named DEFAULT). Temporary macros are recorded by activating the macro toggle button (or the *M* key) to begin recording, and deactivating the button to end recording. Each successive temporary macro overwrites the previous one. Refer to the Macro Record/Stop command documentation (in this *Reference* manual) for information about the temporary macro and the macro toggle button.

F Key (Flip Object)

Flips an object during place and move operations. Not all objects can be flipped. Refer to the Edit Select command (in this *Reference* manual) for more information about flipping objects.

The *F* key also toggles between orthogonal mode pairs; see the Options Configure command for details on orthogonal modes.

G Key (Grid Select)

This key scrolls forward through the list of grid settings. Use Sh*ift+G* to scroll back through the list.

J Key (Enter Coordinate)

Gives focus to the X coordinate box in the status line. From there, you can enter new X and Y coordinates.

L and *Shift+L* Keys (Change Layer)

L cycles downward through the board layer list, duplicating the function of the down arrow on the layer list/combo box on the Status line. *Shift+L* cycles upward through the layer list.

M Key (Record Macro)

Duplicates macro toggle button (**M** button on the Status line) for starting/stopping recording of the *temporary* macro (see *E* key description). Refer to the Macro Record/Stop command (in this

Reference manual) for information about the temporary macro and the Macro toggle button.

O and *Shift+O* Keys (Orthogonal Mode)

O sequences forward through the orthogonal modes during placement of line objects or manual routing; *Shift+O* sequences backward throughout the modes. The *F* key toggles between orthogonal mode *pairs*. Orthogonal modes are set in the Options Configure dialog box. Refer to Options Configure for more information.

Q Key (Draft Mode)

Turns draft mode on and off.

R and *Shift+R* keys (Rotate)

R Rotates objects 90 degrees during Place and Select operations. *Shift+R* rotates objects by the rotation angle specified in the Options Configure dialog box. For more information about rotating objects with Place and Select operations refer to the Edit Select command.

ACCEL Tango PCB allows rotation increments of 90 degress only. Thus both R and Shift+R rotate an object 90 degrees.

S Key (Select)

Shortcut for the Edit Select command (also the toolbar Select button). When enabled, you are able to click on objects to highlight them, then move, rotate, delete, duplicate, and otherwise modify them.

U Key (Undo)

Shortcut for the Edit Undo command, which reverse a *completed* action. Refer to the Edit Undo command for more information. For the *unwind* feature, refer to the backspace key description in this section.

Y Key (Options Layers)

Shortcut for the options Layers command. This key opens the Options Layers dialog.

W Key (Line Width Scrol)

Scrolls forward through the list of line widths established in the design. Use Shi*ft+W* to scroll back throught the list.

X Key (Cursor Style)

Toggles between the three cursor styles: **Arrow**, **Small Cross**, and **Large Cross**.

Y Key (Options Layers)

Shortcut for the Options Layers command. This key opens the Options Layers dialog.

Z Key (Zoom Window)

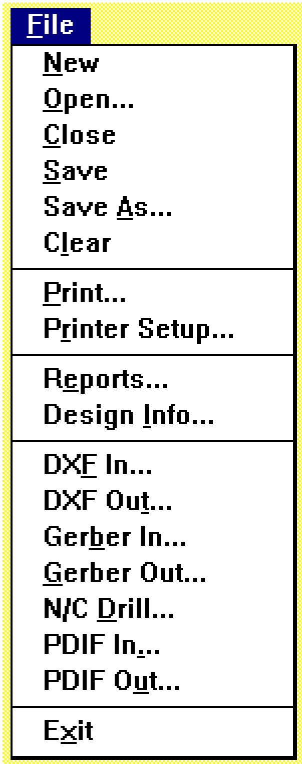
Shortcut for the View Zoom Window command. Just press *Z* and then draw the zoom window; a zoom cursor (magnifying glass) will display until you draw the zoom window. Whatever you surround with the zoom window will fill the screen.

Slash Keys (/ or \)

Stops a route in mid-connection without adding a final copper segment. A backward slash functions identically to a forward slash.

Close

File Menu (PCB)



File New

Opens a window containing a new, untitled window.

The File New command clears the styles, sheet definitions, and sheet sizes for the new design. All design parameters are returned to their default settings.

File Open

Opens an existing file using the Windows standard File Open dialog. If the design isn't currently loaded, this command creates a new window and loads the design. If the design is already loaded, this command creates a new window on the design; it doesn't reload the file from the disk. This is the same as the Window New Window command.

When you choose Open, PCB displays a dialog from where you can choose the directory and filename of the file you want to open. The Directories area displays the current path and a list of your immediate directories from which to change the path. The File Name area lists all the files of the current directory, with the extension specified in the List Files of Type area. Notice that your default file extension is .PCB.

You can use the drag and drop method for opening PCB design files (.pcb) from the File Manager or other Windows file maintenance Utils. Just click on the filename in the utility and drag it to the PCB window or reduced icon and release. The specified PCB design file will open.

PCB can read ASCII files as well.

To Open a File

1. Run the File Open command to open the File Open dialog.
2. Type, or select from the list, the name of the file you want to open in the File Name box.
3. If the file you want is not in the current directory, then either type the directory name in front of the document name, or select the directory from the Directories box. You can move through the directory tree by selecting the appropriate directories from the list.

note:
Global and local attributes in PCB design files are merged when the files are loaded from versions 3.05 and earlier.

---

4. Click **OK**.

To Open a Recently Used File

You can quickly open any of the last four files you worked on; their names appear at the bottom of the File menu. Just open the File menu and click the name of the file you want to open. The most recently opened file will be the one at the top of the list.

ACCEL Tango PCB

If any of the design limits are exceeded, your design cannot be opened using ACCEL Tango PCB.

See also:

[Opening an ASCII File]
[Opening a P-CAD Binary File]

File Close

Closes all windows for the active design.

If the design has been changed but not yet saved, you are asked whether or not you want to save your changes before closing. If you close the last design, it is automatically replaced with a new, untitled design.

File Save

Saves the changes to the design in the active window and creates a backup file (.BAK).

When you select File Save, the file remains open so you can continue working on it, and the backup file is copied. The current file name and location are unchanged by this command. To save a new (untitled) file or the current file to a different name or location, use the File Save As command.

Pending Engineering Change Orders (ECOs) must be appended to an ECO file or discarded when a design file is saved. If there are pending ECOs, the Save ECOs dialog appears.

1. The ECO filename appears at the top of the dialog. It is the last used ECO file. To change it, click the **ECO Filename** button and the ECO Filename dialog appears.
2. The ECO Filename dialog is a standard File Save dialog. Type, or select from the list, the name of the ECO file you want to use in the **File Name** box. Click **OK** to return to the Save ECOs dialog.
3. In the **Comments** box, type any comments that can help document the ECOs.
4. To append ECOs to the ECO file, click the **Append ECOs to File** button.
5. To discard ECOs, click the **Discard ECOs** button. Once discarded they cannot be recovered.

Net Classes

Net classes are saved to binary and ACCEL ASCII design files. Pre-Sequoia binary files will be post-processed after loading so that net classes can be automatically created from nets with CLASS attributes. This will only be necessary for designs that used the CCT router.

ACCEL Tango PCB

If any of the design limits are exceeded, your design cannot be opened using ACCEL Tango PCB.
See also:
Saving an ASCII File

File Save As

Saves the current design to a specified file name and location and creates a backup file (.BAK). You can either name a new file or save an existing file to a new name (if there is an original file, it remains the same).

This command displays the Windows common File Save As dialog, from where you can choose the directory and enter the filename you want to save the changes to. The **Directories** area displays the current path and a list of your immediate directories. The **File Name** area lists all the files of the current directory with the extension specified in the **Save File as Type** area. Notice that your default file extension is .PCB.

To save a file to a name and location:

1. Select File Save As and the dialog is displayed.
2. Type the filename you want to use in the **File Name** box.
3. If the current directory is not appropriate, then either type the directory name in front of the document name, or select the directory from the **Directories** box. You can move through the directory tree by selecting the appropriate directories from the list.
4. Make sure the extension is correct by checking the **Save File as Type** box.
5. Click **OK** to save the file as you have specified.

   If there are pending ECOs, you are prompted to save them. Refer to the File Save command section for detailed instructions.

ACCEL Tango PCB

If any of the design limits are exceeded, your design cannot be opened using ACCEL Tango PCB.
See also:
Saving an ASCII File

File Clear

Clears the workspace and resets the title bar.

When you choose Clear, you will be prompted to save changes to the current file before it is cleared from the workspace. The File Clear command does not clear the style or layer definitions setup by Options Pad Styles, Options Layers, etc. for the previously opened file. This allows you to use template information from an existing file. The command also does not clear the aperture list defined by the File Gerber Out command, allowing you to load a photoplot file without redefining all of the apertures.

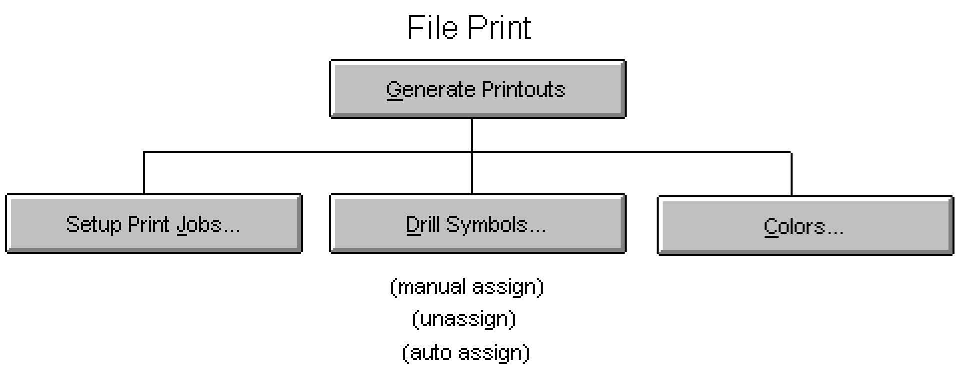
File Print

Prints a copy of the currently displayed PCB file, according to the printing specifications you have set. When you run the File Print command, the File Print dialog appears.

This dialog is used to generate individual or batch print jobs according to a variety of specifications. If you already have print jobs set up, you can generate them from this dialog. Additional dialogs for further specifications are accessible from this main dialog.

ACCEL Tango PCB

If any of the design limits are exceeded, your design cannot be printed using ACCEL Tango PCB.
To learn more, click a button from the following diagram:



Generating Print Jobs

After you have set up the print jobs, drill symbols, and colors, return to the File Print dialog to generate the print jobs. You can do a batch print by highlighting multiple print jobs in the listbox. You can individually highlight or de-highlight print jobs by clicking them in the listbox.

When you click **Generate Printouts**, the highlighted print job(s) are processed and printed.

Close exits the dialog and saves the settings for the File Print dialog and related dialogs.

Setup Print Jobs

The **Setup Print Jobs** button displays the Setup Print Jobs dialog.

In this dialog you can set up multiple print jobs, with each job including specific layers, design items, and other print options.   You can also modify existing print jobs to include (or exclude) layers, items, etc.

The Setup Print Jobs dialog contains the following areas:

Print Jobs
Print Job Selections

Print Jobs

Any print jobs that you have defined in the **Print Name** textbox appear in this listbox. Each print job listed can have its own particular layers, and other options specified. After you have set up a number of print jobs and listed them, you can highlight each print job name and the layers and parameters you specified for that particular job will be reflected in the dialog.

Whatever is in the listbox appears in the Print Jobs listbox in the File Print dialog after you exit (Close) this dialog.

The **Add**, **Modify**, and **Delete** buttons beneath the listbox work in conjunction with what you enter in the **Print Name** textbox. The print name that you enter appears in the list if you click **Add**; whatever print job(s) are highlighted in the listbox or listed in the textbox are deleted when you click **Delete**.

The collection of print jobs defined in the dialog are saved to the design file, possibly to be printed at some other time.

Print Job Selections

The following options appear in the Print Job Selections area:

Print Name	Pads Checkbox
Layers	Vias Checkbox
Layer Sets	Drill Sym Checkbox
Scale	RefDes Checkbox
X and Y Offset	Pad/Via Holes Checkbox
Drill Sym Size	Connections Checkbox
Rotate	Value Checkbox
Mirror	Type Checkbox
Draft	Glue Dots Checkbox
Keepouts Checkbox	Pick and Place Checkbox
Cutouts Checkbox	

After you have setup all the options and specifications for the print jobs, click **Close** to return to the File Print dialog.

Print Name

Print Name is the name you give a print job. When you click the **Add** button, the name you enter here is added to the **Print Jobs** listbox.

Layers

All items on the selected layers are printed in the current print job. Pads and vias must be enabled separately. For example, to include all items on the top layer in the print output, select **Top** in the **Layers** listbox. The **Set All** and **Clear All** buttons beneath it affect the Layer listbox. To print a composite of all layers, click **Set All** to highlight all of the layers, which you can then name as a print job.

When creating output for a specific hole range, you need to select all the layers on which a pad and vias hole has been defined.

Layer Sets

The **Layer Sets** box is used to designate specific, predefined layer sets to print. To select a layer set:

1. Select a layer set from the drop down list box.
2. Click the **Apply Layer Set** button.

   The layers belonging to that set are highlighted in the **Layers** box.

Scale

Scale allows you to increase or decrease the size of the print job by a specific factor. The value you enter affects the X axis and Y axis equally. For example, 0.100 reduces the output by a factor of 10, while 10.00 enlarges the output by a factor of 10.

X and Y Offset

The X and Y Offset adjustment allows you to offset your printout horizontally or vertically by whatever value you specify.   Minus values go left and/or down; plus values go right and/or up.   Default units are the values settings in Options Configure.   You can override the default units by entering the units with the value (e.g., mil, mm, in).

Drill Sym Size

Drill Sym Size allows you to adjust the size of the drill symbol (usually for size reduction), which represents the drill hole location. For example, you would reduce the drill symbol size so that the drill symbols will not overlap in the printout if they are located close to one another.

Rotate

Rotate, when enabled, causes a 90 degree clockwise rotation of the image on the printout. If Mirror is also enabled, the image mirrors before it rotates.

Mirror

Mirror, when enabled, causes the printout to be a mirror image (reverse) of the normal view. Normally, each layer is displayed as seen from the Top layer. A mirror image printout would be as the board appears from the Bottom layer (as if you flipped the board over horizontally and looked at it with the Bottom layer facing up).

Draft

Draft, when enabled, allows an outline rough draft to be printed (normally speeding up the print job). This would be a printout of only the outlines for pads, vias, polygons, copper pours, etc.

Object Checkbox

When this checkbox is enabled, the corresponding object is included in the printout.

Drill Symbols

To define drill symbols in your printer output, click the **Drill Symbols** button to display the Drill Symbol Assignments dialog.

To generate a drill symbol drawing, you must enable all signal and power/ground layers on which the pad is defined. Enabling **Drill Symbols** excludes all copper from this print job and sends just the assigned symbol in its place.

To attach a drill symbol to a hole diameter, highlight both of them and then click **Assign**. You can clear the assignment in the same way by clicking **Unassign**.

If you click **Automatic Assign**, the drill symbol and hole diameter lists will automatically match up from top to bottom, the first hole diameter assigned to the first drill symbol, the second symbol to the second diameter, etc. **Unassign All** clears all of the drill symbol assignments.

Click *Close* to save all of the assignments and return to the File Print dialog.

Colors

You can define your printer output to use colors, grayscale, or monochrome (depending on your printer). Click the **Colors** button to display the Printer Colors dialog.

The Printer Colors dialog has the following functions:

Layer/Item Colors
Display Colors
Defaults Button

Layer/Item Colors

This is a color matrix with objects across the top (columns) and layers along the side (rows). You can specify everything on one layer to be of one color by clicking on a layer button (e.g., **Bottom**) to display the color. By the same process, you can make an item the same color on all layers by clicking an item button (e.g., **Line**). Or you can make an item a specific color on a specific layer by clicking on the square where the item and the layer meet. You can then choose the color for the layer, item, or combination.

Any of these choices displays the Windows standard Color dialog. From there you can perform the color selections; PCB is limited to 16 solid colors. If you select a non-solid color, it automatically maps to the nearest solid "equivalent".

Display Colors

When you select one of the item buttons in the Display Colors area, it displays the same Windows standard Color dialog(s) as in the Layer/Item Colors selection.

When you select a display color for an item, that color applies throughout the design on the printed output.

Defaults Button

The **Defaults** button turns all of the colors to monochrome, a white background with black items. If you don't have a color printer, then this is the recommended setting. If you have color settings, but a monochrome printer, you may get undesirable output when the colors are converted to grayscales.

File Printer Setup

Displays a list of installed printers and allows you to set the current printer. You can also change device-specific parameters. When you run the command, you see the Print Setup dialog.

Select a printer and press the **Setup** button to configure print parameters. Because the parameters are device-specific, the dialog which appears is different depending upon the printer you selected.

To select a printer, highlight it in the **Printers** listbox, and click **Close**.

File Reports

Allows you to output reports with specific output options. These options are saved to the PCB.INI file when you exit the program.

The following paragraphs describe the areas and function of the File Reports dialog.

Filename

Generally, you specify individual reports (report types listed in the Report Options area), which you can choose to output all at once, or one at a time. Each report type has its own extension, the filename determined by the current open design.

Report File Extensions

These extensions cannot be changed. The extensions are used are as follows:

- APR for Aperture Information
- BOM for Bill of Materials
- CPL for Component Locations
- GLU for Glue Dot Locations
- LCT for Library Contents
- PNP for Pick and Place locations
- STA for Statistics.
- ATR for Attributes
- DEI for DRC Error Indicators.

Report Options

Aperture Information lists aperture information such as units used, definitions (D code, type, etc.), and aperture assignments.

Bill of Materials lists the component type, component value (if any), number of components of each type, and reference designator assigned to each component. If you have special components that have hidden reference designators (RefDes) and you want to exclude them from the Bill of Materials, you can explode the component (Edit Explode Component) and delete the RefDes, or just delete the component from the report.

Component Locations lists, for each component, the reference designator and location (the X, Y coordinates). If you have special components that have hidden reference designators (RefDes) and you want to exclude them from the Component Locations, you can explode the component (Edit Explode Component) and delete the RefDes, or just delete the component from the report.

Glue Dot Locations lists the layer and position of all glue dots in the current design.

Library Contents lists all of the components that are listed in all of the open libraries (see Library Setup for opening libraries).

Pick and Place Locations lists the layer and position of all pick and place points in the current design.

Statistics contains a variety of information about the current design, such as primitives count, board dimension, line widths used, layer types used, etc.

Attributes contains component and net attributes.

DRC Error Indicators generates a list of all the DRC error indicators on the board. The report shows the locations and text of all the DRC error indicators on the board. This allows you to generate a hardcopy for all the DRC error indicators generated by Online DRC.

Page Format

These options, when enabled, will be output with whatever report type you have chosen to use.

Use Header and **Use Footer** will include whatever you have specified as header and footer in the header and footer dialog fields.

Design Info will include whatever you entered in the File Design Info command/dialog.

Date/Page will include the current date and page number.

Pagination allows you to create your own pagination (lines per page); don't print from Notepad if you enable this option; use the DOS Print command.

Style Format

Comma Separated puts all information in comma separated format, which is a spreadsheet-readable format.

PCB is a human-readable format, with columns and spaces, etc.

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

Report Destination

Screen sends the output to a file and invokes Notepad, where you can view the report.

File sends the output to a file.

Printer sends output directly to the printer without creating files.

File Design Info

You can enter design information into the dialog that is displayed with this command.

Certain fields that you place in your design will use this information to display data in the respective fields. Title, Author, Date, Time, and Revision number can all be changed.

For example, if you placed a Title attribute (Place Attribute command) in your design as design01, and you changed it in this dialog to design02, then the Title attribute in your design would change to design02. If you had numerous Title attributes placed in your design, they would all be changed by what you enter here.

The information that you specify with this command is saved and restored from design files.

Statistics

The **Statistics** button displays a view-only dialog that gives you statistics for the specific file that you currently have open.

Attributes

Allows you to display and modify design-level attributes. These are attributes that are attached to a particular sheet, rather than a part or net. When you click the **Attributes** button, the Attribute Collection Property Page dialog appears.

You can view, add, modify, or delete a collection of design attributes. The dialog contains a two-column table showing the collection of design attributes. Within a collection, each attributes name and value appear in the column.

- **Changing an Attributes Value:** To change a value, select it and type the new value over the existing value. The name column cannot be changed
- **Adding an Attribute**: To add an attribute, click the **Add** button to open the Attribute dialog. Enter the name and value for the attribute and set attribute properties. Click **OK**, and the attribute is added to the table.
- **Viewing or Changing Attribute Properties:** To view or change an attributes properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the Attribute dialog.
- **To Delete an Attribute**: Select an attribute in the table and click **Delete**, or press the *Del* key.

Attribute Dialog

The following information appears in the dialog:

- **Category Listbox:** Displays a list of all attribute categories, All, Component, Net, Clearance, Router and SPECCTRA. Selecting a category brings up a list of pre-defined attributes for that category.
- **Name Listbox:** Displays all pre-defined attributes for the specified category. The first entry in the list is *User-defined*.

  The currently-selected attribute also appears in the **Name** edit box, unless *User-defined* is selected. In that case, the Name edit box is blank so that you can enter a user-defined attribute name.
- **Name Edit Box:** For user-defined attributes, enter a name for the attribute.

note:

If the dialog is accessed for an attribute that already has a name, then the Category listbox, Name listbox, and Name edit box are filled in, but grayed. If the attribute doesnt have a name, these controls are enabled.

- **Value:** Use this edit box to enter a value for the attribute.
- **Visible:** This checkbox indicates whether or not the attribute is visible.
- **Location:** This area shows the X and Y coordinates of the components reference point.
- **Text Style:** This area lets you select the attribute text style. Text styles appear in the **Text Style** drop down listbox. To change the selected Text Style, click on the text style you want from the listbox. To modify the text style, click the **Text Style** button.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped** Box: This box indicates whether or not the pattern has been flipped.
- **Justification**: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

See also:

Place Field

File DXF In

DXF files can now be loaded into PCB. This allows you to create dimensions, board outlines, manufacturing instructions, artwork, logos, etc., and then import the resulting DXF file into PCB. DXF files generated using AutoCAD® Version 9.0 through 12 or other conforming CAD program may be input into PCB.

Loading a DXF File

To load a DXF file into PCB, run the File DXF In command. The File DXF In dialog appears requesting a file name and layer name.

To load a DXF file into PCB, run the File DXF In command. The File DXF In dialog appears requesting a file name,layer name, and layer.

1. Click the **File Name** button and the following standard File Open dialog appears for you to select a file.
2. Type, or select from the list, the name of the file you want to open in the **File Name** box.
3. If the file you want isn't in the current directory, then either type the directory name in front of the document name, or select the directory from the **Directories** box. You can move through the directory tree by selecting the appropriate directories from the list. Click ***OK*** to exit this dialog.
4. From the File DXF In dialog, type a layer name and select an unassigned layer number. The layer name entered is used to create a non-signal layer where all the DXF items are placed. After the DXF items have been translated and placed onto the new non-signal layer, these items are equivalent to other PCB primitives. The new lines, arcs, text, and polygons can be modified as usual.
5. Click ***OK***.

   The progress of DXF translation is shown by a percentage counter. Some items supported in the DXF language cannot be translated for use by PCB. Other items may fall outside the PCB Workspace. Errors and warning messages are placed in a report file, which you may view at the end of the translation.

Items and Features Supported for Translation

The following items and features are supported for translation:

Header Variables

The AutoCAD® state variables are grouped together at the top of the file in the HEADER section. Supported variables are listed and described below, with default values in parentheses:

$ACADVER	The AutoCAD® drawing database version number (must be 9.0 or higher).
$ANGBASE	The angle zero direction. The DXF coordinate system is rotated by this angle. (0)
$ANGDIR	The angular orientation: 1 = clockwise; 0 = counterclockwise. (0)
$MIRRTEXT	Mirrored text if non zero. (0)

$TEXTSIZE	Default text height.

Tables

The DXF In command does not support the LTYPE, LAYER, or STYLE tables.

The only entries in the BLOCKS section that are supported are the dimension blocks. All dimensions that are translated come from this section.

Entities

The majority of a DXF file is made up of entities. These include lines, arcs, text, block insertions and others. Only two-dimensional entities are supported; z-axis values are ignored. All block insertion entities are ignored. Information embedded in the entities for color and layer are also be ignored.

LINE	DXF LINE entities have infinitesimal width. They are translated into PCB lines one mil wide. All line styles are translated to solid.
ARC	DXF ARCs are translated into PCB arcs of one mil.
CIRCLE	DXF CIRCLEs are translated into PCB arcs with a sweep angle of 360 degrees and one mil.
POLYLINE	DXF POLYLINEs are sequences of possibly tapered, straight and curved lines that are connected end-to-end. These may be open or closed. PCB does not support tapering and only supports normal 2-D (unflagged) DXF vertices. POLYLINEs are translated into PCB arcs and lines with a thickness equal to the initial POLYLINE thickness.
VERTEX	Vertices define DXF POLYLINEs. When translated, they become the defining vertices of the translated PCB item. Only normal 2-D DXF vertices are supported (no spline-fit, curve-fit, 3-D mesh, or other special flags).
SOLID	DXF solids are filled three- or four-sided polygons. They are translated into PCB polygons. Four sided solids that form a PCB "complex" polygon will be ignored.
TRACE	DXF traces are lines with thickness that can be filled or unfilled. They are treated the same as DXF solids and are translated into PCB polygons.
TEXT	DXF text is translated into PCB text. Obliquing angle ("slant") and font name are not supported. The font used for translation is the PCB default font. **Due to the difference in fonts, translated text strings may be of different total width than the DXF version**. The bar over barred text may not align exactly with the text.

The following is a specific (but not comprehensive) list of items that are **not supported**:

- All BLOCKS, except the dimension blocks.
- DIMENSION entities. All dimensions are translated from the BLOCKS section.
- SHAPE, ATTDEF, and ATTRIB entities.
- 3DLINE and 3DFACE entities.
- Curve- or spline-fit vertices or meshes for POLYLINE and VERTEX entities. Tapering

POLYLINEs are also not supported.

- Obliquing angle and font for TEXT entities. The PCB Quality font will always be used.
- Three-dimensional entities and coordinates; thickness for all entities will be ignored, and only the first two values of a coordinate-triplet will be used.
- Color values for individual entities. Color values for entities will depend on the PCB primitive and layer to which the entity is translated.

DXF In Notes

The following is a list of notes that are important or useful when using the DXF In command:

- The EXPLODE command in AutoCAD® can be used to transform blocks into individual entities.
- An important item to be translated from a DXF file is the dimension. AutoCAD® creates a new dimension block every time a dimension is moved, edited, or altered in any way. The user should use the PURGE command to eliminate any unreferenced copies of the dimension blocks created by AutoCAD®. The user should then output the design to DXF format immediately after the PURGE command, before editing or modifying dimensions.

File DXF Out

The File DXF Out command is used to create DXF (Drawing Interchange Format) files of your PCB designs. These files can then be transferred to AutoCAD and other mechanical CAD packages. The output is compatible with AutoCAD® Version 9.0, and above.

When you run the File DXF Out command, the File DXF Out dialog appears.

To use this dialog:

1. Click the **DXF Filename** button and following the standard File Save dialog appears for you to select a file.
2. Type the name of the DXF file to you want to save in the File Name box.
3. Click ***OK***.
4. Select the PCB layers you wish to output using the **Layers** listbox. Each PCB layer is output to a separate DXF layer. The **Select All** and Clear All buttons let you select and clear all the board layers at once.

   You can use **Layer Sets** box to designate specific, predefined layer sets to output. To select a layer set, select a layer set from the drop down list box and click the **Apply Layer Set** button.
5. If you select Output Drill Symbols, you need to specify a size for the resulting drill symbols. The default value is 80 mils (or the equivalent if you're in metric mode). To use a different value, type it in the Size box. If you specify zero, the Drill Drawing layer is output without drill symbols.

   To make or examine drill symbol assignment, click the Setup button. The Drill Symbol Assignments dialog appears. This dialogis explained in the File Gerber Out command section of PCB Reference.
6. DXF polylines are normally used for all lines, arcs, and pad and via shapes, and solids are normally used for polygons. Polylines are filled lines with thickness. Select the Draft checkbox to output DXF arcs, lines, and circles instead of polylines, and polygon outlines instead of DXF solids. Draft mode produces smaller files that process faster, but the drawings are not technically accurate since the lines have no width and areas are not filled.
7. Click the ***OK*** button to generate the DXF output file. While the file is generating, the Status line indicates progress by displaying the current layer being output. While the file is being generated, you can press ***Esc*** or click the right mouse button to cancel the operation.

Component Height Checkbox

When you enable the **Component Height** checkbox, PCB automatically detects valid ComponentHeight attributes assigned to components, and writes corresponding component height geometry to the DXF.

PCB uses the geometric primitives in a components pattern to produce component geometry in a DXF (circles, lines, arcs, and polygons). Pads, vias, text, and signal layer items are ignored. Component height geometry is created in the DXF on layers that are consistent with the source geometry.

In the DXF, the height of an individual component extends upwardly or downwardly from the PCB, depending on whether the component resides on the top or bottom layer.

When the DXF is read into a 3-D CAD system (such as AutoCAD), the specified heights are represented as geometric elements of the proper PCB components when viewed in 3-D mode.

DXF Output Considerations

Layers. Items are output to individual layers, which keep their PCB names.

Blocks. DXF blocks are used to combine individual entities into a common unit, to be treated as a whole by the CAD package, analogous to a part or component. Blocks are used, where possible, to make CAD processing easier, and to reduce the size of the DXF file. For example, a component block has the name of the reference designator; exploding a component block produces text (for attributes) and a pattern block. This in turn can be exploded to produce some single items and padstyle blocks. These can be exploded to produce padshape blocks; which can be exploded to produce solids and polylines.

Polygons. PCB polygons are also represented as blocks containing a collection of three- or four-sided solids. In this way they can be processed as a unit.

Copper Pours. PCB copper pours are represented as blocks containing a collection of lines that outline and fill the copper pour. In this way they can be treated as a unit.

Lines. When not in Draft mode, lines consist of a straight polyline and two round endcaps. Note that due to limitations in how blocks are scaled in DXF, lines are not blocks; the endcaps and polylines are separate from one another. In Draft mode, lines become DXF LINEs with no endcaps.

Arcs. When not in Draft mode, lines consist of a curved polyline and two round endcaps. Note that due to limitations in how blocks are scaled in DXF, arcs are not blocks; the endcaps and polylines are separate from one another. In Draft mode, arcs become DXF ARCs with no endcaps.

Text. Text strings are converted into DXF text strings of the same height, rotation, mirroring, and justification; the AutoCAD® STANDARD font is used. Note that due to the difference in fonts, translated text strings may be of different total width than in PCB.

Pads and Vias. These are blocks of pad shapes, which are in turn blocks of SOLIDs and POLYLINEs. For example, a rounded rectangle is a block consisting of two SOLIDs forming a thick "plus" and four circular POLYLINEs in the corners. A pad is a block containing a stack of what could be different shapes on different layers. When not in Draft mode, these shapes are filled; in Draft mode, only the outlines are represented with lines, arcs, and circles. Pads and vias are not output with holes whether or not you are in Draft mode.

File Gerber In

File Gerber In allows you to load a series of Gerber files. You can load a Gerber file into the editor to check its accuracy. Each file is loaded onto a separate layer.

You can load a Gerber file either into an empty workspace (use File Clear to clear the workspace) or superimpose it onto an existing design. Loading a file alone is usually for checking pad size, line width, or other possible errors. Superimposing a file onto a design is a good way to verify the Gerber file against the design.

You can examine the Gerber file through the workspace display in addition to analyzing the error file displayed in Notepad.

From the Gerber File In dialog, click the **Gerber Filename...** button to access the Windows common File Open dialog (File Gerber In), where you can access the Gerber file.

Access your Gerber file (e.g., chap9.top) and click **OK** to return to the first dialog.

From the first dialog, specify the **Layer Name** and **Layer Number** and enable or disable **View log file upon completion**. The layer name Gerber is provided as a default, but may be changed.

You can load multiple Gerber files onto your design file, each inhabiting its own layer. Therefore, the layer names must be unique. So, the first file loaded would be on the layer Gerber, the next file loaded would receive the default name Gerber1, then Gerber2, etc.

When you load a Gerber file, it automatically creates a non-signal layer named "Gerber", using the next free layer in sequence. (If you have eleven layers, the twelfth is used.)

Normally you would load multiple files to verify that the layers are aligned and registered properly.

warning:

The D Code apertures that are called out by the loaded Gerber file must be present and be defined as they were in the original Gerber file. If a D Code is no longer present, then the program will flag the error. If the D Code is present but its definition has changed, no error will be flagged, and you may get unexpected results. For example, this could occur if you loaded an old Tango Series II Gerber file without first recreating the aperture definitions properly. Our advice is that you should not redefine D Codes at all between the creation of the Gerber file and reloading it for design file/Gerber file compairson.

Aperture definitions are saved and loaded in the design file. Loading a new design file can completely change the current aperture settings. The File Clear command does not clear current aperture definitions, so use File Clear to clear the workspace before loading a newly created Gerber file.

Click ***OK*** and the photoplot file will load.

ACCEL Tango PCB

If the layer limit is exceeded, your design cannot be loaded using ACCEL Tango PCB. ACCEL Tango PCB supports 4 user-defined layers and up to 15 total layers.

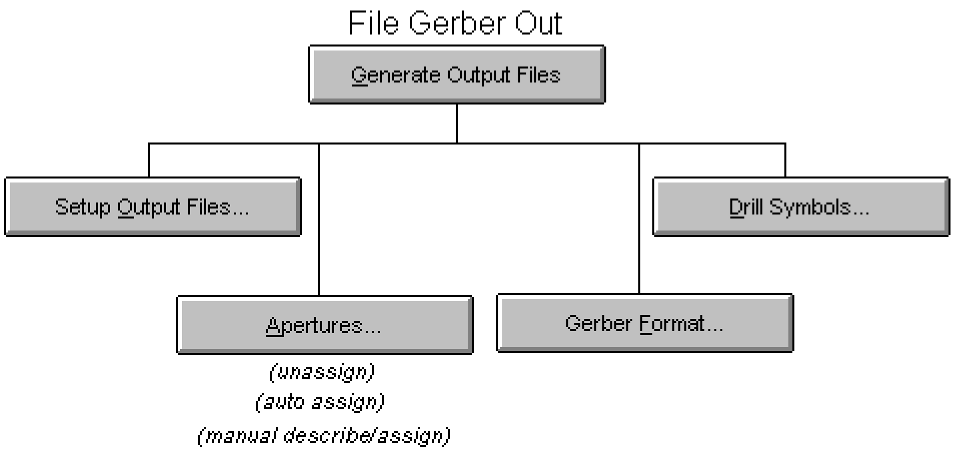
File Gerber Out

The File Gerber Out command allows you to output Gerber files with a variety of options and specifications.   You can cancel this command before the dialog appears (while the program is searching the database for items) by clicking the **Cancel** button or pressing *Esc*.

When you run this command, the File Gerber Out the dialog appears.

There are multiple dialogs you can access from the File Gerber Out dialog buttons.

To learn more, click a button from the following diagram:



Setup Output Files

The **Output File** listbox contains the filenames of the output files you set up. Each set of options you specify (described in the following paragraphs) comprises a separate output file, differentiated by its extension. After you have set options and the extension for a particular file, you can click *Add* to add it to the list. After you have a file list, you can choose to change a file (click *Modify*), or delete it from the list (click *Delete*).

Your output files will have the same base name as the design file, but each with a unique extension. Typical extensions used to differentiate files would be layer-specific, such as TOP, BOT, and TSK (TOP, BOTTOM, and TOP SILK).

The **Output File Selections** area allows you to set options for each output job such as: layer, file extension, X and Y offset, drill symbol size, and items included (Pads, Type, etc.), and the output path of the particular file.

- **File Extension** should be a meaningful name for the output (e.g., .top). Avoid such potentially conflicting extensions as .exe and .pcb. When you click *Add*, the base filename with the extension you specify will be added to the file list.
- **X Offset** and **Y Offset** allow you to set the origin of the plot in a different position than the home position of the plotter. For X offset, enter a negative number to offset the plot towards the left, positive to the right. For Y offset, enter a negative number to offset downward, positive to offset upward.
- **Drill Symbol Size** value determines the sizes of the drill symbols output in the particular file.
- The **Enable/Disable** checkboxes allow you to include or exclude items and attributes from the output.
- **Ref Des**, **Type**, and **Value** checkboxes are typically used for silkscreen generation, with the silkscreen layers.
- **Mirror** is typically used to produce a mirror-image output for bottom layers (bottom, bottom silk, paste mask, solder mask, assembly, etc.) Some fabrication shops prefer to mirror the output themselves, so check with them before using **Mirror**.
- **Pads**, **Vias**, **Pad/Via Holes**, and **Drill Symbols** should be enabled if you want them included in the Gerber output.

  When producing an output file for a specific hole range, you need to select all the layers on which a pad and vias hole has been defined.
- **Output Path** allows you to output the files to a particular location.
- **View error file upon completion** gives you the option of viewing the log file in Notepad.
- The **Layers** box lists all current layers of the design. You can highlight any number of them for output to a particular file.
- The **Layer Sets** box is used to designate specific, predefined layer sets to output. Select a layer set from the drop down list box and click the **Apply Layer Set** button.

To generate a drill symbol drawing, you must enable all layers that match the pad stack (plane and signal). Also, you must enable pads, vias, and drill symbols.

After you have set up your output files list in the dialog, click **Close** and return to the Gerber File Out dialog, which will list the files that you set up for output.

Apertures

To assign/describe apertures automatically or manually, click the **Apertures** button (from the File Gerber Out dialog) to display the Aperture Assignments dialog.

The listbox of this dialog displays the items of the loaded design file and any aperture assignments that may exist for those items. You can click an item line and the item's characteristics will be listed to the right of the listbox. If an aperture is assigned to the item, the aperture characteristics are listed there as well.

To *manually* describe and assign an aperture (or change an existing assignment), double-click an item line, or highlight an item line and click *Assign*, which displays the Describe/Assign Apertures dialog.

Unassign deletes the assignment for the item that is selected in the assignment list.

Unassign All clears all assignments, usually so that you can perform automatic describe and assign functions to all items/apertures.

Additional Information

Auto (Automatic Describe/Assign)
Assign (Manual)
Available Procedures

Auto (Automatic Describe/Assign)

To automatically assign apertures, click the **Auto** button. PCB automatically assigns all apertures that have not been assigned manually.

You can enable the **Pad/Via holes** option to have PCB create apertures with holes, but this is generally not recommended for two reasons:

- Drilling holes through the board could break drill bits if the flashed pad/via holes don't line up through the board exactly.
- Non-hole apertures are usually less numerous; a 50 x 50 *round* with no hole can be used for a 50 x 50 on the Top layer and also for a 40 x 40 via with a 5 mil swell solder mask.

The **Clear current apertures** option is enabled by default. If disabled, the current apertures remain. You generally would want to clear apertures when you auto-assign.

Draw Rotated or Offset Pads/Vias, when enabled, draws pads and vias with non-orthogonal rotations, or offset holes (non-orthogonal being rotations other than 0 or multiples of 90 degrees).

Draw Aperture Size specifies the default draw aperture size for any unassigned apertures, such as drawn drill symbols and polygons.

After you have taken these steps, click *Auto* to automatically describe and assign all unassigned apertures to the appropriate items.

Assign (Manual)

To manually describe/assign apertures, click *Assign* to display the Describe/Assign Apertures dialog. You can also double-click an item line in the previous dialog to display this dialog.

The item name you highlighted on the previous dialog appears in the **Assign Aperture to** field at the top of this dialog. If the item was already assigned an aperture, that aperture name appears highlighted in the **Apertures** listbox. The item characteristics are listed to the right of the listbox, and the aperture characteristics are listed there as well (if an aperture is assigned).

In the **D Code** entry box, you can enter a value in the range of 10 through 999 for the draft code used to select the aperture. The program automatically prefixes the value with the letter **D**.

The **Shape** combo box displays the aperture shape. The following shapes are supported:

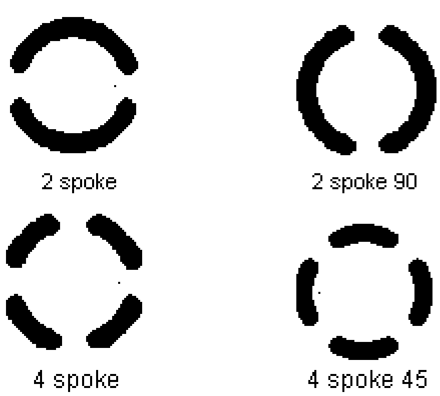
- **Ellipse** is a rounded shape with separately specifiable X and Y dimensions; an ellipse with equal X and Y dimensions is a circle (frequently called a *round*).
- **Oval** is a short line segment with round end caps (half-circles), the radius of which is 1/2 the length of the shortest side; if the X and Y dimensions are equal, this too is circular or round.
- **Rounded Rectangle** contains 1/4 circles on the corners of a rectangle. The 1/4 circle radius is 1/4 the length of the shortest side.
- **Rectangle** shapes are X=width and Y=height.

note:

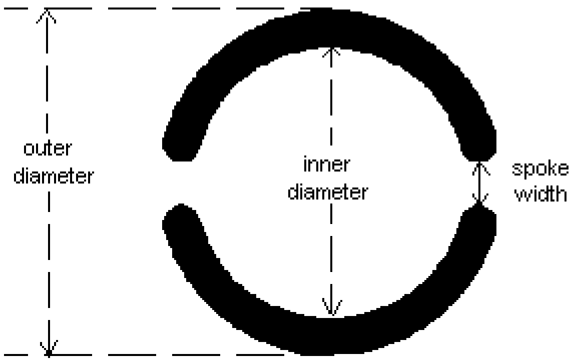
If you want a circle pad, you use the ellipse shape with identical height and width (the example shown is an oblong ellipse). If you want a square pad, use a rectangle with identical height and width.

- **Thermal** apertures are typically used for connection to planes, where heat is a factor. These apertures are constructed from two or four arc segments separated from each other by spokes. The other dimension is the diameter of the outer edge of the arc-segments, the inner dimension measures the inside edge, and the *gap* measures the spoke width, or amount of copper connection on a reverse-screened plane layer. Thermals are always circular, and the size is a function of the pad/via hole size.

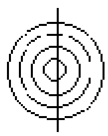
thermal spoke apertures



thermal diameters and spoke width

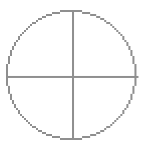


- A **Target** is used for layer registration.



Target

- **Mounting Hole** is a circle shape with a cross in it used to represent a mounting hole.



Mounting Hole

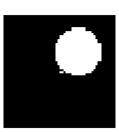
- **Drill Symbol** could be any of the drill symbol shapes assigned in the Drill Symbols dialog.

X Dim and **Y Dim** are the X (horizontal) and Y (vertical) dimensions for the aperture. These values may be entered with your choice of units.

The **H Dia** is the hole diameter for the aperture. This value may be entered with your choice of units. The hole diameter cannot exceed the minimum aperture dimension.

Angle is the amount in degrees that an aperture is rotated.

X Off and **Y Off** are horizontal and vertical offset amounts of the hole in an aperture.



The **Type** combo box lists the aperture types (*type* indicates whether the aperture can be flashed, drawn or both). The type is specified using either **Flash**, **Draw**, or **Flash/Draw**. Flash and Draw are typically specified for use with mechanical or vector photoplotters, because different holders must be used for the two types of apertures. **Flash/Draw** is used with laser photoplotters, which do not have this limitation.

The **Comment** box allows you to enter a comment of up to 32 characters. This is especially helpful for further describing thermal reliefs and drill symbols.

Available Procedures

There are basically four procedures available in this dialog:

- *Create a new aperture* by describing the aperture (filling in the D Code, Shape, etc.) and clicking *Add* to assign it to the item. The Shape and Type combo boxes allow you to select the shape and type of the aperture.
- *Modify an existing aperture* by changing various descriptions, but keeping the same name (D Code), and clicking *Modify*. The assignment remains the same (item to aperture), but the characteristics of the aperture have been changed.
- *Use an existing aperture to create a new one* by changing the name (D Code) of the aperture, and then changing the descriptions of the aperture or using the same descriptions. You would then click *Add*, in effect creating a new aperture.
- *Delete an aperture* assignment/description by simply clicking *Delete*.

Click *Close* to save all of your changes. You then return to the Aperture Assignments dialog, where the assignments you've made are listed. The currently selected aperture is used as the assignment for the item shown in the dialog.

Gerber Format

To access Gerber options, click the **Gerber Format** button (in the File Gerber Out dialog) to display the Gerber Format dialog.

Select the **Output Units** as inches or millimeters for your output file.

Select **Numeric Format** on the basis of the required resolution. The format **4.4** means that there are four digits to the left of the decimal point and four digits to the right. The format **5.3** means that there are five digits to the left of the decimal point and three digits to the right.

The **G54 With Apertures** option is initially disabled. It determines whether or not to send a G54 tool-select code with each command to change apertures.

The **Include aperture definitions** option determines whether definitions and assignments macros are to be included in the output.

Click *Close* to exit the dialog, and the format you specified will be applied when the output is generated.

Generate Gerber Output

After you have set up the output files, aperture assignments, drill symbols, and Gerber format, you can generate the output files by highlighting them from the **Output Files** list box and clicking the **Generate Output Files** button in the File Gerber Out dialog.

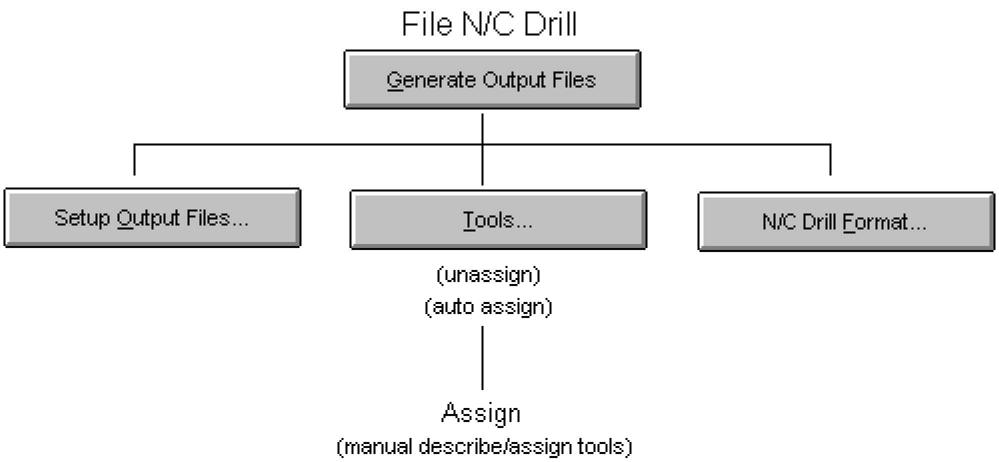
A progress dialog displays the name of the file being converted. When finished, the log file is displayed if the **View error file upon completion** box was enabled in the Setup Output Files dialog.

File N/C Drill

Allows you to output an N/C Drill file for your design in Excellon format. You can cancel this command before the dialog is displayed (while the program is searching the database). Choose File N/C Drill to display the dialog.

This main N/C Drill dialog allows you to output individual or multiple files. Before you generate output, you may need to establish N/C Drill settings in some of the multiple dialogs available from the File N/C Drill dialog.

To learn more, click a button from the following diagram:



Generate N/C Drill Output Files

When you have established all of the options for output files, tool assignments, and other drill settings, the resulting output files will be listed in the **Output Files** listbox. You can click on the filenames individually to enable or disable for output. **Set All** enables all of the files for output (you can then deselect individually). **Clear All** deselects all the files (you can then select files individually). Then click **Generate Output Files** to generate drill files according to what you have specified.

Setup Output Files

When you click **Setup Output Files**, the Setup Output Files dialog appears.

The filenames for the output files you set up here are determined by the individual extensions you give them (e.g., *filename*.NC1, *filename*.NC2). Most of the time only one file is used for N/C Drill, so that you have the design filename as the root name, and the extension as .NCD. This is to differentiate it from the regular design file and Gerber files in the same directory (.PCB, .TOP, .BOT, etc.)

The **Layers** listbox fills with defined signal layers; no non-signal layers appear.

When producing an output file for a specific hole range, you need to select all the layers on which a pad and vias hole has been defined. In this way it is possible to generate files for blind and buried vias. If your board contains only through-hole pads/vias (not blind or buried vias), then only one file is necessary, and it should contain all of the signal and plane layers.

You can perform four basic procedures for output files from within this dialog:

- *Create an output file*. Specify an extension (e.g., NC1) and the X, Y offset settings. Select layers to be included by clicking on the layer names individually to highlight them, or use the **Set All** or **Clear All** button to highlight all or deselect all, respectively. For example, if you set all layers, you can then deselect one or two layers that you want to exclude. Click *Add*.
- *Modify an existing output file*. Select (highlight) a filename from the list. Keep the same extension, but change the settings accordingly, including X, Y offset, and layers to be included. Click *Modify*.
- *Create a new output file from an existing one.* Select (highlight) a filename from the list. Change the extension of the file, and either change settings and included layers, or leave them alone, as appropriate. Click *Add*.
- *Delete an output file from the list.* Select (highlight) a filename from the list. Click *Delete*.

Specify the destination of your output files by specifying the path in the **Output Path** textbox.

You can also enable or disable the option to **View error file upon completion**.

Click **Close** to save your output files settings and return to the main File N/C Drill dialog.

Tools (Assignments)

To assign N/C Drill tools, click the **Tools** button in the File N/C Drill dialog to display the Tool Assignments dialog. In this dialog you can unassign tools, and then either automatically or manually describe/assign what remains.

To manually describe/assign tools, click on (highlight) a line, then click **Assign** to display the Describe/Assign Tools dialog. (Or you can double-click on a hole/tool line to display the dialog.)

To individually unassign a tool, select a tool assignment in the list to highlight it, then click **Unassign**.

To unassign all tools (usually before you automatically assign), click **Unassign All**.

To automatically describe/assign all remaining unassigned tools, click **Auto**. When you enable the **Clear current tools** option, the tools will automatically clear when you use **Auto**.

Click **Close** when your descriptions and assignments are set.

Additional Information:

Assign (Manual)

Assign Manual (N/C Drill)

Click *Assign* to display the Describe/Assign Tools dialog, where you can manually describe/assign tools.

The hole name of the hole or tool you highlighted on the previous dialog appears at the top of this dialog in the **Assign Tool to** field.

All tools are listed in the listbox; if the hole already has a tool assigned to it, then that tool is highlighted, and the tool's **Tool code** and **Diameter** values appear in the edit boxes. If the hole has no tool assigned (no tool is highlighted) then the contents of the edit boxes contain the next available tool code and hole diameter.

In the **Tool Code** text box, you can enter a value in the range of 1 through 80 for the tool code used to select the drill bit. The program automatically prefixes the value with the letter **T** and fills in the leading zero for codes **T01** through **T09**.

In the **Diameter** textbox, you can enter a value to determine the diameter of the drill bit.

There are four procedures available in this dialog:

- *Create a new tool assignment* by describing/assigning the tool (filling in the T Code and Diameter) and clicking *Add* to assign it to the hole.
- *Modify an existing tool assignment* by changing the Diameter, but keeping the same T Code, then clicking *Modify*. The assignment remains the same (tool to hole), but the diameter of the tool has been changed.
- *Use an existing tool to create a new one* by changing the name (T Code) of the tool, and then using the same diameter and hole assignment. You would then click *Add*, in effect creating a new tool assignment.
- *Delete a tool* assignment/description by simply clicking *Delete*. The tool assignment must be highlighted to delete it.

Click *Close* to save changes. You are then returned to the Tool Assignments dialog, where the assignments you've made will be listed.

N/C Drill Format

To establish N/C Drill format settings, click **N/C Drill Format** (from the File N/C Drill dialog) to display the N/C Drill Format dialog.

For Units, if you select **Inches**, the units are in inches and the format is automatically set to 2.4 (two digits to the left of the decimal point and four digits to the right).   If you select **Millimeters**, the format is automatically set to 4.2 (four digits to the left of the decimal point and three digits to the right).

Select the appropriate **Output Code Type** and **Zero Suppression** options.   Check with the fabrication house for the appropriate settings.   These options are saved in your PCB.INI file.

After you have established settings in N/C Drill Format, click *Close* to save settings and return to the File N/C Drill dialog.

File PDIF In

Allows you to load PCAD.PDF format design files.   When you run this command, the PDIF File Name dialog appears.

1. Select File PDIF In to open the dialog.
2. Type, or select from the list, the name of the file you want to open in the **File Name** box.
3. If the file you want is not in the current directory, then either type the directory name in front of the document name, or select the directory from the **Directories** box.
4. Click **OK**.

Refer to the chapter Using P-CAD files for more information.

ACCEL Tango PCB

If any of the design limits are exceeded, your design cannot be loaded using ACCEL Tango PCB.
For additional information, refer to the PCB Users Guide.

File PDIF Out

The File PDIF Out command is used to create PDIF files of your PCB designs. When you run the File PDIF Out command, the PDIF File Name dialog appears.

PDIF File Export

2. Type the filename you want to use in the **File Name** box.
3. If the current directory is not appropriate, then either type the directory name in front of the document name, or select the directory from the **Directories** box. You can move through the directory tree by selecting the appropriate directories from the list.
4. Make sure the extension is correct by checking the **Save File as Type** box.
5. Click *OK* to save the file as you have specified.

For additional information, refer to the Using PDIF document.

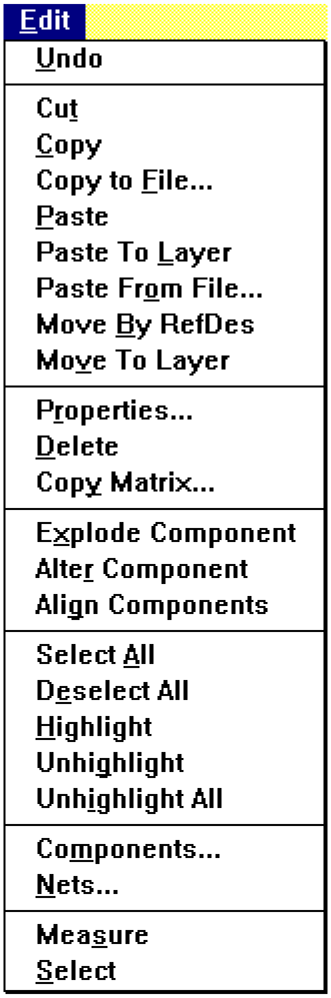
File Exit

Exits the PCB program.

If any open designs have been modified since the last save, you are prompted whether you want to save the changes to each design.

The program writes information to the PCB.INI file when you run Exit.   This information, which will apply to subsequent PCB sessions, consists of parameters and settings such as workspace size, units and values set in Options Configure, report file settings from File Reports and Utils DRC, etc.

Edit Menu (PCB)



Edit Undo

Undo reverses your last completed action to the design file.

You can undo Place commands, Delete, Copy Matrix, and Alter Component, and modifications such as move, rotate, and flip (actions performed using the Select tool). Many File commands, such as File New, Save, Print, etc. cannot be undone. If an action cannot be undone, Undo appears grayed on the Edit menu.

You can only undo a finished Place command. For example, if you are placing an object that requires more than just one click in the workspace (such as drawing lines, polygons, arcs), you must finish the segment or arc before you can undo it.

To unwind line segments (delete the previous segment) before you finish the series of segments, use the backspace key.

This function can be enabled with the Undo button from the toolbar.

See also:

Delete

Edit Cut (Ctrl+X)

Cut removes objects/items from your design and saves them to the Windows clipboard, from where you can paste them into another design or to another location within the current design.

You must enable the Select tool (Edit Select) and have at least one item selected to run this command; otherwise the menu item is grayed and the Ctrl+X shortcut key is inoperative.

Once items are in the clipboard, you can either immediately paste them to another design location, or save them to a clipboard file (.CLP). Use File Save in the Windows Clipboard application to create a clipboard file. You can paste a clipboard file into your design file. The clipboard format in this case is specific to PCB and cannot be used by non-PCB applications.

You can cut multiple objects by using multiple select and block select operations. Refer to Edit Select for information.

Cutting Objects from Nets

When you Edit Cut to remove objects from nets, you can get a variety of results, depending on what you cut and the makeup of the net you remove it from. The function of smart nets is to maintain certain connections when objects such as copper connections, unrouted connections, and net nodes are removed. In general, the following can occur.

- If you remove free copper (no connections), then the copper disappears and no connection compensation occurs.
- If you remove a node, then the remaining nodes are still part of the net, and there will be compensation to maintain connections between the remaining nodes.
- If you remove a component, you are removing nodes from all nets to which the component's pads were connected. The connectivity feature of PCB reconnects the remaining nodes in each net in the most efficient way.
- If two nets become merged, and one net is a plane net, then the plane layer net takes precedence and the merged net is a plane net.
- If you remove a connection from the middle of a net, the net is split. One portion retains the net name and the other is given a new net name.
- If you remove a connection that isolates a pad from the rest of the net, you end up with a disconnected node.
- If you remove a copper segment that is part of a net, that segment is not removed, but instead becomes a connection (in effect unrouting the net). In this case, the net remains intact, although changed.

Edit Copy

Allows you to copy objects/items to the clipboard, from where you can paste them to another design, or to another location within the same design.

You must enable the Select tool (Edit Select) and at least one item selected to run this command; otherwise the menu item is grayed and the Ctrl+C shortcut key is inoperative.

You can use Ctrl + left mouse button (a drag-and-drop operation) to copy within the same design.

Once items are in the clipboard, you can either immediately paste them to another design location, or save them to a clipboard file (.CLP). Use File Save in the Windows Clipboard application to create a clipboard file. You can paste a clipboard file into your design file. The clipboard format in this case is specific to PCB and cannot be used by non-PCB applications.

Refer to the Edit Paste command for rules and parameters for pasting objects.

Edit Copy to File

Allows you to copy selected objects to a block file, from which you can paste them at a later time using the Paste From File command.

You must enable the Select tool (Edit Select) with at least one item selected to run this command; otherwise the menu item is grayed.

When you run the Copy to File command is selected, the Edit Copy to File dialog box appears.

The **Directories** area displays the current path and a list of directories for changing the path. The **File Name** area lists all the files of the current directory, with the extension specified in the **Save Files as Type** area. Notice that the default file extension is .BLK.

Specify a location and name of the block file to which you wish to copy the selection. You can copy multiple objects by using multiple select and block select operations. Refer to the Edit Select command section for more information.

Edit Paste (Ctrl+V)

Allows you to paste objects/items from the clipboard into your design file.

When you choose Edit Paste, your cursor becomes an action pending cursor (a *crosshair*-style shape), until you click in the Workspace to paste the item(s). Items are pasted at the cursor position.

The Paste command handles only clipboard items that are in PCB format.

If any of the design limits are exceeded, the object/item cannot be pasted in your design using ACCEL Tango PCB. See ACCEL TangoPCB features.

Paste Item(s)

1. Choose Edit Paste, and click in the workspace in the appropriate location.
2. A ghosted shape of the item appears in the workspace until you release the mouse button. But before you release the mouse button, you can drag the item to a more precise location and then release.
3. The area you are pasting in must contain available space for the item you are pasting. The computer will beep if the object is too large for the space or if you are attempting to paste too close to the edge of the workspace.

Pasting On Layers

If an item is layer-specific, then it will be pasted onto the same layer that it was cut/copied from, regardless of the current layer setting. If the target layer doesn't exist, then the item(s) will not be pasted. To paste to a different layer, use Paste to Layer.

When pasting a component from one design to another (or placing a component from a library), if components of the same name but from a different library already exist in the destination design, the component cannot be pasted/placed due to the probable conflict in pin assignments. This conflict could also occur when components from a Series II board are mixed with components of the same name in a PCB library. In effect, the first instance of the component name establishes the standard.

When you paste vias, pads, and text (usually in a component) from a different design that contain styles that have the same names but different data than those in the current design, the incoming style names will be bracketed to indicate the style conflict. The new (bracketed) style names will be added to the list of available styles in the current design. Refer to the Edit Modify, Options Pad Style, and Options Text Style commands for object style information. If you paste a component that has the same RefDes as a component already in the design, an A is appended to its RefDes (e.g., U10A).

Edit Paste to Layer

Allows you to paste items to a different layer from what they were cut or copied from, within the same design or to a different design. This feature includes single, multiple, or block selection cuts and copies.

The Edit Paste To Layer command accepts only clipboard data that are in PCB format. Clipboard data from other applications cannot be pasted into the PCB Workspace.

- If you block select items that reside on different layers, e.g., one line on the Top layer and one on the Bottom, they will both be pasted onto the current layer.
- Multi-layer items, such as pads, will remain as multi-layer.
- When items are ghosted for pasting, you can press the L key to toggle between layers before you do the final paste.
- You can paste to layer from a clipboard file (.CLP) as well as directly from the clipboard. This clipboard data cannot be loaded or pasted to or from any other Windows utility; it is PCB-specific.
- When pasting a component from one design to another (or placing a component from a library), if components of the same name but from a different library already exist in the destination design, the component cannot be pasted/placed due to the probable conflict in pin assignments. This conflict could also occur when components from a Series II board are mixed with components of the same name in a PCB library.
- When you paste vias, pads, or text from a different design that have the same style names but different data than those in the current design, the incoming style names will be bracketed to indicate the style conflict. This would normally occur with components, which contain vias, pads, and text. The new (bracketed) style names will be added to the list of available styles in the current design. Refer to the Edit Modify, Options Pad Style, and Options Text Style commands for object style information.

Edit Paste from File

Allows you to paste items from a block file into the current file. The block file must have been previously created using the Edit Copy to File command. When selected, the Edit Paste from File dialog appears.

The **Directories** area displays the current path and a list of directories for changing the path. The **File Name** area lists all the files of the current directory, with the extension specified in the **List Files of Type** area.

Select the block file containing the item(s) you wish to paste. Once the file is selected, this command works like the Edit Paste command.

If any of the design limits are exceeded, the object/item cannot be pasted from the Clipboard using ACCEL Tango PCB. See ACCEL TangoPCB features.

Edit Move to Layer

This command allows you to select a number of objects, and then move the objects to another layer. This facility will enhance the your ability to clear out congested areas for routing.   Additionally, if you have accidentally placed objects on the wrong layer, you can move them to the correct to the correct layer.

To Move an Object

To move one or more objects to a different layer:

1. Select the object or objects.
2. Change to the layer where you want the objects moved.
3. Run the Edit Move to Layer command or use ***Shift-T***.

   All the single-layer objects are moved to the current layer.   You will be returned to the Select tool and all the moved objects will be selected.

Restrictions

- If an object has net information it can move between signal layers only.   Otherwise it can move to any layer.
- Multi-layer objects will not be moved (components, glue dot points, keepouts, pads, pick and place points, reference points, or vias).   Objects that can be moved are single layer objects (arcs, copper pours, cutouts, keepouts, lines, polygons, attributes, fields, and text).
- If multi-layer and single-layer objects are selected simultaneously the tool will ignore the multi-layer objects and only move the single-layer objects.
- Single-layer objects on different layers will all be moved to the current layer.
- If the single-layer objects are of mixed net and non-net types and you are moving them to a non-signal layer, a warning message will appear telling you that only the non-net objects will be moved.   You can cancel this command at this time.
- The Move To Layer performs auto-insertion of vias with a slight difference compared with the Route Manual command.   A via will not be inserted if a line segment is already connected to a pad or via. If the existing pad or via is incorrect for maintaining physical connectivity, the line segment will not be moved.
- When moving copper traces between layers, vias are not added for free copper, only for net copper.   The net attribute VIASTYLE will be honored as it is in Route Manual.
- When multiple objects are moved and the physical connectivity cannot be maintained for some of the objects (e.g.,if a via cannot be added), a warning message appears prompting you to continue or cancel the command.   If you press ***Cancel***, the objects moved before the cancel will not be moved back. If you press ***Continue***, the object with the error will not be moved and the tool will skip to the next object.
- If a copper pour is currently poured, it will be unpoured before the move and automatically repoured after a move.   If the pour is not correct, use Modify Copper Pour to repour the pour.   If the pour is not poured, it will remain unpoured after the move.   A pour's net connectivity will be maintained.

Edit Properties

Displays the Properties dialog for the selected object(s). This dialog lets you query and modify the selected objects properties. Except when selecting components and nets, you must use the Select tool to select the object(s) before you can run Edit Properties.

The Properties dialog that appears is specific to the object you select. If multiple objects are selected, they must all be of the same type (e.g., arcs), otherwise the command is grayed and no dialog will appear. If the objects are of the same type, the changes you make apply to *all* selected objects.

Right Mouse Button to Query/Modify

You can also run the Properties command from a popup menu. Select an object and click the right mouse button to bring up a pop-up menu. The items available from this pop-up menu vary depending on the object you select.

Double Click to Query/Modify

You can also run the Properties command by double clicking on an object to open that objects Property dialog, if the option is enabled in the Options Preferences dialog.

To learn how to query or modify an objects properties, click an object from the list:

Component Properties	Cutout Properties
Line Properties	Plane Properties
Arc Properties	Text Properties
Pad Properties	Attribute
Via Properties	Field Properties
Polygon Properties	Connection Properties
Copper Pour Properties	DRC Error Indicator Properties

Plane Properties

When you select a cutout and run the Edit Properties command, the Cutout Properties dialog appears with the Plane tab selected.

The Net tab on the Connections Properties dialog provides access to Net information.

The Plane Properties dialog presents the following information:

- **Net:** Use the drop down list box to assign the plane to an existing net.
- **Boundry Width:** This box define the line width for the Polygonal outline. The default is the current defaut line width. Change the width by typing over the default value.

Net Plane Color Button

Select a net plane color, by clicking the Net Plane Color button and selecting a color from the Color dialog.

Connection Properties

When you select a connection and run the Edit Properties command, the Properties dialog appears with the Connection tab selected.

For the connection Start and End points, you see these query fields:

- **Net Name:** The net name of the connection.
- **X and Y coordinates:** The X and Y location of the start and end point.
- **Node:** The node name to which this connection is connected.
- **Type:** The type of start or end point shows what the connection is connected to (e.g., pad).

The Net tab on the Connections Properties dialog provides access to Net information.

Copper Pour Properties

When you select a copper pour and run the Edit Properties command, the Properties dialog appears with the Style tab selected.

This dialog allows you to establish or change the characteristics of a copper pour object. A typical scenario would be to draw multiple copper pour outlines (Place Copper Pour), then select them all in a multiple select action, and modify them all at once.

When you click *OK*, the copper pour shape fills in (if **Poured** was enabled). The copper pour polygon will remain selected (in the select highlight color) until you click outside of the shape.

warning:

You can mistakenly split the copper pour in half (or more than two pieces) by modifying lines or polygons that were inside the pour region to where they are longer than the copper pour polygon. In that case, the electrical connection would be severed.

On the Style tab, you'll have following options available to you:

Line Options

Line Width determines the width of the lines used in filling and hatching.

Line Spacing determines the separation between fill or hatch lines. If you selected **Solid** for the Fill Pattern, then the **Spacing** value is grayed.

Backoff Option

The **Backoff** option allows you to set the distance between the copper pour and any objects that may be *inside* of the copper pour polygon. This option also backs off from any objects that are *outside* the copper pour polygon if they are too close to it. The copper pour backoffs from any copper item that is not in the net associated with the pour. The backoff options takes the objects thickness into account.

If you choose the **Fixed** radio button, you can specify a value for this distance.

If you choose **Use Design Rules**, the backoff distance for a specifc net can be set using design rules (see the Options Design Rules command). Backoff clearances are fixed at the greatest of the Line to Line or Line to Pad clearance amount set for the current layer in Options Design Rules dialog.

State Options

Poured Unpoured and **Repour** are the fill states of the Copper Pour. Repour can be selected to force an already filled pour to recalculate its islands.

This tab also allows you to select Fill Patterns, and to set the Backoff Smoothness.

The following tabs on the Properties (Copper Pour) dialog provide access to additional information:

Connectivity
Island Removal

Net

Fill Patterns

Select one of the following **Fill Patterns**:

- **Solid** copper fill will actually be banded (striped) with lines.
- **Horizontal** fills the pour with horizontal lines (no hatching).
- **Vertical** fills the pour with vertical lines (no hatching).
- **45-degree Cross** is *cross-hatched* diagonally, at a 45-degree angle (like X).
- **90-degree Cross** is *cross-hatched* horizontally and vertically, at a 90-degree angle (like +).

Thermals

The **Thermals** options determine whether to use thermals, and if you do, what type.

- **None** specifies that the copper will pour right over pads and vias (no thermals) that belong to the same net as the copper pour (i.e., direct connect).
- **45** and **90** specify 45- or 90-degree thermals.

Backoff Smoothness

These options allow you to set the smoothness of backoff polygons.   There are three objects:

- **Low** specifies eight- to 10-sided polygons.
- **Medium** specifies 12- to 14-sided polygons.
- **High** specifies 16- to 18-sided polygons.

Connectivity

When you select the Connectivity tab, the Properties dialog appears with the following options:

Thermals

The **Thermals** options determine whether to use thermals, and if you do, what type.

- **None** specifies that the copper will pour right over pads and vias (no thermals) that belong to the same net as the copper pour (i.e., direct connect).
- **45** and **90** specify 45- or 90-degree thermals.

Spoke Width

The **Spoke Width** option allows you to specify a value for the width of the thermal spokes.

Net Combo Box

The **Net** combo box allows you to list the net that you want the copper pour to connect to. The copper pour will backoff from all copper items not in this net. If no net is specified, the copper pour backs off from all copper objects inside the pour region except other "no-net" copper pours.

The following tabs on the Copper Pours Properties dialog provide access to additional information:

Style

Island Removal

Net

Island Removal

When you select the Island Removal tab, the Properties dialog appears with these three Automatic Island Removal objects, which allow you to set the smoothness of backoff polygons:

- **Minimum Area** specifies a minimum area an island can have before being removed. Enter a minimum value (in square units) in the text box.
- **Interior** specifies that all islands in a pour that dont have an edge in common with the perimeter of the copper pour are removed.
- **Do not repour** performs the island removal without repouring the pour. This is a very fast way to remove unwanted islands since you do not have to wait for the pour to regenerate.

Cutout Properties

When you select a cutout and run the Edit Properties command, the Properties dialog for cutouts appears.

The Properties dialog for cutouts shows you the following information:

- **Layer:** The layer on which the cutout appears.
- **Points:** The X and Y location of each point in the cutout polygon.

Field Properties

When you select a field and run the Edit Properties command, the Fields Properties dialog appears.

Query Fields

The Properties dialog for fields lets you view the following:

- **Value:** The field value entered in the File Design Infor dialog.
- **Rotation:** The Rotation field shows the rotation amount if the field has been rotated.

Name

The field name appears in the **Name** drop down listbox. You can change the name by selecting a new field from the list of field name.

If you change the field name, its value changes accordingly.

Location

The X and Y coordinates of the selected field appear in the **Location** box. You can move the field, by typing new coordinates.

Justification

Nine buttons allow you to change justification by setting the *reference point* of the field. For example, if you enable the middle button, the field reference point (the lower-left corner) moves to the center of the bounding rectangle.

Text Style

Text styles appear in the drop down listbox. To change the **Text Style**, click on the text style you want from the **Text Style** listbox.

Clicking the Text Style button displays the Options Text Style **dialog.**

From this dialog you can add, delete, rename, or edit text styles.

Layer

The layer on which the field is placed appears in the **Layer** drop down listbox. You can change the layer by selecting a new field from the list of field name. @@IS THIS GONE??

Arc Properties

When you select a free arc and run the Edit Properties command, the Arc Properties dialog appears.

Query Fields

For the free arc Center, Start, and End points, you see:

- **X and Y coordinates:** The X and Y location of the Start and End point.
- **Tangent Slope Angle:** The tangent slope angle of the arc.

Change Arc Properties

You can only change these properties if the arc is not part of a net and is not connected to other objects.

To change the arcs Start Angle: use the scroll buttons (up and down arrows) to scroll through arc start angles or type in the new value.

To change X and Y coordinates of the Center Point: type over the existing X and Y values in the Center Point box.

To change the arcs Sweep Angle: use the scroll buttons (up and down arrows) to scroll through arc sweep angles.

To change the arcs Radius: type a new radius over the existing value.

To change the arcs line Width: type a new width over the existing value.

Line Properties

When you select a free line and run the Edit Properties command, the Line Properties dialog appears.

This dialog shows you the **Length** of the selected line.

Change Line Properties

You can change line properties as follows:

To change the lines Width: type a new width over the existing value.

To change the lines End Points: type new X and Y coordinates over the existing values. You can only change end points if a line is not part of a net and is not connected to other objects.

Pad Properties

When you select a free pad and run the Edit Properties command, the Pad Properties dialog appears with the Pad tab selected.

The Net tab on the Pad Properties dialog provides access to Net information.

Query Fields

The Properties dialog for free pads shows you the following:

- **Net Name:** The name of the net associated with the selected pad.
- **Tool Code:** The tool code used to select the drill bit for the corresponding hole diameter. See the File N/C Drill command for additional information.
- **Flip:** The **Flip** box indicates whether or not the pad has been flipped.
- **Rotation:** The Rotation field shows the rotation

Change Pad Properties

You can change pad properties as follows:

Pad Number: Changing the pad number is useful when you only want to change one or two pads. If you want to renumber a series of pads, use the Utils Renumber command for smoother and faster action.

Location: Enter new X and Y coordinates for the pad.

Pad Style: You can change the style of the selected pad by selecting one of the existing pad styles in the **Pad Style** listbox.

Clicking the **Pad Styles** button displays the Options Pad Style dialog. From this dialog you can modify a pad style.

Free Polygon Properties

When you select a free polygon and run the Edit Properties command, the Properties dialog appears.

The Properties dialog for free polygons shows you these two fields:

- **Area:** The area of the polygon.
- **Points:** The X and Y location of each point in the free polygon. You can type over the existing values.

Text Properties

When you select a text and run the Edit Properties command, the Text Properties dialog appears.

From this dialog, you can change the text content, justification, and style. You can also modify any non-default text styles.

Query Fields

The following fields are available for viewing only:

- **Flip:** The **Flip** box indicates whether or not the text has been flipped.
- **Rotation:** The Rotation field shows the rotation of the text.

Changing Text Properties

To change the Text: Type over the text displayed in the **Text** box and click *OK*. For multi-line text, *Enter* creates a line break. You can enter a maximum of 2,000 characters.

While the **Text** edit box has focus, you can use *Ctrl+V* to paste text from the Windows clipboard.

To change the text Location: The X and Y coordinates of the selected text appear in the **Location** box. You can move the field, by typing new coordinates.

To change the text Justification: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

To change the Text Style: Click on the text style you want from the **Text Style** listbox.

Text Style

Text styles appear in the drop down listbox. To change the **Text Style**, click on the text style you want from the **Text Style** listbox.

Clicking the Text Style button displays the Options Text Style **dialog.**

Attribute

When you select and attribute and run the Edit Properties command, the Attribute Properties dialog appears.

The following information appears in the dialog:

- **Category Listbox:** Displays a list of all attribute categories, All, Component, Net, Clearance, Router and SPECCTRA. Selecting a category brings up a list of pre-defined attributes for that category.
- **Name Listbox:** Displays all pre-defined attributes for the specified category. The first entry in the list is *User-defined*.

  The currently-selected attribute also appears in the **Name** edit box, unless *User-defined* is selected. In that case, the Name edit box is blank so that you can enter a user-defined attribute name.
- **Name Edit Box:** For user-defined attributes, enter a name for the attribute.

note:

If the dialog is accessed for an attribute that already has a name, then the Category listbox, Name listbox, and Name edit box are filled in, but grayed. If the attribute doesnt have a name, these controls are enabled.

- **Value:** Use this edit box to enter a value for the attribute.
- **Visible:** This checkbox indicates whether or not the attribute is visible.
- **Location:** This area shows the X and Y coordinates of the components reference point.
- **Text Style:** This area lets you select the Ref Des, Value, and Type text style. Text styles appear in the **Text Style** drop down listbox. To change the Text Style, click on the text style you want from the listbox.
- **Layer**: This combobox shows the name of the layer on which the attribute is located. To change the layer, select a new one from the drop down list.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped** Box: This box indicates whether or not the pattern has been flipped.
- **Justification**: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

Component Properties

When you select one or more patterns and run the Edit Properties command, the Component Properties dialog appears with the Pattern tab selected.

The Component Properties dialog allows you to observe properites for the selected component and to modify certain component properties. The selected component appears in a view area of the dialog.

Pattern Tab

The following information appears in the dialog:

- The **Ref Des** box shows the reference designator name. To change the reference designator, type a new value in the **Ref Des** field. If you selected more than one component, this value cannot be changed.
- The **Value** box shows the components value. To change the value, type a new value in the **Value** field.
- The **Type** drop down list box shows the component type.
- The **Libraries** box lets you select a library when you want to change the type of component.
- The **Visibility** area contains checkboxes indicating whether the selected component(s) have visible, invisible, or undetermined Ref Des, Value, and Type attributes.

  If a box is checked, the attribute is visible. If the box isn't checked, the attribute is invisible. If the box is grayed, then the attribute either does not exist (e.g., there is no Value attribute for the selected component), or there is a conflict between multiple components selected (e.g., the attribute on one component is visible, but is invisible on another).
- The **Text Style** area lets you select the Ref Des, Value, and Type text style. Text styles appear in the **Text Style** drop down listbox. To change the Text Style, click on the text style you want from the listbox.
- The **Pattern** field displays the pattern name. To change the pattern, type a new pattern name in the box. Changing the pattern in this dialog doesn't change the pattern attached to the component in the library.
- The **Location** area shows the X and Y coordinates of the components reference point.
- The **Flip** box indicates whether or not the pattern has been flipped.
- The **Rotation** field shows the rotation amount if the pattern has been rotated.
- The **Glue Dot Locations** listbox shows a list of all glue dot locations. Glue Dots are used to hold components in place until they are soldered during manufacturing.
- The **Pick and Place Locations** listbox shows a listing of all pick and place locations. Pick and Place points provide reference points in directing the *pick and place* mechanism (or *auto insert*) in manufacturing (picking up the component and placing it on the board).

To replace a component, click the following topic:

Replacing a Component

Clicking the Text Style button displays the Options Text Style dialog.

In addition to the Pattern tab, the following tabs on the Component Properties dialog provide access to additional information:

Pattern Pads Tab
Component Tab
Component Pin Tab
Attributes Tab

Replacing a Component

To replace the component(s) associated with this pattern with another component from any open library:

1. To select a component from another open library, select a new library from the **Libraries** listbox.
2. Select a new component from the drop down listbox.
3. Click *OK* to replace the component or components with the new component.

   The new component is placed at the same Reference Point and same Rotation as the old.

   The net connectivity is maintained after the swap.

   If connectivity can't be maintained, a warning message appears and the component is swapped.

note:

Using the Properties function to swap a component can result in changes to the netlist if the pin designators on the replacement component are not the same as those on the original component.

If a warning message indicates that netlist changes have occurred, not only may some netlist nodes and their corresponding from-to connections be missing, but the net names may be removed if they result in a single node net. Additionally, any intelligent copper connected to the pads that are no longer netlist nodes are stripped of their netlist information. Use Edit Undo if you want to undo the component swap.

Component Tab

When you select the Component tab, the Component Properties dialog appears showing component information.

This dialog shows information for the component or components you selected.   It shows information on a gatebygate basis.   This information is display-only; it cant be modified from this dialog.

To show information for a different gate, select the gate from the **Gate Number** drop down list box.

Component Pins Tab

When you select the Component Pins tab, the Component Properties dialog appears. Use this dialog to look at pin information for the component pins within the symbol or component.

The following information appears:

- **Pad #:** The number of the corresponding pad on the attached pattern.
- **Pin Des**: The pin designator of each pin in the component.
- **Gate #**: The part number defines the part that the pin is associated with. In multi-part components, the parts are uniquely numbered from 1 through n.
- **Pin #**: The number of the corresponding pin on the attached symbol.
- **Pin Name:** The pin name associated with that pin designator.
- **GateEq**: The gate equivalence column defines which gates are equivalent. All gates with the same GateEq number are defined to be equivalent. This information is used by PCB Schematic when automatically incrementing reference designators (e.g., Place Part and Utils Renumber commands) and by PCB to determine which gates can be swapped during manual or automatic gate swapping.
- **PinEq**: Indicates which pins within a gate are logically equivalent and may be swapped using the Utils Optimize Nets pin swap commands. The pin equivalence values must be non-zero and identical for a swap to occur between two pins. Non-swappable pins are indicated with a zero value.
- **Elec Type:** The electrical type of the pin.

Attributes Tab

Allows you to display and modify component attributes for the selected component. When you click the Attributes tab, the dialog appears.

You can view, add, modify, or delete a collection of component attributes. The dialog contains a two-column table showing the collection of component attributes. Within the collection, each attributes name and value appear in the column.

- **Changing an Attributes Value:** To change a value, select it and type the new value over the existing value. The name column cannot be changed
- **Adding an Attribute**: To add an attribute, click the **Add** button to open the Attribute dialog. Enter the name and value for the attribute and set attribute properties. Click **OK**, and the attribute is added to the table.
- **Viewing or Changing Attribute Properties:** To view or change an attributes properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the Attribute dialog.
- **To Delete an Attribute**: Select an attribute in the table and click **Delete**, or press the *Del* key.

Attribute Dialog

The following information appears in the Attribute dialog:

- **Category Listbox:** Displays a list of all attribute categories, All, Component, Net, Clearance, Router and SPECCTRA. Selecting a category brings up a list of pre-defined attributes for that category.
- **Name Listbox:** Displays all pre-defined attributes for the specified category. The first entry in the list is *User-defined*.

  The currently-selected attribute also appears in the **Name** edit box, unless *User-defined* is selected. In that case, the Name edit box is blank so that you can enter a user-defined attribute name.
- **Name Edit Box:** For user-defined attributes, enter a name for the attribute.

note:

If the dialog is accessed for an attribute that already has a name, then the Category listbox, Name listbox, and Name edit box are filled in, but grayed. If the attribute doesnt have a name, these controls are enabled.

- **Value:** Use this edit box to enter a value for the attribute.
- **Visible:** This checkbox indicates whether or not the attribute is visible.
- **Location:** This area shows the X and Y coordinates of the components reference point.
- **Text Style:** This area lets you select the attribute text style. Text styles appear in the **Text Style** drop down listbox. To change the selected Text Style, click on the text style you want from the listbox. To modify the text style, click the **Text Style** button.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped** Box: This box indicates whether or not the pattern has been flipped.
- **Justification**: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

Pattern Pads Tab

When you select the Pattern Pad tab, the Component Properties dialog appears. This dialog allows you to change the pad style of specific pins within the component and edit the pad style itself.

Query Fields

The following information appears (for display purposes only):

- **Location:** The X and Y coordinates of the selected pad.
- **Flip:** The **Flip** box indicates whether or not the pad has been flipped.
- **Rotation:** The Rotation field shows the rotation amount if the pad has been rotated.
- **Net Name:** The net name for the net to which the pad is attached.
- **Tool Code:** The tool code used to select the drill bit for the corresponding hole diameter. See the File N/C Drill command for additional information.

This dialog has two listboxes. To learn about them, click on one of the following topics:

Pads Listbox
Pad Styles Combo Box

Pads Listbox

The **Pads** listbox lists the pin numbers of the pads in the selected component.   The information fields to the right display information about the highlighted pad.

The **Set All** and **Clear All** buttons can be used to highlight and unhighlight all pads in the listbox.

Pad Styles Combo Box

The **Pad Style** list box allows you to perform single or multiple pad editing. The pad numbers are highlighted in the **Pads** listbox and you can either change to another pad style for those pads, or you can click *Pad Styles* to display the Options Pad Style dialog, allowing you to modify an existing pad style, rename it, or create a new one based on an existing style.

Click **Apply** to apply any modifications to the selected pad or pads.

Via Properties

When you select a via and run the Edit Properties command, the Properties dialog for vias appears.

Use this dialog to view or change the style of the selected via and to change its location.

The Net tab on the Via Properties dialog provides access to Net information.

To learn about this dialogs query fields, click the following topic:

Via Properties Query Fields

To learn about changing pad properties, click the following topic:

Changing Via Properties

To learn about the Text Style button, click the following topic:

Via Styles Button

Via Properties Query Fields

The following fields are available for viewing only:

- **Net Name:** The name of the net associated with the selected via.
- **Tool Code:** The tool code used to select the drill bit for the corresponding hole diameter. See the File N/C Drill command for additional information.
- **Flip:** The **Flip** box indicates whether or not the via has been flipped.
- **Rotation:** The Rotation field shows the rotation of the via.

Changing Via Properties

You can only change these properties if the via is not part of a net and is not connected to other objects.

To change the via location: The X and Y coordinates of the selected via appear in the **Location** box. You can move the field by typing new coordinates.

To change the via style: Click on the via style you want from the **Via Style** listbox.

Via Styles Button

Click *Via Styles* to display the Options Via Style dialog for via styles. From this dialog you can add a new style based on an existing style, modify a non-default via style, or delete a non-default via style.

Modify (View) DRC Errors

Using the Edit Modify command or Modify from the Select popup menu allows you display a dialog from which to view DRC error/violations through indicators visible in your design. Each indicator represents an error or violation detected in the design rule checking pass (Utils DRC). You can block select error indicators and cycle through them by error number.

To generate DRC error indicators, use the Utils DRC command.

To block select DRC error indicators, use the Options Block Selection command to include the indicators in a block select, then use Edit Select to perform the block select.

To display DRC error indicators, use the Options Display command, click **Misc...** and click **Show DRC Errors**.

View DRC Errors

1. Select DRC error(s) either individually or through a block select.
2. Click the right mouse button and select Modify from the popup menu, or choose Edit Modify to display the View DRC Errors dialog.
3. The dialog is view-only, therefore you are limited to reading error information, or scrolling through the errors if you have selected multiple errors in the design.

   Use the **Next** and **Previous** buttons to scroll forward and backward through the errors.

Edit Delete

Deletes all *selected* objects. Use the Del key as a shortcut.

This command does not cut the data to the clipboard (as does Edit Cut). As there is nothing to paste, the only way to reverse Delete is to use the Edit Undo command.

You can also access Delete from the popup menu (select an object, then click the right mouse button to display).

To Delete Objects

1. Choose Select from the Edit menu (or use the Select button on the Toolbar). Click on the object you want to delete.
2. Choose Delete from the Edit menu, press the *Del* key, or click the right mouse button and select Delete from the popup menu. If you mistakenly delete an object, choose Undo from the Edit menu to reverse the delete action.

   This operation can also be performed with multiple objects by using multiple select and block select operations. Refer to Edit Select.

Deleting Objects from Nets

When you delete objects from nets, you can get a variety of results, depending on what you delete and the makeup of the net you delete from. The function of smart nets is to maintain certain connections when objects such as copper connections, unrouted connections, and net nodes are deleted. In general, the following can occur.

- If you delete free copper (no connections), then the copper disappears and no connection compensation occurs.
- If you delete net copper, then it will be replaced with connections to maintain the net.
- If you delete a node, then the remaining nodes are still part of the net, and there will be compensation to maintain connections between the remaining nodes.
- If you delete a component, you are removing nodes from all nets to which the component's pads were connected. The connectivity feature of PCB reconnects the remaining nodes in each net in the most efficient way.
- If you delete a connection from the middle of a net, the net is split. One portion retains the original net name and the others are given new system-generated names.
- If you delete a connection that isolates a pad from the rest of the net, you end up with a disconnected node that is no longer part of any net.
- If you delete a copper segment that is part of a net, that segment is not deleted, but instead becomes a connection (in effect unrouting the net). In this case, the net remains intact, although changed.

Related Topics

Select
Undo

Edit Copy Matrix

Duplicates all the selected objects according to the parameters you specify. The objects must be selected first before Copy Matrix can function.

In the Edit Copy Matrix dialog, the **Number of Columns** and **Number of Rows** determine the number of X (horizontal) and Y (vertical) duplications, respectively, of a selected object.

Column Distance and **Row Distance** allow you to enter a value, in current units, to determine the spacing between the duplicated objects. With Column, a positive value will duplicate to the right, a negative value to the left. With Row, a positive value will duplicate up, a negative value down. For example, if you specify 200 mil for Column Distance and 200 mil for Row Distance (and specify 3 rows and 3 columns), the result is a matrix with 9 objects 200 mils apart.

The values represented in Distance will default to **mm** (millimeters) or **mil**, depending on what you have set in Options Configure (your current units). You can specify a measurement value (overriding Options Configure) by typing in **mil, mm, cm**, or **in** after the numerical value.

To Duplicate an Object(s)

1. Choose Select from the Edit menu. Select the object(s) you want to duplicate by clicking and releasing over the objects to highlight them.
2. Choose Copy Matrix from the Edit menu, and the dialog appears.
3. In the **Number of Columns** box, specify how many duplications you want to perform horizontally. In the **Column Distance** box, enter a value to determine the spacing between duplications and in which direction (positive=right, negative=left) to duplicate.
4. In the **Number of Rows** box, specify how many duplications you want to perform vertically. In the **Row Distance** box, enter a value to determine the spacing between duplications and in which direction (positive=up, negative=down) to duplicate.

   You will receive an error message if what you specify for your duplication is too large to appear the workspace
5. Click **OK**. If your duplication is unsatisfactory, select Undo to reverse the action and try again.

See also:

Select
Undo

Edit Explode Component

This command allows you to convert a component back to its basic primitives, creating a collection of editable graphic objects. When you explode a component, the collection of objects is no longer a component or a pattern.

This feature is useful for modifying an existing component or creating a new component from an existing one. After you explode the component, you can then perform changes to the objects such as adding more pads, changing line size or thickness, renumbering pads, etc.

warning:
Make sure that the component you are about to explode has not been flipped.

To Explode a Component

1. Select the component you want to explode by using the Select button from the Toolbar or with the Edit Select command.
2. While the component is selected (in the Select color), choose Explode Component from the Edit menu.
3. The component then becomes a collection of modifiable objects. To create a component again, you must save the objects as a pattern (Library Pattern Save As command), and then define the pattern as a component in the Library Manager.

To alter an existing component in place without having to save its pattern to a library, use the Sub Select feature.

Refer to the Library Pattern Save As command for more information.

Edit Alter Component

Allows you to select certain component items and subsequently move, rotate, flip, and (in some cases) delete them.

With this editing process you can alter certain component characteristics either for aesthetic reasons or manufacturing improvement, such as avoiding any co-location problems during manufacturing (e.g., through-holes and silkscreen paint). The rules/restrictions are as follows:

- Pads cannot be selected, and therefore cannot be edited in any way.
- RefDes, Type, Value and Ref Point (reference designators and reference points) cannot be deleted.
- If you want to hide the RefDes, Value, or Type, modify the component (select and Edit Modify), and disable the Visibility options. To hide Glue Dot and Pick and Place points, disable their respective display options in Options Display (Misc).
- You cannot undo any alter actions until the operation is complete.

Alter a Component

1. Run the Edit Select command to select the component you want to edit. Then run the Edit Alter Component command. You are now in a temporary editing mode, signified by the *crosshair* cursor shape.
2. Zoom in sufficiently. Change layers if necessary. You can select items individually only if you are on the correct layer, otherwise you need to use a block select. The settings in Options Block Selection can affect your results here.

   Editable items typically reside on the Top Silk layer, except for Ref Points, which aren't layer items, but this depends on how the pattern was originally created.
3. After you have selected individual item(s), you can then move, delete, or otherwise alter them (according to the restrictions mentioned previously). Press Esc or click the right mouse button to end the editing mode.

Edit Select All

Selects *all* items on the current sheet.

Edit Deselect All

Deselects *all* items previously selected.

Edit Highlight

Highlights the selected item or items in one of the highlight colors chosen through the Options Display command. Successive invocations of this command cycle through the defined highlight colors. If you do not want to use two highlight colors, set both colors to be the same in the Options Display dialog box.

If PCB and PCB Schematic are both running, and if the **DDE Hotlinks** checkbox in the Options Configure dialog box is checked in *both* applications, then component and net highlight information is communicated between the two applications. Highlighting a net in one application highlights the corresponding net in the other application; highlighting a pattern in PCB highlights the corresponding symbol in PCB Schematic and vice versa.

When you run this command, the selected items are drawn in the highlight color until they are unhighlighted. The selected color overrides the highlight color, so you won't see the highlights until the items are deselected.

You can also access this command by selecting an item or items, clicking the right mouse button to bring up the popup menu, and choosing Highlight.

Edit Unhighlight

Removes the highlighting from the selected item or items and restores the normal object colors.

If PCB and PCB Schematic are both running, and if the **DDE Hotlinks** checkbox in the Options Configure dialog box is checked in both applications, the highlights are removed from the selected item or items in *both* applications.

You can also access this command by selecting an item or items, clicking the right mouse button to bring up the popup menu, and choosing Unhighlight.

Edit Unhighlight All

Select the Unhighlight All command to remove the highlight from all items on all layers in the design and to restore the normal object colors.   This command applies to all highlighted objects, regardless of whether they are selected or not.

If PCB and PCB Schematic are both running, and if the **DDE Hotlinks** checkbox in the Options Configure dialog box is checked in *both* applications, the highlights are removed in *both* applications.

Edit Nets

Allows you to display, hide, and rename nets, display net information, select net items, and edit net attributes. These functions are not undo-able.

When you run the Edit Nets command, the Edit Nets dialog appears.

Nets Listbox

The **Net Names** listbox contains the names of all nets in the active design. You can select individual or multiple nets in the listbox. Once selected, you can highlight and unhighlight nets and jump to a node in the net.

Nodes Listbox

The **Nodes** listbox contains the names of all nodes in the net that is outlined by the focus cursor in the **Net Names** listbox. If you select more than one node, this listbox is blank.

Set All Nets/Clear All Nets

These buttons allow you to select or deselect all of the nets displayed in the **Nets** list box.

Set Nets by Attr

When you click the **Set Nets by Attr** button, the Set By Attribute dialog appears. This dialog lets you find those nets for which attributes have been defined.

If you select an attribute from the list box and click **OK**, you are returned to the Edit Nets dialog. All nets that have that attribute defined are highlighted.

If you select an attribute and click **Change Value**, you can set a specific attribute value for that attribute name. When you return to the Edit Nets dialog, all nets with the selected value for the attribute are highlighted.

You can select multiple attribute names and values. When you return to the Edit Nets dialog, only those nets with all of the selected attribute names and values are highlighted.

To select all attribute names and current values, click **Set All**. **Clear All** deselects all attributes. The **Clear current items** checkbox allows you to control whether or not nets matching the selected attributes are added to, or replace, the highlighted nets in the Edit Nets dialog. If this checkbox is enabled, only nets having the selected attributes are highlighted (i.e., the **Nets List** is cleared first). Otherwise, those nets are added to the list of highlighted nets.

Show or Hide Connections

When you choose **Show Conns** or **Hide Conns**, the connections for the net highlighted in the list box are either displayed or hidden on the screen. An asterisk next to a net name means the connections are visible. The display of nets does not affect the routing of the net.

Edit Attrs Button

To learn about net attributes, click the following topic:

Net Attributes Tab

View Attrs

When you select one or more nets from the Nets listbox, the **View Attrs** button becomes active. Click View to open the Windows Notepad utility and display a list of attribute names and values for each selected net.

In conjunction with the **Set Nets By Attributes** button, the View option makes it easy to find nets with the same attribute value, and then view all attribute values for those nets.

Rename

When you highlight a single net name in the **Nets** listbox, the **Rename** button becomes active. Click **Rename** and the Net Name dialog will display.

Type in the new name and click **OK** to rename the net and return to the Edit Nets dialog.

Info

When you highlight a single net name in the **Nets** listbox, the **Info** button becomes active. You can display the Net Info dialog with the **Info** button.

Net Info displays detailed information about length and characteristics of the selected net. It only measures the X and Y distances, not depth (such as via length to another layer). Arc length is included (accurately) in the calculation of connection and copper lengths. The Manhattan length is an approximation of the final routed length of a diagonal connection.

Select

You can select a net or multiple nets in the design by highlighting the net names in the listbox in the Edit Nets dialog, then clicking the **Select** button (the button is enabled only if a net is chosen). The nets in the design will highlight.

Net Selection Mask

The performance of Select Net is directly affected by the settings in Options Block Selection. For example, you could configure Options Block Selection to allow you to select line segments in a net while ignoring the vias (it ignores the inside/outside block specifications), allowing you to modify line widths for all segments in a net.

This feature is useful for unrouting a design (net by net) by highlighting routed copper items and deleting them. Be careful to disable selection of connections when unrouting nets. If a connection is deleted, the corresponding net is changed or destroyed.

You can also select nets through the Select popup menu. Select an item that is part of a net (Edit Select), click the right button, and choose Select Net from the popup menu. The complete net attached to the selected item will be selected and highlighted.

In the Edit Nets dialog, click **Close** to exit the dialog.

Set Nets By Node Count

This feature lets you select all nets with a node count between the range you specify in the corresponding **Min** and **Max** edit box.

Net Attributes

Allows you to display and modify component attributes for the selected component.

When you click the Attributes tab, the dialog appear.

You can view, add, modify, or delete a collection of net attributes. The dialog contains a two-column table showing the collection of net attributes. Within the collection, each attributes name and value appear in the column.

- **Adding an Attribute**: To add an attribute, click the **Add** button to open the Attribute dialog. Enter the name and value for the attribute and set attribute properties. Click **OK**, and the attribute is added to the table.
- **Viewing or Changing Attribute Properties:** To view or change an attributes properties, select an attribute from the table and click the **Properties** button (or double-click the attribute) to open the Attribute dialog.
- **To Delete an Attribute:** Select an attribute in the table and click **Delete**, or press the *Del* key.

Attribute Dialog

The following information appears in the Attribute dialog.

- **Category Listbox:** Displays a list of all attribute categories, All, Component, Net, Clearance, Router and SPECCTRA. Selecting a category brings up a list of pre-defined attributes for that category.
- **Name Listbox:** Displays all pre-defined attributes for the specified category. The first entry in the list is *User-defined*.

  The currently-selected attribute also appears in the **Name** edit box, unless *User-defined* is selected. In that case, the Name edit box is blank so that you can enter a user-defined attribute name.
- **Name Edit Box:** For user-defined attributes, enter a name for the attribute.

note:
If the dialog is accessed for an attribute that already has a name, then the Category listbox, Name listbox, and Name edit box are filled in, but grayed. If the attribute doesnt have a name, these controls are enabled.

- **Value:** Use this edit box to enter a value for the attribute.

The following items appear grayed in the dialog:

- **Visible:** This checkbox indicates whether or not the attribute is visible.
- **Location:** This area shows the X and Y coordinates of the components reference point.
- **Text Style:** This area lets you select the attribute text style. Text styles appear in the **Text Style** drop down listbox. To change the selected Text Style, click on the text style you want from the listbox. To modify the text style, click the **Text Style** button.
- **Rotation**: Shows the rotation amount if the pattern has been rotated.
- **Flipped** Box: This box indicates whether or not the pattern has been flipped.
- **Justification**: Under **Justification** are nine buttons, which allow you to change text justification by setting the *reference point* of the text string. For example, if you enable the middle button, the text reference point (the lower-left corner) moves to the center of the bounding rectangle.

To learn more about attributes, click on the following topics:

Attribute Description for Autorouting
Clearance Attributes

See also:

Edit Nets dialog

Close

Attribute Description for Autorouting

The following attributes are recognized by the autorouter. All other predefined net attributes are ignored for routing.

Attribute	Description
WIDTH	Overrides global line width settings for the selected nets. A valid line width should be entered as the value.
VIASTYLE	Overrides global via style settings for the selected nets. An existing via style name should be provided.
MAXVIAS	Defines the maximum number of vias that can be placed for this net. Valid values are 0 (no vias) - n (any specific number).
RIPUP	Overrides the global ripup setting for the selected nets. Valid values are No, 0 and False. All indicate that the net should **not** be ripped up.
NOAUTOROUTE	Indicates that the selected nets will not be routed. You can use Yes, 1, and True to override a default ripup of No
AUTOROUTEWIDE	Indicates that the selected nets are scheduled as WIDE passes when you select passes manually. You can use Yes, 1, and True to override a default ripup of No.

Close

Clearance Attributes

Along with other pre-defined net attributes, which are available from the **Names** list box, the following clearance attributes are available:

- PADTOPAD
- PADTOLINE
- LINETOLINE
- PADTOVIA
- LINETOVIA
- VIATOVIA
- CLEARANCE

By adding one of these attributes to a net and assigning it a value, thematching global default clearance is overridden for DRC and auto-routing.The value may have a suffix to define the units. If the units are left off, then the current global units are used. If the clearance value can't be converted to a valid number for DRC or auto-routing, then the attribute is considered undefined and the corresponding global clearance is used. The last pre-defined attribute, CLEARANCE, defines a clearance value for all object pairs in this net.

For example, if the clearance between every object pair in a net is the same value, then the CLEARANCE attribute can be used to store that value, rather than creating six clearance attributes with the same value.

ACCEL PRO Route honors the clearances for all object pairs, but doesnt use the clearance attributes defined in specific nets. The log produced by PCB Route includes the clearance values used for all object pairs. QuickRoute clearances are set to approximately 1/2 of the routing grid. The SPECCTRA autorouter uses net specific clearance attributes, net class clearance attributes, and class to class clearance attributes.

Edit Measure

Measures the X distance, Y distance, and total distance between two points. Use the ruler button on the Toolbar as a shortcut for the Edit Measure command.

You can measure vertical, horizontal, and diagonal distances and the results will be displayed on the status line. The measurements are displayed either in mils or millimeters, depending on current settings in Options Configure.

Measure is a mode, meaning that if you were placing or selecting objects, when you use Measure you will exit the mode to go into measure mode. After you have measured, you will need to restart whatever mode you were in to resume editing.

To Measure

1. Select Measure from the Edit menu (or use the Toolbar button).
2. Move the cursor to the first point of your measurement in the workspace. Click and drag to the end point of the measurement.
3. Before you release, look at the results on the status line for X distance, Y distance, and T (for total) distance (in mm or mils). When you release the button the results disappear.

note:

Measuring with the mouse does not snap to grid. If you want to get a measurement strictly between grid points, you must use the arrow keys and space bar.

Edit Select

Edit Select is a tool which allows you to perform many functions and commands with previously placed objects and items. The Select tool can also be enabled from the Toolbar button (arrow).

Select Actions

single-, multiple-, or block-select

move, resize, rotate, flip, copy, modify, highlight, unhighlight, and delete

Select Commands

Edit Cut, Copy, Copy to File, Paste, Paste from File, Paste to Layer, Select All, Deselect All, Delete, Highlight, Unhighlight, Copy Matrix, Modify, Explode Component and Alter Component are operational when the Select tool is enabled. Net Info and Select Net are available from the Select popup menu (select the item, then right mouse click to access) or through the Edit Nets command.

Select actions are possible only if an object is selected. For example, you cannot move an arc unless it selected; you cannot modify a line unless it is selected.

Information included in this section only covers the mouse/cursor actions for Select:

For keyboard equivalents to standard mouse functions in PCB, refer to Keyboard.

For the Edit menu command descriptions, refer to their respective command sections.

Selecting Objects

To single select, click a single object; all other selected objects are deselected. You must be on the appropriate layer.

To multiple select, first select a single object, then hold down the *Ctrl* key and click on additional objects/items. The selected objects are surrounded by a selection box, which increases as you add items to the multiple selection. Click again on selected items (still using the *Ctrl* key) to deselect them individually. If you release the *Ctrl* key and click anywhere other than one of the selected objects, all are deselected.

To deselect, click an empty area of the workspace to deselect all items outside the selection region or run the Edit Deselect All command.

To block select, you click, hold and drag the cursor to create a selection box surrounding a block of objects, then release the button. You can add objects to the block selection individually by doing a multiple select (see above paragraph).

If you have **Outside Block** enabled (in the Options Block Selection command), then the selection will occur outside of the selection block. If you have the **Touching Block** option enabled, a block selection will include everything inside and touching the selection block.

A block selection mask can be used. Objects can be filtered or masked in a variety of ways, depending on how you set up the selection options. Use the Options Block Selection command for altering or setting the selection options.

To sub select, hold down the Shift key and click with the mouse. This allows you to select a single object which is part of another object (e.g., a single pad in a component or an island in a copper pour).

When Objects Overlap

When objects overlap, it may seem difficult to select an *underlying* object. Continue clicking *without moving the cursor* and Select cycles through all objects underneath the cursor. The spacebar is easier to use than the mouse in this situation (pressing the spacebar twice equals click and release for the left mouse button). For multiple select, press the *Shift* key, then click the left mouse button without moving the mouse.

Moving and Copying Objects

To move an object, you first need to select it, then click on the object and drag the cursor (with the object attached) to the new location. Release to place the object.

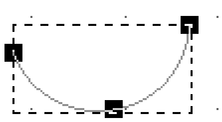
If you are moving multiple objects within a selection box, click anywhere in the selection box and drag; all the selected objects in the box will follow. Release the button to place the objects.

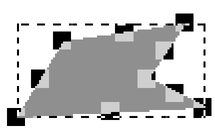
You can copy objects in the same manner; after you select the object(s), hold the Ctrl key down and drag a copy of the object(s) to where you want to place it. When copying a component, the RefDes (reference designator) will change in the copy, and connections will not copy. Any copper that is copied will become free copper that is not associated with any net. To cancel a move or copy in progress, click the right mouse button while the left mouse button is still depressed.

Resizing Objects

You can resize a selected object by clicking one of its handles, and dragging to stretch the object. The resize function varies for the different objects.

For example, to resize an arc you click one of the endpoint resize handles and drag the endpoint to increase the sweep angle. To resize a polygon, you can grab one of its vertex handles and move it to change the polygon.





When you move a handle that is on an edge between two vertices, a new vertex is created (allowing you even more reshaping). You can delete a vertex by moving it to an adjacent vertex and releasing.

Lines, Arcs, Polygons, Copper Pours, Keepouts, and Cutouts can be resized with the Select mode resize function.

Rotating and Flipping

Select an object. Press R to rotate 90 degrees counterclockwise. To flip the object in the X direction (about the Y axis), press F while the object is selected

Shift+R will rotate the object by the value specified in the **Rotation Increment** field of the Options Configure dialog (default is 45 degrees).

This function works on multiple- or block-selected objects as well.

ACCEL Tango PCB allows rotation in 90 degree increments only.

warning:

When you flip a pad or component, pad characteristics on Top, Top Silk, Top Paste, and Top Mask layers are swapped by corresponding characteristics on the Bottom, Bottom Silk, etc. layers. All other layers are left alone. Perform the operation cautiously.

Edit Properties

While the Select tool is enabled, run Edit Properties after an object is selected. Or, use the Select popup menu by clicking on the object to select it, then clicking the right mouse button to display the popup menu, then choosing Properties from the menu. Either way will display a dialog for the selected object.

Modifiable objects include Components, Lines, Arcs, Pads, Vias, DRC error indicators, Copper Pours, and Text. Each entity is enabled in the same manner, but has its own particular Modify dialog and subsequent results. Refer to the Edit Modify command documentation for information on all modifiable objects.

Net Info

Net Info is available from the Select popup menu, accessed by selecting an item that is associated with a single net, then clicking the right button. Net Info will either display a dialog or a message stating that items from multiple nets were selected.

Net Info displays detailed information about length and characteristics of the selected net item. It only measures the X and Y distances, not depth (such as via length to another layer). Arc length is included (accurately) in the calculation of connection and copper lengths. The Manhattan length is an approximation of the final routed length of a diagonal connection.

To access Net Information by net name, use the Edit Nets command.

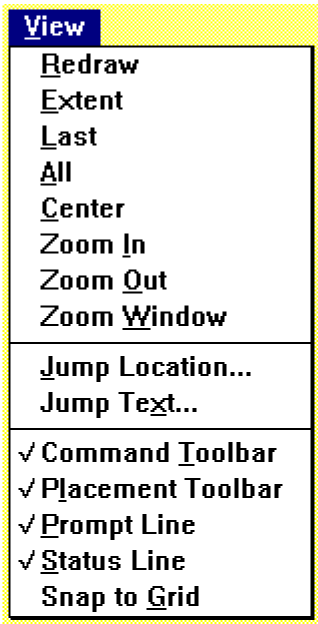
Select Net

After you have selected one or more objects that are associated with nets, this command will select all of the items in each of the nets. It will select everything in a net except component pads. If you select a component, then choose Select Net, it will select all net items that emanate from the pads on the component.

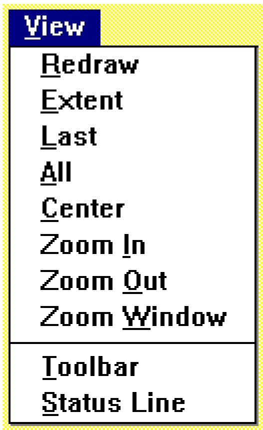
This feature is useful for unrouting a design (net by net) by highlighting routed copper items and deleting them. Be careful to disable selection of connections when unrouting nets. If a connection is deleted, the corresponding net is changed or destroyed.

The Options Block Selection command will operate in conjunction with Select Net, except that it ignores the inside/outside/touching block specifications.

View Commands (PCB)



View Commands (Route)



View Redraw

Clears everything in the workspace to the background color and then repaints the screen. View Redraw is good to use when you have leftover traces and shapes from moving or deleting objects; Redraw erases leftover graphics.

When you redraw, the items that reside on the current layer are drawn last. Refer to Options Layers for more information about layers.

This command also causes co-located top and bottom layer items to be drawn in the correct order. For example, if the current layer is Top, all Top layer SMT pads of an edge connector will be drawn on top all Bottom layer SMT pads in the edge connector. Other View/Zoom commands do not necessarily draw co-located pads in the correct order.

To interrupt a redraw in progress, click the right mouse button or press Esc.

View Extent

Displays the extent of all objects placed in the workspace.

PCB computes and draws the workspace such that all placed objects on enabled layers are visible. Disabled layers are ignored.

View Last

Redraws the previous view.

There is no previous view until you run at least one View command that changes the view area. If you run View Last multiple times, you will toggle between the last two views.

Scrolling, centering, and redrawing do not affect the previous view.

View All

Redraws the screen with the entire workspace shown.

The workspace size displayed is determined by the **Workspace Size** option set in Options Configure. If you want to make the workspace smaller to fit the board you are working on, then size it accordingly using Options Configure. Options Configure is available from the PCB menu bar when the router is not running. The scroll bars are not displayed at this zoom level.

View All is the default view when you start up PCB with an empty workspace.

View Center

Redraws the screen using the cursor as the relative center point.

The cursor becomes a magnifying glass shape, signifying the zoom mode and prompting you to click in the workspace; the point where you put the cursor and click in the workspace will become the center of the screen.

View Center doesn't interfere with the mode in which you are operating, it is a temporary mode. For example, if you are in Place Pad mode and then run View Center, wherever you put your cursor and click will become the center of the screen. Then if you click again, you will place a pad, resuming the previous mode.

The selected point may not be in the center of the screen if you are near a workspace boundary.

C key

The shortcut key for View Center is C. Using C is useful for panning across the workspace. You don't need to click the mouse button, just move the cursor to the point you want centered and press C.

View Zoom In

Zooms in by the magnification factor value set in Options Configure.

When you select View Zoom In, the cursor becomes a **zoom cursor**, taking on the shape of a magnifying glass, prompting you to click for the center point of the zoomed area. The cursor location becomes the center of the zoomed-in area. You must re-invoke the command for every zoom action. To cancel the zoom after the zoom cursor appears, click the right mouse button or press Esc.

plus (+) key

An easier way to zoom in is to use the plus (**+**) key as a shortcut. Your cursor location will become the center of the zoom action; you don't have to click in the workspace. The keypad plus key also works.

View Zoom Out

Zooms outs by the factor value set in Options Configure.

When you select View Zoom Out, you are prompted to click for the center point of the zoomed area. The cursor position becomes the center of the zoomed-out area. You must re-invoke the command for every zoom action. To cancel the zoom after the zoom cursor appears, click the right mouse button or press Esc.

minus (-) key

An easier way to zoom out is to use the minus (**-**) key as a shortcut. Your cursor location will become the center of the zoom action. The keypad minus key also works.

 View Zoom Window

Zooms to an area of the workspace that is specified by a zoom window. A zoom window is a rectangle you click and drag to create in the workspace; the window you create will fill the screen.

To Zoom through a Window

There are three ways to invoke the zoom window: the Z key, the Toolbar zoom button, or the View Zoom Window command from the menu bar. In any case, after it is invoked, you must draw the zoom window in the workspace.

1. After you enable the zoom window function, the cursor becomes a magnifying glass shape until you click and drag to create a rectangular shape around a specified view area.
2. Release when you have completed your "window". Whatever you surround with the zoom window will be enlarged on the screen (the window will fill the screen).
3. The last active tool is still active after you do the zoom window action. To cancel the zoom action once the zoom cursor appears, click the right mouse button or press Esc.

note:

You must drag the cursor to create a zoom window. If you click and release in the workspace without dragging the cursor, the program responds with a beep, and does not zoom in.

Edit Components

Allows you to edit components within your design and to jump to a particular component. Also allows you to highlight components and to highlight nets attached to a particular component.

Select Edit Component to display the dialog.

Components Listbox

The **Components Names** listbox contains the names of all components in the active design. You can select individual or multiple components in the listbox. Once selected, you can highlight and unhighlight components and attached nets and jump to a component

Set All /Clear All

If you want to select all components in the **Components** listbox, click the **Set All** button. If you don't want any components selected, click the **Clear All** button.

Properties

The Properties button accesses the Component Properties dialog for the selected component or components. See Component Properties for details.

Highlight/Unhighlight

The **Highlight** button highlights one or more components selected from the **Components** listbox in one of the highlight colors chosen through the Options Display command. Successive invocations of this command cycle through the two defined highlight colors. If you do not want to use two highlight colors, set both colors to be the same in the Options Display dialog box.

When you run this command, the chosen components are drawn in the highlight color until they are unhighlighted. The selection color overrides the highlight color, so you won't see the highlights until the components are deselected.

If PCB and PCB Schematic are both running, and if the **DDE Hotlinks** checkbox in the Options Configure dialog box is checked in *both* applications, then component highlight information is communicated between the two applications. Highlighting a component in PCB highlights the corresponding part in the PCB Schematic.

The **Unhighlight** button removes the highlighting from the selected components.

Highlight an Attached Net

You can highlight nets which are attached to the components selected in the **Components** list box.

1. To highlight attached nets, select one or more components from the **Component** listbox. (You can use the **Set All** button if you want to select all components in the list.)
2. Click the **Highlight Attached Nets** button. The attached nets are highlighted with the highlight color set using the Options Display command.
3. To remove a highlight, select one or more components from the **Component** listbox.

4. Click the **Unhighlight Attached Nets** button.

Jump to a Component

1. Select *one* component from the list.
2. Click **Jump** to jump to the specified component. The specified component appears in the center of your Workspace.

View Jump Location

Positions the cursor to a specified location (X, Y coordinates). You can also use the J key to enter coordinates on the status line.

If you are zoomed in, this command pans the workspace to the specified location, attempting to center the location. Your current zoom setting is not changed by the jump location panning (except View Last will be updated). If the specified location is already visible on the screen, no panning is necessary.

The units used for the location value (mil or mm) are determined by the setting in Options Configure. Select Options Configure to override the default settings.

The location is also based on the Options Grids setting, either Absolute or Relative; e.g., if your grid setting is in Relative mode, then the location is a Relative coordinate. Also, you can use negative coordinate values when in Relative mode. Refer to the Options Grids command documentation for more information.

To Jump to a Location

1. Select Jump Location from the View menu.
2. When the dialog is displayed, specify the **X** and **Y** coordinates (in the text boxes) of where you want your cursor to be in the workspace.
3. Click **OK**.

View Jump Text

Allows you to sequence through all text strings to locate a specific combination of letters (i.e., words). The character string that you specify could be part of a word or a complete word.

Case Sensitive Search, if enabled, searches for text matches based on case. If you disable this option, text case is ignored in the search.

Search Entire Design searches all layers for the specified text string, and search only the current layer when you disable the option.

To Jump to Text

1. Select Jump Text from the View menu to display the dialog.
2. Type the text you want to search for in the text box. Notice that the **OK** button is grayed out. Click **Search**.
3. If the matching text is not already on the screen, the program pans the screen at the current zoom level to locate the text and center its reference point. (You can set the text reference point by setting the justification with the Options Text Style command.)
4. After PCB finds the first instance of the specified text string (highlighting it in the highlight color), the Search button changes to **Next**, and if you click it the program will find subsequent instances of the same text string. You can continue to cycle through all occurrences of the search pattern until you click **Cancel** or **OK**.

   The **OK** button is no longer grayed; if you click it, the program moves the cursor to the highlighted text and the highlighting disappears.

   Highlighted text may be obscured by the dialog; move the dialog to view the text if necessary.

Placement Toolbar

With this command you show or hide the Placement Toolbar.   The toolbar gives you quick access to the most frequently used Place commands.

Disabling the command increases the space within the applicable window.

The setting of the Placement toolbar visibility is saved to your PCB.INI file when you exit the program, and restored when you restart it.

View Toolbar

With this command you show or hide the System Toolbar.   The toolbar gives you quick access to the most frequently editing commands (such as the Select tool).

Disabling the command increases the space within the applicable window.

The setting of the toolbar visibility is saved to your PCB.INI file when you exit the program, and restored when you restart it.

View Prompt Line

Allows you to either show or hide the Prompt line.

The Prompt line is useful in that it gives instructions on what to do in certain modes. For example, while you are in one of the Place object modes, it tells you what to do next, depending on what action you have already taken.

The current settings for the visibility of the Prompt line will be saved to your PCB .INI file when you exit the program, to be applied to subsequent sessions.

View Status Line

Allows you to either show or hide the PCB Status line. The Status line provides status information and allows you to change layers and execute temporary macros.

A check mark alongside the command indicates that the Status line is visible. Disabling the command increases the space within the applicable window.

The setting of the Status line visibility is saved to your PCB.INI file when you exit the program, and restored when you restart it.

View Snap to Grid

Creates a snappy cursor, meaning that the cursor can only move from grid point to grid point, as opposed to a free floating cursor. The benefits of a snappy cursor are mainly a question of personal preference-- the user may be accustomed to a snappy cursor from other program applications. Also, a snappy cursor can create a predictable point of reference and placement when moving and rotating objects.

At times the you may want to disable this command (and cause the cursor to float freely) due to the pick and select limitations. For example, a line you want to select does not run true along grid points; a snappy cursor only allows you to pick on the line where the line intersects a grid point. Or, you may need to get to an off-grid pad.

View Toolbar (Route)

Use this command to show or hide the ACCEL PRO Route toolbar.

When you begin the routing process by running, the Toolbar changes. The autorouting Toolbar consists of graphical display buttons (icons) that correspond to commonly used commands in ACCEL PRO Route. These icons appear just below the menu bar on the ACCEL PRO Route screen. Their display is controlled by selecting the View Toolbar command.

These buttons include some commonly used commands, in the following order (from left to right in the Toolbar):

- Route Info
- Route View Log
- Route Cancel
- View Zoom Window

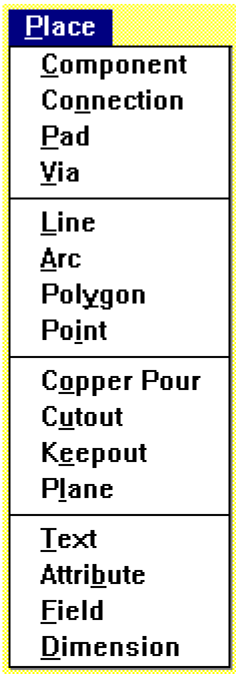
View Status Line (Route)

When you begin the routing process, the Status line changes. The Status line area provides routing status information and summary pass performance data. During each pass, the Status line is updated as each connection is attempted. Information is presented as follows:

Pass Name: (AA/BB:CC) DD% (EEEE/FFFF)

Pass Name	The name of the current routing pass.
AA	Number of connections, nets, or fanouts completed for this pass.
BB	Number of connections, nets, or fanouts scheduled this pass.
CC	Number of connections, nets, or fanouts deferred for later passes.
DD%	Percent of all connections completed.
EEEE	Number of all connections completed.
FFFF	Total number of connections on the board.

Place Commands (PCB)



Place Component

Places a named component at a specified location.

Placing a component requires that the component and its corresponding pattern have already been created and assigned to a library. Also, the library must be open before the component can be accessed and placed (refer to the Library Setup command for information about accessing libraries).

With this command you can browse the component display.

Place a Component

1. Select Place Component from the menu, or use the component pushbutton on the Toolbar.
2. Click in the workspace to display the dialog. You are now in the component selection mode of Place Component.

   The **Component Name** listbox shows a list of component names of whatever library you have accessed. You can display the component in the dialog with the **Browse** button. You can also specify the **RefDes** and **Value** for the component you are placing. The **Library** combo box shows the current library and allows you to switch to a different library.
3. In the **Component Name** listbox, find the component you want to place and highlight it. If you want to view it, click **Browse** and the dialog will stretch to the right with the component view.

   If the component you want to place is not listed in any open libraries (the **Library** combo box lists them), you will need to run the Library Setup command.
4. Specify a reference designator in the **RefDes** box.

   If you leave this blank, the program uses the default RefDes prefix assigned to the Component at component creation time. If you do not assign a number, the program automatically assigns the next available number.

   Specify a value in the **Value** box (e.g., electrical values for resistors and capacitors; typically blank for logic components).

   Click **OK** when you have filled in the desired information.
5. You are now in the place mode of Place Component. Move the cursor to the workspace location where you want to place the component. Click to place it. Or you can click down to make a ghost, then drag and drop (release) to place it more accurately. (An alternate method for drag-and-drop is Alt + left mouse click, then release the Alt key. You can then move the component freely with the mouse without having to keep the button depressed.) To cancel ghosting of a component, click the right mouse button.

   You can rotate a component while placing it by pressing R while the mouse button is depressed. See the rotate/flip section (following) for more information.

   You can continue to place the same component by clicking in selected locations, and each component will be given a unique reference designator. Also, each component will have displayed its type, name, and any other attributes previously specified (from the Place Attribute function).
6. If you want to place a different component, you can press the right mouse button or Esc to exit the current place mode. As you are still in Place Component mode, you can click in the workspace to display the dialog and select another component to place (as in step 3).

Rotate/Flip

To rotate or flip a component after it has been placed, select it, and press R to rotate or F to flip while the component is selected.

For rotation, the default increment rotation is 90 degrees, which is activated by the R key. You can use Shift+R for a rotation of 45 degrees (default) or for a different (custom) rotation. Use the Options Configure command to set the rotation increment for Shift+R.

You can also flip or rotate a component as you are placing it, by keeping the button down while pressing F or R (or Shift+R). For rotation, the result of the placement rotation will apply to the next component you place. R and Shift+R still apply, as described above. For example, you are placing a component and you rotate it 25 degrees before you finish it (Shift+R, with 25 degrees set in Options Configure). Then you place another component of the same type; it will be automatically placed at the same 25 degree angle without any rotation action. Therefore, you can place multiple components at the same angle by rotating only the first component you place.

If you click the right button to end the temporary mode and then click the left button to place another component, the rotation memory will not apply; the rotation memory applies only to the temporary placement mode.

note:

Components of the same type can be placed only if they have the same pin mapping (i.e., pin designator to pin name to pad number). In other words, if you place the "same" component from different libraries, the first instance of the component type will establish the standard pin mapping for that type of component. Any components of that type placed subsequently have to conform to the pin logic of the first or they will be unplaceable.

When you place components that include vias, pads, and text that are of styles that have the same names but different data than those in the current design, the incoming style names will be bracketed to indicate the style conflict. The new (bracketed) style names will be added to the list of available styles in the current design. Refer to the Edit Modify, Options Pad Style, and Options Text Style commands for object style information.

Place Line

Places a line (or series of line segments) of the current line width on the current layer. To change the current line width, use the Options Current Line command.

Place a Line

1. Select Line from the Place menu (or click the line button on the Toolbar).

   While you draw a line or segments, the cursor is displayed as a crosshair shape. When you finish the line segments, the cursor returns to its normal shape.

2. With the cursor in the workspace, click and hold the button at the starting point, then drag the line to its second point and release to place the line. You can continue with connected segments in the same manner.

   You can use Alt + left mouse button instead so you don't have to hold the button down while dragging line segments.

   While you drawing a line or segments, the cursor is displayed as a crosshair shape. When you finish the line or segments, the cursor returns to its normal shape.

3. To finish the line or line segments, click the right mouse button or press Esc. Then you can begin another line beginning at a new location.

   You will remain in Place Line mode until you enable another mode.

If you use the Undo command (Edit Undo), it will undo only the last finished line or line segments (finish by pressing Esc or right button).

Status line measurement: The Status line information display area shows line measurements for delta X and delta Y while you are dragging a line segment. When the line segment is finished, the total length measurement of the segment(s) is displayed, each subsequent segment being included in the total measurement, and each segment with a delta X and delta Y length while is being dragged.

Orthogonal Modes (for Line Segments)

Orthogonal modes use lines that are horizontal, vertical, and at 45-degree angles. The available orthogonal modes are provided as mode pairs. The O key cycles through the mode pairs and the F key toggles between the pair that is current. You can enable or disable the orthogonal modes in Options Configure.

For placing lines, only the linear orthogonal modes are available (no arcs).

Place Arc

Places an arc or circle of the current line width on the current layer. With this command you can create arcs of varying length and radius and circles of varying radius.

To change the current arc width (line width of the arc), use the Options Current Line command.

Arcs are partial circles. Arcs and circles are constructed counter-clockwise; the click (down) and release (up) define the start and end point of the arc, therefore a stationary click/release comes full circle and defines a circle. In this case, the second click and drag moves the center point away from the defined point on the circumference.

To Place an Arc

1. Select Place Arc or the arc button on the Toolbar. Move the cursor into the workspace to where you want the starting point of the arc.

   While you draw an arc, the cursor is displayed as a crosshair shape. When you finish the arc, the cursor returns to its normal shape.

2. Click and drag to where you want the end point of the arc to be. Release and the start and end points are established as a 180-degree arc.

   The arc will sweep counterclockwise as you place it. (e.g., left-to-right the arc sweeps up; right-to-left the arc sweeps down, etc.) If the start and end points are the same (i.e., you click and release without dragging) a circle will be created.

3. After the start and end points are established, click and drag the cursor to define/alter the center point, thereby increasing or decreasing the sweep angle (radius) of the arc.

   You can flip the arc (swapping the end points) by pressing F while the arc is still unfinished. When you release the mouse button, the arc is permanently placed.

   To cancel ghosting of an arc, click the right mouse button.

Misc. Arcs

To move, resize, rotate, flip or perform other types of changes to the arc (after it is permanently placed), use the Edit Select tool.

To rotate or flip an arc, select it, click and hold in the center of it as if to move it (not on the handles), and press R to rotate or F to flip while the button is depressed.

For rotation, the default increment rotation is 90 degrees, which is activated by the R key. You can set a different rotation increment with Options Configure. To activate what you have set in Options Configure, use Shift+R.

To move the arc, click within the selection box (not on the handles) and drag the arc.

To resize, click and move the center handle to change the center point or move the start or end point to change the sweep angle.

Place Pad How to

Places a pad of the current style. Pads do not belong to a particular layer. To place a surface pad, for example, the current style must be set up to have a shape defined only on a surface layer.

Use the Options Pad Style command to define new pad styles and to set the current Pad Style. Use the Free Pad Properties (Edit Properties) command to change pad styles or pad numbers of already placed pads.

To learn how to rotate and flip a pad, click the following topic:

Rotate or Flip a Pad

To learn how to renumber pads, click the following topic:

Renumbering Pads

See also:

Options Pad Style
Free Pad Properties

Close

Place a Pad

1. Select Pad from the Place menu (or use the toolbar pad button) and move the cursor to where you want to place the pad.
2. Click the mouse to place the pad.   You can click, drag, and release for more accurate placement with a ghosted outline.   You can rotate or flip a pad while placing it.

   To cancel ghosting of a pad, click the right mouse button.

Rotate or Flip a Pad

To rotate or flip a pad *after* it has been placed, select it and press *R* to rotate or *F* to flip.

The *R* key rotates the pad 90 degrees, *Shift+R* rotates by the rotation increment set in Options Configure.

You can rotate a pad *as you are placing it*, and whatever angle is the result of the rotation will apply to the next pad you place. You must keep the button down while you are rotating it (before the pad is permanently placed). For example, you are placing a pad and you rotate it 25 degrees before you finish it (*Shift+R*, with 25 degrees set in Options Configure). Then you place another pad; it will be placed at the same 25 degree angle *without any rotation action*. If you decide to rotate the second pad, it will increment 25 degrees more, resulting in a 50 degree angle. Therefore, you can place multiple pads at the same angle but only have to perform the rotation action on the first pad.

To flip a pad as you are placing it, press *F* while the pad is ghosted before final placement. When you flip a pad, the hole range does not flip with it.

Pads and vias share the same rotation *memory*. In other words, if you place and rotate a pad at 90 degrees, then immediately place a via, that via will be placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select-and-rotate actions that take place after object placement.

Renumbering Pads

If you want to number the pads you placed, you must enable the Select tool and then run the Utils Renumber command. A dialog appears where you can choose Pad Num as the type, then specify the start value and increment value. Then you can click on each pad manually and they will be numbered in sequence. You can also use the Modify Pad function to assign a pad number (Select pad, Modify pad). Refer to the Utils Renumber and Modify Pad commands for more details in this manual.

Place Dimension

Places a dimension at a specified location.

Dimensions are thin lines used to show the extent and direction of dimensions.   A break in the dimension allows you to display numeric values.

Not available for ACCEL TangoPCB.

Placing a Dimension

1. Select Place Dimension from the menu, or use the dimension icon on the toolbar.
2. Click in the workspace to display the Place Dimension dialog.
3. Choose a dimension style from the Styles drop down list box:

- **Point to Point:**   Measures the distance between two points.
- **Baseline:**   Measures the distance between a single reference point and subsequent points.
- **Leader:**   A dimensioning line extending from a piece of text or symbol to the dimensioned object. It can be used for dimensioning or notation.
- **Center mark:**   Is a cross-hair marking of the center of an arc or circle.
- **Radius:**   Measures the radius of an arc or circle.
- **Diameter:**   Measures the diameter of a circle.
- **Angular:**   Measures the angle between two objects with a common center reference point.

4. Use the **Orientation** area to set the dimensions orientation to be either **Horizontal** or **Vertical**.
5. Use the **Text Orientation** area to set the orientation of the text in between the dimension lines to be either **Horizontal** or **Vertical**.
6. Use the **Units** area to choose the measuring unit for the dimension.
7. Use the **Symbol** area to choose the type of notation for a Leader style dimension.
8. Set a symbol size using the **Symbol Size** box.
9. Set a center size for the center of the symbol using the **Center Size** box.
10. In the **Line Width** box, set a line width for the dimension lines.

    (After you disable the Place Dimension tool, the cursor returns to its normal shape.)
11. Indicate whether to display the dimensions units and diameter symbols by enabling the appropriate checkboxes.
12. Click **OK** to return to the workspace.
13. Point in the workspace to place the dimension.

    To place a Point to Point dimension click an existing object, then click a second object.
14. Select the location for the dimension text and lines.

    If the dimension lines and text cannot fit between the extension lines, you are asked if you want the dimension text outside the extension lines.   If you say No, the text remains in between the lines.
15. Continue placing dimensions as necessary.   To stop, press **Esc** or click the right mouse button.

Rotate

To rotate a dimension after it has been placed, select it, and press *R* .

For rotation, the default increment rotation is 90 degrees, which is activated by the *R* key. You can use *Shift+R* for a different (custom) rotation. Use the Options Configure command to set the rotation increment for *Shift+R*.

Place Via

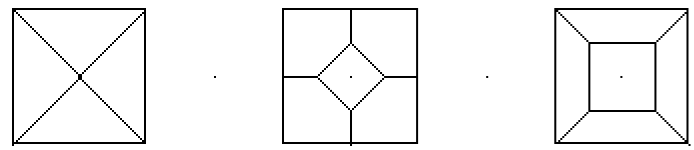
Places a via of the current style. Vias do not belong to a particular layer. To place a buried via, for example, the current style must be set up to have a shape defined only on a buried layer.

Use the Options Via Style command to define new via styles. Use Via Properties (Edit Modify) to change styles of already placed vias..

Vias are almost identical to pads in the way that they are placed, rotated, flipped, and edited. Refer to the Place Pad command instructions for detailed information.

Place Point

Place Point allows you to specify a Ref Point, Glue Dot, or Pick and Place point on a pattern object before it is saved to a library.



Reference Point: You need to specify a reference point on a pattern object before it is saved to a pattern library (Library Pattern Save As). The Ref Point will appear in the same color as what is specified for the 1x grid color in Options Display.
When you place a reference point on an object, the component (or pattern) will move with the cursor at the reference point. Components are flipped and rotated about the reference point. For most components, select a reference point in the center of pad #1.

Glue Dots are used to hold components in place until they are soldered during manufacturing. A glue dot can be placed as part of a pattern before saving the pattern to a library. Use the File Reports command to get a listing of all Glue Dot locations.

Pick and Place points provide reference points in directing the pick and place mechanism (or auto insert) in manufacturing (picking up the component and placing it on the board). Use the File Reports command to get a listing of all Pick and Place locations.

note:
Glue Dots and Pick and Place points appear as free points, not associated with a pattern, unless you save them as part of a pattern in a library.

To Place a Point

1. Select Place Point or click the reference point button on the Toolbar.
2. Move the cursor to the location on an object or with a collection of objects. Click to place the Point. (Or, click, drag and release to place the reference point more accurately.)
3. You can move a Point by selecting it and dragging it to a new location.

   To cancel ghosting of a point, click the right mouse button.

note:
Pick and Place points and Glue Dots can be flipped to the Bottom layer. They use the line color from the layer that they are on.

Displaying/Hiding Points

You can choose to display or hide the placed Glue Dot and Pick and Place reference points. Select the Options Display command, click the **Misc...** button, and then either show or hide them with the appropriate radio button selections. You can also choose to include or exclude these items from your printer output (File Print) by specifying the items in the Setup Print Jobs dialog.

Place Polygon

Places a polygon fill on the current layer.

To Place a Polygon

1. Select Place Polygon or click the polygon button on the Toolbar.

   While you draw a polygon, the cursor is displayed as a crosshair shape. When you finish the polygon, the cursor returns to its normal shape.

2. Put the cursor at the starting point of where you want your polygon. Click and drag to the second point and release.

3. Click and drag to a third point and you have a triangle. Any subsequent polygon points will be connected by a line to the first point you selected, e.g., the fourth point of a polygon is connected to the first point automatically.

note:

Complex polygons are not allowed. A complex polygon is a self-intersecting or self-crossing polygon. Polygon sides can touch each other but not cross each other.

4. When you have established all your points in creating the polygon, click the right mouse button (or press Esc) to finish and fill the polygon.

Draft/Outline Display Mode

Polygons can be drawn and printed as outlines. They are drawn in outline form when the Draft Mode (Options Display Misc command) option is enabled. They are printed as outlines when the Draft option is enabled in File Print, Setup Print Jobs.

Rotate/Flip

You can rotate or flip a polygon after you have placed it. Select the polygon and press R to rotate or F to flip while the polygon is selected.

The R key rotation is 90 degrees. You can use Shift+R to rotate to whatever rotation increment is set in Options Configure (a customizable rotation).

Alter the Shape

You can alter the shape (move, add, or delete vertices) by selecting the polygon, then clicking and dragging one of the handles. When you move one of the handles that lies between two vertices, a new vertex is created. To delete a vertex, grab its handle and move it to an adjacent vertex and release.

Place Copper Pour

Creates a copper pour outline on the current layer.

A copper pour is a polygonal area that the user specifies, which will be filled by copper in the board manufacturing. A copper pour will automatically back off from items in different nets.

To Place a Copper Pour

1. Select Copper Pour from the Place menu or click the copper pour button on the Toolbar.
2. Move the cursor into the workspace to where you want the first point of the copper pour to be. Click for the first point and drag to the second point and click to establish it. Place the third point in the same manner and you can see the outline of the copper pour so far. Click and drag to establish subsequent points. "Complex" copper pour polygons (self-intersecting or self-crossing) are not allowed.
3. Click the right mouse button to terminate the placing of the outline and establish the outline of the copper pour shape.
4. Now you will need to access the Modify Copper Pour dialog to establish the characteristics of your copper pour. Whatever you specify here will only apply to the selected copper pour.

   Select the copper pour outline (with the Edit Select tool), then choose Edit Modify. The Modify Copper Pour dialog is displayed.
5. For **Pour Pattern**, you can choose between a number of options. **Solid** is, of course, a solid pour (there is banding, but it overlaps to create a solid). **Horizontal** and **Vertical** are banded or striped, according to what you specify for Line Width and Line Spacing. **45 Cross** and **90 Cross** are both cross-hatched, but at different angles, also according to line width and spacing.

   For the line properties **Line Width** determines the size of the banding (solid), hatched, or cross-hatched lines. **Line Spacing** determines the space between lines, measured from the edge of one line to the edge of the next. If you selected **Solid** for the Fill Pattern, then the **Spacing** value should be designated as a small negative number (e.g., -1), so there will be no spaces interfering with the solid fill that you specified.

   The **Pour Backoff** option allows you to specify a value for the distance you want between the copper pour and any objects that may be inside of the copper pour polygon. The copper pour will backoff anything that is not specified in the Net combo box (see the explanation for Net).

   The **Spoke Width** allows you to specify the width of the track segments between pads and copper pour.

   The **Poured** and **Unpoured** radio buttons allows you the option of turning off or on the graphic fill display (if you are working quickly, the graphic fill can be time-consuming to display, so you can disable it).

   The **Thermal** options allow you to control the electrical connections to pads of the same net.

   The **Net** combo box allows you to select the net that you want the copper pour to connect to. The copper pour will backoff from any copper items that are not in the same net as the pour. If no net is associated with the pour, it backs off from all copper items except "no-net" pours.

warning:

You could mistakenly split the copper pour in half (or more than two pieces) if the copper pour surrounds items that are longer than the copper pour polygon. In that case, the electrical connection would be severed.

---

6. Click **OK** and the copper pour shape will fill in, if the **Poured** option is enabled.

Rotate/Flip

You can rotate or flip a copper pour after you have placed it. Select the pour and press R to rotate or F to flip while it is selected.

The R key rotation is 90 degrees. You can use Shift+R to rotate to whatever rotation increment is set in Options Configure (a customizable rotation).

AutoPlowing Pours on Disabled Layers

If you modify a component inside a pour on a disabled layer, the pour is autoplowed.

Place Cutout

To Place a Cutout

To place a cutout in your design:

1. Select Cutout from the Place menu or click the cutout pushbutton on the Toolbar.
2. Move the cursor into the workspace to where you want the first point of the cutout to be. Click for the first point and drag to the second point to establish the first edge of the polygon cutout. Click and move to place the third point in the same manner, and the outline of the cutout should be visible. Click and drag to establish subsequent points. "Complex" (self-intersecting or self-crossing) polygons are not allowed.
3. Click the right mouse button (or press Esc) to establish the cutout polygon; it will appear as an outline. When the copper is re-poured, the area defined by the cutout will not be filled.

Rotate/Flip

You can rotate or flip a cutout after you have placed it. Select the cutout and press R to rotate or F to flip while the cutout is selected.

The R key rotation is 90 degrees. You can use Shift+R to rotate to whatever rotation increment is set in Options Configure (a customizable rotation).

Place Keepout

Allows you to create a barrier to either keep in or keep out certain processes on a specific area of the board. You can place either a line keepout or a polygon keepout.

Keepouts are non-electrical items that only affect the autorouter; they are ignored by Design Rule Checking, nets, and copper pours.

First use Current Keepout to set up what type of keepout you want to place (line or polygon) and on what layers (all or current). Place Keepout will reflect whatever you have specified in Current Keepout.

Line keepouts work like Place Line. Polygon keepouts work like Place Polygon.

Place Plane How to

Creates a split plane outline on a plane layer.

Not available for ACCEL TangoPCB.

A split plane is a created by placing a polygon outline on a plane layer. The area within the polygonal outline is the split planes copper. A split plane is associated with a single net. When you select the net, all split planes in that net are selected.

Split planes appear and are printed in negative video.

To learn about connectivity issues with split planes, click the following topic:

Split Plane Connectivity

To learn how to rotate and file split planes, click the following topic:

Rotating and Flipping Split Planes

Close

Placing a Plane

1. Select Plane from the Place menu or click the plane icon on the toolbar.
2. Move the cursor to the first point of the plane. Click for the first point and drag to the second point and release to establish it.

   Place the third point in the same manner. (You can see the outline of the plane so far.)

   Click and drag to establish subsequent points.
3. Click the right mouse button to terminate the placing of the outline and establish the outline of the plane shape.

   The plane is drawn using the polygon color defined for that plane layer.

   It is illegal to have two planes that intersect, because connectivity to the planes will not be correct. Design Rule Checking (DRC) will report this error if you enable Plane Violations in the Utils DRC dialog.
4. Now you need to access the Modify Plane dialog to establish the characteristics of your plane.

   Select the plane outline (with the Select tool), then choose Edit Modify.
5. Use the drop down list box or type a new value to assign the plane to an existing net.
6. Define the line width for the Polygonal outline. The default is the current defaut line width. Change the width by typing over the default value.
7. Select a net plane color, by clicking the Net Plane Color button and selecting a color from the Color dialog.
8. Click *OK* and the plane shape fills in.

Split Plane Connectivity

For a pad to electrically connect to a split plane, the pad and the split plane must be in the same net and the pads hole must *intersect* the split plane. If the pads hole partially intersects the split plane, the pad is still considered electrically connected to the split plane, but DRC reports this partial connection as a warning.

Net connectivity is maintained with split planes, just as net connectivity is maintained with plane layers. For example, if two pads are electrically connected because they both connect to the same split plane, then a connection line is not necessary between the two pads. If two pads are electrically connected to the same split plane, ACCEL PCB assumes that copper in the split plane is continuous between these two pads.

Rotating and Flipping Split Planes

You can rotate or flip a plane after you have placed it.   Select the pour and press *R* to rotate or *F* to flip while it is selected.

The *R* key rotation is 90 degrees.   You can use *Shift+R* to rotate to whatever rotation increment is set in Options Configure (a customizable rotation).

See also:

Options Configure command

Place Text

Allows you to enter/place text on your design. To define the style of text you want to place, use the Options Text Style command first.

To Place Text

1. After you are in Place Text mode, click on the workspace to display the Place Text dialog.
2. In the **Text** box, enter the text that you want to place in the design. If you want to enter multiple lines (e.g., a list), you can start new lines in the **Text** box by using a carriage return (the Enter key).

   While the Text edit box has focus, you can press *Ctrl+V* to paste text from the Windows clipboard.

   In the **Justification** box, you can specify a change in the text reference point. The default reference point for text is the lower-left corner. You can modify the reference point before you place the text, or for already placed text (e.g., center rather than lower-left). The reference point defines the cursor location on text placement, and also the axis on which text will rotate (see rotate/flip section below). There are nine different reference point options.

   You can change the current text style by highlighting one in the **Style** box.

   You can modify a non-default text style (or view the default text style) by clicking **Edit Styles**. The Options Text Style dialog is displayed. Refer to Options Text Style for detailed information on adding, modifying, viewing, renaming, and deleting text styles.
3. After you click **OK** (or press Enter) and exit the dialog(s), click and hold the mouse button, and move the cursor to drag the box (containing the text) to where you want to place it. When you release the button the text will be placed, with the cursor as the reference point. The text will reflect the content, style, and justification you specified in the Place Text dialog.

   If you want to rotate or flip a text string before you place it, press R (to rotate) or F (to flip) before you release the button. See the following section for more information.

Rotate/Flip

To rotate or flip text after it has been placed, select it and press R to rotate or F to flip while the button is depressed.

The R key rotation is 90 degrees. You can use Shift+R to rotate to whatever angle you set in Options Configure (a customizable rotation).

You can flip or rotate text as you are placing it. For rotation, whatever angle is the result of the rotation will apply to the next text you place. For example, you are placing text and you rotate it 25 degrees before you finish it (Shift+R, with 25 degrees set in Options Configure). Then you place more text; it will be placed at the same 25 degree angle without any rotation action. (If you decide to rotate the second text string, it will increment 25 degrees more, resulting in a 50 degree angle.) Therefore, you can place multiple fields at the same angle but only have to perform the rotation action on the first text string.

Attributes, text, and fields share the same rotation memory. In other words, if you place and rotate text at 90 degrees, then immediately place a field, that field will be placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select-and-rotate actions that take place after object placement.

Text Summary

The text you enter is case-sensitive; what you enter is what you will get.

If you want to enter multiple lines (e.g., a list), you can start new lines in the text box by using the Enter key.

You can change the text justification by specifying one of nine justification points in the Place Text dialog or Modify Text dialog.

With Modify Text (select the text, click right button, and click on Modify in the popup menu), you can change the content of already placed text. In either of these dialogs, you can change the text style, add/delete a text style, or modify the text justification. To change or add/delete a text style, click **Edit Styles**, which activates the Options Text Style dialog, where you can add, modify, view, rename, or delete text styles.

note:

You cannot delete a text style currently in use.

Related Topics

Options Text Style

Place Attribute

Places an attribute according to the **Name** and **Value** options you select in the dialog.   This command allows you to place an attribute within a collection of objects comprising a pattern or component.

The predefined Names are: **Description**, **NoSwap**, **Part Number**, **RefDes, SwapEquivalence**, **Type,** and **Value**.

The Value box specifies the attribute definition, e.g., the actual filename rather than just the place holder *Filename*.

To edit an attribute after it's been placed, run the File Design Info command and click the **Attributes** button.

Placing an Attribute

1. Choose Place Attribute, or click the attribute toolbar button.
2. Click in the workspace to display the Place Attribute dialog.
3. Choose an attribute category from the **Category** list.   All pre-defined attributes for the category appear in the **Name** list.
4. Select an attribute from the **Name** list.   This name, unless you selected user-defined, appears in the **Name** edit box.
5. For user-defined attributes, type an attribure name in the **Name** edit box.
6. Type a value for the attribute in the **Value** edit box.
7. Set attribute properties.
8. Click **OK**.
9. Move the cursor into the workspace and click, drag and place the attribute.   Before you release the button to place it, you can move, rotate, or flip the placement box (see following section).

Rotate or Flip After Placement

To rotate or flip an attribute after it has been placed, select it and press *R* to rotate or *F* to flip while it is highlighted in the select mode.

The *R* key rotation is 90 degrees.   You can use *Shift+R* to rotate to whatever increment you set in Options Configure (a customizable rotation).

You can rotate or flip an attribute *as you are placing it*.   For rotation, whatever angle is the result of the rotation will apply to the next attribute you place.   For example, you are placing an attribute and you rotate it 25 degrees before you finish it (*Shift+R*, with 25 degrees set in Options Configure).   Then you place another attribute; it will be placed at the same 25 degree angle *without any rotation action*.   You can place multiple attributes at the same angle by rotating the first attribute as you place it.

Attributes, text, and fields share the same rotation *memory*.   In other words, if you place and rotate text at 90 degrees, then immediately place an attribute, that attribute will be placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select and rotate actions that take place after object placement.

If you don't specify a value for an attribute, the attribute key name appears in brackets, e.g., {Type}.

Place Field

Places a field containing design information such as date, time, author, etc.

The information that is displayed when you place fields is determined by what you have specified in the File Design Info dialog (accessed through the File Design Info command). You can place a field from a selection of field types including: date, current date, time, current time, author, revision, filename, title, and drill symbol. Current date and current time are taken from the computer's clock. If you just select date then you must specify the date in the File Design Info dialog. The same is true for time (as opposed to current time).

All information except current time and current date must be specified in the Design Information sheet, otherwise you will place a generic field, e.g., {author} rather than {W. Shakespeare}.

To Place a Field

1. Select Place Field and click in the workspace. A dialog is displayed with a combo box containing a list of field types.
2. From the list in the combo box, select what kind of field you want to place. Click **OK**.
3. Move the cursor in the workspace to where you want to place the field; click to place it. If you click again, the Place Field dialog will be displayed so that you can choose another type of field for placement.
4. You can rotate a field as you are placing it by keeping the mouse button depressed and pressing R for 90 degrees (default) or Shift+R for a rotation value that you specify in Options Configure.

Rotate/Flip

To rotate or flip a field after it has been placed, select it and press R to rotate or F to flip the field while it is highlighted.

The R key rotation is 90 degrees. You can use Shift+R to rotate to whatever increment you set in Options Configure (a customizable rotation).

You can rotate or flip a field as you are placing it. For rotation, whatever angle is the result of the rotation will apply to the next field you place. For example, you are placing a field and you rotate it 25 degrees before you finish it (Shift+R, with 25 degrees set in Options Configure). When you place another field; it will be placed at the same 25 degree angle without any rotation action. If you decide to rotate the second field, it will increment 25 degrees more, resulting in a 50 degree angle. Therefore, you can place multiple fields at the same angle but only have to perform the rotation action on the first field.

Attributes, text, and fields share the same rotation memory. In other words, if you place and rotate text at 90 degrees, then immediately place a field, that attribute will be placed at a 90-degree angle. This rotation memory derives only from rotation action during placement, not from select and rotate actions that take place after object placement.

Place Connection

Places a connection between pads. Vias are not supported. Connections do not belong to a particular layer and do not have a user-definable width.

To perform a connection, you must have already placed free pads and/or component pads. The connected pads become nodes of a net.

You need to change layers to place connections between surface pads that are on different layers. For example, switch the current layer to Top, begin the connection on one of the Top layer pad, change layers to the Bottom and complete the connection to a bottom layer pad.

To Place a Connection

1. Choose Place Connection or use the connection button on the Toolbar as a shortcut.
2. Click on a pad and draw a line by dragging to another pad and releasing. You will be prompted for a net name (a default is provided). Enter the net name (or accept the default net name as shown below) and click **OK**.
3. You can continue placing connections in the same manner, from pad to pad (node to node). If the connection shares the same node as a previous connection, then they become part of the same net and you will not be prompted for a new net name.

   If you place a connection between nodes that don't share a node with an existing net, you will be prompted for a new net name (or you can use the automatically incremented default net name, e.g., NET00001).

Free Pads

You can place connections to free pads, but the free pads will not appear in the netlist when you generate a netlist (Utils Generate Netlist).

Unused Component Pads

Components can have unused pads that a electrically inactive (that is they have no designator). You cannot place a connection to an unused component pad.

Merging Nets

If you attempt to place a connection between two nets, in effect merging nets, you will see the Merge Nets dialog. From there you can choose to name the merged net one of the two net names that are being merged, or give it a new name.

For routing of the connections you make, use the Route Manual or Route Interactive commands.

To optimize the connections for minimum length, use the Utils Optimize Nets command.

Route Commands (PCB)

PCB supports multiple autorouters. Prior to version 2.50, ACCEL PRO Route was the only autorouter available for PCB designs. But with release 2.50, PCB now supports the following autorouters:

- QuickRoute
- PRO Route 2/4
- ACCEL PRO Route
- CCT SPECCTRA (valid for SP10, SP4P, and SP2 products)

The look and feel of the user interface is consistent for all autorouters. The menu structure is similar for all routers. The supporting dialog boxes are router-sensitive, so that router-specific data such as pass selection are different for each autorouter. The Route Autorouters dialog is the dialog that lets you select and configure a router as well as start the routing process.

Route Menu

Route Autorouters

When you run the Route Autorouters command, the Route Autorouters dialog appears. The Route Autorouters dialog lets you choose the autorouter you want to use to route your designs and to set autorouter options for the selected autorouter. You can also start and restart the autorouting process from this dialog.

The **Autorouter** combo box defaults to Quick Route, the embedded router. To select a different router, click the down arrow and choose a router from the list. Notice that dialog changes depending on the router you select. To get information about a specific router, click on a router from the list below.

Supported Routers

ACCEL Quick Route
ACCEL PRO Route
CCT SPECCTRA
ACCEL Route 2/4

ACCEL QuickRoute

When you select Route from the **Autorouter** combo box, the Route Autorouters dialog appears as follows.   Although there are differences in functionality, QuickRoute dialogs will look familiar to ACCEL PRO Route users.

Using Quick Route, you can perform the following functions from the Route Autorouters dialog:

Start
Restart

Strategy Box

Strategy File
Output PCB File
Output Log File
Load
Save
Set Base

Layers
Net Attrs
Passes
Via Style
Routing Grid
Line Width

Error Messages Box

Error Messages

See also:

Constraints

During routing, commands are available from the following Route menus:

View Menu
Route Menu
Options Menu

ACCEL PRO Route

When you select PRORoute from the Autorouters combo box, you access ACCEL PRO Route. Several buttons, which now appear in the Route Autorouters dialog, are simply shortcuts to PCB commands and functions: **Clearances**, **Layers**, **Net Attrs**, **Line Width**, **Via Style** and **Route Grid**. By pressing any of these buttons, you are taken to PCB dialogs which should already be familiar to you.

The remaining strategy options instruct the router in its operation. All of these settings are saved to, and can later be loaded from, a strategy file using the **Save** and **Load** buttons. This gives you an easy way to test (and preserve) several strategies for optimum results. Although default file names and strategy values are provided, you can override these names and values.

Using ACCEL PRO Route, you can perform the following functions from the Route Autorouters dialog:

Start
Restart

Strategy Box

Strategy File
Output PCB File
Output Log File
Load
Save
Set Base

Design Rules
Layers
Net Attrs
Passes
Line Width
Via Style
Routing Grid
Via Grid Multiple

Options Box

Options Box
Simaltaneous Class Routing

Copper Share Box

Copper Share

Error Messages Box

Error Messages

CCT SPECCTRA Router

The SPECCTRA autorouters (SP10, SP4P, and SP2) work differently than other PCB autorouters. There is no real-time, record-oriented data exchange mechanism (like PRO Route); instead, the SPECCTRA autorouter runs in batch mode, driven by a command file called a DO file. You set up a DO file using the Route Autorouters dialog. When you start running SPECCTRA, PCB runs the autorouter in a Windows DOS shell.

You must save your design in ACCEL ASCII format. PCB then translates this format into the native SPECCTRA format. Upon termination of the autorouter, PCB merges the original ACCEL ASCII file with the routes produced by the autorouter. The resultant file is your routed design.

If you start running SPECCTRA and the design is not currently in ACCEL ASCII format,you are prompted to save the design in ASCII format or abort the routing operation.

Route Autorouters Dialog

When you select the SPECCTRA autorouter, the Route Autorouters dialog dynamically changes to simplify DO file creation, building net classes and specifying and line options. The Strategy File button changes to a DO File button to reflect the fact that SPECCTRA uses a different command file format.

DO File Button

The DO File is an ASCII file that contains SPECCTRA commands that execute in sequence to control autorouting. It includes all data needed by the autorouter to route the board. The DO filename initially appears as the same name as the current design file, but with an .DO extension. This is the default filename.

Output PCB File Button

The **Output PCB File** button displays a dialog in which you can specify the name and location of your output design file after it has been routed. Here too, a default name has been provided. The letter R (for routed) precedes the current design filename. The last character is dropped if the new name exceeds eight characters. The default file extension is .PCB.

The default name can be overridden by clicking the **Output PCB File** button. The dialog which appears allows you to specify the name and location of your routed PCB board. Again, this is a Windows common dialog, with a familiar look and feel of other Windows dialogs used for opening, retrieving or creating files.

Output Log File Button

In addition to the output file, the SPECCTRA autorouter generates a report file at the end of the routing session, detailing the results of the session. The **Output Log File** button displays the Select Output Log File dialog, in which you can specify the name and location of your report file. This dialog is a Windows common dialog, with a familiar look and feel of other Windows dialogs used for opening, retrieving or creating files.

Load Button

Select the **Load** button to restore saved DO files. Choose the DO filename, then click Load. The DO file is then read into memory and may be edited as text or through the DO Wizard.

Save Button

Anytime after you have selected a DO filename, you can save the file by clicking Save. The DO file is also saved automatically when you start the route.

Set Base Button

The **Set Base** button returns the DO and output files to their default filenames. This is a simple way to go back and start over again when assigning filenames. The default names are derived from the design filename, including the full path.

DO Wizard

The easiest way to create or manipulate a DO file is by clicking the **DO Wizard** button. The SPECCTRA DO File Wizard dialog appears. The SPECCTRA DO File Wizard dialog is an intelligent DO file editor that makes creating DO files more efficient.

From this dynamic dialog you can select DO commands, enter information, and add information to the DO file without having to edit DO file text directly. The DO Wizard has an **Auto Create DO File** button that allows you to quickly create a DO file complete with current grid, line width and layer settings. The DO commands chosen closely match those of other PCB autorouters. The DO Wizard creates commands with the correct syntax. Refer to your SPECCTRA User's Guide and Reference Manual for a complete discussion on DO file options and use.

Edit as Text

You can click the **Edit as Text** button to bring up the file in a text editor. The DO file created or modified in this manner is used verbatim as input to the SPECCTRA autorouter.

Net Classes Button

The option lets you define a group of nets that share common rules. Collections of nets sharing the same rules are referred to as a net class.

When you click the **Net Classes** button, the Net Classes dialog appears.

This class editor allows you to create named net classes using pre-defined SPECCTRA autorouter clearance rules and then assign nets to that class. You can also add user-defined attributes to the net classes for your own use.

For net classes, you can specify general clearance rules. These rules can be further refined by specifying clearance rules for pairs of objects, like pad to pad clearances or line to via clearances. To set net class to net class rules, use the Options Design Rules dialog in ACCEL PCB.

The precedence order used is (from lowest precedence to highest precedence) :

- Default design (includes layer specific clearances).
- Net class rules.
- Net class to net class rules.

- Net rules.

Within the net class rules and net rules, object-pair clearance rules have precedence over pad, line, and via clearance rules.

To create named net classes:

1. Enter a class name in the **Classes** box.
2. Click **Add**.
3. Select unassigned nets and click the **Add** button (bottom of dialog) to add the nets to the class.
4. Use the **Edit Attributes** button to assign one or more attributes to this new net class. (Refer to the Edit Nets command section for details.)

Command Line Button

When you click the Command Line button, the SPECCTRA Command Line dialog appears.

The SPECCTRA program is run in a Windows DOS shell and must be launched with the correct command line arguments. The SPECCTRA Command Line dialog gives you control over the exact manner in which the autorouter is run.

The settings shown here are the recommended defaults. Note that DOS command lines are limited to a maximum of 128 characters. If that limit is exceeded, an error message is displayed in the DOS box or by Windows and the router is not started. The edited command line is saved for the next run. These checkbox options are described in the CCT manual.

The Autorouting Process

The SPECCTRA autorouter can run in graphics mode or text-only mode, but in either mode it runs in a Windows DOS shell. In graphics mode, the DOS shell does not run in the background under Windows 3.x. In text-only , the DOS shell runs in the background. The graphics mode autorouter runs only in the foreground when the Windows DOS shell is full screen. Therefore, once you start the CCT autorouter in graphics mode, focus switches to the autorouters DOS shell where it must remain until the router is finished. While the autorouter is running, this message appears if you switch back to PCB during the route:

SPECCTRA Autorouter is running. PCB has been suspended until Routing Completes.

An autorouting menu and Toolbar do *not* be appear; they are not needed. PCB simply waits for the SPECCTRA task to complete. PCB knows when the autorouter is done and proceeds with the second part of the translation process.

Log File

The log file is a combination of output from the PCB-to-SPECCTRA translator, the SPECCTRA autorouter output, and the SPECCTRA-to-PCB translator. Post-route processing embeds the status file produced by the autorouter (and specified in the DO file) into the log file only if the System LogFile name is provided in the command line. Thus, the Route View Log command in PCB provides you with post-route information.

Manual File Translation and Autorouting

The PCB interface to the SPECCTRA autorouter is fully automatic. You can also "manually" invoke the SPECCTRA autorouter outside of PCB, for example to run it on an alternate machine.

The steps involved in manually autorouting a PCB design with SPECCTRA are as follows:

1. Create a DO file and set up net classes as described above.
2. Save the unrouted PCB design in ASCII format (a .PCB extension is assumed).
3. Open a Windows DOS box and from the DOS prompt change to the PCB installation directory:

   cd\PCB
4. Run the Tango-to-SPECCTRA translator using the following command line arguments:

   ACCL2SP <design.pcb> -o <design.dsn>
5. The output of step 4, <design.dsn>, is a SPECCTRA design file. Autoroute it as described in your SPECCTRA documentation, using the DO file created in step 1.
6. The "routes" output from SPECCTRA must now be merged into the original PCB design. Assuming the routes to be in a file named <design.RTE>, run the SPECCTRA-to-Tango translator using the following command line arguments:

   SP2ACCL <design.rte> -orig <design.pcb> -o R<design>.PCB
7. If no command line arguments are given, the translators prompt for input.
8. The file R<design>.PCB is the routed design and may be loaded into PCB for review.

note:

The tranlators, ACCL2SP and SP2ACCL, use conventional DOS memory and have limits on the size of the input file. The entire ACCEL ASCII file is read into memory before translation. Therefore, your computer may run out of memory with large designs.

Two additional translators are also supplied: ACCL2SPW and SP2ACCLW. These are the exact same translators built to run as native Windows applications. Since Windows applications allocate from a virtualized extended memory space, these versions of the translators work with very large designs.

ACCEL Route 2/4 Router

ACCEL Route 2/4 is a reduced functionality version of the PCB autorouter. This version limits the complexity of the design file that may be submitted to the PCB autorouter. This product enables the user to route boards with one or two signal layers with no pin count limitation, or route boards with up to four signal layers with a limit of 4000 pins (net nodes) in the design. Designs exceeding both limits will not be routed by the autorouter. The availability of this interface, as with the full feature interface, is controlled by the PCB security key.

Documentation for the ACCEL Route 2/4 autorouter, subject to the limits described above, can be found in the PCB Using Route manual.

See also:

ACCEL PRO Route

Strategy File

The Strategy File is a collection of settings that you set up in Route Autorouters. It includes the name of the router you are using and all data needed by the autorouter to route the board. The strategy file name initially appears as the same name as the current design file, but with an .STR extension. This is the default file name. You can, however, save route strategies to a file having any name you wish to assign.

Strategy File Button

To choose an alternative strategy file, use the **Strategy File** button. The **Strategy File** button displays the Select Strategy File dialog in which you can select the name and location of a strategy file you want to save or load. This is a Windows *common* dialog, with the familiar look and feel of other Windows dialogs used for opening, retrieving or creating files.

Output PCB File Button

The **Output PCB File** button displays a dialog in which you can specify the name and location of your output design file after it has been routed. Here too, a default name has been provided. The letter R (for routed) precedes the current design file name. The last character is dropped if the new name exceeds eight characters. The default file extension is .PCB.

The default name can be overridden by clicking the **Output PCB** button. The dialog which appears allows you to specify the name and location of your **routed** PCB board. Again, this is a Windows *common* dialog, with a familiar look and feel of other Windows dialogs used for opening, retrieving or creating files.

Output Log File

In addition to the output file, ACCEL PRO Route and Quick Route generate a report file at the end of the routing session, detailing the results of the session.

The **Output Log File** button displays the Select Output Log File dialog, in which you can specify the name and location of your report file. This dialog is a Windows *common* dialog, with a familiar look and feel of other Windows dialogs used for opening, retrieving or creating files.

Valuable information about your PCB design, routing strategy and route preference data is provided.

Before routing starts, the input PCB is analyzed. General information as well as your routing strategy and router-selected options are written to the log file. After each pass, per-pass and total routing statistics are written to the file. When routing is completed, summary information is written to the file.

Refer to View Log for additional information.

Load Button

Select the **Load** button to restore saved strategies. Choose the strategy file name, then click **Load**. The design and routing data is updated to the saved values. You will see these changes in the Route Autorouters dialog.

Save Button

Anytime after you have selected a strategy file name, you can capture the current strategy by clicking **Save**. The strategy file is also saved automatically when you start the route.

Set Base Button

The **Set Base** button returns the strategy and output files to their default file names. This is a simple way to go back and start over again when assigning file names. The default names are derived from the design file name, including the full path.

Layers

The **Layers** button on the Route Autorouters dialog provides a shortcut for activating the Options Layers command and dialog.

You can add, delete, enable, disable and modify the routing bias of a layer.   Changes made for autorouting apply to PCB.

ACCEL PRO Route supports true multi-layer autorouting on up to 30 signal layers including Top, Bottom and user-defined layers.   The router seeks to make connections on any of the enabled signal layers; it is not restricted to routing on layer pairs.

ACCEL PRO Route supports any number of plane layers, up to a total of 99.   The nets to be connected to planes are defined in PCB when the netlist is first loaded.   These can also be defined manually when a plane layer is created.

ACCEL PCB provides Top, Bottom, Board and several non-signal layers automatically.   Additional signal layers (e.g., MID1-MID8) and power and ground plane layers are automatically defined when a Tango PCB or PCB PLUS (Series II) board is loaded.

For additional information, refer to Options Layers.

Net Attrs

Clicking the **Net Attrs** button from the Route Autorouters dialog displays the Edit Nets dialog.   This dialog allows you to display or hide nets, edit or view net attributes, change net names, get net information or select information by net.   Setting net attributes is important for routing, because it lets you override the global values on a net-by-net basis (for example: the line width and via style).

See the Edit Nets command for details.

Quick Route

QuickRoute supports the following net attributes.   These attributes override the default settings for each net to which they are attached.

AUTOROUTEWIDE = <TRUE/FALSE>

VIASTYLE = <via style name> for wide nets

WIDTH = <routing track width in current units>

NOAUTOROUTE = <TRUE/FALSE>

MAXVIAS = <number of vias> for maze route only

Limitations

- Only a single via style is allowed across all non-wide net classes.   The current routing via as specified in the Route Autorouters dialog must be geometrically identical to any VIASTYLE attribute that exists for a net that is not routed with the wide pass.
- For nets routed with the wide pass, each of the net attributes VIASTYLE, WIDTH, and AUTOROUTEWIDE must be specified.   There can be a different via style or line width for each wide net.
- Global ripup and the RIPUP attribute are not supported.
- The MAXVIAS attribute is supported for maze routing only.

Refer to the Edit Nets command for details.

Passes

The **Passes** button allows you to enable different types of routing passes. When you click the **Passes** button, you see the Passes Selection dialog. All passes except Route Cleanup and Via Minimization default to ON. From this dialog you have the following options:

Wide Line Routing
Horizontal
Vertical
L Routes (1 via)
Z Routes (2 vias)
C Routes (2 vias)
Any Node (2 vias)
Maze
Any Node (Maze)
Route Cleanup
Via Minimization

Via Style

A via style defines a stack of shapes for each layer or layer type that make up the via. This option indicates the via style to be used for vias added by the autorouter. The shape and size of the item on each layer may vary. By default, shape definitions are explicitly provided for top and bottom layers, signal layers (not specifically defined), plane layers (not specifically defined) and non-signal layers (not specifically defined). However, you can add shapes for **any** layer. Additionally, with the exception of the default style, which is read-only, pad definitions for any layer can be modified.

You can add, delete, or edit via styles by using the series of available dialog boxes. This command also sets current via style for the Place Via command for PCB and can be accessed from the Options Via Style command.

note:

With this command you are defining the global defaults for nets as they relate to a via style. For wide routes only, you can override these defaults on a net-by-net basis by adding the VIASTYLE attribute with the Edit Nets command.

You set the default routing via by setting the desired via to be current, marked with an asterisk (*).

note:

Do not confuse the default routing via with the default via style. The default via style, which is seen as **(Default)** in the Options Via Style dialog, is a single style provided as part of PCB. It may not be changed or deleted. The default routing via is the via style that the autorouter uses to route all nets, except ones with the VIASTYLE attribute set to override this default. The default routing via may be the default via style, **or it may be any other style you choose.**

You may create a new via style or modify an existing one directly from this dialog. Additionally you can view the default via style and delete an existing via style. See Via Properties for more details.

Quick Route Limitations

- Only simple via and pad styles are supported.
- Only a single via style is allowed across all non-wide net classes. The current routing via as specified in the Route Autorouters dialog must be geometrically identical to any VIASTYLE attribute that exists for a net that is not routed with the wide pass.
- The routing via can be no larger in diameter than twice the current routing grid. In the case of non-uniform routing grids, the smallest of the individual grid values is the limit.
- There is no via grid multiple. Vias are placed on the routing grid.

Error Messages

Use the **Error Messages** box to set where you would like error messages to appear.

You can direct the messages to appear on the screen only, only in the Output Log File, or both (the default setting). If you choose **Output to Log File**, routing continues uninterrupted, because you do not have to respond to error messages.

Start Button

Clicking this button starts the routing process. Several changes occur to the screen:

- The Menu bar changes to offer route-specific commands.
- The Toolbar icons are replaced with icons for route commands.
- The Status line displays each step of the routing process.

The autorouter is then initialized, the board is prepared for routing, data is transferred to the router and the router analyzes your design. During this time, the Status line keeps you informed of all activities. Routing begins, pass-by-pass, and the Status line displays each completed line as it is routed.

Remember, you are in Windows. While routing you have access to other Windows applications. Note however, that the autorouter is resource-intensive; performance of the router and of other programs is affected as the demand for resources goes up. The Pause command can be used to temporarily stop the autorouter and relinquish resources for other tasks, if this is necessary.

The router routes the board using the current design and route strategy information. Your board must be saved prior to routing to ensure a known base.

If you run the command after having edited a strategy file that contains an existing output filename, the program prompts you to **overwrite** the output file or **cancel**. In this way, you can avoid overwriting files you may wish to keep.

Commands available during the routing process are:

- Display (colors, layers, items, cursor shape, display mode, etc.). Options Display
- Info (current routing status display) Route Info
- Zoom (various zoom levels and types). View Commands
- Cancel to stop or suspend a route. Route Cancel
- Pause/Resume. Route Pause and Route Resume
- View Log to obtain a snapshot of the output log file during routing. Route View Log
- Help.
- Prompt, status and Toolbar line display control. View Commands

After interrupting routing to run one of the above commands, the routing process continues.

note:

ACCEL PRO Route doesnt performs a DRC on pre-routes. You should execute the PCB Utils DRC command to check pre-routes before the board is routed by ACCEL PRO Route.

Quick Route Constraints

- Only simple pad and via styles are allowed.
- Only a single via style is allowed across all non-wide net classes. The current routing via as specified in the Route Autorouters dialog must be geometrically identical to any VIASTYLE attribute that exists for a net that is not routed with the wide pass.
- For nets routed with the wide pass, each of the net attributes VIASTYLE, WIDTH, and AUTOROUTEWIDE must be specified. There can be a different via style or line width for each wide net.
- The routing via can be no larger in diameter than twice the current routing grid. In the case of non-uniform routing grids, the smallest of the individual grid values is the limit.
- The allowed routing grids are 10mil, 12.5mil, 16.7-16.6-16.7mil, 20mil, and 25mil. Metric grids are not supported, even in mm mode.
- The routing line width cannot exceed the grid value. In the case of non-uniform grids, the smallest of the individula grid values is the limit.
- The NOAUTOROUTE net attribute is supported.
- Disabled layers are not sent to the router and a warning message is generated if disabled signal layers have copper on them. The routes are discarded.
- Checkpointing is not supported.
- Global ripup and the RIPUP attribute are not supported.
- The MAXVIAS attribute is supported for maze routing only.
- Copper sharing is not an option, the router shares copper with vias.
- The router always uses orthogonal routing.
- There is no auto pass nor auto grid selection.
- There is no via grid multiple. Vias are placed on the routing grid.
- The log file does not contain entries for options that are not supported removed. However, the format of the log file is the same.
- There can be no more than four plane layers.

Options

The Options area of the Route Autorouters dialog lists the following options.

Auto Grid

Allows ACCEL PRO Route to select the best grid, based on the board to be routed and the design rules you specified. When the **Auto Grid** option is enabled, the autorouter ignores the manual grid setting.

Ripup

When enabled, this option allows ACCEL PRO Route's reconstruct algorithm to remove and re-place pre-routed lines during the Iterative and Manufacturing passes.

This option should be disabled only when you have pre-routed lines that must remain intact as you routed them in PCB. ACCEL PRO Route will leave those pre-routes alone. However, if you have no sacrosanct pre-routes, always enable this option, giving ACCEL PRO Route the maximum flexibility to converge on the best design in the shortest time. You can also control ripup per net using the **Net Attrs** button.

ACCEL PRO Route never rips up a line without re-placing it, with the original or a new topology. So you will never lose previously routed lines.

Diagonals

The **Diagonals** option allows you to enable diagonal routing, which performs 45-degree entry and exit from a pad as well as 45-degree diagonal lines elsewhere on the board. If this option is disabled, no diagonal routes are added to the PC-board.

ACCEL PRO Route provides a great deal of flexibility with regard to the routing of 45-degree lines. If the diagonal routing option is enabled, ACCEL PRO Route will make 45-degree routes, taking into account the type of grid in use. The 45-degree routes are placed during the routing passes.

For SMT or particularly dense designs, disabling diagonals until the board is fully routed may help improve performance. If the router is having trouble completing the board, disable **Diagonals** and run the design through the Iterative passes. After the board routes to 100%, run the output PC-board through the Manufacturing passes of ACCEL PRO Route with diagonals enabled to allow the router to shorten traces and to clean up the board.

The benefits of diagonal routing are that it reduces the number of vias and connection inches, produces more manufacturable results and creates a more aesthetically pleasing design.

Checkpoint Interval Minutes

ACCEL PRO Route saves a checkpoint file at the end of every routing pass and on request by means of the Route Cancel command during routing. You can also direct the program to save a checkpoint file at a specified time interval according to the value set in the **Cpt interval** textbox. The default setting calls for a checkpoint every 120 minutes.

The checkpoint file can spare you the disappointment of lost routes and costly re-routing time in the event of power failure or other system disruption. The checkpoint file is saved with the .CPT extension and may be used to resume the routing session with the Route Restart button.

Copper Share

The **Copper Share** option allows you to specify how the autorouter will do copper sharing; using lines and vias, or vias only. You can also disable the option. When line and via sharing is enabled, ACCEL PRO Route routes connections to the closest line or via in the same net, rather than being restricted to pin-to-pin routing. Via-only copper sharing allows connections to be made through a via, but not directly to a line.

Copper sharing, or "T-Routing," helps to achieve higher completion rates and reduces the amount of copper length on the finished board design. This in turn improves the reliability and operational characteristics of the finished product.

Passes

Allows you to enable different types of routing passes or set the program to automatically select the optimum passes to be run. You also set the number of iterative (rip-up) and manufacturing passes to be run, and indicate if manufacturing passes should be forced if the board isn't 100 percent routed.

You may run up to 20 Iterative passes and ten Manufacturing passes. Sets of (Wide Initial, Wide Comprehensive) or (Memory, Initial, Comprehensive, Exhaustive) passes are run for each net class.

The passes may be broken down into three phases:

- *Constructive*, which includes all of the options from **Wide Via Fanout** through **Exhaustive** in the Pass Selection dialog. These passes function only to add lines and vias to the board.
- *Iterative*, which includes local and global ripup passes. These passes rip up and re-route existing connections to make room for routing other ones.
- *Manufacturing Improvement,* which includes **Manufacturing** and **Final Manufacturing** passes. These passes clean up the board for manufacturing. If the board is not 100% routed, these passes arent run unless you enable the **Force Manufacturing** pass option.

The autorouter schedules passes for each net class. A net class is a collection of nets with the same characteristics (e.g., width and via style). If multiple net classes are defined, one "set" of passes is scheduled for each class.

For example, you would have 2 net classes if the default is 12 mil lines and 40 mil vias, and your power and ground nets are given override attribute values indicating 20 mil lines and 50 mil vias. The router may schedule wide initial and wide comprehensive passes as the constructive passes to complete the power and ground nets. Memory, initial, comprehensive and exhaustive passes might be scheduled for all other nets.

The autorouter automatically schedules iterative passes only for the default net class. This is usually advantageous since the non-default net classes are typically routed first (when the board is empty) and do not need to be repeatedly ripped up. It is a time-saving decision. Alternatively, all net classes are scheduled for iterative pass routing when you manually select that pass. This gives you more control to finish your board.

All net classes are scheduled for the number of manufacturing passes chosen in the **Pass Count** box. The total number of scheduled passes cannot exceed 100.

If you decide to control the autorouter by manually selecting passes, the following description of individual passes may be helpful.

Wide Via Fanout (SMD)

During the Wide Via Fanout pass, wide lines are routed from SMD (e.g., power and ground) pads to vias. Fanouts ensure that all routing layers have access to the surface pads. They are typically made on boards with more than two signal layers. To schedule this pass manually, you must set the AUTOROUTEWIDE attribute for the nets to be fanned out as wide nets. Click the **Net Attrs** button in the Route Autorouters dialog to access this option

Via Fanout (SMD)

During the Via Fanout pass, lines are routed from SMD signal pads to vias. Fanouts ensure that all routing layers have access to the surface pads. They are typically made on boards with more than two signal layers.

Wide Initial

During the Wide Initial pass, bus-width power and ground lines and other wide lines that require not more than three vias are routed in a particular arrangement. This simplifies later signal line routing. During this pass, layer directionality is strongly enforced. To schedule this pass manually, you must set the AUTOROUTEWIDE attribute for your wide nets. Click the **Net Attrs** button in the Route Autorouters dialog to access this option. See the Net Attrs section for additional information.

Wide Comprehensive

The Wide Comprehensive pass is similar to the Wide Initial pass except that layer directionality is loosely enforced and connections to any number of vias are made. This pass is designed to complete power and ground routing. To schedule this pass manually, you must set the AUTOROUTEWIDE attribute for your wide nets. Click the **Net Attrs** button in the Route Autorouters dialog to access this option.

Memory

During the Memory pass, easily made connections on a single layer are completed. These are generally data bus connections to memory arrays. The end points must be located so that they can be joined by lines that are very nearly horizontal or vertical within 100 mils or within the defined grid cell. This pass only permits vias between surface pads.

Initial

During the Initial pass, easily made connections requiring no more than three vias are completed. Layer directionality is strongly enforced and no diagonals are permitted.

Comprehensive

During the Comprehensive pass, more difficult connections requiring no more than six vias are made. Segments directed away from the targets are permitted and layer directionality is loosely enforced.

Exhaustive

During the Exhaustive pass, the most difficult connections are attempted, using whatever means are necessary within the constraints of the design rules. An unlimited number of vias and segments directed away from the target are permitted. Layer directionality is very loosely enforced.

Iterative (rip-up)

The iterative passes are the heart of this reconstruct router. They perform the complex tasks of locating blocking traces and reconstructing alternative paths for them. This allows previously un-routed connections to be made. During this process, the entire board is analyzed to get the highest completion rate possible.

ACCEL PRO Route supports up to ten Iterative passes per net class. Two types of Iterative passes exist for different conditions: *local* and *global* rip-up. The autorouter selects local rip-up of the board if it is 98% complete or if three global rip-up passes have been run consecutively. Otherwise, global rip-up is selected.

A local rip-up pass is similar to an Exhaustive pass in that unlimited numbers of vias and segments directed away from the target are permitted, and layer directionality is very loosely enforced.

Global rip-up is more complex. Like local rip-up, the autorouter permits an unlimited number of vias and, again, layer directionality is very loosely enforced. However, if you watch the Status line during a global iterative pass routing, you will notice that statistics are provided twice for a single pass: once to re-route nets and once to retry remaining open connection. The first time, the router works through all nets and the second time, through all connections.

note:

You specify the number of iterative passes in the **Pass Counts** box. Iterative passes should not be manually selected to route a board that has not been, or will not be, sent through the constructive passes. Performance will degrade because of the considerable complexity of the Iterative pass algorithm. Constructive passes work effectively to do the initial routing and should not be bypassed.

Manufacturing

The manufacturing passes are your cleanup passes; they optimize routing for manufacturing. Because the function of the iterative passes is to complete open connections, these passes place tracks and vias in a way that leaves space for subsequent tracks and vias. Whereas these passes do a great job at completing the open connections, they leave your board far from being manufacturable. That task is left for the manufacturing passes. You specify the number of manufacturing passes to run for each net class in the **Pass Counts** box, up to a maximum of 10.

Other functions performed by the manufacturing passes include the spreading of lines, removal of acute angles and improved pad entry (through the corners of square pads and the ends of rectangular or oval pads). True memory patterns of a single layer are generated. These same routing characteristics provide improved electrical performance and result in aesthetically pleasing designs.

Line segments are moved to other layers to help separate the signals, and diagonals are added to shorten line length. The pass also utilizes a concept known as *copper sharing* to combine lines in the same net.

The manufacturing passes are not intended to improve the autorouter's completion percentage and are generally not enabled until the board is finished.

Final Manufacturing

During the Final Manufacturing pass, routing is refined for manufacturing. The same changes seen during manufacturing passes are seen here, but the changes tend to be more subtle. For instance, if diagonal routing is permitted by the design rules, additional 45-degree corners may be incorporated during this pass.

Force Manufacturing Pass

The **Force Manufacturing Pass** check box allows you to force the router through the manufacturing passes even if the board is not 100% routed. Disable this option if you want routing to stop before the Manufacturing passes if your board isn't completely routed, so that you can try a new grid or make some other design change that might help achieve 100% completion.

Auto Pass Selection

Allows ACCEL PRO Route to select which passes to use during routing. If you want to select passes manually, disable this option and enable the specific pass selections in the Manual Pass Selection area of the dialog. When you enable this option, The autorouter automatically selects constructive, iterative and manufacturing passes that are appropriate for your board. The number of iterative and manufacturing passes scheduled come from the **Pass Counts** boxes (discussed below).

Once the router achieves 100% completion, it skips any remaining constructive or iterative passes. For this reason, you may want to set the Iterative pass count to 10, allowing the router the maximum opportunity to complete the board.

Pass Counts

Maximum Iterative Passes and **Manufacturing Passes**: The maximum iterative and manufacturing pass counts instruct the autorouter on the number of passes to schedule. These numbers are used whether you select passes automatically or manually. However, there is an important distinction for the way iterative counts are used. For Auto pass selection, the autorouter schedules only the default net class for the selected number of iterative passes. For manual passes (assuming the **Iterative Pass** check box is enabled), the autorouter schedules all net classes for the number of iterative passes requested in the **Pass Count** box.

The default numbers are 5 Iterative passes and 2 Manufacturing passes. Acceptable ranges for both are from 0 to 10.

Line Width

You can set the line width for the routed lines by clicking the **Line Width** button on the Route Autorouters dialog. This presents the same menu as if you had chosen the Options Current Line command from the PCB menu. Click the **Line Width** button to display the dialog. Enter the value in the **Line Width** text box (or choose from the options in the combo box) and click **OK**.

Refer to the Options Current Line command for details.

Route Grid

Defines a routing grid if the **Auto Grid** option has **not** been selected. The current, absolute or relative grid is used for routing. In both cases, the autorouter automatically locates to optimal grid origin for routing. When the route is complete, the **Relative Grid Origin** box is updated with the routing grid origin, and the routing grid used is added as the current Relative Grid Spacing. The units used are determined by the setting in Options Configure.

Mode

To specify a routing grid when the **Auto Grid** option is disabled in the Route Autorouters dialog, you may set either mode (**Absolute** or **Relative**).

Grid Spacing: Uniform/Non-Uniform

You specify routing grid spacing in the **Grid Spacing** text box. The desired value may be selected from the Grids list, but you are not limited to using the grids listed there. You specify your own custom grid by typing it in the **Grid Spacing** text box, and clicking **Add**.

The grid spacing defines the spacing pattern of the grid points. The grid pattern may be uniform (equal spaces between all lines) or it may be non-uniform to allow one or more lines to pass between component pads, while keeping the number of grid points to a minimum.

Uniform grids are generally used for relatively simple, low-density PC-boards. The illustration below shows 60 mil pads spaced 100 mils apart, 50 mil grid spacing (using the hatched grid option), with one routing connection running between the pads.

Non-uniform grids are typically required on complex, high-density PC-boards. The main benefit of the non-uniform grid over the uniform grid is performance: it produces far fewer grid positions without the loss of route paths. Thus, completion time is shortened and less memory is required, without losing any freedom in placement.

For example, if you have 100 mils between grid points, you can set non-uniform grid spacing to 40-20-40, which would allow two grid points between pads spaced 100 mils apart. With 8 mil line widths and clearances and 56 mil pads, the autorouter could place two lines between the pads.

note:

Remember to set up your grids correctly before you place your components. Place the components on a relatively coarse grid (e.g., standard DIP components on an absolute 100 mil grid). You can subsequently choose a finer grid spacing for routing purposes (such as non-uniform as described above). The router is able to route to off grid components, but there is a performance penalty in speed and perhaps completion.

Additional Information

Routing and Via Grids

Via Grid Multiple

This option limits via placement during routing to keep routing channels open. It is most useful when fanning out vias for surface mount devices. If you allow vias to be placed at every point, they can block routing channels, reducing completion. The value you specify here is a multiple of the manually selected routing grid.

For non-uniform routing grids, the via multiple is applied to the sum of the non-uniform spacing values. For example, if you have a 40-20-40 mil, non-uniform, routing grid, and type a 2 as the **Via Grid Multiple**, then the autorouter allows via placement every 200 mils (2 times the sum of 40 + 20 +40). This option is valid only when you disable the **Auto Grid** option in the Route Autorouters dialog.

note:

The **Visible Grid Style**, **Relative Grid Origin** and **Delete Grid** options are also available from the Options Grid dialog. Because they aren't required for routing, they are not discussed here.

Design Rules and Routing Grids

There is a direct correlation between your design rules (e.g., line width, clearances, via styles) and the routing grid. Together, these values have an enormous effect on completion and performance of ACCEL PRO Route.

You are always responsible for setting the design rules because they are determined by your own particular fabrication requirements. ACCEL PRO Route, on the other hand, will assist by automatically selecting the best routing grid to go with your design rules. The default values supplied (12 mil for all instances) are satisfactory for general purpose commercial, high volume, low priced production.

If you choose your own routing grid, be certain you consider the design rules before selecting a routing grid, and vice versa.

See also:

Routing and Via Grids

Restart Button

Restarts ACCEL PRO Route from a checkpoint file. When you cancel a route and choose the suspend option, the router saves your route in a checkpoint file, so that you can continue it at a later time. ACCEL PRO Route also saves a checkpoint file at the end of each pass and after the amount of time specified with the **Cpt Interval** option on the Route Autorouters dialog.

Checkpoint files have the same path and base name as your design file, but have a .CPT extension. They are automatically deleted if the route completes the scheduled passes or is prematurely terminated when you select any of the Stop options from the Route Cancel command.

To restart the route from this checkpoint file, you must first load the original, unrouted input board in PCB. Then, select the Restart button from the Route Autorouters dialog. All of the lines and vias placed before the checkpoint was made, exist as part of the checkpoint file, along with your original routing and design strategy. This data is used to restore the autorouter to its state at the time the checkpoint file was created.

note:

The checkpoint file used for restarting is deleted when the route completes, when you cancel after restarting or when a new checkpoint file is created. For this reason, you should save the checkpoint file before restarting if you intend to use it again.

Routing resumes automatically. You don't have the chance to change your design and routing strategy, once you have clicked the Restart button in the Route Autorouters dialog. If you want to make strategy changes, use the Route Cancel command to stop and save your board as an output PCB file. After you make strategy changes, run this command to begin routing the updated design.

Route View Log

Allows you to view the log file derived from your routing session. ACCEL PRO Route generates a comprehensive report file at the end of the routing session, detailing the results of the session. The report is presented in file viewer selected in the Options Configure dialog. Viewing the log file does not interrupt routing.

Valuable information about your PCB design, routing strategy and route preference data is provided.

Before routing starts, the input PCB is analyzed. General information as well as your routing strategy and router-selected options are written to the log file. After each pass, per-pass and total routing statistics are written to the file. When routing is completed, summary information is written to the file.

The repor file contains the following information:

Headers and Footers

The name and version number of the router being used is found in the header. The footer displays the date and time the report was created. You can define your own header and footer using the File Reports command in PCB. This information is stored in the PCB.INI file.

General

The report provides a list of the input PCB, output PCB and strategy file names, your selected units, the available memory and the route start time. The report also lists the routing grid and via grid settings, the checkpoint interval, rip-up pre-routes, diagonal routing and copper sharing options, the iterative and manufacturing pass counts and whether manufacturing passes should be run if the board is not completely routed.

Layer Settings

Provides a listing of the selected layers, indicating their directional bias (horizontal, vertical). Net names are provided for plane layers.

Net Classes

PRO Route determines net classes for nets with the same WIDTH, VIA STYLE, etc. This area of the log provides a listing of each net class and their defined width, via padstack and the maximum number of vias. The autorouter routes all nets belonging to the same net class together in a pass.

Pass Settings

Lists the scheduled passes and identifies the net classes to be routed during each. Some scheduled passes may not be run, as explained in the next section.

Pass Performance

For each routing pass completed, the report file lists a count and percentage of the lines scheduled, completed and deferred during the pass, and for the entire run so far. Also reported are vias and copper length that were added or deleted during the pass as well as for the entire routing session.

Final Board Statistics

Final board statistics lists the total number of pads on the board, the number of equivalent 16-pin ICs (EICs), the dimensions and area of the design, the density (in square units per EIC; the lower this number is, the denser the board), the vias added during the routing process, the total number and length of routed lines (and percentage of routed lines to total lines), the total number of un-routed lines (and percentage of un-routed lines to total lines) and the total execution time for the routing process.

note:

> If you abort the routing process, the report file reflects the final board statistics only up to the point of termination. A warning message appears in the report file, indicating the type of termination request made (stop and save or stop and don't save).

Route Miter

Not available for ACCEL TangoPCB.

Converts 90-degree corners on routed connections into 45-degree mitered corners or 90-degree arcs, depending on which option you select. The 90-degree corner must be true horizontal and vertical segments and have the same line width. This command only works on 90-degree tracks; it does not work on 45-degree tracks or arcs.

1. Select Route Miter or the Toolbar button, and the Route Miter dialog appears.

   Once you have selected the corner style option from the Route Miter dialog you can miter any number of corners with that style. After you finish mitering, click the right mouse button or Esc to cancel the mode.

2. Click and hold on the 90-degree corner that you want to miter. Drag to create the mitered corner. Don't release the mouse button until you have the proper length of mitering.

Route Manual

Allows you to manually route and/or reroute existing connections or net copper items on multiple layers.

Before you route you must have already placed components and placed connections between the component pads (net nodes). For information about components and connections, refer to Place Component and Place Connection. You can also get connections by loading a netlist (see Utils Load Netlist).

Also before you route, you should have all the layers enabled that you will be using during your routing procedure. To see the list of enabled layers, use the layer combo box on the Status line. You can change between these enabled layers (making each layer the current layer, in turn). Use the Options Layers command to alter the list of enabled layers (enable, disable, add to, delete, modify), or as an alternate way to change the current layer.

note:

The program does not allow routing on a non-signal layer. If you try to change to a non-signal layer while you are in Route Manual mode, an error message will be displayed.

Manually Route a Connection

1. Zoom in so that the connections you want to route are large enough to select. Select Route Manual.
2. Click directly over the connection and drag to where you want to route the connection. The new segment will be drawn in the current layer color. The new copper segment will start from whatever end of the connection it is closest to.

note:

An alternate method for click-and-drag is to press the Alt key, then click the left mouse button (Alt + left mouse). You can then drag the routed line anywhere without having to keep the mouse button depressed.

3. You can continue routing by clicking at points where you want the route to go. Click to select a route point on the connection line, drag the line to the new location. Select the next point and do the same thing. Continue this process until the connection route is satisfactory.

   To complete the routing of a segment, click the right button.

Slash Keys

To leave a partial route, which is represented by a connection line, press the forward slash key "/".

When the **Optimize Partial Route** option in the Options Configure dialog is enabled, pressing the slash key during manual routing terminates the route and causes the guide connection line to connect to the nearest net endpoint.

Status Line Information

The right side of the Status line displays measurement information while you are manually routing. For example, it displays the delta X and delta Y measurements of a ghosted segment while you are dragging, and the total length of the segment being routed (as shown below) when you release. The measurement does not include already routed or yet-to-route portions of a net.

Routing Between Layers

You must switch layers if you attempt to route from, or to, a pad on another layer from the current layer. Use the L key as a shortcut to toggle between enabled layers. A via of the current style will automatically be inserted when you change layers.

Crossing a Blockage

To route any segments where you need to cross a blockage, change to the bottom layer (use the *L* key or the layer combo box). A via is automatically placed to connect the layers and the line from that point reflects the new color (of the layer). Again, whenever you click, it becomes a point where you can create a routing angle or change direction.

Unwind

If you make a mistake, you can press the backspace key, and the previous segment disappears (unwinds). With each press of the backspace key, a previous segment or via is unwound.

Right Mouse Button

To route the remainder of a connection (all the way to the next node), click the right mouse button. You can then click (left button) over another connection segment to begin routing it.

If the current endpoint of the trace being routed is point-to-point with an object belonging to the same net (pad, via, line, or arc) pressing the right mouse causes the route to be recognized as completed, ending the route. The routing *guide connection* (the remaining connection displayed during manual routing) is not replaced with a line trace to the original guide connection destination point when the right mouse button is pressed. The guide connection will be updated as appropriate.

The exception to this behavior occurs when routing to a pad that is the endpoint of the current guide connection. Pressing the right-mouse button completes the route to the pad center.

Orthogonal Modes (O key)

You can press the O key to toggle the orthogonal modes while placing lines during manual routing. You can enable/disable certain orthogonal modes in the Options Configure dialog. The available orthogonal modes are provided as three mode pairs, with a total of six modes (three types, with two variations on each type). The O key cycles through the three mode pairs and the F key toggles between the pair that is current. Refer to the Options Configure command for information.

90/90 Line-Line

Both lines are either horizontal or vertical (displayed perpendicular to each other). For long, the first segment is always longer than the second. For short, the first segment is shorter. You can toggle between the two with the F key.

45/90 Line-Line

You can toggle between the two modes with the F key. The first mode makes the first segment display at a 45-degree angle and the second segment is either horizontal or vertical. The second mode makes the first segment either horizontal or vertical and the second segment is displayed at a 45-degree angle.

90/90 Arc-Line

You can toggle between two modes with the F key. The first mode makes the first segment display as an arc and the second segment is either horizontal or vertical. The second mode makes the first segment either horizontal or vertical and the second segment is displayed as an arc.

On-line DRC

Online DRC checking is performed after each mouse or keyboard click fixes the end of a segment during manual routing. Each routing click can cause up to two trace segments (either lines or arcs) and a via to be placed in the design; all newly created items are checked for DRC violations. When a violation is detected, a beep is generated and a DRC error indicator is placed at the approximate location of the violation; no error message appears. Multiple violations result in two beeps and multiple error indicators.

When a connection is completed, the number of DRC error indicators generated for the connection appears in the Status Info box on the status line, and a report of the violations is presented automatically if you checked the View Report checkbox in the Options Configure dialog and there is at least one violation. This report includes a summary count of the violations created during the routine of the connection, plus the information from the individual DRC error indicators.

The clearance values used for clearance testing of pads, vias, and lines on a particular signal layer are taken from the Options Design Rules dialog. This dialog is discussed in the Clearances section above.

Associated DRC error indicators are automatically deleted when unwinding an offending trace or via, and when undoing a completed connection with violations. Otherwise, DRC error indicators are not automatically removed when the violation is corrected.

Use the Options Configure dialog to enable online DRC checking.

Overlapping Connections

For routing overlapping connections, you may need to temporarily route one connection out of the way to access the desired connection. If you can't unwind or undo the temporary route, you can select and delete it and it will return to its prior existence as a connection.

When you complete a route on a pad and a diamond shape appears, then you have reached the connection destination. The diamond symbol represents a zero length connection. This is helpful if you can't distinguish between overlapping connections.

Polygons

Polygons maintain full net information and correctly remove connections after the polygon is added to a net. You can add the polygon by running the Utils Reconnect Nets command or the Utils Load Netlist command with the **Reconnect Copper** options enabled.

T-Routing

Manual routing of a trace to non-endpoints of an existing trace in the same net completes a net connection. The new trace completes a net connection if it has been point-to-point routed, or crosses the center line of an existing line trace belonging to the same net. Moving an existing trace so that it is point-to-point routed, ends on, or crosses the center line of another trace in the net also completes the net connection.

If the routed trace endpoint lies on the center line of the existing trace, the existing trace is split into two traces, each having an endpoint at the intersection. If the trace being moved or routed crosses an existing trace, the trace being moved or routed is also split into two traces having endpoints at the point of intersection.

Any redundant connections in the net are removed.

Line to arc, or arc to line routing where the endpoint of the trace being routed is point-to-point routed behaves as described above except that there is no automatic trace splitting performed to either trace.

Manual routes that intersect with traces belonging to other nets create on-line DRC and Utils DRC errors.

Routing to Free Copper

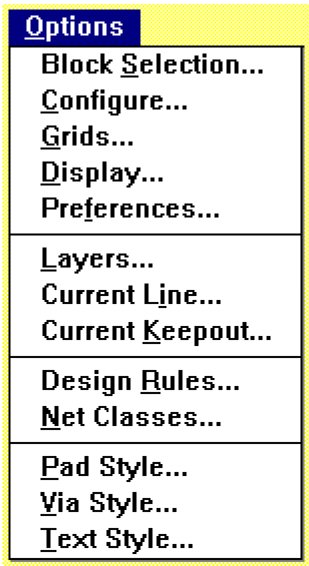
Manual routing to the center of a pad, via, or the endpoint of a line that is not part of any net adds that item to the net being routed. This occurs as each route operation is completed, without running the Utils Reconnect command or the **Reconnect Nets** option in the Utils Load Netlist command.

To add other objects (arcs and polygons) to the net, run Utils Load Netlist with the Reconnect Copper option enabled, or use the Utils Reconnect Nets command after the routes intersecting free copper have been completed.

Stop Routing

You can temporarily stop routing a connection by pressing the forward or back slash keys, leaving an unrouted connection attached to the last routed segment.

Options Commands (PCB)



Options Block Selection

Defines selection filter for block selects (see Edit Select). You can configure the block select feature to select all objects, objects by type, or even objects with specific parameters. And you can specify these selection criteria layer by layer, e.g., certain types of objects for one layer, another type for another layer, etc. The selection possibilities can be as specific, limited, or as general as you choose.

For example, if you wanted to change 10 mil lines to 15 mil lines on 3 layers, you could do it very easily with this command in conjunction with Edit Select and Modify Line, without affecting any other line widths on these or any other layers; this is not to mention ignoring all other objects within the block selection.

Enable Options Block Selection to display the Options Block Selection dialog. From this dialog you can set criteria for selection of specific objects and layers.

Select Mode Box

Inside Block is the default setting, which means that all items inside the selection block will be included, according to the selection criteria you establish in this dialog.

Outside Block allows you to select outside of the block select (outside of the selecting rectangle). All of the selection criteria that you specify in the Options Block Selection dialog will function outside the block rather than inside, which is the default.

Touching Block allows you to select anything that is touching or inside the selection block that you draw. This feature is more inclusive (it selects more items) than either **Inside** or **Outside Block**.

Items Area

All of the objects listed in the dialog can be specified individually whether they will be included in a block select or not. If you want to include only one or two items in your selection, then first click **Clear All** to disable all items, then individually enable the specific items (with the check boxes). If you want to exclude only one or two items in your selection, then first click **Set All** to enable all items (if they are not already enabled), then individually disable the specific items you want to exclude (with the check boxes).

The enable/check boxes have three states: checked (included), blank (excluded), or grayed (masked).

Items Buttons (Selection Mask Dialogs)

Those items with corresponding **Property** buttons (e.g., Port) have three states:

- included (checked).
- excluded (blank).
- masked (grayed).

This feature allows you to narrow your selection further by setting specific properties as selection criteria for any item with a corresponding **Property** button. To access these property masks, set the item to its third state, the masked (grayed) state, in the items area. This allows access to the items button. For example, to create a selection mask for lines, you click the **Line** button and a smaller dialog appears (Line Selection Mask) where you can set the search criteria.

If you designate 50 mil for the Line Width, then when you do a block select, only 50 mil lines will be selected within the block.

Selection Mask Parameters

This is a basic summary of the parameters you can work with in the Selection Mask dialogs.

For **Arc** and **Line**, you can specify line width of arcs and line segments for a block select (refer to Free Arc Properties or Free Line Properties).

For **Component**, you can specify component type, reference designator, and value. For example, you can include a specific component of a certain value, and the block select will include only those of that value (refer to Pattern Porperties). Type, reference designator, and value support wild card characters: ? to match any single character, and * to match a sequence of zero or more characters. For example, a RefDes value of **U?** matches all components with a two character RefDes string beginning with U.

Notice that **Component**, **Global Attr**, **Pins**, the **Visibility** options appear in the dialog, but are disabled because they are not valid options for block selection.

For **Copper Pour**, you can specify Pour Pattern, line properties, and even the specific net to include in the selection criteria (refer to Copper Pour Properties).

For **Pad**, you can include Pad Number and Pad Style in the selection criteria (refer to Free Pad Properties).

For Plane, you can specify a net and and boundary width (refer to Plane Properties).

For **Text**, you can use the mask feature to search for a specific text string in your block select (refer to Free Text Properties).

For **Via**, you specify the via style to include in the block select (refer to Via Properties).

Layers

The **Layers** area of the dialog allows you to select any combination of layers for the selection mask. When you choose a layer, it becomes "highlighted" or colored. You can individually select or deselect layers by clicking on a succession of layer names and they will become part of (or be excluded from) the selection list.

If you want to exclude only a small number of layers, click **Set All** (the default setting) and individually choose the layers to exclude. To include only a limited number of layers, click **Clear All**, then individually choose the layers to include in your block selection.

This is a scenario for selecting and modifying all lines of a width of 12 mil and changing them to 15 mil.

1. Choose Options Block Selection to display the dialog.
3. Click **Clear All** to disable all of the items in the **Items** area (the **X** disappears from each box).
4. Click the **Line** enable box until the check box becomes grayed. The **Line** button then becomes ungrayed.
5. Click the **Line** button to display the Line Selection Mask dialog. Enter **12 mil** in the textbox. Click **OK**.
6. In the **Layers** area, click **Set All** to allow selection on all layers.
7. Do a block select of the complete design. Only the 12 mil lines will be highlighted.

8. Choose Edit Modify. Enter 15 mil in the textbox. Click **OK**.

   All 12 mil lines in the design will be converted to 15 mil.

Related Topics

For information about determining Styles, refer to the Options Pad Style, Options Text Style, and Options Via Style commands.

For more detailed instructions on the use of the Mask Selection dialog, refer to the respective Modify Object commands. The Modify dialogs are identical in both format and function.

For information about block selection, refer to the Edit Select command.

Options Configure

When you select this command, the Options Configure dialog is displayed, where you can set the PCB editor parameters.

Units

You can alter your display units between mils and millimeters with this option.   Dimensions are not altered, only the unit of measurement of dimension.   A **mil** equals 0.001 inch or .0254 mm.   A **mm** equals 0.001 meter.

This setting will affect all dialogs, reports, Status line displays, etc. containing measurements.   For example, setting **Units** to **mm** causes all dialogs to display measurements in millimeters.   These units can be overridden in many command settings.

Workspace Size

You can set your workspace size to the specified dimension.   If you change units, then the values in this box will change to reflect the units (although the size will remain constant).

Orthogonal Modes

Orthogonal modes are for use while routing (Route Manual) or placing lines (Place Line) using lines that are horizontal, vertical, at 45-degree angles, or with arcs.   The available orthogonal modes are described below.

- **90/90 Line-Line** creates true 90 degree angles, long and short.
- **45/90 Line-Line** creates 45/90 and 90/45 angles.
- **90/90 Arc-Line** creates a combination straight line and arc, in long and short.

For placing lines, only the linear orthogonal modes are available (no arcs).

Refer to Route Manual for more details about the orthogonal modes.

ECOs

Engineering Change Orders are generated when you enable the recorder using the Utils Record ECOs command.   From the Options Configure dialog, you can select a format for these ECO files.   This format applies to the active design.   For more information, refer to the Utils commands section.

The ECO file is generated in a ECO format or a Was/Is format.   Click either the **ECO Format** or the **Was/Is Format** radio button to select a format.

ECO format records full ECOs.   The Was/Is format records only   RefDes changes.

ACCEL Tango PCB supports only Was-Is ECO format.

Online DRC

An Enable Online DRC checkbox is included in a new Online DRC group box in the Options Configure dialog; checking this box enables Online DRC. A View Report checkbox has also be added to the group box; if it is checked, then a report of violations created during routing is automatically presented after each connection is complete. For new designs, both checkboxes default to disabled.

The Online DRC function checks for clearance violations or shorts of traces (both lines and arcs) and vias that are added to a design during manual routing. Online DRC performs a check of the traces and vias against all items, including copper pours, on signal layers to determine if a DRC violation occurred. A violation occurring on a non-signal layer (such as a via intersecting a silk layer item) is not detected with Online DRC.

When Online DRC is enabled, the following DRC checks are included: Netlist Violations, Clearance Violations, and Text Violations. The following DRC options are not available with Online DRC: Netlist Compare, Unrouted Nets, Silk Screen Violations, Unconnected Pins, and Copper Pour Violations. The following actions explicitly do not invoke online DRC checking: placement (other than during manual routing), moving, copying, and editing of signal or non-signal items.

Although you can enable or disable the Online DRC feature at any time, the feature defaults to disabled when you create a new design. The current Online DRC setting is saved to a design file and restored when a design is loaded.

A button for Online DRC is included on the toolbar. When the button is depressed, Online DRC is enabled; otherwise, it is disabled. You can also enable and disable Online DRC from within macros.

Refer to Route Manual for more details about online DRCs.

Connection Options

The **Optimize Partial Route** checkbox is used to indicate whether the guide connection is to be optimized for Manhattan length after manual routing. When enabled, pressing the slash key during manual routing terminates the route and causes the guide connection line to connect to the nearest net endpoint. This optimization doesnt occur if the checkbox is not checked, or if the net being routed has an Optimize=no attribute.

The **Optimize After Delete** checkbox is used to indicate whether any connections added after a trace is deleted should be added in an optimized manner. If it is not necessary to add a connection to maintain connectivity, none will be added. If a connection is necessary, and either the checkbox is not checked or the net has an Optimize = no attribute, the connection is added at the same location as the deleted trace.

Autopan

Allows you to adjust the amount of autopanning that will occur when you move the cursor to the edge of the screen with the arrow keys. An autopan of 25 will move anything at the edge of the screen 25% nearer to the center of the screen; 50 will move fringe objects to the center, etc.

AutoSave

Allows you to enable the AutoSave feature, which regularly saves your files at a user-defined interval. The design is only saved when you select a menu item or a toolbar item. AutoSave wont be performed during autorouting or if a tool is busy.

The following options are available:

- **Enable AutoSave:** Click this checkbox to turn on the AutoSave feature.
- **AutoSave Time Interval:** Enter the time between saves. AutoSave uses a rolling backup to save files, incrementing each subsequent autosave file.
- **Purge Previous Backups:** When enabled, the option causes all backups saved from the previous design session to be deleted when you begin a new design session.
- **Number of Backup Files:** Allow you to specify the number of design files to be archived before file names are reused. This must be a number between 1 and 99.

Rotation Increment

The rotation value you set here will be activated when you select an object and use Shift+R to rotate it. Values must be between 0 and 360 degrees, with tenth degree resolution.

You can also use the R key to rotate a selected object by 90 degrees; R is not affected by what you specify in this dialog.

Zoom Factor

Allows you to adjust the amount of zoom that will occur when View Zoom In or View Zoom Out is enabled. A factor of 2.00 will double (or halve) the size of objects in the workspace, etc. Zoom factors must be greater than 1.00.

File Viewer

Allows you to define the viewer to be used for viewing reports, log files, error reports, etc. Enter the application name (e.g. Notepad). If the application is in a directory that is *not* included in your AUTOEXEC.BAT path statement, include the complete pathname here.

DDE Hotlinks

If checked, this option enables exchange of hotlink data with PCB Schematic. Hotlink data consists of highlighting and unhighlighting commands for components and nets. The state of the **DDE Hotlinks** option is saved in the PCB.INI file.

Solder Mask Swell

Allows you to set the solder mask swell value globally for all pads on the Top and Bottom solder mask layers. You can override the global swell value by specifying a pad style for the Top Mask and Bottom Mask layers (Options Pad Style).

A Solder Mask Swell resists the solder mask on the mask layers, in effect allowing solder to be placed both on the pad and the swell around it. The swell that is created will conform to the shape of the pad increasing the pad shape by the swell value. The swell value is added to the pad as a radius, enlarging the pad shape in all directions; therefore, if you have a 60 mil diameter pad and add a swell of 10 mil, the total diameter of the swell would be 80 mils. When you define a pad size for the Mask layer (creating a local swell value), the global swell value is ignored.

You can view the increased size by making the Mask layer current, and then redrawing (View Redraw) to see the results.

Paste Mask Shrink

Allows you to set the paste mask shrink value globally for the Top and Bottom Paste layers. You can override the global value with a local value by specifying a reduced pad size for the Top and Bottom Paste layers (Options Pad Style).

A Paste Mask Shrink reduces the size of the area that paste will be applied to when components are attached to the board in manufacturing (to prevent paste from squeezing out when it shouldn't). The shrink is a radius value and should be figured as such when subtracting it from the diameter value of a pad. For example, a 60 mil pad with a 10 mil shrink value will create a 40 mil diameter entity. When you define a pad size on the Paste layer (creating a local shrink value), the global shrink value is ignored.

You can view the reduced size by making the Top Paste layer current, and then redrawing (View Redraw) to see the results.

Plane Swell

Allows you to set the plane swell value. This becomes the global plane swell setting. You can override the global value by enabling and setting the **Local Swell** option in Options Pad Style.

A Plane Swell is the gap between the copper on the plane layer and the hole on a pad or via which is not connected to the plane. A plane swell would not be applicable to pads or vias that are thermally or directly connected to the plane.

Auto Plow Copper Pours

When you enable the **Auto Plow Copper Pours** option, copper pours affected by new copper generated by the Manual Route or Interactive Route tool will autoplow when the route **Completes** or **Suspends**.. Plowing can create more islands if you splinter existing islands, yet Automatic Islands Removal is still performed using the settings you established for that copper pour.

Options Grids

Defines current editing grid. The dialog contains numerous grid options you can choose from, and allows you to set up custom grid settings. Grid settings are saved in your design file.

The units used are determined by the **Units** setting in Options Configuration.

Mode

You can specify **Absolute** when you want the grid origin point to be the lower-left corner of the workspace. You can specify **Relative** to allow any point as an origin point.

A typical setting is to set your **Absolute** mode to 100 mil grid spacing, **Relative** to 25 mil grid spacing. In this way you can toggle the Grid button on the PCB Status line between 100 mil and 25 mil. (The relative grid origin should be set to Absolute 0,0 for this to work)

Visible Grid Style

The **Visible** enable box allows you to either display or not display grids. When you enable this option, the grid style options are available. **Dotted** pinpoints grid points, while **Hatched** draws lines along the grids to show grid intersections (like graph paper).

Relative Grid Origin

You can specify your X and Y relative grid origins by entering the coordinates. You must have the **Prompt for Origin** option disabled for it to work.

If you switch to **Relative** grid in the dialog and the **Prompt for Origin** dialog is enabled, you will be prompted for the new origin point after you click **OK**. You will be prompted every time you change from Absolute to Relative Grids using the grid toggle button or the A key.

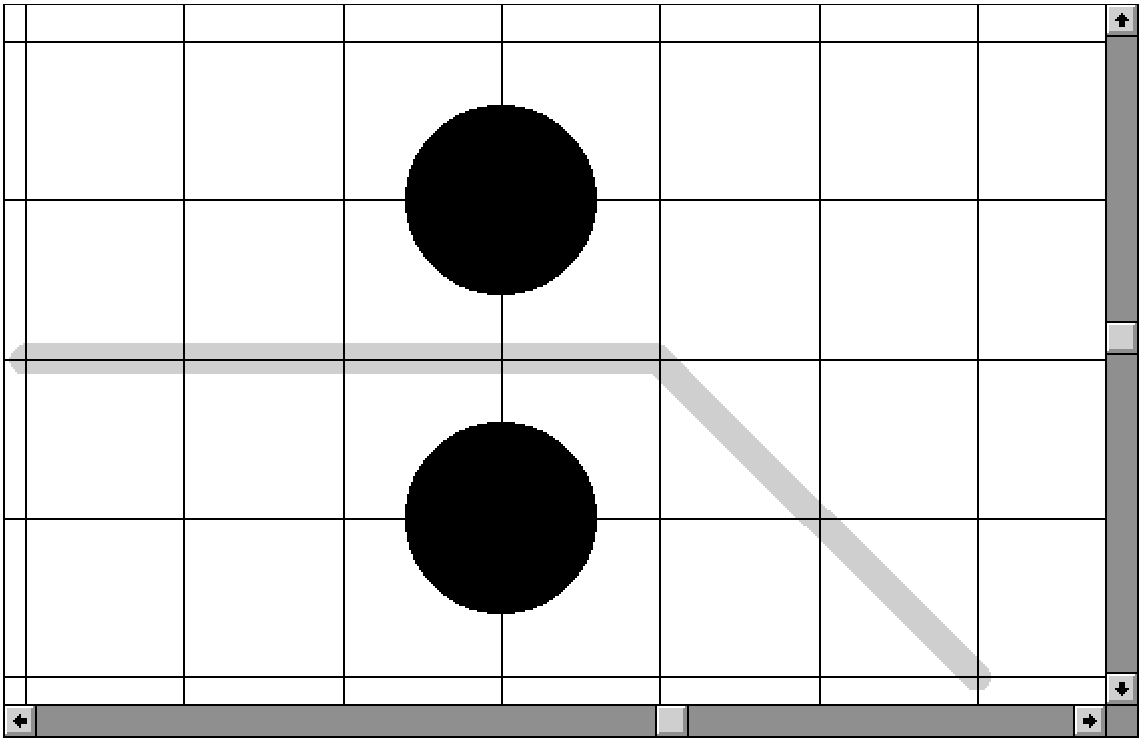
Grid Spacing: Uniform/Non-uniform

You can select appropriate values for grid spacing for specific modes in the Grids listbox. You are not limited to using the grids in the listbox; you can specify your own custom grid spacing in the Grid Spacing textbox, then click **Add** to add it to available choices in the listbox. To delete a grid-spacing value, highlight it in the **Grids** listbox and click **Delete**.

The routing grid defines the spacing pattern of the grid points. The grid pattern may be uniform (equal spaces between all lines) or it may be non-uniform to allow one or more lines to pass between component pads, while keeping the number of grid points to a minimum.

Uniform grids are generally used for relatively simple, low-density PC-boards. Non-uniform grids are typically required on complex, high-density PC-boards to achieve completion of the connections in the minimum amount of time.

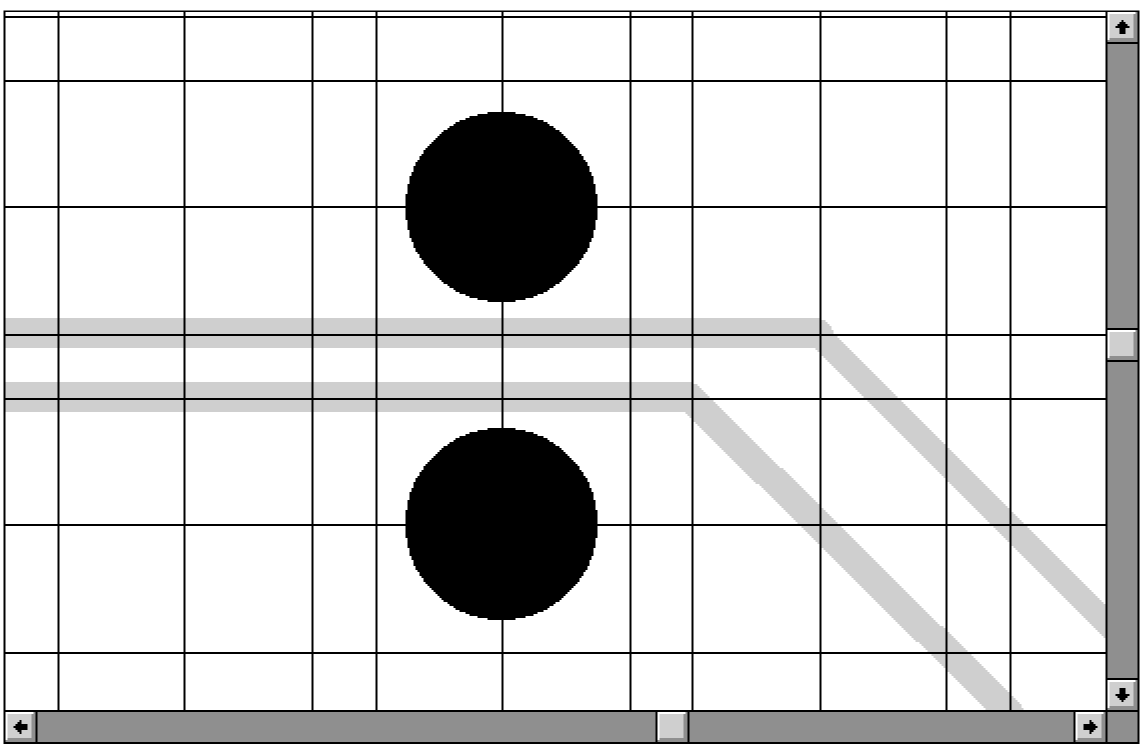
Uniform Grid Spacing



The main benefit of the non-uniform grid over the uniform grid is performance: it produces far fewer grid positions without the loss of route paths.   Thus, completion time is shortened and more freedom in placement is allowed.

For example, if you have 100 mils between grid points, you can set non-uniform grid spacing to 40-20-40, which would allow two grid points between pads spaced 100 mils apart.

Non-uniform Grid Spacing



note:

Remember to set up your grids correctly before you place your components.   It is important that you place the components on the appropriate grids (e.g., standard DIP components on 100 mil absolute).   You can subsequently alter your grid spacing for routing purposes (such as non-uniform as described above).

Grid Toggle: Abs (or Rel) Button

You can easily toggle between absolute and relative grid settings with the grid toggle button or G key. The button appears as **Abs** for absolute grid spacing and **Rel** for relative grid spacing.   Your absolute and relative grids are determined by what you have set in the Options Grids dialog.   For example, if you have Relative set to 25 mils and Absolute set to 100 mils, the Status line will read 25 mil when you are in Relative grid and 100 mil when you are in Absolute grid.

If you use the grid toggle button or the G key to switch to Relative, you will be prompted for the origin point if you have the **Prompt for Origin** option enabled in Options Grids.   The crosshair cursor (the cursor displayed when the system is awaiting input) is displayed; when you click in the workspace, that becomes the relative origin point (X=O, Y=O) and the cursor returns to normal.

Options Display

Defines color preferences, cursor style, and other display-items. When you select Options Display, the dialog is displayed, where you can set your colors.

Whatever colors you establish here will be saved in your .INI file.

Layer/Item Colors

This is a color matrix with objects across the top (columns) and layers along the side (rows). You can specify everything on a layer to be of one color by clicking on a layer button (e.g., Bottom) to display the color. By the same process, you can make an object the same color on all layers by clicking an item button (e.g., Line). Or you can make an object a specific color on a specific layer by clicking on the square where the object and the layer meet. Any of these choices will display the Windows common Color dialog. The scroll bars allow you to access up to 99 layers along the side.

Color Dialog (Windows Common)

Click on a layer/item button and the Windows common Color dialog is displayed, where you can choose the color for the layer, object, or combination. Simply click on one of the colored squares, click **OK**, and the color will be applied.

note:

In the Color dialog, there is a **Define Custom Colors** button which displays another Color dialog for dithered colors. These dithered colors are not currently supported in PCB; only the Basic Colors displayed in the squares are supported. If you choose a dithered (non-solid) custom color, it is automatically mapped to the nearest basic solid color.

Display Colors

These buttons determine general PCB display colors, regardless of layer or item colors. You must be careful that your display colors do not conflict with layer or item colors that you specify; for example, if you set background color to be the same as line color, your lines will not display.

Notice that there are two highlight colors. Successive invocations of certain commands (e.g., View Nets) cycle through the two colors selected here. If you do not want to use two highlight colors, set both colors to be the same.

These commands also display the Color dialog, where you can select your display color.

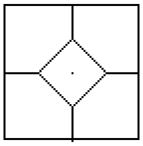
Misc... Button

The Misc button brings up a dialog with numerous display options for you to enable or disable, as follows:

Glue Dots

The **Show** radio button globally enables the display of Glue Dot points, which are used to hold surface-mount components in place until they are soldered during manufacturing. The **Hide** radio button globally disables their display. A glue dot can be part of a pattern in pattern creation. Glue dots appear as free points in the reports unless they are saved as part of the pattern.

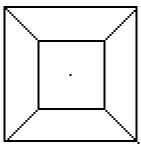
You must click **OK** in the main Options Display dialog for the enable/disable to take place.



Pick and Place

The **Show** radio button globally enables the display of Pick and Place points. Pick and place points provide reference points in directing the *pick and place* mechanism (or *auto insert*) in manufacturing (picking up the component and placing it on the board). A pick and place point can be part of a pattern when you create it. The **Hide** radio button globally disables the display of pick and place points. Pick and place points appear as free points in the reports unless they are saved as part of the pattern.

You must click *OK* in the main Options Display dialog for the enable/disable to take place.



DRC Errors

The **Show** radio button enables the display of DRC Errors in your design. Design Rule Check error flags contain information concerning design errors and the layer of the error (Utils DRC). The **Hide** radio button disables the display of DRC errors Refer to Edit Modify, View DRC Errors command for information on accessing DRC error information by way of an error flag.

Cursor Style

Allows you to change the shape of your cursor to **arrow**, **small cross**, or **large cross**. The large cross stretches horizontally and vertically to the edges of the PCB screen.

Miscellaneous Checkboxes

In this section of the dialog, you have the following options:

Drag by Outline

Allows you to drag objects around the screen more quickly by only displaying the bounding box of the objects that are being moved. This only affects the move and copy operations of the Edit Select command.

Draft Mode

Displays a thin (single-pixel) outline of pads, vias, and text.

Displays segmented, outlined representations of arcs, lines, and any line segment objects (i.e., polygons, cutouts, etc.) Draft Mode can help in faster redraws and also allows you to view any segment overlaps.

Scroll Bars

Enables or disables the display of Windows scroll bars in the active window.

Display Pad Holes

Enables or disables the display of any pad holes in the active window..

Display Pin Designators

Enables or disables the display of the pin designator. The pin designator appears just above the pad center (the top half of the pad). You must be zoomed in sufficiently for the designators to be displayed.

Display Pad Net Names

Enables or disables the display of the pad net names just below the pad centers (bottom half of the pad). You must be zoomed in sufficiently for the net names to be visible.

Display Plane Indicator

Enables or disables whether to indicate which plane the pad or via is attached to, by putting a cross over the pad or via in the line color of the plane layer it is attached to. If there are multiple planes associated with a pad or via, the indicator color will match the line color of a randomly selected attached plane layer.

Translucent Drawing

This checkbox is used for selecting whether or not items are drawn in translucent mode.

Silkscreen in Background

This checkbox is used for selecting whether the silk layers are drawn in the foreground or background.

Options Preferences

Defines keyboard, mouse, and toolbar preferences used to set up the application.

The Options Preferences dialog, which appears when you run the command, has three tab:

- Keyboard Tab
- Mouse Tab
- Toolbar Tab

Keyboard Tab

The Keyboard tab, which appears when you access the dialog, lets you customize shortcut key assignments for menu commands, shortcut key commands, and macros.

- **Command Type:** Choose the type of command for which you want to change shortcut key assignments.
- **Menu Commands/Shortcut Commands/Macros:** Select the command or marcro you want to add a shortcut key assignment to or from which you want to remove a shortcut key assignment.
- **Current Keys:** Displays the existing key assignments for the command or macro you select in the **Menu Commands/Shortcut Commands/Macros** box.
- **Press a Shortcut Key:** Press the keys you want to assign to the selected command or macro. You can press the CTRL or SHIFT key plus any other combination of numeric or alphabetic keys and function keys.

  If the shortcut is currently assigned, the current assignment appears in the **Current Binding** field just below this box.
- **Assign**: Assigns the key appearing in the **Press a Shortcut Key** box to the selected command or macro. If the shortcut is currently assigned, the current assignment disappears
- **Remove**: Removes the key you select in the Current Keys box.
- **Key File:** Allows you to select or create a key binding file to use with this application. When the Select Key File dialog appears, select the file you want to use. The current key file appears at the top of the dialog, and is written to the application .INI file.
- **Default:** Restores original default shortcut key assignments to all commands or macros.

Mouse Tab

The Mouse tab lets you customize certain mouse behaviors.

- **Ctrl/Shift Behavior:** Allows you to choose which keys to use (*Ctrl* or *Shift*) for multiple (extended) selections and which to use for Sub Selections.
- **Cycle-Picking Threshold:** The number of pixels you can move the mouse during cycle-picking.
- **Double-Click Displays Properties:** When enabled, this option allows you to double-click an object to bring up the Properties dialog for that object.

Toolbar Tab

The Toolbar tab lets you choose the location of the Placement toolbar.

- **Horizontal:** The toolbar appears just below the Command toolbar.
- **Vertical Left Side**: The toolbar appears vertically along the left of the workspace.
- **Vertical Right Side:** The toolbar appears vertically along the right of the workspace.

How to

Options Layers

Allows you to view layers and make modifications to layer properties.

When you run the Options Layers command, the Options Layers dialog appears with the Layers tab selected.   This dialog has a Sets tab which is discussed below.

To learn more about a function, click on the appropriate topic from the following list:

Layer Sets
Layers
Line Color
Routing Bias
Type
Buttons

To add a layer, delete a layer or make a layer current, click on the following topic:

Adding a Layer

Layer Sets

Layers sets let you group layers to control the selection, display, and printing of your design, Gerber output and N/C Drill output, DXF output.

Current Layer

The **Current Layer** drop down listbox displays the name of the current layer for the selected layer set. You can change the current layer by selecting a different layer from the drop down list.

Layers

The **Layers** box displays a list of all layers in the design.

Set Name

The **Set Name** box allows you to add a layer set name.

Layer Sets

The **Layer Sets** box displays a list of all layer sets in the design.

Set Contents

The **Set Contents** box displays a list of all layers in the currently selected layer set.

Adding or Modifying a Layer Set

Follow these instructions:

1. To add a new layer set, type a new name in the **Set Name** box and press **New**. The newly-added layer set is empty.
2. To add a layer to the set, select the layer set from the **Layer Sets** box. Select the layer to be added from the **Layers** box and click **Add**.
3. To remove a layer from a layer set, select the layer set from the **Layer Sets** box. Select a layer from the **Set Contents** box and click **Remove**.
4. Click **Enable Layers** to save the layer set.
5. To delete a layer set, select it and click **Delete**.

Layers

The following layers areas are found in the Options Layers dialog:

Layers

In the **Layers** box are listed all existing layers of your design. The letter codes between the layer name and layer number represent the following:

Type:	**S** = Signal, **P** = Plane, **N** = Non-Signal
	E = Enabled, **D** = Disabled
Routing Bias:	**A** = Auto, **H** = Horizontal, **V** = Vertical (signal layers only)

Layers can be moved up and down the list using the **Move Up** and **Move Down** buttons.

Layer Name

Shows the name of the currently selected layer.

Layer Number

Shows the number of the currently selected layer.

L key and Status Line

You can use the *L* key or the layer box on the status line (combo box and scroll arrows) as a shortcut for selecting the current layer. The current layer is identified by name and color on the Status line. Use *Shift+L* to scroll backwards through the enabled layers.

Line Color

Shows the line color for the current layer.

Routing Bias

The routing bias of a signal layer indicates the preferred routing direction. Layer bias is strictly adhered to in certain passes (e.g., Memory) and loosely adhered to in others (e.g., Iterative). The router recognizes the bias only as a guideline when placing lines.

Automatic Bias Selection

The directional bias for the layer is determined by ACCEL PRO Route. If you set the **Auto** selector for the layer, ACCEL PRO Route automatically selects the directional bias for that layer. The bias is calculated following a number of guidelines:

- Typically, half of the layers are biased horizontally and half vertically, in an alternating pattern, adjacent to one another.
- *Pre-routed copper*. The program tries to conform to any pre-routed copper lines.
- *Bias selection*. If no pre-routes exist, layers are biased arbitrarily according to the following rules:

1. If there are an even number of layers, odd-numbered layers are horizontal and even-numbered layers are vertical.
2. If there are an odd number of layers, ACCEL PRO Route tests whether the PC-board's x-dimension is greater than or less than the y-dimension. If x is greater than y, odd-numbered layers are horizontal and even-numbered layers are vertical. If **x** is less than **y**, odd-numbered layers are vertical and even-numbered layers are horizontal.

The default layer directionality can be overridden as necessary by manually enabling either the Horizontal or Vertical option.

ACCEL PRO Route's automation with respect to layers is limited to selecting directionality, not the number of layers used. If you enable four signal layers, ACCEL PRO Route uses all four while routing.

There is no algorithm applied during the manufacturing passes to specifically move lines to reduce the number of layers back to something less than what you set. It is best to start the routing process with as few layers as possible, adding more layers only if the completion percentage is unacceptably low or the via count is unacceptably high.

Manual Bias Selection

To manually select the layer bias, enable desired signal layer(s), and the routing direction on each of those layers (**Vertical** or **Horizontal**).

Any of the layers may be horizontal or vertical, but the number of layers in either direction is typically balanced in pairs, e.g., two horizontal, two vertical; or three horizontal, three vertical. Long, narrow boards make up one exception. For these boards, more layers should be biased in the longer direction to enable more paths for routing, from one end of the board to the other.

Another example of when you should select layer direction manually is when you want to follow the direction of the wave solder flow. There are also certain other designs which would benefit from an uneven number of layers being run in either direction due to the type and placement of their components, but they are generally few and far between.

Type

These options are applied to a new layer (Add).   You cannot modify the Type of an existing layer.

Buttons

The **Move Up** and **Move Down** buttons allow you to arrange layers. You can arrange the layers in your design to reflect the actual layer order of your board. However, signal and plane layers must be between the Top and Bottom layers.

The **Add** button adds to the list whatever **Layer Name** you have specified, with the **Layer Number**, **Routing Bias,** and **Type** specified.

The **Modify** button allows you to change the routing bias on a signal layer and to change the net name of a plane layer. *To modify the routing bias*, select a signal layer, choose a routing bias option and click *Modify*. *To modify a plane layer net name*, select a plane layer, click *Modify* and enter the new name in the Plane Layer Net Name dialog.

You can rename a layer although you can't change the Type or layer number. You can also reassign a plane net name.

The **Delete** button deletes the highlighted layer. You can delete any user-defined layer that is empty. You cannot delete a predefined layer.

note:

When you delete a plane layer, the ratsnest does not automatically return. You need to re-optimize to get the ratsnest back. Also, when you delete a layer, components are processed and stripped of any items that existed on the deleted layer; this is done without warning.

When you delete a layer that a pad/via styles hole range is dependent on, an error message appears explaining the dependency and which style you should correct. You are not allowed to delete a layer until there are no styles that are dependent on it.

The **Current** button makes current any highlighted layer (The current layer indicated by an asterisk).

note:

The current layer cannot be disabled. Also, items on disabled layers dont appear and can't be edited.

The **Enable** and **Disable** buttons respectively enable or disable whatever layer that you have highlighted.

Enable All / **Disable All** enables or disables all existing layers regardless of what is highlighted.

Adding a Layer

When you add (create) a layer, you must give it a unique name, specify a layer number (one that is not already occupied), a routing bias, and a layer type. When you create a plane layer, you must give it a net name.

1. In the Options Layers dialog, type the new layer name in the **Layer Name** text box and specify the **Layer Number** to enter the Add mode. The **Add** button is grayed until you specify layer name and number.
2. If you are adding a signal layer, click on the **Routing Bias** you want to use (used only by the autorouter).
3. Click the layer **Type** you want.
4. Click **Add.** The new layer will be listed in the listbox, with the layer type (S, P, or N), E for enabled, and the layer number you specified.

   Once a layer has been created/added, you cannot change its layer number or type.

   Signal/plane layers are placed just above the Bottom layer. Non-signal layers are placed at the bottom of the list.
5. Use the **Move Up** and **Move Down** buttons to arrange the layers in the appropriate order.
6. If this is a plane layer, you are prompted for a net name.

ACCEL Tango PCB designs are restricted to a maximum of four user-defined layers and 15 total layers. Therefore, when using ACCEL Tango PCB, do not exceed this number when pasting components into your design.

Deleting a Layer

To delete or enable/disabl a layer, click the layer name in the list box (to highlight it) and click on the appropriate button at the right (e.g., Current, Delete, etc.).

You can delete any *added* layer (you cannot delete a predefined layer). Also, you cannot delete a layer that contains objects; you must first delete all items from the layer. To modify a layer, click the layer name, make the desired changes and click **Modify**.

Making a Layer Current

To make a layer current, select it from the **Current Layer** drop down list box.

The **Enable All** and **Disable All** buttons enable and disable all layers on the board. The Current layer cannot be disabled.

Options Current Line

Sets the current line width for the Place Line, Place Arc, Route Manual, and Route Interactive commands and for routed lines. Whatever line width was set in PCB will be in effect for ACCEL PRO Route unless you change the value here. A change made here is reflected in PCB commands. The Options Current Line setting doesn't affect placing of line segments for polygons, copper pours, keepouts, connections, etc.

When the dialog appears, the **Line Width** editbox with the current line width value. The current line width is highlighted in the listbox.

To choose a new line width value, select a new value from the listbox.

To add a new line width value, type the new value in the **Line Width** editbox and click **Add**. The new value is inserted in the listbox before the highlighted value. To add a new value and make it the current value in one step, type the new value in the **Line Width** edit box and click **OK**.

To delete a value, highlight it in the listbox and click **Delete**.

Overriding Default Units

The units displayed as default in Options Current Line are determined by what is set using the PCB Options Configure command. To switch units between millimeters and mils, use Options Configure. You can also override the default units by typing in an explicit suffix (e.g., **mm**) in the Options Current Line dialog. As noted previously, you can also override the line defaults on a net-by-net basis with the Net Attrs command.

To change the line width (thickness) of existing lines and arcs (routed lines and arcs as well), use the Edit Modify command.

Options Current Keepout

Keepout allows you to designate certain areas of you design as off limits to certain processes, e.g., routing.   What is specified here is used by the Place Keepout command.

Options Current Keepout allows you to set up the type (line or polygon) and layers (current or all) of keepout barriers you want to use with the Place Keepout command.   **Line** or **Polygon** are the choices for **Style**, and you can select either **Current** or **All** for **Layers**.   When you specify **All**, then you can place one object that applies to all signal and plane layers.   **Current** restricts keepouts to only certain layers (the current layer).

Options Design Rules

When you access the Design Rules dialog, it appears with the Global tab selected.

The Default tab shows global default clearance rules.

The clearance values of the layer you select appear in the **Pad to Pad**, **Pad to Line**, **Line to Line**, **Pad to Via**, **Line to Via**, and **Via to Via** boxes. If you have a variety of settings and you click on two layers that contain conflicting values, the box is blank. The value you enter in the box is applied to the selected layer(s) when you click the Update button.

Clearance Rules

When a clearance rule for a specific object is requested (e.g., DRC), the design rules category are searched in the following order of priority

Class to class rules.	(highest priority)
Net rules.	
Net class rules.	
Global rules.	(lowest priority)

Within each category, the clearance rules are searched in the following order of priority:

1. Object pair clearance rules (e.g., **Pad to Line** clearance).
2. Clearance rules.

The order of evaluation matches the order of evaluation used by the CCT SPECCTRA Router. ACCEL PRO Route uses only global clearance rules.

ACCEL DRC ignores CCT SPECCTRA Router clearance rules that have been added as attributes.

Net Class Tab

The Net Class tab allows you to specify clearance rules for pads, vias, lines, and object pairs (such as pad to pad or line to via). The dialog lists all net classes and the rules associated with each.

To edit a rules Properties, double-click the rule in the **Rules** listbox.

To add, delete or edit rules, click the **Edit Rules** button to access the Attributes dialog. (Refer to the Edit Nets command section for details.)

Net Tab

The Net tab you to specify clearance rules for a specific net in the design. The dialog lists all nets and shows the rules associated with the net you select from the **Nets** listbox.

To edit a rules Properties, double-click the rule in the **Rules** listbox.

To add, delete or edit rules, click the **Edit Rules** button to access the Attributes dialog. (Refer to the Edit Nets command section for details.)

Class to Class Tab

The Class to Class tab allows you to specify class to class clearance rules.   The dialog lists all class to class definitions and the rules associated with each.

To create a Class to Class definition , select nets classes from the two **Net Class Name** listboxes, and click **Add Definition**.   Once you have created a definition, it can be deleted using the **Delete Definition** button and modified using the **Modify Definition** button.

To edit a rules Properties, double-click the rule in the **Rules** listbox.

To add, delete or edit rules, click the **Edit Rules** button to access the Attributes dialog.   (Refer to the Edit Nets command section for details.)

Clicking the Design Rules button in the Utils DRC dialog or the Route Autorouters dialog also brings up the Options Design Rules dialog.

Options Pad Style

Sets current pad style (a.k.a., *pad stack*) for the Place Pad command. You can add, delete, purge unused pad styles, and edit pad styles and set hole ranges using the series of dialogs available.

A pad style (or pad stack) is a collection of pad information concerning pad shape relative to layer, hole size, offset, etc. A pad style can be simple, or it can be complex.

Select Options Pad Style to display the Options Pad Style dialog.

You cannot modify or delete the Default pad style. When the Default pad style is selected, the **Modify** buttons become **View (Simple)** and **View (Complex)**.

To learn how to view the default pad style, click the following topic:

Viewing the Default Pad Style

To learn how to add a new pad style, click the following topic:

Adding a Pad Style

To learn how to modify a simple pad style, click the following topic:

Modifying a Simple Pad Style

To learn how to modify a complex pad style, click the following topic:

Modifying a Complex Pad Style

To learn how to delete a pad style, click the following topic:

Deleting a Pad Style

To learn how to purge unused pad styles, click the following topic:

Purging Unused Pad Styles

To learn how to rename a pad style, click the following topic:

Renaming a Pad Style

To learn how to set a pad style hole range, click the following topic:

Setting a Pad Style Hole Range

Purging Unused Pad Styles

1. Click *Purge Unused Styles*.

   All unused pad style definitions are deleted from the current PCB design.

Viewing the Default Pad Style

1. Select Default from the list of pad styles.
2. Click *View Simple* or *View Complex*. (View appears instead of Modify when Default is highlighted).

   The read-only View Default Pad Style dialog appears.

Simple Pad Style

A ***simple*** pad style must meet the following criteria:

- Shapes can be defined only on the built-in Top, Bottom, (Signal), (NonSignal), and (Plane) layers. No other custom layer definitions are allowed.
- The shapes and dimensions on the Top, Bottom, and (Signal) layers must either be identical (a uniform thru-pad), be defined on the Top layer only (a top surface pad), or be defined on the bottom layer only (a bottom surface pad).

To qualify as a simple, uniform thru-pad, the signal layer shape must be an ellipse, an oval, a rectangle, a rounded rectangle, a target, or a mounting hole. If a signal layer shape is an ellipse, oval, rectangle, or rounded rectangle, the (Plane) shape must be a Direct Connect or a 45-degree, 4-spoke Thermal.

To qualify as a simple, top surface pad, the shapes must have zero width and zero height on the Bottom, (Signal), (NonSignal), and (Plane) layers. The Top shape must be an ellipse, an oval, a rectangle, or a rounded rectangle.

To qualify as a simple, bottom surface pad, the shapes must have zero width and zero height on the Top, (Signal), (NonSignal), and (Plane) layers. The Bottom shape must be an ellipse, an oval, a rectangle, or a rounded rectangle.

- If there is a hole defined for the pad, the X and Y hole offsets must both be zero.

Complex Pad Style

Any pad style that does not meet *all* of the criteria to be a simple pad sytle is considered a ***complex*** pad style.

Adding a Pad Style

To add a pad style, click the **Copy** button.   The Copy Pad Style dialog appears.

Type the new pad name (e.g., `mypad`) and select a pad style to copy it from (e.g., *(`Default)`).   Click *OK* to return to the Options Pad Style dialog.

Modifying a Simple Pad Style

To modify a simple pad style, select a simple pad style and click the **Modify (Simple)** button. The Modify Pad Style (Simple) dialog appears. The **Modify (Simple)** button is grayed if you select a complex pad style.

Use this dialog to set the width, height, hole diameter and shape, select a pad type. For thru type pad styles, you can select either a thermal or direct plane connection. You can also set the plane swell value; either enable the **Use Global Swell** option (set in Options Configure), to use the global plane swell setting in Options Configure, or disable it and specify a **Local Swell** option, overriding the global Options Configure setting.

Modifying a Complex Pad Style

To modify a complex pad style, select a complex pad style and click the **Modify (Complex)** button. The Modify Pad Style (Complex) dialog appears.

The values represented will default to **mm** (millimeters) or **mil**, depending on what you have set in Options Configure (your current units). You can specify a measurement value (overriding Options Configure) by typing in **mil, mm, cm**, or **in** after the numeric value.

The **Pad Definition** area allows you to add a new pad definition to the Layers list, or to modify an existing pad definition for a layer.

The **Hole** and **Plane Swell** areas refer to all layers and are not affected by the Add, Modify, or Delete functions.

The **Add**, **Modify,** and **Delete** buttons in the Pad Definition area of the dialog work as follows:

1. *To add a pad definition* to the **Layers** listbox from the **Layer** combo box in the **Pad Definition** area: highlight a layer name in the Layer combo box, set the diameter, spoke width, shape, then click *Add***.** The new layer is added to the list.
2. *To modify a pad definition* for one of the layers in the **Layers** listbox: highlight a layer name in the listbox; then you can change pad shape (**Shape** options), and (in the case of thermal pads) define the dimensions for **Outer Dia** and **Inner Dia** and **Spoke Width;** then click *Modify*.

   You can view the pad definition of a layer by clicking on the layer name in the listbox. The pad definitions are listed by shape, width, and height in the **Pad Definition** area.
3. To delete a pad definition, highlight a layer in the listbox and click *Delete* (a reverse action to number 1 above). If the layer is predefined (e.g., Top), it cannot be deleted.

To learn about the some of the standard pad shapes and thermal spoke pads, click the following topic:

Standard Pad Shapes and Thermal Spoke Pads

To learn how to set the Plane Swell option, click the following topic:

Plane Swell Option

Standard Pad Shapes and Thermal Spoke Pads

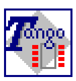
The following shapes are supported:

- **Ellipse** is a rounded shape with separately specifiable X and Y dimensions; an ellipse with equal X and Y dimensions is a circle (frequently called a *round*).
- **Oval** is a short line segment with round end caps (half-circles), the radius of which is 1/2 the length of the shortest side; if the X and Y dimensions are equal, this too is circular or round.
- **Rounded Rectangle** contains 1/4 circles on the corners of a rectangle. The 1/4 circle radius is 1/4 the length of the shortest side.
- **Rectangle** shapes are X=width and Y=height.

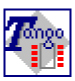
If you want a round pad, you use the ellipse shape with identical height and width (the example above is an oblong ellipse). If you want a square pad, use a rectangle with identical height and width.

Spoke width, and the inner and outer diameters, are specifications for *thermals*. There are four thermal pad styles, which appear as follows.

thermal spoke pads



thermal diameters and spoke width



The **Hole** and **Plane Swell** areas apply to all layers of the pad and are not part of the Add/Modify/Delete function for pad definitions. Specify the **Hole** dimensions and the X and Y offset. **X Off** and **Y Off** are horizontal and vertical offset amounts of the hole in an aperture.

You cant add a hole range for a pad style with a 0 inch hole diameter.

offset aperture hole

Plane Swell Option

With the Plane Swell options you can set the plane swell value; either enable the **Use Global Swell** option, to use the global plane swell setting in Options Configure, or disable it and specify a **Local Swell** option, overriding the global Options Configure setting.

Click *OK* and all of the changes or settings will be saved and applied.   All pads that currently use this style will be immediately updated.

Deleting a Pad Style

To delete a pad style:

1. Select a non-Default pad style name in the listbox.
2. Click *Delete*.

   The pad style name disappears from the list.

note:
You cannot delete the default pad style, and you can only delete a style if it is not currently used by a pad.

Renaming a Pad Style

1. Click the **Rename** button.

   The Rename Pad Style dialog appears:
2. Type a new pad style name in the **New style name** listbox.
3. Click *OK*.

Setting a Pad Style Hole Range

To set a pad style hole range, click *Modify Hole Range*. The Options Modify Pad Hole Range dialog appears.

This dialog allows you to set up the hole ranges for pads. Only those styles with a hole diameter is greater than zero appear.

The Hole Range Layers list box reflects the layer ordering of your board as you established in the Options Layers dialog.

To set a hole range:

1. Select a pad from the **Styles** list.
2. Select the begining and ending layers for the hole range.
3. Select another pad and repeat the process.
4. Upon completion, click *OK*.

Options Via Style

Sets current via style for the Place Via command.   You can add, delete, rename or edit via styles by using the series of available dialog.

Select Options Via Style to display the dialog.   Note that Quick Route allows only simple pad styples.

note:

Via styles are almost identical to pad styles in the way that you add, edit, modify, view, delete, and rename them in PCB.   Refer to Options Pad Style for all the details about via styles.

The only difference between pads and vias is that vias don't support Mounting Hole and Target shapes.

See also:

Options Pad Style

Options Text Style

Sets current text style for the Place Text command and allows you to add, delete, rename, or edit text styles by using the series of available dialogs. The text styles you create or edit here are available when you enable the Place Text command, or when you want to modify already placed text with the Edit Properties command.

The Default style can't be deleted, renamed, or modified. The other three default styles can be modified, but not deleted or renamed.

Choose Options Text Style to display the Options Text Style dialog.

Adding Text Style

To add a text style, click **Add** and the Add Text Style dialog appears.

1. *S*elect the style of text you want to base the new style on.
2. Specify the text style name you are adding (e.g., `busstyle`).
3. Click **OK** and the Text Style Properties dialog appears.

Text Style Properties

Remember, you cannot change the properties of Default text style; you can only query it.

To query and edit text style properties:

1. Highlight the text style.
2. Click **Properties** and the Text Style Properties dialog appears (shown below).
3. From this dialog, you can modify the **Height, Thickness,** and **Font** fields for non-default fonts

Renaming Text Style

To rename a text style:

1. Highlight the *non-default* text style you want to rename.
2. Click Rename. The Rename Style dialog appears.
3. The new text style appears in the listbox.
4. Specify the new name by typing over the existing name.
5. Click **OK**.

Deleting Text Style

To delete a text style:

1. Highlight the *non-default* text style you want to delete.
2. Click **Delete**.

   A message appears asking you to confirm your deletion.

3. Click **Yes,** and the highlighted text style disappears from the list.

note:
You cannot delete default text styles or a text style that is currently in use.

Text Style Properties Dialog

The Text Style Properties dialog appears when you click the **OK** button in the Options Add Text Style dialog or the **Properties** button in the Options Text Style dialog.

This dialog lets you add and modify font properties of the selected (non-default) text style.

The values represented appear as **mm** (millimeters) or **mil**, depending on what you have set in Options Configure (your current units). You can specify a measurement value (overriding Options Configure) by typing in **mil, mm, cm** or **in** after the numeric value.

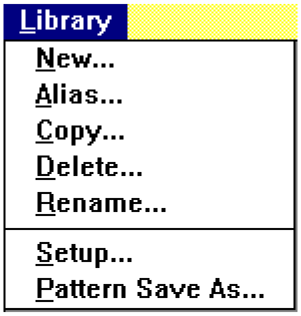
You can set the following stroke font properties.

- **Height:** The fonts height.
- **Thickness:** The text thickness..
- **Font:** Choose between **QUALITY**, **BASIC**, or **LCOM** fonts. QUALITY and BASIC fonts are interchangeable. Basic is simpler and therefore draws faster. LCOM is a serif font (a little fancier).

Font Examples

When you click **OK**, the new (or modified) text style is available for assignment.

Library Commands



Library New

Allows you to create a new library. The new library is empty; it has no components, patterns, or symbols.

When you choose Library New, the Library New dialog is displayed, which is a Windows common dialog (File New). In the dialog, you can specify the filename of your new library.

Library Alias

An alias is an alternate name for an item (component or pattern). You can create multiple equivalent names for the same item with this command.

When you create aliases for an item, it is not the same as creating copies or renaming. For copying or renaming, see the respective Library commands.

Aliases allow you flexibility in using a variety of naming conventions for components or patterns, without renaming them. For example, what PCB calls an SN7400N, you may want to use a generic alias of 7400. Or, if you are using components from a vendor using a particular naming convention, and you want to continue using that system, you can use alias names and display them on your design as such.

The library that you use in the execution of Library Alias, Library Delete, or Library Rename will remain current if you re-invoke any of the commands during the same PCB session.

Create an Alias

1. Choose Library Alias to display the Library Alias dialog.
2. If the appropriate library is not current, then click the Library... button and the Library Select dialog is displayed.

   The Library Select dialog is a Windows common dialog (File Open).

   Set up the library you want, click **OK**, and the Alias dialog is redisplayed.
3. Specify the **Alias Item** (**Component** or **Pattern** radio button). The **Symbol** radio button is grayed in PCB.
4. Click the item button (e.g., **Pattern...** or **Component...**) to display the Library Browse dialog. Highlight the item you want from the list and click **OK**.
5. You have now returned to the first dialog. Enter the new alias for the item in the **New Alias** edit box, and click Add to append it to the Aliases list. Click **Close** when finished.

note:

If you click **Close** before **Add**, then the dialog will close without completing the add alias action.

Library Copy

Copies an item from one file to another (either in the same or in a different library). If you want to copy a component with its associated Schematic symbols, use the Library Copy command in the Library Manager. PCB cannot mange symbol data.

It's important to note that a library part consists of a component section (type, reference designator, etc.), a pattern section (the PCB graphics), and a symbol section (the Schematic graphics). Generally, you will need to copy the component and its pattern when copying between different libraries (notice that the **Copy Item** section of the dialog has a choice between **Pattern** and **Component**). When you copy a component, you will be prompted as to whether you want to include its associated pattern; you would normally respond **Yes**. When you copy a pattern, no components will be included in the copy.

The dialog allows you to select source library and item name as well as destination library and item name.

The source library and destination library that are used with Rename will remain current if you re-invoke the command during the same PCB session.

If you are copying a component or pattern but are not changing its name, you can leave the **Destination Name** box blank.

Copy a Pattern or Component

1. Select the Library Copy command to open the dialog.
2. Select the **Copy Item** (Component or Pattern radio button) for your copy action. The **Symbol** radio button is grayed in PCB. If you are copying a component, make sure you also copy its pattern (if desirable) when copying from one library to another.
3. Click the **Source Library** button. The Library Select dialog is displayed, from which you can select the source library of your copy action.

   The Library Select dialog is a Windows common File Open dialog.

   After you choose the library from Library Select, the source library name is displayed in the Library Copy dialog.
4. Click the item button (**Pattern** or **Component**) and a list of items in the library are displayed in the Library Browse dialog. Select the item you want to copy, and the item name will be displayed in the Library Copy dialog.
5. Click the **Destination Library** button. The Library Select dialog is displayed, from which you can select the destination library of your copy action. The library name is displayed in the Library Copy dialog.
6. Specify the **Destination Name** for the new item. If you want to use the same name, you can leave the textbox blank.
7. Click the **Copy** button and all the boxes (except the Source library path and the Destination library path) become blank. This way you can continue to copy items between the same source and destination libraries.
8. Press **Close** to exit the dialog.

note:
If you click **Close** before **Copy**, then the box will close without completing the copy action.

Library Delete

Deletes a library item.

This command deletes the item in name only, if it has aliases. The alternate names (aliases) still exist unless you delete them. If the item has only one name and you delete it, then the item itself is deleted from the library.

The library that you use in the execution of Library Alias, Library Delete, or Library Rename will remain current if you re-invoke any of the commands during the same PCB session.

warning:

If you delete a pattern, then all of the components in the library that reference that pattern will be without structure, and would therefore be unplaceable. Normally you would want to delete a pattern alias only, which is not dangerous.

To Delete from a Library

1. Choose Library Delete to display the dialog.
2. Select the **Delete Item** type (Component or Pattern radio button) for your delete action. The **Symbol** radio button is grayed in PCB.
3. Click the **Library** button. The Library Select dialog is displayed, where you can select the library from which you want to delete an item.

   The Library Select dialog is the standard Windows File Open dialog in both form and function.

   The library you selected in Library Select is displayed in the Library Delete dialog.
4. Click the item button (**Patterns...** or **Components...**) and the items within the displayed library will be listed in the Library Browse dialog. Select one and it will then be listed in the Library Delete dialog.
5. Click the **Delete** button and the item box becomes blank. This way you can continue to delete items from the same library.
6. Click **Close** to exit the dialog.

note:

If you click **Close** before **Delete**, then the box will close without completing the delete action.

Library Rename

Renames a pattern or a component.

The library that you use in the execution of Library Alias, Library Delete, or Library Rename will remain current if you re-invoke any of the commands during the same PCB session.

warning:

If you rename a pattern, then all of the components in the library that reference that pattern by the original name will have no pattern reference, and will therefore be unplaceable. If you want to use a different naming convention for a pattern, then create an alias for the pattern (Library Alias command) and use that alias name. Likewise for components: if you want to use a different naming convention, using aliases is much safer than renaming.

To Rename a Library Item

1. Select the Library Rename command to display the dialog.
2. First select the **Rename Item** type (**Component** or **Pattern** radio button). The **Symbol** radio button is grayed in PCB.
3. Click the **Library** button to display the Library Select dialog, where you can choose the library to access.

   The Library Select dialog is Windows File Open common dialog. The library you select in Library Select is displayed in the Library Rename dialog.
4. Click the item button (**Pattern...** or **Component...**) and the items within the displayed library will be listed in the Library Browse dialog. Select one and it will then be listed in the Library Rename dialog.
5. In the **New Name** section type the new name of your item, then click **Rename**. Both the old name and new name disappear if the rename action is successful. Then you can continue renaming items in the same library.
6. Click **Close** to exit the dialog.

note:

If you click **Close** before **Rename**, then the box will close without completing the rename action.

Library Setup

Opens libraries from which you can access components.   The libraries that you open in Library Setup will then be available for the following functions:

The Place Component and Utils Load Netlist commands use the open library list to place components.

The Library Pattern Save As command saves patterns to the open library list.

When you want to place a component, the library file where the component resides must be open.

To Setup a Library

1. Run the Library Setup command to display the dialog.

   The dialog will list any libraries that are already open in the **Open Libraries** listbox.

2. To add another library to the list, click **Add** to display the Library File Listing dialog, which is the Windows File Open common dialog.   From there you can access the library directory to select a library file.

   When you select a file from the Library File Listing (and click **OK**), that filename appears in the Open Libraries listbox in the Library Setup dialog.

3. To remove a library from the list, select the library name from the **Open Libraries** listbox and click **Delete**.

4. Click **OK** and the libraries that you have specified are now open and accessible for component placement or saving patterns.

When you create a pattern or a component, you will need to have a library to save it to.   The list of open libraries is saved to the PCB.INI file and therefore saved for subsequent PCB sessions.

Library Pattern Save As

Saves a pattern to a library.

This command allows you to group a selected collection of objects as a pattern, and save the pattern to a library. From the library you can later attach the pattern to a component. You must have a library already open to save a pattern to it (use the Library Setup command to list and open component libraries).

To Save a Pattern

1. Use the block select function (Edit Select, draw a selecting rectangle) to include all objects you want to be included in your pattern.
2. While the objects are highlighted, choose Library Pattern Save to display the dialog.
3. Choose a library (highlight it) that you want to save the pattern to, then specify a pattern name. If you want to just create a pattern (not a component), click **OK**.
4. To automatically create a component that corresponds to the pattern, enable the **Create Component** option before you click **OK**.

   An auto-created component has pin designators which match the pin numbers, and the pin names are blank. Give the component a new name when in the Save Component As dialog; otherwise the component will be named the same as the pattern name.

note:

The **Create Component** option should be used only for components where pin numbers and pin designators can numerically match, as in DIP components. A CAP component does not have numerically matching pin designators and pin numbers, and therefore cannot be created correctly with this option.

If the box is left unchecked, the pattern is created without a component; to create a component from this pattern otherwise, you would use the Library Manager to build the component and attach this pattern to it.

Utils Commands

Utils Renumber

Renumber is a triple-function command for assigning pad numbers to free pads, and assigning reference designators to components manually or automatically.

For components, you can also specify designator templates (e.g., U) and increment values (e.g., 1) for reference designators (U1, U2, etc.).

Renumber Reference Designators

1. Select Renumber to display the Renumber dialog.
2. Enable **RefDes** and the rest of the choices in the dialog will be displayed accordingly (for Ref Des), such as the **Starting RefDes** (e.g., U1) and **Increment Value** (e.g., 1). You also have the option of **Manual** or **Auto**, **Top to Bottom** or **Left to Right**.
3. If you enable the **Manual** option, you are in a temporary mode of assigning numbers, so every left-button click on a component for **Manual** renumbering will increment the value next to the designator. Notice that the **Top to Bottom** and **Left to Right** options are unavailable.

   You can enter the starting value in the **Starting Ref Des** box. For example, if you entered the value U100 in the **Starting Ref Des** textbox, then specify the **Increment Value** as 10, the first component you click on would be designated U100, the next one U110, then U120, etc. If you started at U000 and incremented by 1, the first would be U000, the second U001, etc.

   Click the right-button or press Esc to end the temporary mode of Utils Renumber.
4. For the **Auto** option, you must have multiple components selected before you use this command (**Auto** is grayed if you don't). After you click **OK,** the reference designators will automatically increment according to your specification. Only the selected components will be renumbered. For example, you have selected three components positioned horizontally that are numbered U1 through U3, you renumber them with the **Increment Value** of 2, automatically (**Auto**) and enabled **Top to Bottom**. When you click **OK**, they will renumber U1, U3, and U5, in row order.

Renumber Pads

1. Select Renumber to display the Renumber dialog.
2. Enable **Pad Num**; the rest of the choices in the dialog will change accordingly. Then specify **Starting Number**, and **Increment Value**, as appropriate. **Manual** is the only choice for pad numbering; **Auto** is grayed out.
3. You are in a temporary mode of assigning numbers, so every left-button click on a pad will number a pad.

   For example, the first pad you click on would be number 1 (if **Start Value** was specified as 1), the second pad number 2 (if the **Increment Value** was specified as 1). As you click on a pad while in the Renumber mode, it highlights to show that a number has been assigned.

   The Status line information area displays the pad number every time you (re)number a pad.

   You can use the unwind feature to reverse the renumbering process. The backspace key will unwind the renumbering.
4. Click the right-button or press Esc to end the Renumber temporary mode.

note:

The renumber feature is sensitive to layers when renumbering pads. You must be on the appropriate layer for the pad to renumber it. For example, the current layer must be Top for a top

SMT pad.

Utils Force Update

This command replaces all components of a given type with the first component of that type in an **open** library. When you run the Utils Force Update command, the Utils Force Update dialog appears.

Use the Force Update command if you encounter a component cache error after using the Library Manager to change a component that is already placed in a PCB design. You must update all occurrences of that component in the design. This must be done before you can place any more instances of that component.

Select a component, or use the **Set All** button to select all components and press the **Update** button. The command looks in all open libraries in the order they are listed in the Library Setup list for the replacement components.

The **Maintain Rotation** option allows you to maintain of rotation of any rotated components. It does not maintain rotations for components in designs loaded from Tango Series II.

Updating a component copies all the components attributes to all of the patterns representing that component.

Utils Record ECOs

The Utils Record ECOs dialog allows you to record Engineering Change Orders (ECO). Select the **ECO Recorder** radio buttons to enable or disable the ECO recorder.

If there are pending ECOs, you are prompted when a design is saved on whether to append the pending ECO to the current ECO file.

This function can also be activated using the ECO icon on the Toolbar. When this function is activated (either from the Toolbar or the Record ECOs dialog), the Toolbar icon is depressed.

Types of ECOs

The following types of ECOs can be recorded:

- RefDes change (Was-Is).
- Net name changes.
- Additions, deletions, and modifications of components and parts.
- Additions and deletions of nets.
- Additions and deletions of net nodes.
- Additions, deletions, and modifications of attributes.

The format of the ECO file is determined by the setting in the Options Configure dialog. Full PCB ECO files have an .ECO extension, and Was/Is ECO files have a .WAS extension.

Utils Import ECOs

Imports an ECO file and applies the ECO changes to the current design file. The ECO file is created in PCB Schematic to capture schematic changes that impact your design. When you run this command the Utils Import dialog appears.

ECO Filename

1. Click the **ECO Filename** button to open the ECO Filename dialog.
2. The ECO Filename dialog is a standard File Open dialog. Type, or select from the list, the name of the file you want to open in the **File Name** box. Click **OK**.

   .ECO files are assumed to be full PCB format; .WAS files are assumed to be Was/Is format.

Preview ECOs

When you select an ECO filename and then click the **Preview ECOs** button, you can view ECOs, if there are any, before importing them. The ECOs are displayed in Windows Notepad.

Utils Export ECOs

Allows you to save ECOs to the ECO file at any time, without saving the design file. If there are pending ECOs, the Utils Export ECOs dialog appears when you run this command.

View Pending ECOs

When you click the **View Pending ECOs** button, you can view pending (outgoing) ECOs which are still stored in memory. The pending ECO data are written to a temporary ASCII file and displayed in Windows Notepad. The format displayed is either full or Was/Is, depending on the setting in the Options Configure dialog.

Save ECOs Now

1. To save pending ECOs, click the **Save ECOs Now** button. A warning message appears.

   It is recommended that you save ECOs at the same time you save your design using the File Save command. If you save ECOs without saving the design, your file and the ECOs may not match. That is, the ECOs might not reflect the current state of the design.

2. To continue, click **Yes**.

   The Save ECOs dialog appears.

3. The ECO filename appears at the top of the dialog. It is the last used ECO file. To change it, click the **ECO Filename** button and the ECO Filename dialog appears.

4. The ECO Filename dialog is a standard File Open dialog. Type, or select from the list, the name of the file you want to open in the **File Name** box. Click **OK** to return to the Save ECOs dialog.

   Full ECO files must have a .ECO extension, and Was/Is files must have a .WAS extension.

5. In the **Comments** box, type any comments that can help document the ECOs.

6. To append ECOs to the ECO file, click the **Append ECOs to File** button.

7. To discard ECOs, click the **Discard ECOs** button. Once discarded they cannot be recovered.

Utils DRC

Allows you to set up Design Rule Checking. This process makes sure that all electrical connections in the design layout match the connections in the netlist; the program also checks to see if minimum clearances have been maintained throughout the design.

DRC uses the precedence rule order to determine the proper minimum clearance rule between two design objects. (See Options Net Classes for details.) The DRC report includes a list of the net class clearance rules.

Select the Utils DRC command to display the following dialog.

To learn how to select a report file for saving DRC information, click the following topic:

Saving DRC Information

To learn about the various DRC report options, click the following topic:

DRC Report Options

To learn about the View Report and Annotate Errors options, click the following topic:

View Report and Annotate Errors

To learn how to specify DRC clearance options, click the following topic:

DRC Clearance Options

Saving DRC Information

Click the *Filename* button to display the Design Rule Check Report dialog (a Windows common File Save As dialog).   From this dialog you choose a DRC report file to which you can save the report information.

DRC Report Options

When enabled, these options will be included on the DRC report.

Netlist Compare compares a Tango format netlist file with the current nets in the design.

Netlist Violations enables electrical checking against the netlist within the design. Items are considered to be physically conntected if they overlap or have a clearance of 0 mil. Items that can be physically conntected to one another are arcs, polygons, pads, lines and vias. Copper Pour is not subject to Design Rule Checking because it has its own checking feature. Note that although text can be checked for clearance violations, it is not considered to carry current. If there are no nets in the design, this option is ignored.

Unrouted Nets enables reporting of any nets that are currently unrouted (unrouted connections still exist in the design). The report includes a warning for those objects that are not point to point routed. The warning includes the location of the objects

Clearance Violations enables air-gap clearance checking. If disabled, no clearance errors will be reported.

Text Violations enables air-gap clearance checking of text on the signal layers. If you have placed text on the signal layers, you should enable this option to make sure that the text does not short to other copper. When checking for clearance violations, the bounding rectangle of the text is used.

Silkscreen Violations enables checking on pad/via to silkscreen violations. Silkscreen on pads on top layer can interfere with soldering process; on vias it can cause paint dripping or collecting in unwanted areas.

Unconnected Pins enables the reporting of all pins that are not connected to other pins. This includes all of the single-node routes as well as pins that are not connected to anything at all.

Copper Pour Violations enables reporting of unflooded copper pour entities, copper pours that are not part of a net, copper pour island clearance violations, unconnected islands, and thermal connections that have clearance violations.

Only pour outlines are reported. Thus if there is via that shorts in the middle of a pour, it is not reported.

Drilling Violations enables connectivity checking of pads/vias through their attached layers, utilizing the layer ordering and hole range for determining where the pad/vias begin and end.

Also enables checking for layer separation, interference between holes, collocated holes, and vias existing solely on a signal layer.

Plane Violations enables reporting of overlapping planes, invalid pad and via copper connections, connected pad and vias that are not electrically connected to the plane, isolated areas of copper in a plane.

View Report and Annotate Errors

View Report

The **View Report** checkbox enables the screen display of the DRC report file when the DRC is complete.

Annotate Errors

The **Annotate Errors** checkbox enables the creation of DRC error indicators, which will be displayed on your design. These indicators can then be selected for viewing of error information. The error information is determined by the other error/violation options that you enable in design rule checking.

To view the error associated with an error indicator, select it and run Edit Modify. Their display is controlled by the Options Display (Misc) command; the selection criteria for DRC error indicators (for block selecting) is determined by the Options Block Selection command. Refer to the respective command sections for further information.

The DRC error indicators can be output to a report, which lists all of the locations and the errors that they are indicating.

When you are finished setting up the DRC options, click *OK* to begin the design rule checking process. Error indicators are cleared when you run Utils DRC.

DRC Clearance Options

You can specify the rules for the design check in the DRC Clearances dialog. Click the **Clearances** button to display the Options Clearance dialog.

The dialog lists the enabled layers of the currently loaded design file. The default clearances of 12 mil (.3 mm) are shown for each layer and between each item.

Set All will select (highlight) all layer names with items in the **Layers** listbox. **Clear All** deselects them all. You can select or deselect one or more layer names with items by clicking on them individually in the listbox.

The clearance values of whatever you select appear in the **Pad to Pad**, **Pad to Line**, **Line to Line**, **Pad to Via**, **Line to Via**, and **Via to Via** boxes. If you have a variety of settings and you click on two layers that contain conflicting values, the box will be blank. Whatever value you enter in the box will be applied to the selected layer(s) when you click the **Update** button. Nets with clearance attributes defined will override the global default clearance values for DRC. The report produced by the DRC includes clearances specified for specific nets.

Click **Set Defaults** to return all layer/item settings to 12 mil clearances.

Hole to Hole Clearance is a design-wide clearance setting (rather than layer specific for the other clearances). If you change it, the value will apply when you click *OK* .

Click *OK* when you have finished setting up the design rules (and return to the main DRC dialog). From the main dialog, click *OK* to begin the design rule checking process.

Utils Load Netlist

Loads a netlist from a .NET file into your design. If you have a board outline created, the components will be placed directly above the board outline. If there is no board outline, the components will be placed into the lower-left corner of the workspace.

Load a Netlist

1. Choose Load Netlist, a dialog is displayed.
2. To choose a netlist file, click the **Netlist Filename** button. This will display the Netlist File dialog, which is a Windows common File Open dialog.
3. Use the **Netlist Format** field and combo box to select the correct *source* file format for the netlist file so that the PCB file filter can read the format.

   Your choices are Tango format and ACCEL ASCII format. Tango format is the standard Tango format. ACCEL ASCII format includes attributes attached to each net and component in the netlist.
4. If you select ACCEL ASCII, four options for attribute handling are available: attributes may be merged with current design attributes, favoring either the netlist or the design attribute if the attribute exists in both; attributes may be ignored; or all design attributes may be removed and replaced with those present in the netlist.
5. When you select a file from the Netlist File dialog, you will return to the Netlist File Load dialog, where the filename will appear. Click *OK* and the netlist from the file will load onto your design.

Loading a Netlist on an Existing Board

You can load a netlist onto an existing board (which already has items), with the following rules and results:

- Components with matching RefDes must also have matching Types. If there is any conflict with the component section of the netlist, the load is aborted.
- Any existing components on the board that are not part of the netlist will be preserved.
- Any components in the netlist being loaded that are new to the design will be added.
- For components that include, pads and text that are of styles that have the same names but different data than those styles in the current design, the incoming style names will be bracketed to indicate the style conflict. The new (bracketed) style names will be added to the list of available styles in the current design. Refer to the Edit Modify, Options Pad Style, and Options Text Style commands for object style information.
- The net information and connections of the existing design will be replaced by the new net information.
- If you enable the **Reconnect Copper** checkbox, an analysis done by the program may find nets that are shorted. Shorted nets are listed in the error log file.
- When a new net is found to be open, a connection will be created. When extra connections, or misconnections, are found, a warning message is displayed.
- In the event that old copper and new nets match, no connection lines will appear because the nets are already routed.

Optimize Nets

The **Optimize Nets** checkbox can be used to inhibit the automatic optimization of the ratsnest connections. If optimization is disabled, the connections are made in the order given in the netlist. Disabling this option also speeds up the netlist load process since the time-consuming optimization phase is skipped. Once the netlist is loaded, you can run the Utils Optimize Nets command at any time.

Reconnect Copper

The **Reconnect Copper** checkbox can be used to inhibit the automatic "reconnect" of existing copper on the board. If reconnect is disabled, the net connections match the netlist exactly, and all existing copper on the board is converted to "free" (non-intelligent) copper. Free copper isn't associated with any net. If there is existing copper on the board, and the reconnect option is enabled, the reconnect phase can take a long time. Therefore, we recommend that you only disable the reconnect option if you want to quickly load a netlist into a board with existing copper, and you are willing to accept having free copper.

When the **Reconnect Copper** checkbox is enabled, Utils Load Netlist generates PCB connections between those objects that are physically connected but not point-to-point routed.

warning:

> If you disable reconnect, it is possible that existing free copper on the board will short one or more nets together. This can only be detected by running the Utils DRC command.

After you have loaded the netlist, you can select the components and connections either as a group (a block select) or individually (single select) to move them into their appropriate positions in the design.

Check for Copper Sharing

If the **Check for Copper Sharing** option is enabled, line to line trace intersections and line traces crossing the center of a pad or via are detected. Intersecting line traces are split at the point of intersection and lines are split at pad or via centers, creating point-to-point routes.

Utils Generate Netlist

Creates a netlist from the combination of connections you have established (or loaded) in a design file.

Free copper (non-net connections) will not generate a netlist. Free pads which are a valid part of a net are not listed as net nodes in the netlist.

Generate a Netlist

1. With your design file open, choose the Generate Netlist command to open the dialog.
2. To choose a netlist file, click the **Netlist Filename** button. This will display the Netlist File dialog, which is a standard Windows File Open dialog.
3. If you want to generate your netlist to a different format, use the **Netlist Format** textbox and combo box to specify the *destination* format.

   Your choices are Tango format and ACCEL ASCII format. Tango format is the standard Tango format. ACCEL ASCII format includes attributes attached to each net and component in the netlist.
4. Enable the **Include Library Information** checkbox if you want an optional library section to be written to the netlist.

   Library information is read by PCB but not processed; it is merely informational.
5. Click *OK* in the Netlist Generate dialog and the netlist will be created with the filename and netlist format you have specified.

Utils Compare Netlist

Allows you to compare the current nets of the design you have in memory with the netlist file that you specify. This useful feature verifies the integrity of your design against the original netlist. For example, if you mistakenly deleted a component or net, this command will show you the discrepancy. This command compares logical net information, i.e., it works even if there are open (unrouted) nets. Use the Utils DRC command to check existing copper.

Compare a Netlist

1. Choose Utils Compare Netlist to display the dialog.
2. Click the **Netlist Filename** button to display the Netlist File dialog (a File Open common dialog). From here you can access the netlist filename of the original netlist which you loaded into your design file.

   Also choose a **Netlist Format** which matches the format of the netlist file.
3. Click **OK** and any discrepancies will be displayed on the screen.

Utils Optimize Nets

Pin and gate swapping allows you to re-arrange connections logically in order to produce shortened or more direct routings, or dispersed routings to ease congestion in certain areas. This function is performed using the Utils Optimize Nets command.

Generally, you apply a gate or pin swap on logical connections created as a result of placing components in PCB. You can convey gate and pin swaps, which result in net list modifications, back to PCB Schematic from PCB with the ECO utility. (See the Utils ECO command sections in the PCB reference manual.)

Rules for Pin Swapping

The rules for pin swapping are:

1. Pin swapping occurs within a gate between logically equivalent pins. The equivalence values must be non-zero and identical for a swap to occur between two pins. Swapping is allowed between non-equivalent pins, but only after you give confirmation (a forced swap). Forced swaps can occur only during a manual pin swap; they do not occur during automatic pin swapping.
2. No swapping occurs if net copper is connected to either pad.
3. Swapping does not occur if a net connected to the pin has the **OPTIMIZE** attribute set with a value of "NO".
4. Swapping does not occur if the component has the **NOSWAP** attribute set to Yes.

Rules for Gate Swapping

The rules for gate swapping are:

1. The gates must be logically equivalent.
2. The gates must be of the same component type with equivalent value if swapping across components. The component values must be the same so that gate swapping with discrete parts works correctly. (For example, a gate swap between two RES components, one with a value of 50 ohm and the other 100 ohm, isn't allowed).
3. The swapping of gates between components can be further restricted by use of the swap eligibility attribute (**SWAPEQUIVALENCE**). For example, one circuit might be used to monitor another circuit on the board. It would defeat the purpose of the monitor circuit to have its gates swapped with the gates in the other circuit. By setting the components in the monitor circuit to have a swap eligibility attribute different than the other circuit, swapping can be kept from occurring between the two circuits.
4. Swapping does not occur if the component has the **NOSWAP** attribute set to Yes.
5. Swapping does not occur if net copper is connected to either of the gates.
6. Swapping does not occur if any net connected to the gate has the **OPTIMIZE** attribute with a value of "NO".

Utils Optimize Nets Command

The Optimize Nets command allows you to select **automatic** or **manual** pin and gate swapping. **To run this command, you must enable the Select tool.**

Run the Utils Optimize Nets command to display the Utils Optimize Nets dialog.

The dialog defaults to **Auto**, with the **Gate Swap** and **Pin Swap** options disabled. To perform only a minimization of the nets connection, click ***OK***.

When you click ***OK***, connections are rearranged to shorten the total connection length. If **Gate Swap** or **Pin Swap** are checked the gate and pin swaps occur in accordance with the rules found later in this section.

Automatic Pin and Gate Swapping

1. Be sure that that **Auto** radio button in the **Method** box is selected.
2. For gate swapping, click the **Gate Swap** box.
3. For pin swapping, click the **Pin Swap** box.
4. To perform gate and pin swapping in the entire design, click the **Entire Design** radio button.
5. To perform gate and pin swapping on selected objects, click the **Selected Objects** radio button. This button is grayed out unless you have selected objects before running the Utils Optimize Nets command.
6. Click ***OK***.

   The Not-Undoable Operation Warning message appears. If you press No, you are returned to the Optimize Net Dialog. If you press **Yes**, the Optimize Nets Progress dialog appears.

   The Optimize Nets Progress dialog provides you with the current and cumulative status of the command. You can stop the command at any time without losing the accumulated results.

   The **Current Status** box describes the current activity. The current activity can show swapping gates or pins, or minimizing connection length. Any swaps that occur follow the swapping rules.

Entire Design/Selected Objects

If you selected the **Entire Design** radio button on the Utils Optimize Nets dialog, the following actions occur:

- If you selected **Gate Swap** or **Pin Swap**, pin or gate swapping occurs between any components in the design following the swapping rules outlined in the section entitled "When components and net connections are selected" below. All nets are guaranteed to not have their connection lengths increase.
- Minimum length optimization occurs for all nets.

If you selected the **Selected Objects** radio button, the following actions occur:

When only components are selected:

- If you selected **Gate Swap** or **Pin Swap**, swapping is restricted to pins and gates among the selected components. Swapping occurs only to improve (decrease the manhattan connection length of) those nets with connections to the selected components' pads. Other nets' connections may increase in length.
- Minimum length net optimization occurs on the connections between the selected components.

When only net connections are selected:

- If you selected **Gate Swap** or **Pin Swap**, swapping occurs only to improve (decrease the manhattan connection length) the selected connections' nets. This means that all gates and pins are eligible for swapping, as long as a selected net improves from the swap. Un-selected connections may increase in length.
- Minimum length net optimization occurs on the selected connections.

When components and net connections are selected:

- If you selected **Gate Swap** or **Pin Swap**, swapping is restricted to pins and gates among the selected components. Swapping occurs only to improve (decrease the manhattanconnection length) the selected connections' nets. Those un-selected connections may increase in length.
- Minimum length net optimization occurs on the selected connections.

Manual Gate Swap

You can use the **Manual Gate Swap** options to perform gate swapping manually on selected pads. To perform manual gate swapping,

1. Check **Manual Gate Swap** button on the Utils Optimize Nets dialog.
2. Click the **OK** button. The Optimize Nets dialog disappears, and the cursor changes to the "more points" shape. If you click the right mouse button or press the **ESC** key, the command is aborted and you are returned to the **Select Tool**. The cursor shape returns to the original shape.
3. Click on a pad and all pads of that gate, along with the connections that are attached to those pads, appear highlighted in highlight color one. If you click the right mouse button or press the **ESC** key, the command restarts at the initial state, with all of the pads and their connections being unhighlighted.

   All the pads of the **eligible equivalent gates** (as defined by the gate swapping rules), and their connections, appear highlighted in highlight color two .

note:
The current layer must be a layer that the pad is defined on for the pad to be selected. Otherwise you hear a beep.

---

4. Click a pad. The pad and its connections appear in the first highlight color. If the selected pad is not an equivalent pad, a warning message appears stating: "The selected pad is not recommended for swapping, proceed with caution." This action constitutes a forced swap. The Swap Pins dialog now appears.

   The dialog presents you with a list of the affected nets and their new connection lengths. The manhattan connection length is calculated by determining the manhattan length after an optimization. The percent changed is the percentage of the difference of the connection lengths before and after swapping. Below the list box the total for all of the nets appears.

   If net copper is attached to any pads of the two selected gates, then the **Swap** button is grayed and a warning appears in the dialog. Nets that have copper attached are preceded by an asterisk. The warning tells you that net copper is attached to a gate's pad, and it must be removed before swapping can occur. Free copper is ignored.
5. To perform the gate swap, click **Swap**.

   All of the nets attached to the first gate areswapped with all the nets attached to the second gate. The dialog disappears and the cursor returns to its original shape, ready for the next gate swap. All of the highlighted pads and connections are unhighlighted.
6. Click **Cancel** to cancel the operation.

   The dialog disappears, no swap occurs, and the pads of the second gate, with their connections, revert to highlight color two. You can now select another gate for swapping.

Manual Pin Swap

You can use the **Manual Pin Swap** options to perform pin swapping manually on selected pads. To perform manual pin swapping,

1. Check **Manual Pin Swap** button on the Utils Optimize Nets dialog.
2. Click the ***OK*** button. The Optimize Nets dialog disappears, and the cursor changes to the "more points" shape. If you click the right mouse button or press the ***Esc*** key, the command is aborted and you arereturned to the **Select Tool**. The cursor shape returns to the original shape.
3. Click a pad. That pad, along with the connections that are attached to that pad, appear highlighted in highlight color one. All of the **eligible equivalent pads** (as defined by the pin swapping rules), and their connections, appear highlighteded in highlight color two . If you click the right mouse button or press the **ESC** key, the command restarts at the initial state, with all of the pads and their connections being unhighlighted.

note:
The current layer must be a layer that the pad is defined on for the pad to be selected. Otherwise you hear a beep.

---

4. Click a pad. The pad and its connections appear in highlight color one. If the selected pad is not an equivalent pad, a warning message appears stating the specific discrepancy, and asks you to confirm the selection. This action constitutes a **forced swap**. The Swap Pins dialog now appears.

   The dialog presents you with a list of the affected nets and their new connection lengths. The manhattan connection length is calculated by determining the manhattan length after an optimization. The percent changed is the percentage of the difference of the connection lengths before and after swapping. Below the list box the total for all of the nets appears.

   If net copper is attached to either of the two selected pads, then the **Swap** button is grayed and a warning appears in the dialog. Nets that have copper attached are preceded by an asterisk. The warning tells you that net copper is attached to one of the pads, and it must be removed before swapping can occur. Free copper is ignored.
5. To perform the pin swap, click **Swap**.

   All of the net connections attached to the first pad swap with all the net connections attached to the second pad. The dialog disappears, and the cursor returns to its original shape, ready for the next pin swap. All of the highlighted pads and connections are unhighlighted.
6. Click **Cancel** to cancel the operation.

   The dialog disappears, no swap occurs, and the second pad with its connections, in highlight color one, reverts to highlight color two. You can now select another pad for swapping.

Impact on the PCB Library Mgr

Gate Equivalency maintenance: Currently, the Library Manager allows you to set the gate equivalency of pins in the same gate to be different. This causes gates to be non-equivalent by ambiguity. To correct this problem, when you change a part number, or gate equivalence for some gate, the spreadsheet updates the gate equivalence field of the other pins of that gate to match. Existing components with this problem can be read in but cannot be saved.

Pin Equivalence: Pin equivalence is allowed only between pins with the same electrical type. The electrical types Unknown and Passive are considered the same for purposes of pin equivalence.

See also:

For more details about modifying attributes, refer to the Place Attribute and Edit Modify commands.

Utils ACCEL Schematic

If ACCEL Schematic is installed on your computer, this command runs ACCEL Schematic. If ACCEL Schematic is not running, it is launched. If Schematic is already running, it becomes the active application. Refer to your ACCEL documentation for additional information.

Utils ACCEL Lib Mgr

If ACCEL Library Manager is installed on your computer, this command runs ACCEL Library Manager. If ACCEL Library Manager is not running, it is launched. If it is already running, it becomes the active application. Refer to your Library Manager documentation for additional information.

Macro Commands

Macro Record/Stop

Starts and/or stops a named macro recording. After you record a macro, you can assign it to a key or to a menu.

To Record a Macro

1. Choose Macro Record/Stop to display the dialog.
2. Enter the name of the macro you want to record and click **OK**. The dialog will close and whatever actions you take will be recorded to the named macro (e.g., bionic).
3. When you have finished the series of actions to be your macro, choose the Macro Record/Stop command again. A dialog is displayed where you have the choice of ending the macro recording altogether (click **Stop**), or stopping the macro in mid-recording to perform some other action (click **Pause**) that you want to be recorded.

   To resume a paused macro recording, choose the Macro Record/Stop command again, and the same dialog is displayed, except with a **Resume** button instead of Pause. You can then pick up where you left off and finish your macro.

   Two shortcuts to end recording a macro are to either click the **M** button below the workspace (its other uses are explained in detail below), or press the M key.

Temporary Default Macro Record (M Toggle Button)

The Macro toggle button (**M** button located on the Status line) allows you to create temporary (default) macros on the fly. Typically, you would use this default macro for a short time within a design process (e.g., repeatedly placing a combination of lines, duplicating the same lengths and angles).

Only one default macro can be available at a time; each time you record a temporary macro, it overwrites the previous one.

To Record a Default Macro:

1. Click the **M** button on the Status line to begin recording; it displays a red colored background when it is recording. Press the M key as an alternate way to begin recording.
2. Perform whatever actions you wish to temporarily record.
3. The stop recording, click the **M** button (or press the M key) again; the colored background disappears when it stops the temporary recording.
4. Press the E (execute) key to playback the temporary macro. The actions you recorded will repeat each time you press E.

   A default macro can be paused and resumed just like a named macro, but you must use the Macro Record command to do it (see previous instructions).

note:

The name of the temporary macro is DEFAULT, so don't create any macros by that name.

If you want to create a more permanent macro, then you need to name it and record it with Macro Record/Stop. You can rename DEFAULT with the Macro Rename command.

Macro Delete

Deletes a named macro.   The Macro Delete dialog is displayed when you choose this command.

Select the macro you want to delete from the **Macro Name** combo box, or type the name of the macro. Click **Delete** and the macro will disappear from the list.

Macro Rename

Allows you to rename any created macros, including DEFAULT.

When you rename DEFAULT, it is no longer a temporary macro, but a saved, named macro.

When you select this command, the Macro Rename dialog is displayed.

Renaming a Macro

Click on the **Old Name** combo box to select the name you want to rename.   Then enter the new name in the **New Name** textbox.   Click **Rename**, then click **Close**.

Macro Run

Plays back a named macro.   The Macro Run dialog is displayed when you choose this command.

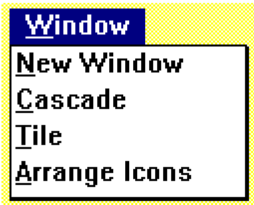
To run the macro, simply enter the macro name by typing it in or selecting it from the combo box, then click **OK**.

Run Temporary Macro

For running the temporary macro, use the E key.   To record a temporary macro, press the M key, do a series of actions, press the M key again to end recording.

This command is also handy for viewing your list of macros before you run one.   If you use a macro frequently, you can assign it to a key (Macro Assign To Key) as a shortcut.

Window Commands



Window New Window

Allows you to open additional windows for the active design. You can move independently in each window, making it easy to compare different parts of the same design.

A number identifying the window is added to the file name in the Title bar and at the bottom of the Window Menu.

Window Cascade

Arranges all open windows so that the window tiles are visible.

All windows overlap, starting in the upper-left corner of your Workspace.   You can see each window's title, making it easy to switch between windows.

Window Tile

Arranges all open windows so that all windows are visible.

Windows are resized and arranged side-by-side so that all windows are visible and none overlap.

Window Arrange Icons

Arranges design file icons (minimized windows) in the main application window.

Design files can be minimized into icons using the Control menu or by clicking the down arrow in the upper-left corner of the screen.   This command arranges these icons so that they are evenly spaced and don't overlap.

You can open one of these *design icons* by double clicking it or choosing Restore from the icons Control menu.

Libraries

This section provides important information about the structure and characteristics of PCB libraries, including:

- The relationship between components and patterns
- Using aliases instead of renaming
- Pad numbers vs. pin designators.

Components and Patterns

PCB libraries contain both components and patterns. A pattern is the basic graphical structure that is used in the creation and display of a component; a component contains the logical and electrical data. A pattern by itself contains no information except the shape of its graphic display. PCB cannot use either a component or pattern by itself, but needs a component/pattern combination. When a component is placed it references a pattern, which is attached to it for graphical structure. Numerous components can reference the same pattern. The component and the pattern it references must both reside in the same library for the component to be placed.

It is very likely that a library will have multiple components using the same pattern. This saves space and makes global edits of a pattern very efficient but also potentially dangerous, as it affects all components based on that pattern. The graphical structure (shape, size, number and shape of pads) for the different components will be the same, but the electrical information for each one is unique.

Clearly, renaming or deleting a pattern could have a profound impact on a library. If you change a DIP14 to a D14, for example, all components that reference the DIP14 pattern are now left hanging without a pattern. The pattern is referenced by name only, and only when the component is placed or copied.

The graphical pattern DIP14 only needs to be stored as one entity. When you place a component, the component in effect locates the named pattern structure and imports it into the design along with the component information.

You could have a PCB library containing 40 components and 25 patterns, and it would be complete as long as all the patterns that the components reference exist in the library.

Copying, Renaming, and Using Aliases

When you copy a component to another library, you generally need to copy its pattern too. The pattern copied with the component needs to be in the same library as the component, otherwise the pattern cannot be copied and the component cannot be placed.

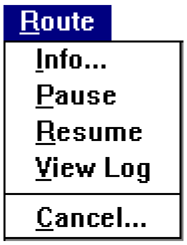
As mentioned previously, you can rename components and patterns, but this is an action that must be undertaken with care. If you prefer to use a different naming convention for your components and patterns, you can create aliases for them without having to alter the original name. Aliases are preferable for a number of reasons: it's a safe way to use a variety of naming conventions, there is no danger of making global mistakes (which is possible when you rename), and the flexibility of using aliases allows the component or pattern to be referenced by any of its aliases.

Pad Numbers vs. Pin Designators

An important fact to understand about components and patterns is that *pad numbers do not equal pin designator values*, except sometimes by a convenient coincidence. Pad numbers are just identifiers for pads, used to cross-reference with the pin designators when assigning the pin designator values. Using an ordered, linear, pad numbering system just makes identifying pad numbers easier, but you could just as easily number your pads randomly and still assign pin designator values to them.

In the component/pattern combination, patterns contain only pad numbers, whereas components contain pin designators and the association between pin designators and pad numbers. Each pin designator references one pad number. The spreadsheet interface is where the you can map or associate the numbers with the designators.

Route Commands (Route)



- Info
- Pause
- Resume
- View Log
- Cancel (PCB Route)
- Cancel (Quick Route)

Info

Displays the Information dialog box that shows up-to-the-minute statistics on your computer system, including available memory, disk size and disk space free.

The top line for each pass provides information for the pass: the pass name and number, the number of connections, fanouts or nets scheduled, deferred and completed, the percentage completed, routing time and vias added or removed. Note that the numbers of items scheduled, deferred and completed differ from pass to pass. For via fanouts, the number of items fanned out is given. During the constructive passes (Wide passes, Memory, Initial, Comprehensive and Exhaustive), connections are being referenced. Local Iterative and Manufacturing rip-up passes also refer to connections. However, global rip-ups for Iterative and Manufacturing pass types go through two phases. First nets are given, and later in the pass connections are referenced.

The second line contains overall routing statistics through completion of the pass. This includes the total connections routed, a total percentage of connections completed, overall routing time and the total number of vias.

Routing stops while you are obtaining this information. Remember to close the Information dialog after you have finished reviewing it.

Pause

Temporarily stops the routing at the point where the command is invoked. While paused, you may change your view, obtain routing information or on-line help, or cancel the route.

Computer resources are temporarily relinquished during pause allowing other memory-intensive applications to run.

Resume

Resumes a route that has been paused.   Routing starts at the point at which it was paused.

Cancel (ACCEL PRO Route)

The Route Cancel command provides options that let you terminate a route before it is completed and control certain aspects of the route.

The Cancel Route dialog presents these options:

Stop routing and save. Stops routing and saves an output PCB file with the name provided in the Route Autorouters dialog. Associated checkpoint files are deleted.

Stop routing and do not save. Stops routing and does not save an output PCB file. The input design file is restored to its original state. Associated checkpoint files are deleted.

Stop routing at the end of this pass. Stops routing at the end of the current pass and saves an output PCB file with the name provided in the Route Autorouters dialog.

Skip this pass and continue. Skips the current pass and continues routing at the next scheduled pass. The autorouter does not save a checkpoint file or an output PCB file.

Checkpoint route and continue. Creates a checkpoint file to save the current routing status and continues routing.

Suspend route. Saves a "snapshot" of the current routing session in the checkpoint file and terminates the route.

A checkpoint file can be restarted in the same pass you stopped the route, by selecting the Route Auto Restart command from the menu bar. (If you are part way through a pass, you will restart at the beginning of that pass.)

Cancel (Quick Route)

The Route Cancel command provides options that let you terminate a route before it is completed. The Cancel Route dialog presents these options:

Stop routing and save. Stops routing and saves an output PCB file with the name provided in the Route Autorouters dialog. Associated checkpoint files are deleted.

Stop routing and do not save. Stops routing and does not save an output PCB file. The input design file is restored to its original state. Associated checkpoint files are deleted.

Options Commands (Route)

Routing Fine Points

Pre-routed Connections

ACCEL PRO Route accepts any pre-routes whether they are added manually or created as a result of autorouting. ACCEL PRO Route recognizes pre-routed nets so that they are not routed a second time. Pre-routes can be ripped up if the Ripup option is enabled in the Options area of the Route Autorouters dialog. However, pre-routes on a disabled layer never are ripped up, even if ripup is allowed. The router has knowledge of items on disabled layers so that it can avoid placing vias inadvertently through existing geometry. It may choose to tap into connections on a disabled layer to complete a net in an optimal manner.

Additionally, you can have the autorouter fix individual nets (not ripped up), overriding the global ripup option by setting the RIPUP Net Attribute to No, 0 or False for those nets. This is a very powerful feature and is suggested for preserving pre-routed lines. It allows the autorouter to reconstruct any lines that were previously placed automatically, while leaving your manual placed routes alone.

If a free pad or via is pre-routed to a node in a net, the free pad or via is added to that net. This is useful for connecting surface-mount components and edge connectors to the Power and Ground planes.

If a pre-routed connection consists of several line segments, always try to place the segments (and vias) on the anticipated routing grid.

Lines and arcs with a width that is different from the line width for the net are not ripped up.

If an area needs to be filled with copper, use polygon fills instead of placing a number of crisscrossing line segments. Copper pours should be done after routing has been completed.

These practices reduce the amount of memory required to process the pre-routes and speed the processing of pre-routes.

Keepouts

A keepout defines an area of the board where ACCEL PRO Route and PCB Quick Route does not route. The selected autorouter applies keepouts as lines or polygons on the current layer or on all layers depending on what you specified using the Options Current Keepout command. A line segment, as opposed to a polygon fill, is really a "don't cross", so lines can both restrict routing from a section and confine routing within a border.

You can create a keepout on a layer by placing keepout lines or a keepout polygon fill on the current layer.

note:

You must be sure not to touch these keepout items with lines or pads that belong to a net, otherwise the router treats them as a part of the net.

One example illustrating the use of keepouts on all layers is to place lines around a pre-routed analog section of a board. This would prevent all digital lines placed by ACCEL PRO Route or Quick Route from entering the analog area.

You might want to use keepouts on signal layers to prevent the router from routing under the pads of an edge connector. To do this, create polygon keepouts beneath the edge connector on all mid-layers. Alternately, place line or polygon keepouts between the edge connector pads. This has the benefits of affecting all layers at once and preventing the router from placing lines on the Top and Bottom layers between or around the pads.

You can save keepouts with a component. For instance, to prevent the autorouter from using the space beneath a component, place a keepout polygon over the area to be restricted. Select the keepout with the other objects being saved with the pattern. Then use the PCB Library Patter Save As command to store the keepout with the pattern.

note:

Do not cover pad centers with keepouts, as this prevents the router from routing to or from the pad.

Off-grid Items

If a pad is not centered on the selected routing grid, the program routes to the nearest grid point and then adds a small line segment between the grid point and the center of the pad. *Off-grid pads and line segments tend to block subsequent routes more than on-grid items, and they may also lower the completion percentage.* Additionally, the router must maintain rows and columns of extra point locations. This increases memory requirements and reduces router performance. Although the router is able to work with off-grid items it is best to place your components on the intended routing grid or some multiple thereof.

The selected autorouter converts arcs to line segments. To prevent clearance violations, these segments are short. The side effect is that these segments are off-grid, and a problem similar to the one just described occurs. If possible, place arcs after autorouting has been completed.

Plane Connections

ACCEL PRO Route and PCB Quick Route derives its plane information from the layer definition. You can specify the nets to be connected to the planes when the layers are created. You can also modify the net name using the Current Layers dialog accessed from the Route Autorouters dialog or by using the Options Layers command in PCB.

Power and Ground nets set in Tango PCB or PCB Plus (Series II) boards are automatically carried forward to PCB when they are loaded.

If a pad belongs to the plane net but is not connected to it, ACCEL PRO Route or Quick Route may connect it by adding a line to a via or pad that is connected to the plane (i.e., perform via fanout on SMD).

As explained earlier in the section, a free pad or via that is pre-routed to a node in a net becomes a new node in that net. Pre-routes are taken into account during the processing of plane nets. A free pad or via that is pre-routed to a node in one of the plane nets is considered a fanout to the node and processed like any other node in that net.

Connecting Surface Pads to a Plane

On a printed circuit board, you can usually find two types of pads: *Through-hole pads* (which belong to all PC-board layers) are used for component packages with leads that pass through all board layers. *Surface pads* (which belong to either the Top or Bottom layer) are used for surface-mount components and edge connectors.

You can connect through-hole pads directly to any plane. Surface pads, however, require that you connect the pad to a plane by means of a free via or free pad. Note that vias may be ripped up, but pads are never touched.

For boards with more than two layers (when the auto pass selection is enabled), ACCEL PRO Route or Quick Route automatically connects the surface pads, including pads for SMDs and edge connectors, to a via and then to the plane providing that:

- The surface pads are part of a plane net.
- The surface pads are not already connected to a through hole pad.

Additionally, pads are automatically connected to planes for two-layer boards under special conditions such as when a top and bottom surface pad, attached to the same net, are co-located. This work is done in the Wide Via Fanout and Via Fanout passes. You may also choose to schedule these passes manually.

Direct Connections

PCB, TangoRPO Route, and Quick Route display an **X** in the center of a pad to indicate that the pin is connected to a plane.

The **X** lets you tell at a glance whether a pad is connected to a plane. But how can you tell to what plane the pad is connected? By running the Options Display command, you can assign separate colors to each plane. A good practice is to use red for a Power plane (red = hot = power) and green for a Ground plane (green often signifies ground connections). Alternately, select the via or free pad and the net name appears on the Status line.

note:

Plane vias added by the router are connected to the plane with a thermal relief or connected directly as defined by the via style assigned to that plane net. You can see the type of connection by displaying the plane layer.

Surface Pads

While ACCEL PRO Route and PCB Quick Route permit you to set the orientation of routes on any layer, a typical double-sided board is routed with horizontal lines on the Top layer and vertical lines on the Bottom layer. This convention makes it difficult to route a set of surface pads that are arranged horizontally on the Top layer or vertically on the Bottom layer.

For example, consider access to surface pads on the Top layer of an edge connector which is along the lower edge of the board. It is difficult to complete connections to the middle pads of the edge connector (all pads other than the first and last) since there is no easy access to these pads with horizontal lines. Setting the layer bias (to vertical in this case) may help.

To ensure high rates of completion in these cases for boards with more than two layers, ACCEL PRO Route and Quick Route automatically connect the surface pads to a via, providing that:

- The surface pads are part of a net.
- The surface pads are not already connected to a through hole pad.

The via or pad then becomes a potential target on any layer (with any bias) for routes to the surface pad. If you have a board with several mid-layers, select layer biases to favor the direction of your surface mount pads or connectors. Also, if you are manually selecting passes, run a fanout or wide fanout pass to create better access to your surface pads.

Improving Performance

Performance in an autorouter is defined as the minimum amount of time required to produce the desired result. The desired result, in turn, is a printed circuit board design which is complete, manufacturable, electronically sound and aesthetically pleasing.

The total time to complete the desired board includes any time-consuming, manual clean-up required after the autorouter completes its end of the task. Thus, completion percentages and the quality of the routes accomplished are a key component in performance.

ACCEL PRO Route is a very efficient and intelligent autorouter, and under most conditions it will produce better results with its default routing strategy than with a user-defined strategy. However, under certain conditions the program may encounter problems, including failure to reach 100% completion and slower-than-expected performance. This section contains information and tips about how to deal with these problems.

Routing Completion

Since there is no way to predict in advance what the results will be for a particular circuit design, the recommended procedure is to let ACCEL PRO Route start with the default routing strategy, in particular, let the autorouter define the grid and pass selections.

When to Abort the Default Strategy

Progress of the run should be monitored using the Route Info command and information provided on the Status line. The critical point comes at the end of the Constructive passes, *just before the start of the Iterative passes*. The decision to continue the run using the default strategy or to abort the run and change the strategy should be guided by the completion rate at that point. The general rules below provide guidelines to help make this decision.

Check the total board completion percentage just before the start of the Iterative passes.

- **Less than 90 percent.** The probability of achieving 100% completion is low. Abort the run and try another alternative.
- **90 to 92 percent.** The probability of achieving 100% completion is about 50/50. Either abort the run and try another alternative, or continue to monitor the router's progress. If little or no improvement is noted, abort the run and try the suggested alternatives.
- **Greater than 92 percent.** The probability of achieving 100% completion is very good but not assured. Continue with the default strategy to the end of the run.

If it becomes necessary to abort the default strategy, one or more of the following actions or factors can be changed in an attempt to improve the results.

- Change routing grid.
- Change via grid.
- Change line width or clearances.
- Change via size.
- Change pad sizes.
- Increase number or change the direction of layers.
- Adjust the component placement and make sure components are on grid.
- Disable diagonal routing until the Manufacturing passes (for SMDs).
- Add curved lines after autorouting has been completed.

View the Output Log File

To help you decide which changes to try, look at the output log file. ACCEL PRO Route generates a comprehensive report file at the end of the routing session, detailing the route strategy and results of the session. This file is named in the Route Autorouters dialog, and it may be viewed by selecting the Route View Log command during or after a route session.

If ACCEL PRO Route fails to achieve 100% completion, this report file can be of real use to you in determining how best to alter the design rules or fine-tune the routing strategy to achieve 100% routing completion.

Choices made by the router are indicated with an asterisk (*) in the first section of the output log file . Examine these to make sure that the values are reasonable. The second section provides per pass history. Compare completion percentages against prior runs to determine the best strategy.

If you examine the file during routing or cancel the routing process, the report file reflects the final board statistics only up to the point of termination. When you cancel the route, a cancel warning message appears in the report file.

The manufacturing passes typically produce diminishing returns. If you notice that results are satisfactory before all manufacturing passes are run, you may benefit by skipping the remaining passes and continuing to final manufacturing.

Once the board is routed to 100 percent, the router skips any remaining constructive or iterative passes. However, it runs all manufacturing passes, so you will want to select the **Manufacturing Pass Count** carefully. Three manufacturing passes are generally adequate.

Changing Design Parameters

On any board that gets poor completion rates, you should check the following items:

- Design rules.
- Line width.
- Via size.
- Pad size.
- Number of layers.
- Grid spacing.
- Component arrangement.
- Net attributes.
- Layer bias.

Check the report file to verify that each of the above listed items has the value you expect. In particular, if this is a fine line circuit, is there room to fit two (or more) lines between pins at the selected spacing? Is the via pad size the size you expect?

If all seems correct, consider reducing one of these items if your manufacturing techniques will permit. In particular, small reductions in the via size can often produce large gains if a line can be placed one grid point closer to a via. The advantages of permitting more lines between pins and greater line packing are obvious.

The advanced design of ACCEL PRO Route permits it to effectively handle a wide range of autorouting requirements using its default strategy. Under normal conditions, ACCEL PRO Route can compensate for occasional deviations from good design practices. However, there is a limit where ACCEL PRO Route can no longer deal with components that are randomly placed.

For the best results, components should be placed on the board in an orderly fashion. Where possible, components should be placed with their pad centers on a 100-mil grid, and if this is not possible, the pad centers should be placed on the intended routing grid. The other standard placement considerations of locating components to minimize line length and reduce congested areas will also improve ACCEL PRO Route's results.

A good strategy for locating congested areas is to run ACCEL PRO Route through the Constructive passes and look at the remaining open connections (the rat's nest). A display of the uncompleted connections at this point shows problem areas and may suggest improvements to component placement.

Grid Selection

The routing grid defines the spacing pattern of the grid points. The grid pattern may be uniform (i.e., equal spaces between all lines) or it may be non-uniform to allow one or more lines to pass between component pads, while keeping the number of grid points to a minimum.

- *Uniform grids* are generally used for relatively simple, low-density PC-boards and often for SMD boards.
- *Non-uniform grids* are typically required on complex, high-density through-hole, and sometimes surface-mount, PC-boards to achieve 100% completion of the connections in the minimum amount of time.

ACCEL PRO Route's support of both uniform and non-uniform routing grids enables it to handle virtually any board density and any combination of routing grids. The main benefit of the non-uniform grid over the uniform grid is performance: it contains far fewer grid positions without the loss of route paths. Thus, completion time is shortened and more freedom in placement is allowed. ACCEL PRO Route easily handles both packages with different pitches on the same board and boards with mixed technologies (through-hole and SMD).

Selecting the Proper Grid

Grid selection is important for obtaining optimum router performance. The autorouter can intelligently make this selection for you, and we suggest you use the auto grid option first. However, since all of your special needs can't be anticipated, you have the option of setting grids manually.

With this flexibility comes the added responsibility of carefully planning the pad sizes, line widths and component placement grid so that you can get the most out of ACCEL PRO Route.

The first step is to determine your board manufacturer's capabilities and to verify the yields available with various line densities. This helps you determine the best design rules to use when placing components and pre-routing, and on the autorouted lines in ACCEL PRO Route.

Planning in advance is absolutely necessary. Determine your routing and placement grids early in the layout cycle; they should be known before placing components. *We cannot emphasize the importance of proper component placement prior to autorouting*. Changing grids during placement can result in many off-grid components and pads, thus lowering completion rates.

Always place components on a grid that is an integral multiple of the intended routing grid. For example, place components on a 25-, 50- or 100-mil grid, if you intend to autoroute with a 25-mil grid. In general, it is best to try to place your components on a 100-mil grid, as all of the routing grids are evenly divided into 100-mil. You should follow similar guidelines for metric boards.

If a pad is not centered on the routing grid, the program routes to the nearest grid point and then adds a small line segment between the grid point and the center of the pad. *Off-grid pads and line segments block subsequent routes more than on-grid items and may lower the completion percentage.*

If possible, place arcs on signal layers after autorouting. The autorouter converts arcs to line segments, and the ends of these lines often fall off-grid.

Gridded vs. Gridless Routers

ACCEL PRO Route offers the best of both gridded and gridless autorouters. ACCEL PRO Route is a gridded autorouter. But unlike typical gridded routers, however, ACCEL PRO Route's support for both uniform (50, 25, 20, 16.7, etc.) and non-uniform (40-20-40, 42-16-42, etc.) routing grids, combined with its ability to route off-grid enable it to handle virtually any combination of design rules and pad sizes.

Non-uniform gridded routing allows multiple lines to be routed between pads for higher completion rate while still utilizing the speed advantage of the grid. To achieve the same result with a uniform grid, a finer grid must be selected, thus making the process much slower and more memory intensive.

Once the grid is selected, it is set for the routing session. However, ACCEL PRO Route solves localized blockage problems by routing off-grid, returning the line to the grid pattern when the component and line placement allows.

Thus, the non-uniform-grid and off-grid capabilities of ACCEL PRO Route provide the same benefits that gridless routers offer, but require much less time to route a PCB. At the same time, the autorouter benefits from being grid-based (versus gridless) because gridded routers produce clean, attractive boards similar to those designed manually by experienced layout professionals.

Automated Grid Selection

When you enable the **Auto Grid Selection** option in the Route Autorouters dialog, the autorouter automatically selects the best routing grid based on the type and placement of components on the board, their pitch (spacing between pads), your selected line width(s) and design rule clearances. The **Grid** button in the Route Autorouters dialog is disabled.

ACCEL PRO Route begins by establishing a grid pattern over the routable area of the PC-board which is enforced during the entire routing session. The grid establishes pathways for routing.

The use of a grid is still the most efficient way to autoroute a circuit board. However, with ACCEL PRO Route there is no requirement that items on the your design must be on the grid. The program routes off the grid under two conditions:

- To reach a pad.
- To place a line between two pads.

ACCEL PRO Route automatically calculates the optimum routing grid using either of two sets of rules: through-hole pad rules or surface pad rules.

- *Through-hole pad rule*s: ACCEL PRO Route assumes that the through-hole pad-to-pad distance (e.g., 100 mils) is the usual spacing between pads. It then calculates how much space would be left between two pads with the most common through-hole pad size. It calculates the maximum number of lines that can be packed into this space, given the line width and clearances.
  If no lines fit between two pads, the routing grid is set to 50. If only one line fits between two pads, the routing grid is set to 25. If more than one line fits between two pads, ACCEL PRO Route defines a routing grid that accommodates the maximum of lines between them.
  ACCEL PRO Route attempts to space the channels between the pads so that the distance between these channels divides evenly into 100. If this is not possible, the channels between the pads are packed as tightly as the line widths and clearances will allow.
- *Surface Pad Rules*: ACCEL PRO Route uses a proprietary algorithm. If results are not as desired, manually select a grid that packs the maximum number of lines evenly into the normal spacing pads.

The decision to use one or the other of these rules is based on the ratio of surface pads to through-holes.

- If the surface pad through-hole ratio is equal to or greater than 2:1, surface pad rules are used.
- If the ratio is less than 2:1, through-hole pad rules are used.

To determine the ratio, ACCEL PRO Route scans the input design file to count the number of through-hole pads and surface pads.

When routing lines, ACCEL PRO Route follows your specifications for the allowable routing area, line widths, clearance tolerances and other design variables.

Because of the intelligence built into ACCEL PRO Route, we suggest that first-time or novice users stick with ACCEL PRO Route's experience and not try to design their own routing strategy. In fact, experienced users will find that in the large majority of cases, the default strategy of ACCEL PRO Route produces better results than if user strategies are employed.

However, if at the end of the Exhaustive pass, completion is less than 92%, the router may have difficulty completing the design with the current settings so you may want to try your hand at selecting your own grid.

Manual Grid Selection

You can select the routing grid (and via grid) manually. The grid you select is used throughout the entire routing session; it is not changed on a pass-by-pass basis.

With SMD designs, the via size is a more critical element than with through-hole designs. This is because there is generally a via placed by many SMT pads in the via fanout passes. These vias may later be eliminated during the manufacturing passes, but in the meantime they present an obstacle to routing. This is particularly true for SMD designs where the via is larger than the width of the average surface pad.

The various grid options are primarily intended to improve the completion rate during the constructive and iterative routing passes. However, you can take advantage of certain grid combinations to improve the results of the manufacturing passes as well.

Here are three examples: After routing the board with the 42-16-42 grid, use the 42-8-8-42 grid or the 17-16-9-8-8-9-16-17 grid during the manufacturing passes. After routing the board with the 40-20-40 grid, use the 40-10-10-40 grid during the manufacturing passes. This gives the reconstruct algorithm more grid positions to use while cleaning up lines, spreading tracks and reducing vias in the manufacturing passes.

Since the grid selected is for the entire routing session, you could try to re-initiate routing and select just the manufacturing pass to enable these special grids as described above.

Via Grid

If you manually select the routing grid, you may also select the via grid to be a multiple of the routing grid. If the routing grid spacing is non-uniform, the autorouter applies the via grid multiple to the sum of the spacings (e.g., if the route grid is 38-12-12-38, and the via multiple is 2, vias can be placed every 200 mils). The via grid is selected for you if you have enabled the **Auto Grid Selection** option.

The purpose of having a separate via grid from the routing grid is to allow for higher completion. Again, it is particularly important on SMD boards where the router does a fanout of vias from the SMD pads. By having a courser grid set for vias than for routes (a Via Grid Multiple greater than one), vias are placed further away from the SMD pads, and from each other. This prevents a wall of vias from forming, which would block subsequent routes. Coarser via grids may be required for manufacturability or testability reasons. Many bed-of-nails testers require vias to be on a 100-mil grid.

ACCEL PRO Route uses the selected via grid as a recommendation, but is not bound by this selection. A "costed via grid" is used by ACCEL PRO Route when placing vias. This means that placement of vias on the grid is encouraged, but is not mandatory. Placement of vias off of the via grid only occurs as a last resort to achieving 100% completion.

Memory Management

PCB is a powerful design tool that provides you with the ability to create large, complex designs. The single non-design factor, which may limit autorouting of these complex boards is the amount of available Random Access Memory (RAM).

Certain memory requirements are fixed. A minimum of 7.6 Mb is required to begin routing. This includes memory for DOS, Windows, PCB and the router itself. Grid data requires additional memory. The amount of this additional memory varies depending on board size, the number of signal layers and the selected routing grid.

You can use the following formula to compute the approximate additional memory requirements for ACCEL PRO Route on your board:

Memory Requirement =(2+3L)(XG)(YG)

Where:	L	= number of signal layers
	XG	= number of grids in X
	YG	= number of grids in Y

For Example:

If L=4 Layers

Grid= 40-20-40; (3 grids/100 mils, or 30 grids/inch)

Board Size= 12.3 inches by 8.2 inches

Then:

Add'l Memory Req'd	= (2+ 3*4)(30*12.3)(30*8.2)
	= (14)(369)(246)
	= 1,270,836 bytes

If the ACCEL PRO Route memory requirements exceed available physical memory, some solutions are to reduce the number of grid points (i.e., select a coarser or non-uniform grid), reduce the number of layers, or obtain more physical memory. Also try removing any TSRs, including Smartdrive. Generally, you will want to run Windows using Smartdrive. However, the effect of not using it during routing isn't significant. Thus removing Smartdrive to obtain sufficient memory is a viable solution.

Remember that the non-uniform grids such as 40-20-40 use far less memory than the 20-mil grid, and yet still achieve the same number of lines between pads. Use a non-uniform grid, where possible. Also, off-grid pre-routes and components and arcs on signal layers can significantly increase the amount of memory required.

ALLOC Option

The ALLOC option provides you with a way to limit the amount of memory available to ACCEL PRO Route. This may be useful to run other Windows applications at the same time. Use the Windows PIF Editor, and enter the following in the Optional Parameters field:

/ALLOC:<Kbytes>

This option limits the router to allocate only the amount of memory specified. Thus /ALLOC:8000 would allow the router to allocate only 8 megabytes of memory.

The value, in kilobytes, should be less than or equal to the total amount available. If the value exceeds the memory available to the program, the maximum amount is allocated.

Improved Completion Rates

Modifying the default grid strategy can be an effective way to improve the percent of completion on your board. Here are some suggestions for both SMD and through-hole designs.

SMD Boards

For SMD boards, there are two different approaches that may be tried to improve completion rates. These include using a finer grid and selecting a coarser via grid. Both of these alternatives are discussed below.

Try a Finer Grid

The default strategy for SMD boards is to select a grid primarily based on considerations involving vias. Most surface pads (except in memory arrays) require at least one via in order to be connected. On a fine grid, the vias may block more routing channels. Occasionally, however, better results will be achieved by selecting a grid equal to the line width plus the line-to-line clearance, permitting maximum packing of lines.

Therefore, you may want to re-route your board to see if a finer grid improves the completion rate. To experiment, set your grid and route through constructive passes.

When selecting a grid it is best to use a grid that fits exactly into the pitch of your through-hole components.

Select a Via Grid

Typical design rules do not allow a line to go between two vias on the pitch of the SMDs even though the vias themselves may fit on the pitch. For example, suppose the SMD pitch is 50 mils, the line width is 10 mils, the via size is 40 mils and all clearances are 10 mils. Under these conditions, two vias can be placed 50 mils apart, but no lines can go between these vias.

When the router does a fanout from the SMD pins there will be a tendency for walls of vias to build up next to the SMDs. If this seems to have occurred, use of a 100-mil via grid will force the vias on adjacent pins to opposite sides of the SMD. In this case, selecting a via grid multiple of two accomplishes this goal.

On boards with only two signal layers, a coarser via grid should probably be used for all passes. With designs having more than two signal layers, experiment with using a coarse via grid for just the fanout passes or for all passes except the Final Manufacturing passes.

Through-Hole Boards

Through-hole circuit designs require different remedies than SMDs to solve completion problems.

Through-Hole Fine Line

On fine line designs, ACCEL PRO Route typically selects a non-uniform grid that permits the maximum number of lines between two pins on 100-mil centers. This choice is primarily made for performance reasons since ACCEL PRO Route can be as much as three times faster on a 40-20-40 grid as on a uniform 20-mil grid and use considerably less memory. The negative side of this choice is that in open areas of the board, maximum line packing will not occur. Using a finer grid that still permits the same number of lines between pins usually increases completions.

One special situation needs to be noted. If the design rules are 8-mil lines and clearances with 60-mil pads, ACCEL PRO Route will choose a 42-16-42 grid for through hole designs. If you decide to try a finer grid you might be tempted to use a 16.7 grid. However, this grid only permits one line between IC pins. A better grid choice in this situation is the union of 42-16-42 and 16.7, i.e., 17-16-9-8-8-9-16-17. A choice that would not effect router performance would be to select a uniform 20-mil grid, with the pad-to-line clearance reduced. The technique of making a new grid that is a union of two others is equally feasible for metric boards.

It is recommended that the via grid multiple be set to 1 for passes other than the manufacturing passes if this strategy is attempted.

One Line Through-Hole

On one line through-hole designs it is difficult to improve on the results of the default passes and grids. Occasionally, if the design rules will permit, use of a finer grid to pack lines closer will gain connections. For example, if the line widths are 10-mil and the line-to-line clearance is 10-mil, you could try a 20-mil grid.

Completing Your Design

After the routing passes conclude and the output files are generated, you may still find uncompleted connections. Your options are to either:

- Modify any or all of the routing parameters or modify the component placement and try autorouting again.
- Finish the remaining connection by hand.

Both procedures are explained below.

Iterative Approach

The first step in finishing the board with the iterative approach is to examine the connections left unrouted by the autorouter. Your choices at this point are:

- To move components for better placement before re-routing.
- Enable signal or planes layers (using Options Layers) that were not previously enabled, then re-route.
- To immediately return to ACCEL PRO Route and try new design rules, different passes, more layers or finer grids.

When viewing the unrouted connections in PCB, look for areas of obvious congestion that can be fixed by moving components around the board. If you find such bottlenecks, move the components on the board, maintaining routed connections if possible without creating shorts, to relieve the congestion. Then, feed the board back into ACCEL PRO Route.

ACCEL PRO Route is fast enough that you can cycle through the following procedure several times before completing the board manually in PCB:

1. Autoroute the board only through the constructive passes (i.e., set Iterative and Manufacturing passes to zero if you are allowing the router to select passes or manually select just the constructive passes).
2. Move the components to minimize the density of unrouted connections and to move off-grid components onto the routing grid, or make other strategy changes discussed above.
3. If necessary, repeat steps 1 and 2. Once the board routes to more than 92% before the Iterative passes, continue to the next step.
4. In ACCEL PRO Route, route the board that was output from step 2 through the maximum number of Iterative passes (10) and choose any number of Manufacturing passes (up to 10). Do not select the Force Manufacturing Passes option if you do not want the router to clean up the board before it is 100% completed.
5. If necessary, continue to follow steps 2 and 4.

One final idea that you may want to try is to run your unfinished board through one or more Manufacturing passes, allowing the router to remove vias and line length before sending it through the Iterative passes again. This could actually make routing more difficult, but the technique has been known to work.

When should you attempt this iterative approach and when is it time to manually complete the board? There are no hard and fast rules, but the following conditions favor the iterative approach:

- **The components can be moved.** If design constraints do not require fixed component locations, try moving the components before routing.
- **The input PCB contains few, if any, pre-routes.** This simplifies the process of moving components on the input PCB.

- **The routing grid may be changed.** If you have selected a grid for placement such as 50-mil, you may be able to switch routing grids from say 25-mil to 16.7-mil without adding a lot of off-grid components. The finer routing grid may produce better results but must be within the fabrication capabilities of your board manufacturer.
- **More signal layers may be added.** This is a fast way to increase completion percentage, if you can afford the expense of multi-layer board fabrication.
- **Add Plane layers.** Power and Ground layers, or other plane net layers, can be added. This allows connections using thermals and eliminates lines on signal layers for these nets.
- **The routing process is relatively short, for the potential gain in completion.** If routing takes over an hour and only leaves a few connections unfinished, it might be faster to complete them manually. On the other hand, if routing takes a few minutes and leaves 20 or more connections unrouted, it is probably worth a few iterations to reduce the number of unrouted connections.
- **The completion improves with each iteration.** After a while, the iterative process yields diminishing rates of return. If this happens, it is time to back up one step and make another iteration.

Iterative Passes

There are two types of iterative passes: global and local. You can't select them directly, but you can guarantee a mixture of pass types. Although global iterative passes are more comprehensive in handling the most complex connections, a mixture of global and local passes often provide the best results.

The autorouter defines scheduling for these passes. At most, a maximum of three global passes are completed in a row. Specifically the rules are:

- If 98 percent or more complete, the autorouter runs a local pass.
- If less than 98 percent complete, it runs a global pass, unless three global passes have just been run.

For designs that are very dense or complex, select at least four passes by setting the Iterative Pass Count. This ensures a mix of global and local strategies. Before concluding that the router is unable to proceed, allow it to complete these passes.

Manual Approach

Given the guidelines above, when you feel that ACCEL PRO Route has done the best job possible, it is time to manually complete the remaining connections in PCB, using the Route Manual command or Route Interactive command.

Backing Up Files

The autorouter generates checkpoint files after each pass, after the time interval you set in the Route Autorouters dialog, and when you suspend the route using the Route Cancel command. These files have the same base name as your design, with a .CPT extension. The file is deleted when the router normally terminates. For safety, protect your checkpoint file (and your previous work) by making a copy of it before restarting.

If you want to save multiple strategies for one design, save each strategy under a unique file name.

We strongly recommended that you make backup copies of your finished designs and designs-in-progress. Hard disks occasionally crash and everyone at some time or another issues the command **DEL *.*** from the wrong directory! You can take every precaution to avoid these problems but it never hurts to prepare for them. Archiving all of your design files is inexpensive and takes very little time but it can be a lifesaver when a problem occurs.

And here's one final tip. If you are iteratively running your board through ACCEL PRO Route, save subsequent output files using different, meaningful names. If you move the components or make other changes that result in a worse completion rate than the previous PCB file, you will always have the previous PCB file available in reserve.

Routing Grid

The **Routing Grid** combo box lists the fixed list of grids allowed by Quick Route.

Design Rules and Routing Grids

There is a direct correlation between your design rules (e.g., line width, clearances, via styles) and the routing grid. Together, these values have an enormous effect on completion and performance of PCB Quick Route.

You are always responsible for setting the design rules because they are determined by your own particular fabrication requirements. PCB Quick Route, on the other hand, will assist by automatically selecting the best routing grid to go with your design rules. The default values supplied (12 mil for all instances) are satisfactory for general purpose commercial, high volume, low priced production.

If you choose your own routing grid, be certain you consider the design rules before selecting a routing grid, and vice versa.

Line Width

The **Line Width** scroll box lets you select aa legal line width whose minimum value is 0.1 mil and maximum value is a function pf tje Routing Grid selection. The routing line width can't exceed half the grid value.

For example, if you select a 25 mil grid, the Line Width option varies from 0.01 mil (.01 mm) to 12 mil (.30mm) in 0.1 mil (0.01mm) increments. If you type too large a value, the scroll box automatically self-adjusts to its maximum value when you move to another field. Use the up and down arrows to scroll through valid values.

Selection Reference Point

PCB offers the ability to place a selection reference point. The selection reference point is used with all Select operations, such as moving, copying, rotating, flipping, or pasting. However, a selection reference point is not required for these operations.

The selection reference point is saved to the clipboard or to a block file and automatically restored when pasting from the clipboard or block file. If the selection reference point is off-grid when a move operation begins, then it is automatically snapped to the nearest grid point and all the selected objects move the same relative distance. The selection reference point is automatically erased when all the objects are deselected.

To Place a Selection Reference Point

1. Select the object or objects.
2. Click the right mouse button which displays a pop-up menu.
3. Select the menu item Selection Point.
4. Click and drag the left mouse button to ghost the selection reference point.
5. Clicking the right mouse button will cancel the placement of the selection reference point.
6. Dragging the selection reference point over pads (within a pattern or free), vias, or ref points (within a pattern or free) , causes the selection reference point to snap to that objects center point. The shape of the selection reference point changes from a square to a diamond when it snaps to an objects center point. If the selection reference point does not snap to an object, then it moves to the nearest grid point.
7. Release the left mouse button to stop ghosting the selection reference point. The selection reference point is redrawn in the selection color.
8. To move a selection reference point, repeat the steps to place a selection reference point.

This function can be used for radial placement (e.g., of components). To do so:

1. Select the object to be radially copied.
2. Move the selection point to the point about which the object is to be rotated (right mouse down, and release the selection point).
3. Holding the *CTRL* key down, select the object, moving the mouse slightly so that the cursor snaps to the selection point (this copies the object), then release the mouse button and *CTRL* key.
3. Type *R* or *Shift-R* to rotate the copy.

Edit Align Components

Components can be aligned around a selection reference point either horizontally or vertically, and as an option, equally spacing the components. Or, if a number of components are off-gird, these components can be aligned back on-grid.

- The alignment of components is undoable.
- The alignment of components has full macro support.

To Align Components Horizontally or Vertically

1. Select the components to align. Only components can be selected for this command to be enabled.
2. Place a selection reference point. This is the point about which the components will be aligned, either horizontally or vertically.

   Without selecting a selection reference point, the alignment and component spacing options are grayed.
3. Select the command Edit Align Components. The Edit Align Components dialog appears.
4. Select either horizontal or vertical alignment.
5. To align the components with equal spacing, check the Space Equally checkbox and enter the Spacing value. The spacing value is the distance between the reference points of the components.
6. Press **OK** and the selected components will be aligned.

To Align Components To Grid

1. Select the components to align to grid. Only components can be selected for this command to be enabled.
2. Select the command Edit Align Components. The Edit Align Components dialog appears.
3. Press ***OK*** and the selected components will be aligned to grid. Each selected off-grid component will be moved to the nearest grid point.

Wide Line Routing

This pass routes all specified Wide Line nets before executing other passes. Specify wide line nets by adding the AUTOROUTEWIDE attribute to them. The Wide Lines Routing pass makes horizontal or vertical connections only and uses any enabled layer. If you require a wide line that is not horizontal or vertical, pre-route the line in PCB using the desired width. Quick Route maintains this width.

All routing of wide lines is based on the same grid selected for routing single lines. To guarantee the correct clearance, it is necessary for the wide lines to occupy more than one grid point.

When Quick Route performs a clearance check on pre-routed connections, it uses the actual width of the line in its calculations. If Quick Route can't complete the net in the remaining routing passes.

To route the Wide Line nets:

1. Run the board through QuickRoute with only the Wide Lines routing pass enabled. Disable all other passes.
2. If QuickRoute cannot complete the Wide Line nets, finish routing these nets manually using PCB.
3. Run the board though QuickRoute again; this time enable all routing passes. Disable the Route Cleanup and Via Minimization post-routing passes until all nets on the board are completely routed.

Horizontal

This completes simple connections on any layer selected with a horizontal bias, with no vias and minimal deviations from a straight, horizontal line.

Vertical

This completes simple connections on any layer selected with a vertical bias, with no vias and minimal deviations from a straight, horizontal line.

L Routes (1 via)

This pass is formed by the intersection of two lines and one via, forming an L. The lines have minimal deviations from their center lines and may be placed on any two enabled layers with opposite bias (horizontal and vertical) The L can have any orientation. Lines are placed no more than 100 mils outside the rectangle defined by the two endpoints in the connection. Although the pass is enabled by default, it is automatically disabled if at least two layers are not set to route in opposite directions (i.e., horizontal and vertical).

Z Routes (2 vias)

This pass is formed with three lines and two vias, forming an orthogonal Z shape. The Z can have any orientation Lines are placed no more than 100 mils outside the rectangle defined by the two endpoints in the connection. Although the pass is enabled by default, it is automatically disabled if at least two layers are not set to route in opposite directions (i.e., horizontal and vertical).

C Routes (2 vias)

This pass is formed with three lines and two vias, forming a C.   The C can have any orientation   The C route is more flexible than the L or Z routes; it allows lines to be placed more than 100 mils outside the rectangle defined by the two endpoints in the connection.   Any enabled layers may be used to complete the C route.   Although the pass is enabled by default, it is automatically disabled if at least two layers are not set to route in opposite directions (i.e., horizontal and vertical).

Any Node (2 vias)

The previous passes attempted to route only the optimized connections (the set of connections for a net that would minimize the total line length). To attain the highest possible number of completed connections, the Any Node pass analyzes each net and attempts to make a connection between **any** nodes in the net.

Maze Routes

This pass attempts optimum connections only (rather than any node in a net). This pass is not restricted by line orientation. It allows the line orientation to differ from the standard orientation of lines on the board layer, to make turns, and to double back.

The Maze pass inserts vias, as required to complete a connection. Specify the maximum number of vias that are allowed for each connection. The default is 10.

The lines generated by the Maze pass can block channels that you may need to hand-route remaining lines. If so, you might consider an iterative approach; run the board through Quick Route with maze routing *disabled*, hand-route the desired lines in PCB, and then re-route with Maze routing *enabled*.

Any Node (maze)

This pass uses the same routing strategy as the Maze pass. The Maze pass attempts to route only the optimized connections. The Any Nodes (Maze) pass attempts to obtain the highest possible number of completed connections by analyzing each net and attempted to make a connection between **any** nodes in the net.

Route Cleanup

This pass is included to improve a board's aesthetics and manufacturability. The autorouter intentionally hugs lines during routing for efficiency and higher completion rates. The Route Cleanup pass re-routes some of the lines to eliminate extra jogs where possible. This pass uses the concept of copper sharing to combine lines in the same net. The reduction in line segments has the added benefit of reducing the size of your design file.

Via Minimization

This pass attempts to reduce the number of free vias. During this pass, Quick Route checks all the lines connected to each free via. If the lines can be swapped to another enabled layer without violating the design rules, they are swapped and the via is removed.

This pass doesn't affect pads. If you want to ensure that all pre-routed vias remain intact, disable this pass or use a through-hole pad instead of a via for connecting pre-routed lines on opposite signal layers.

ACCEL ASCII Files

A file filter in the **List Files of Type** listbox in the File Open dialog and the **Save File as Type** listbox in the File Save dialog reads ASCII input.   The new file filter is **ASCII Files** (*.pcb), and it will read and write ACCEL ASCII files.

If ACCEL ASCII Files is selected in the file type listbox of the File Save As dialog, a ACCEL ASCII file is generated instead of a PCB binary file.   The output file is a complete design file,and it contains all design data represented in a PCB binary file.   A description of the ACCEL ASCII format can be found in the file **ASCII.WRI**.

Errors and warnings generated while loading an ASCII file are written to a file named ***<design-name>*.log**, which is automatically displayed using the Microsoft Windows Notepad application.

ASCII File Save and File Save As use a temporary file for file generation.   The file is named ***<file>*.pc$**, where file is the is the name you specify in the File Save As dialog or the current design name if you are using the File Save dialog.   If the you cancel the save process or it is interrupted by an error condition, the temporary file will contain the ASCII contents written up to the point where the process was terminated.

Simaltaneous Class Routing

Simultaneous Net Class Routing indicates whether the ACCEL PRO Route autorouter will route all design net classes simultaneously, or route each class in sequence. Simultaneous routing is useful if signals routed during the first passes are prone to block channels necessary for routing subsequent net classes.

A net class is a collection of nets whose attributes cause the router to route that group of nets together. For example, several nets having identical WIDTH and VIASTYLE attributes attached would be viewed as a distinct net class by the PCB autorouter. These nets would be routed in separate passes from other net classes if **Simultaneous Class Routing** is not enabled. If the option is enabled, all net classes are routed in all passes. The net classes chosen by the autorouter are listed in the router log file.

To enable simultaneous class routing, select the **Simultaneous Class Routing** checkbox in the Route Autorouters dialog. The default for this option is "off" since not all boards benefit from simultaneous routing. Simultaneous routing requires significantly more system memory, and is slower than sequential pass routing. Memory usage and the time to complete a route increase directly as the number of net classes increase if this option is checked.

This option is recommended for designs where the Fanout or other route pass places a route for one net class that blocks the router from placing a route for a subsequent net class. Some designs will route with higher completion rates and fewer vias when using the Simultaneous Class Routing option.

Routing and Via Grids

When the routing grid selection is set to **Auto Grid**, the via grid will be equal to the routing grid.

If you manually enter a non-uniform routing grid (Auto is not selected), the via grid used is a multiple of the sum of non-uniform grid values. Normally, the multiple is one, but you can set this to be a multiple value greater than one. For example, a routing grid of 16.7 16.6 16.7 mils will cause the router to set the via grid to the sum of 16.7 16.6 and 16.7: a 50 mil grid. In this example, a multiple of 2 sets the via grid to 100 mils.

The router may offset the routing grid in order to increase routing completion. The design's relative grid origin will be updated by the router to reflect this offset. Switch to relative grid mode to make further editing changes to the design.

Utils Reconnect Nets

The Utils Reconnect Nets command uses the current design netlist to re-establish the net information for copper traces, arcs, pads, polygons, and vias having point-to-point routes to copper in an existing net. When you run this command, the Reconnect Nets dialog appears.

This command is like the **Reconnect Copper** option within the Load Netlist command, except that it uses the current design netlist information instead of an external netlist file.

Point-to-point routing is not required. Routes that are physically connected but not point-to-point routed are indicated by a connection (blue-line).

Shorted nets are indicated in the logfile with a warning. These nets are not merged.

Cutout

A cutout is a polygonal, void area for copper pours which will not be filled with copper when the pour is flooded. Cutouts do not affect solid polygons or keepout polygons. Cutouts are recognized only by copper pours.

Options Net Classes

The command lets you define a group of nets that share common rules. Collections of nets sharing the same rules are referred to as a net class.

When you click the **Net Classes** button, the Net Classes dialog appears.

This class editor allows you to create named net classes using pre-defined clearance rules or pre-defined SPECCTRA autorouter clearance rules and then assign nets to that class. You can also add user-defined attributes to the net classes for your own use.

For net classes, you can specify general clearance rules. These rules can be further refined by specifying clearance rules for pairs of objects, like pad to pad clearances or line to via clearances.

Net classes are transferred from the Schematic design to PCB via the ACCEL-format netlist.

The net class information is written to binary and ASCII design files. P-CAD does not support Net Classes so this information is lost when exporting an ACCEL design to P-CAD format.

To create named net classes:

1. Enter a class name in the **Classes** box.
2. Click **Add**.
3. Select unassigned nets and click the **Add** button (bottom of dialog) to add the nets to the class.
4. Use the **Edit Attributes** button to assign one or more attributes to this new net class. (Refer to the Edit Nets command section for details.)

Edit Move by RefDes

This command allows you to select and move a component by entering its reference designator, without knowing where the component is located in the design.

When you run this command the Edit Move By RefDes dialog appears:

To move a component:

1. Enable the Select tool.
2. Select a reference designator from, or type a value in, the **RefDes** drop down combo box. The list contains all reference designators in the design.
3. Click **OK**.

   The cursor appears as a *crosshair* shape.
4. Move the cursor to the workspace location where you want to place the component.
5. Click to move it to the new location.

   You can click down to make a *ghost*, then drag and drop (release) to place it more accurately. To cancel ghosting (and moving) of a component, click the right mouse button.

Net Tab

A number of objects have net information appearing on a Net tab with the objects Properties dialog. This section describes the information you will find on the Net tab.

Net Name

The **Net Names** box contains the name of the net associated with this pad.

Nodes Listbox

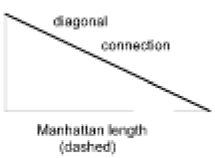
The **Nodes** listbox contains the names of all nodes in the net associated with this pad.

Layers Listbox

The **Layers** listbox contains the names of all layers associated with this net.

Connection Lengths

This area shows the **Manhattan** length, the **Selected** length, and the **Total** lenght of all connections in the net.   The Manhattan length is an approximation of the final routed length of a diagonal connection.



It only measures the X and Y distances, not depth (such as via length to another layer).   Arc length is included (accurately) in the calculation of connection lengths.

Copper Lengths

This area shows the the **Selected** length and the **Total** lenght of all copper in the net.

It only measures the X and Y distances, not depth (such as via length to another layer).   Arc length is included (accurately) in the calculation of   copper lengths.

Counts

The **Counts** area counts the following objects in the selected net:

- Arcs
- Lines
- Pads
- Polygons
- Copper Pours
- Vias

Net Attrs Button

When you select the **Net Attrs**, the Attributes dialog appears.

This dialog show a collection of net attributes for the selected net.   You can add, change, edit and delete attributes for the net.

For details about this function and a complete listing of attribures, see the Edit Nets command below.

DRC Error Indicator Properties

Allows you to view DRC Error Indicator Properties. Each indicator represents an error or violation detected in the design rule checking pass (Utils DRC).

Generating DRC Error Indicators

To generate DRC error indicators:

1. Use the Options Display command to display DRC error indicators. Click **Misc...** and click the **Show** radio button in the DRC Errors Area.
2. Run the Utils DRC command.

Selecting DRC Error Indicators

1. Use the Options Block Selection command to include only DRC error indicators in a block select
2. Run Edit Select to perform the block select.

Viewing Properties

1. Select DRC error(s) either individually or through a block select.
2. Choose Edit Properties to display the Error Properties dialog.
3. The dialog is view-only, therefore you are limited to reading error information, or scrolling through the errors if you have selected multiple errors in the design.

   Use the **Next** and **Previous** buttons to scroll forward and backward through the errors.

Route Interactive

Allows you to run the interacting routing tool. The interactive routing tool provides obstacle avoidance, copper hugging, and intelligent route completion.

Not available for ACCEL TangoPCB.

Routing Connections

To route connections, follow these steps:

1. Select the Route Interactive tool from the menu or from the tool bar.
2. Pick a connection to route. The end closest to the cursor becomes the source point; the other end becomes the destination point.
3. The Interactive Route tool rubberbands uncommitted copper with the proper net width and clearance from the source point to the position of the cursor in the current ortho mode. The rubberbanding copper tracks around obstacles, maintaining the proper clearance for that net, from the point of origin to the cursor position.

   While you are moving the cursor, a connection rubberbands from the cursor to the *second node* of the connection, indicating what remains to be routed.
4. When you click a coordinate that is not the destination node, the previously uncommitted lines are placed on the board.
5. Continue this process until you click on a node that is the other end of the rubberband (this could be a pad, via, line, arc, polygon or copper pour island).

   If the node clicked on is a termination point for the connection, the route completes automatically and you are now free to choose another connection to route or cancel the tool.

Changing Layers

Changing layers adds vias in the same manner as is done with the manual route tool except that once you switch to a new layer, the Interactive Route tool recalculates its internal data structures to reflect obstacles on the new layer.

Blind and buried vias are handled as they are with the manual router. Changing layers while on a through hole pad simply changes layers (if the hole range allows it).

Backtracking

Committing a straight line segment on top of another straight line segment, so called backtracking, acts as an erase operation and the intersection (in the mathematical sense) of the two lines is removed. Vias cannot be erased with backtracking. The user has to perform an unwind operation.

Unwinding

Any segments or vias that are committed can be uncommitted by pressing the *Backspace* "" key or running the Unwind command from the right mouse popup menu (see below).

Slash Key

Routing can be suspended by pressing the slash "\" or "/" key and the unrouted portion becomes a new connection from the suspension point to the end node.

Vias, Layers and Line Widths

You can select vias, layers and line widths during the routing operation through the Options menu or the appropriate hot keys.

Net Attributes

The net attributes WIDTH, VIASTYLE, CLEARANCE, PADTOPADCLEARANCE, PADTOLINECLEARANCE, LINETOLINECLEARANCE, VIATOPADCLEARANCE, VIATOLINECLEARANCE, VIATOVIACLEARANCE will be honored given the appropriate net class hierarchy.

Copper Pours

Copper Pours affected by new copper generated by this tool will autoplow when the route **Completes** or **Suspends**.

Placing Arcs

Arcs can be placed, but obstacle avoidance and hugging are disabled.

Pop-up Menu

Click the right mouse button to display the following popup menu:

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

- **Complete** autoroutes the remaining portion of the route.
- **Suspend**, equivalent to the "slash" key, leaves the unrouted portion as a connection.
- **Cancel** undoes all routing actions and puts the original connection back. The Interactive Route tool remains active.

  Pressing *Esc* is the equivalent to selecting **Cancel** from the right mouse popup menu.
- **Layers** executes the Options Layers command.
- **Via Style** executes the Options Via Style command.
- **Unwind** removes the last line or via that was committed.

The following keys are recognized during the routing process:

Enter	Same as left mouse, commits the current routed series of lines.
Esc	Cancels the entire routing process and returns to connection selection mode.
/ or \	Stops routing the connection and leaves remaining, unrouted portion as a connection.
Backspace	Undoes the last segment placed.

O Cycles the current ortho mode.

F Flips the current ortho mode.

Rerouting Lines

The routing of existing lines is supported by the Interactive Route tool as described above except that once the operation is completed, the tool waits for another line to be selected.

The popup menu behaves the same as when routing connections.

T-Route to Vias

You can use the Interactive Route tool to T-Route into vias that are part of the net that you are routing. To do so, route up to the via, then run the Suspend command.

Edit Alter Components

Allows you to select certain component items and subsequently move, rotate, flip, and (in some cases) delete them.

With this editing process, you can alter certain component characteristics either for aesthetic reasons or manufacturing improvement, such as avoiding any co-location problems during manufacturing (e.g., through-holes and silkscreen paint). The rules/restrictions are as follows:

- Pads cannot be selected, and therefore cannot be edited in any way.
- The RefDes, Type, Value and Ref Point (reference designators and reference points) cannot be deleted.
  If you want to hide the RefDes, Value, or Type, modify the component (Edit Properties command), and disable the Visibility options. To hide Glue Dot and Pick and Place points, disable their respective display options in Options Display (Misc).
- You cannot undo any alter actions until the operation is complete.

Altering a Component

1. Run the Edit Select command to select the component. Then run the Edit Alter Component command.

   You are now in a temporary editing mode, signified by the *crosshair* cursor shape.

2. Zoom in sufficiently. Change layers if necessary. You can select items individually only if you are on the correct layer, otherwise you need to use a block select. The settings in Options Block Selection can affect your results here.

   Editable items typically reside on the Top Silk layer, except for Ref Points, which aren't layer items, but this depends on how the pattern was originally created.

3. After you have selected individual item(s), you can then move, delete, or otherwise alter them (according to the restrictions mentioned previously). Press *Esc* or click the right mouse button to end the editing mode.

note:

You can also use the Sub Select command to select and modify a single part of a component.

ACCEL TangoPCB Features

ACCEL Tango PCB is a Windows-based printed circuit board design system without the full range of advanced design features available with ACCEL P-CAD PCB.

Differences between the two products have been noted throughout this manual with the symbol you see in the left margin. The following chart provides a summary those differences:

Maximum number of components	400
Maximum number of layers	4 user-defined 15 total
Copper pours	only one with up to 32 vertices. 90 thermals, backoff amount restricted to 1 per layer.
Unsupported objects	cutouts pick and place points glue dots
Route Miter	unsupported
Route Interactive	unsupported
Rotation Increment	90 degree only
DBX modify functions	unsupported
Pin and Gate Swapping	unsupported
On-line DRC	unsupported
Split Plane	unsupported
Dimensioning	unsupported
ECOs	Was/Is only

Opening a P-CAD Binary File

To load a P-CAD binary file, follow these steps:

1. Choose File Open. The Open dialog appears.
2. At the **List Files of Type** field, select .pcb Files as the desired file type.
3. In the **File Name** box, enter the name of the desired .pcb file or select it from the file list.
4. Click **OK**. PCB automatically detects that youre loading a P-CAD binary file and displays the P-CAD to ACCEL PCB Layer Mapping dialog.
5. At the P-CAD to ACCEL PCB Layer Mapping dialog box, select the appropriate layer mapping assignments. For details about mapping layers, see the section Mapping Layers from P-CAD Binary and PDIF Files to PCB.
6. Specify a tool table by clicking the **Tool Table** button. The Tool Table dialog box appears,

   At this dialog box, select the desired tool table (.tbl) file and click **OK**. PCB returns you to the P-CAD to ACCEL PCB Layer Mapping dialog box.

   PCB requires the tool table file because it contains hole size information that the P-CAD file does not have. You specify the tool table file to use at the tool table dialog box. If you do not specify a tool table file, PCB assigns two default hole sizes for you. The sizes are 20 mils for pads and 10 mils for vias.

   If you load the design without specifying a tool table file, you must edit the hole size of all pad and via styles that have a hole size other than 20 mils and 10 mils. To change these sizes, open the pcb.ini file, go to the [PDIF] section, and edit the entries `DefaultPadHoleSize` and `DefaultViaHoleSize`.

   If a tool table file is specified but you do not want to load it, uncheck the **Use Table** checkbox. PCB uses the default hole sizes instead.
7. Specify the Cross Reference file to use using the PDIF Cross Reference button. The Cross Reference file contains power and ground pin assignments and attempts to group heterogenous components.
8. Specify how you want PCB to convert polygons by selecting **Copper Pour** or **Polygon**.

   You can convert polygons to copper pours or use them as polygons.

   Select **Copper Pour** because these polygons can have cutouts (voids) and backoff from pads. Also, if you have padstacks with void definitions, you should select **Copper Pour**.

   The only time you might want to select **Polygon** is if the P-CAD polygons have no cutouts (voids) in them. A P-CAD polygon with a void that is imported as a PCB polygon has the same net information. However, voids are ignored, which may cause a shorted board.
9. Specify how you want to convert the pad definitions. You can either convert them from the **Embedded Aperture Table** or from the designs **Pad Graphics**.

   The polygon aperture shapes and the Aperture macros are not supported.

   Select **Embedded Aperture Table** to take full advantage of the aperture shapes and sizes. You should select **Pad Graphics** only if you dont have an aperture table embedded in your design.
10. Click **OK**. PCB loads the selected .pcb file.

    While loading, PCB displays several messages, indicating the systems progress. After PCB loads the file, a message box appears indicating if there were any errors or warnings. If there are none, you can click OK.

    If there are errors or warnings while loading the file, PCB creates a log file *design-name*.log, where *design-name* is the name of the P-CAD binary file you loaded. In this case, a dialog box appears, prompting you to view the log file. PCB displays the log file using the viewer selected with the Options Configure command. The default viewer is Notepad.

It is a good idea to examine the log and correct any errors that show up. For details about any problems that could occur, see Appendix B, P-CAD System Messages.

Master Designer Command Reference

The following lists maps Master Designer commands/operations with their equivalent PCB commands, where equivalents exist.

The bolded terms on the left are the Master Designer commands, followed by their respective equivalent PCB commands/operations, which are non-bolded and indented.

For example:

Master Designer command

Equivalent ACCEL PCB V2.00 Command/Operation

For more information on specific PCB commands, refer to their respective sections by way of the **Contents** or **Search** buttons in this on-line system.

Legend: > = Sub-command or sub-selection

To obtain the ACCEL command equivalent click one of the following topics:

Master Designer Editors

Align > Components

Edit > Align Components (PCB)

Align > Objects

Edit > Shift Select, then Edit > Align Components

Align > Window

Edit > Select by window, then Edit > Align Components

Align > Undo

Edit > Undo (or the U key) to undo last operation

Change Layer (Expand Def.)

Edit > Move to Layer

Chg. Layer > Component

Edit > Select component, change to the identified layer (Top, Bottom, etc.) and use the F key to flip the component to that layer

Chg. Layer > Object

Edit > Move to Layer

Chg. Layer > Objects

Edit > Shift Select, then Edit > Move to Layer

Chg. Layer > Window

Edit > Select a window, then Edit > Move to Layer

Copy > Object

Edit > Copy or Ctrl + Left Mouse Button (Drag & Drop)

Copy > Object> Repeat

Edit > Copy Matrix

Copy > Objects

Edit > Shift Select, then Edit > Copy Matrix or Ctrl + Left Mouse Button (Drag & Drop)

Copy > Trace

Edit > Copy Matrix or Ctrl + Left Mouse Button (Drag & Drop)

Copy > Window

Edit > Select a window, then Edit > Copy Matrix or Ctrl + Left Mouse Button (Drag & Drop)

Critical. Path > Add Gate

not applicable

Critical. Path > Break Link

not applicable

Critical. Path > Change Name

not applicable

Critical. Path > Remove Definition

not applicable

Critical. Path > Remove Gate

not applicable

Delete > Object

Edit > Select, then Edit > Delete (or the Delete key)

Delete > Objects

Edit > Shift Select, then Edit > Delete (or the Delete key)

Delete > Trace

Edit > Shift Select to select individual copper segments, then Edit > Delete (or Delete Key) to delete the selected copper segments. To delete the entire net, Edit > Select a net segment, click the right mouse button to bring up the Pop-up Menu > Select Net, then Edit > Delete (or the Delete Key).

Delete > Undo

Edit > Undo (or the U key) to undo last operation

Delete > Window

Edit > Select a window, then Edit > Delete (or the Delete Key) [Items can be masked before selection by using the Options > Block Selection]

Dimension

Place > Dimension

Display > Measure

Edit > Measure

Display > Control Ratsnest

Edit Nets > Select by Node Count

Display > Long Pan

not applicable

Display > Path/Group Visibility

not applicable

Display > Recall View

not applicable

Display>Store View

not applicable

Draw > 2 Point Arc

Place > Arc

Draw > 3 Point Arc

Place > Arc

Draw > Line

Place > Line

Draw > Polygon

Place > Polygon

Draw > Polygonal Void

Place > Cutout (Doesn't Clear a Placed Polygon, used for Copper Pour)

Draw > Text

Place > Text

Draw > Circle

Place > Arc

Draw > Circular Void

Place > Cutout (Doesn't Clear a Placed Polygon, used for Copper Pour)

Draw > Filled Circle

not applicable Draw > Filled Rectangle

Place > Polygon

Draw > Flash

Options > Pad Style, then Copy Default pad style and select Modify (Simple) or Modify (Complex) to create a new pad style (pad stack)

or

Options > Via Style, then Copy Default via style and select Modify (Simple) or Modify (Complex) to create a new via style (pad stack)

Draw > Rectangle

Place > Polygon

Edit > Add Via

Place > Via

Edit > Add Vertex

Route > Manual

Edit > Attribute

Edit > Select a component, then Edit >Properties > Comp Attrs or Part Attrs

Edit > Delete Segment

Edit > Select a copper segment, then Edit > Delete (or the Delete key)

Edit > Delete Trace

Edit > Shift Select to select individual copper segments, then Edit > Delete (or Delete Key) to delete the selected copper segments. To delete the entire net, Edit > Select a net segment, click the right mouse button to bring up the Pop-up Menu > Select Net, then Edit > Delete (or the Delete Key).

Edit > Delete Vertex

Edit > Select vertex to be removed, then drag to closest remaining vertex.

Edit > Delete Via

Edit > Select a via, then Edit > Delete (or the Delete Key)

Edit > Keyword

File > Design Info > Attributes or Pop-up Menu > Properties.

Edit > Move All

Edit > Select and Drag copper vertices or vias

Edit > Move Segment

Edit > Select and Drag copper segment

Edit > Move Vertex

Edit > Select and Drag copper vertex

Edit > Move Via

Edit > Select and Drag via (will move attached copper segments)

Edit > Net Attr

Edit > Nets and select a net from the list. then click on the Edit Attrs button or

File > Design Info > Attributes

Edit > Packaging Data (Symbol Mode)

Utils > Library Manager > Pattern View

Edit > Pin Type (Symbol Mode)

not applicable (See Pad Style)

Edit > Segment Layer

Edit > Move to Layer

Edit > Trace Width

Edit > Select, then Edit > Properties

Edit > Undo Delete Segment

Edit > Undo (or the U key)   to undo last operation (The backspace key can be used to unwind trace & line segments)

Edit > Wire

Route > Manual

Enter > Component (PCB Editor)

Place > Component

Enter > Polygon

Place > Copper Pour

Enter > Wire

Route > Manual

Enter Attribute

Place > Attribute

Enter > Board Origin

not applicable

Enter > Component Type

Place > Attribute Type keyword

Enter > Jumper (Symbol Mode)

not applicable

Enter > Non homogeneous Pkg (Symbol Mode)

Utils > Library Manger

Enter > Pin Sequence (Symbol Mode)

Utils > Renumber (Accomplished before Library > Library > Pattern Save As

Enter > Pin Type (Symbol Mode)

See Pad Style

Enter > Ratsnest

Place > Connection

Enter > Replace Component

Edit > Select a component or group of like components, then Edit > Properties > Type, to change the selected component to another type from the loaded library (PCB Only)

Enter > Uncommitted Pin

Edit > Select Connection to pin, then Edit > Delete (or the Delete key)

Environ.. > Polygon Wire Clearance

Edit > Select copper pour, then Edit > Modify Copper Pour - Pour Backoff

Environ. > Merge Voids by Layer

Using Options > Block Selection to mask everything except Copper Pours, Edit > Select all Copper Pours on a layer (Edit > Shift Select or Edit > Select a window), then select the Pour Radio Button

Environ. > Merge Voids by Poly

Edit > Select Copper Pour, then Edit > Modify and select the Pour Radio Button

Environ. > Set Minimum Aperture

not applicable

Environment > Display Statistics

File > Design Info > Statistics

Environment > Mask Items

Options Block Selection

Environment > Assign Layer Pairs

not applicable

Environment > Attach Padstacks

not applicable

Environment > Change Units

Options > Configure > Units

Environment > Detail Mode

Not necessary

Environment > DOS Shell

Not necessary

Environment > Edit Aperture Table

File > Gerber Out, then select Apertures button

Environment > Min. Polygon Size

Edit > Select > Copper Pour > Properties > Island Removal Dialog box

Environment > Set Snap Tolerance

Option > Preferences > Mouse > Cycle-Picking Threshold

File > Level Pop

not applicable PCB (See Part & Pattern Creation in the Editors)

File > Clear Database

File > Clear (or File > New)

File > Create Plot File & Hardcopy > Print/Plot or Windows Plot Utility

File > Print (Ctrl + P)

File > Level > Push

Edit > Explode Components & Utils > Library Manager

File > Load

File > Open (Ctrl + O)

File > Load Block

Edit > Paste from File

File > Print

File > Reports

File > Save Block

Edit > Copy to File

File Quit

File > Exit

File Save

File > Save (Ctrl + S)

Fit View

View > Extent

Group > Add Gate

not applicable

Group > Change Name

not applicable

Group > Remove Definition

not applicable

Group > Remove Gate

not applicable

Chg. Layer > Component

Edit > Select Component , then use the F key to flip

Improve Plc > Components

not applicable

Improve Plc > Gates

Utils > Optimize Nets

Last View

View > Last

Move > Attribute

Edit > Alter Component(PCB)

Move > Component

Edit > Select, then Drag component

Move > Critical Path

not applicable

Move > Group

not applicable

Move > Object

Edit > Select, then Drag Object

Move > Objects

Edit >Ctrl Select, then Drag Objects

Move > Undo

Edit > Undo (or the U key) to undo last operation

Move>Window

Edit > Select window, then Drag Objects

Name > Component

Automatically named

Name > Net

Edit > Nets > Rename

Name > Pin (Symbol Mode)

See Part & Pattern Creation in the Editors

Name > Reseq. Ref. Des.

Utils > Renumber > Ref Des Radio Button

Name > Reseq. Window

Utils > Block Selection to mask everything but specific components, then Edit > Select Window & Utils > Renumber

Pan (Shift + Mouse Button 1)

View > Center (C key)

Pan > Enter Coordinates into status line

View > Jump Location

Placement > Automatic Placement

not applicable

Placement > Define Barriers > Delete

not applicable

Placement > Define Barriers > Delete

not applicable

Placement > Define Barriers > Enter

not applicable

Placement > Define Lattices >

not applicable

Placement > Define Lattices >

not applicable

Placement > Define Lattices > Enter Spacing

not applicable

Placement > Define Lattices > Specify Clearance

not applicable

Placement > Enter Cutlines

not applicable

Placement > Fix > Component

not applicable

Placement > Fix > Window

not applicable

Placement > Histogram > Configure

not applicable

Placement > Histogram > Define Grid

not applicable

Placement > Histogram > Generate Statistics

not applicable

Placement > Histogram > Reset Merit Factor

not applicable

Placement > UnFix > Component

not applicable

Placement > UnFix > Window

not applicable

Query > Component - By Name

Edit > Components > Jump

Query Net > By Name

View > Nets > Jump To Node

Query > Apertures

File > Gerber Out, then select Apertures button

Query > Component

Edit>Select, double-click on a Component or Edit Select Component and choose Properties from the Pop-up Menu (Right Mouse Button)

Query > Critical Path

not applicable

Query > Group

not applicable

Query > Net

Edit > Select, double-click on a Net or Edit Select Net and choose Properties from the Pop-up Menu (Right Mouse Button)

Query > Net Pins

not applicable

Query > Net Vias

not applicable

Query > Object

Edit > Select double-click on an object or Edit Select an object and choose Properties from the Pop-up Menu (Right Mouse Button)

Query > Padstack

Edit Select Component, then Edit > Modify > Pins > Edit Style > Modify (Simple) or Modify (Complex)

Query > Pin

Edit > Select, Component, then Edit > Modify > Pin

Query > Polygon

Not necessary

Query > Polygon Window

Not necessary

Query > Trace

Edit > Select copper segment, then Edit > Nets > Info (provides length of entire copper and length of remaining connections)

Query > Via

Edit > Select via, then Edit > Properties

Recall View

not applicable Redraw

View > Redraw

Rotate > Component

Edit Select a component, then use the R key

Rotate > Object

Edit Select an object, then use the R key

Rotate > Objects

Edit Select a window, then use the R key

Rotate > Undo

Edit > Undo (or the U key) to undo last operation

Rotate > Window

Edit Select a window, then use the R key

Swap > Component

not applicable

Swap > Gate

Utils > Optimize Nets > Manual or Auto Gate Swap

Swap > Pin

Utils > Optimize Nets > Manual or Auto Pin Swap

Swap > Undo

not applicable PCB for Utils > Optimize Nets > Automatic, but can use Edit > Undo for Utils > Optimize Nets > Manual

View Layer

Options Display

View Window

View > Zoom Window (Z Key)

>Merge Polygon Voids

Edit > Select a polygon pour, then Edit > Properties > Poured Radio Button

Master Designer Utilities

Design Maintenance

not applicable in

System Configuration

Options > Configure

Report Generator

File > Reports

Netlist Comparison

Utils > Compare Netlist

Engineering Change Order ~ Back Annotate

Export ECOs (Back Annotate)

Report Editor

Not Necessary. Reports can be directly output to the screen or a user preferred file viewer can be selected in Windows to view previously generated reports.

PCB Tools

Editor Configuration

Options Menu

PCB Editor

ACCEL PCB

Autorouter

Route > Autorouters

Design Rules Check

Utils > DRC

SPECCTRA Router

Route > Autorouters > SPECCTRA

PDIF File Reader

File > PDIF In

PDIF File Writer

File > PDIF Out

Hardcopy or Windows Plot Utility

File > Print or File > Gerber Out

Report Generator

File Reports

Drill

File N/C Drill

Auto-Insertion

See Place > Point > Pick and Place or Glue Dot

Netlist Conversion

Utils > Load Netlist (.ALT Format)

EZPlot

File > Print or File > Gerber Out

Report Editor

Not Necessary. Reports can be directly output to the screen or a user preferred file viewer can be selected in Windows to view previously generated reports.

Interfaces

Add Program

Not required under Windows, use ALT - TAB

SPICE Circuit Writer

Utils > Generate Netlist > PSpice

Viewlogic Interface

See Viewlogic Interface

EDIF Netlist Writer

Utils > Generate Netlist > EDIF 2.0

DXF File Reader

File > DXF In

DXF File Writer

File > DXF Out

Report Editor

Not Necessary. Reports can be directly output to the screen or a user preferred file viewer can be selected in Windows to view previously generated reports

System Configuration

Text Editor

User Defined File Viewer

Printer Port

File>Print Setup

PCB Editor Configuration

Rotate Padstacks (check box)

not applicable

Rotate Text in Four Directions (check box)

not applicable

Allow Adding Uncommitted Pins (check box)

Not Required - a connection between pins must be defined before user can connect a wire to a component pin

Use Display List (check box)

not applicable

Display Trace Length (check box)

not applicable

Auto Clear Flash (check box)

Not Required - Automatic

Load Active PCB (check box)

not applicable

On-line DRC (check box)

Options > Configure > On-line DRC (check box)

Display Verbose Warnings (check box)

Not Required

Dynamic Ratsnest Display (check box)

Options > Configure > Connection Options (check box)

Edit Design Rules File

not applicable - Integrated into PCB Design File

Edit Check Pass File

not applicable - Integrated into PCB Design File

Function Key Filename:

Options > Preferences > Keyboard

Auto Save Configuration

Options > Configure > Autosave

Pin Size:

not applicable (see explanation of Pad definitions)

Via Size:

not applicable (see explanation of Pad definitions)

Flash Size:

not applicable (see explanation of Pad definitions)

Sub Select Command

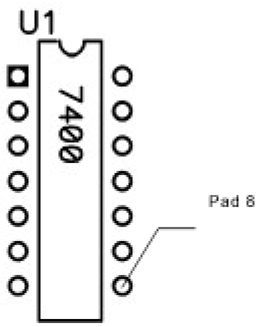
The sub select feature lets you select a single part of an object. Once selected, you can view and, in some cases, modify properties for the item selected. Certain silk items can be moved, rotated, and flipped.

For example, you can select a component pad and bring up a Properties dialog for that pad. Then you can change the pad style for the selected pad.

note:
You can use the Options Preferences Mouse tab to set the *Ctrl* key as the key to use for a sub selection.

Example

To select pad 8 in a 7400 component:



2. Hold down the *Shift* key and left mouse button, and click the pad.

   The pad, and not the entire component, will be selected.