# Package Editor User Guide

# Package Editor User Guide

# Table of Contents

## Package Editor Purpose.

P ackage Editor (PackEdit) is a Windows application for designing 2D physical package models for electronic components. These package models are used for creating PCB footprints, and for verifying component fit prior to board fabrication. Package models can additionally be used for automated programing of Pick and Place assembly machines by assembly houses.

All PCB design tools include some type of footprint editor, but all footprint editors lack the means for defining the metal contact areas where component pins touch down.  Footprints are strictly models to be placed on boards sent to a board house. In contrast, an assembly house needs component placement information. Correct placement verification is impossible without this contact area definition of where a component's leads will land on a board's footprint. PackEdit addresses these assembly needs.  PackEdit is used to design 2D package models based on data structures termed pins. Pin definitions and their locations are required to show how well, or how poorly, a PCB footprint has been designed for a particular package. PackEdit enables PCB footprints to be created from the package pins' geometry. PackEdit thus provides a means to export quality footprints to PCB design tools.

Index Designs has created a database, called PackData.mdb, of thousands of package models and ships it with all three tools:  PartSync, BOM Builder and PackEdit.  While the ability to define package models is important, what's also required is a means to search for existing packages in the package model database. PackEdit provides a customized search engine for locating existing packages, using certain conventions for common names and units. Many equivalent package names are cross-referenced within the PackEdit search engine and measurement units are converted to a common base unit. PackEdit's searching is intelligent and not simply a text compare. For example, a lookup on the search string "soic 8 pin 50mil pitch" will find SSOP, TSSOP and similar package name variations, along with pitch dimensions described in the database as any of 0.05 inch, 50 mil or 1.27mm pitch.

A variety of package parameters can be saved in each package model. In the PackEdit graphics window, these parameters are presented in a graphic WYSIWYG (what you see is what you get) manner, much like a graphical spreadsheet. All package model data is saved in a simple Microsoft Access database (.mdb) file which is readable via many database tools. PackEdit also provides for backing up and restoring individual, selected, or all packages.

When PackEdit executes, a connection is made to the default database using standard Windows database tools (e.g. ADO, JET). Two windows are created, one for backup and restore, and another, the Pick A Package window, showing a large list of existing packages. Selecting an existing package will display that package on the righthand side's graphics window. A selected package can then be opened in the Package Editor using the Edit button.

**Initial Screens**

When PackEdit opens, the previous database connection is restored. Below, you can see the use of a shared drive (P:\) which points to a "Production" shared folder on a NAS (Network Appliance Storage) device. Immediately after installing PackEdit, the default package database location will be:

 C:\ProgramData\IndexDesigns\PackEdit\Data\PackData.mdb

…but if more than one user will be using package data, the use of a shared drive is recommended. The storage device MUST be a Microsoft Network share or one which uses the SMB protocol. Under Linux, a SAMBA server share can be used to provide the Windows Network SMB protocol.

If a new path and filename is entered, click the Connect Button to connect. In the "Backup Source and Dir" region, selecting the "Backup All" radio button, followed by a click on the Backup button will bring up a file dialog for selection of a file name for output. The "Days Old" parameter selects packages from the current time back to so many days old. These packages will be written to the output file.

When the "Restore All & New" radio button is selected, the "Pre-Delete All Data" checkbox is enabled. Checking this box will result in all existing data being deleted PRIOR to reading the selected file. It is suggested that backups of the current package database .mdb file always be done prior to any restore.

All backup and restore functions work with simple text files (.txt). As a last gasp to recovering corrupt files, these .txt files can be opened in a ttext editor. Extreme care is required when inspecting or editing these .txt files.
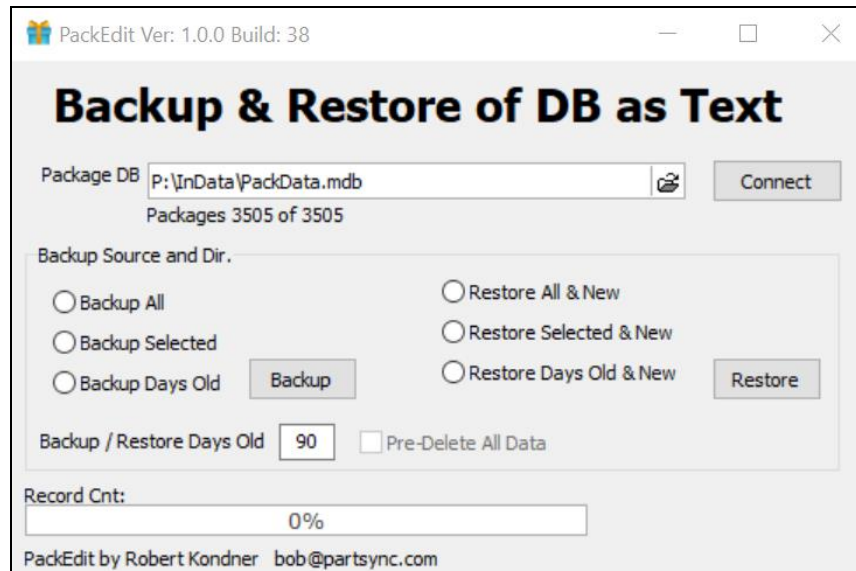
Once a new database file is selected, the "Connect" button is clicked to open the .mdb database. After the database is opened, the packages in the database are listed along the left side of the "Pick A Package" screen. A package can be selected with a mouse click, and that package is displayed on the righthand side's graphics window. The figure below shows an 0204 package selected using a mouse click on the 0204 line on the left. The top left search description field enables entry of a string for package searches. Checking Age Sort will sort packages by creation date, with the newest packages shown first. Search results will organize packages by how well they match the search description. Best matches are shown first, with blank lines displayed between groups with different search result weights. Show All displays all packages and sorts them alphanumerically by package name.

On the righthand side of the Pick A Package screen, several key parameters are shown, along with buttons for Edit, Delete and Write PCB Decal. Note that there is no "Undo" for the delete function.
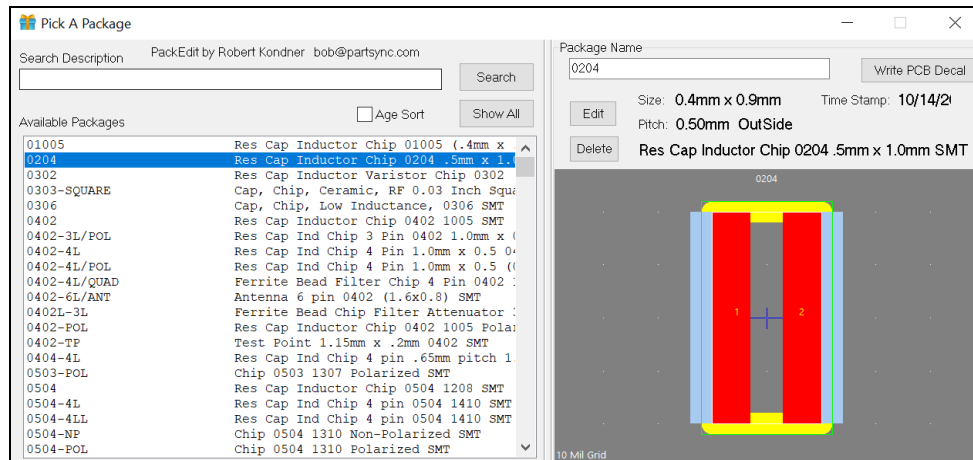
A search finds a number of closely matching packages. It is important to include search text using the words pin and pitch, such as "8 pin" and "50mil pitch", as that is what we wish to find. The search engine is not based on text searches, so be sure to enter important parameters. Note that both mm, inch and mils are translated and converted into a common searchable base unit.   The figure below shows the resulting list after on searching on "SOIC 8 pin 50mil pitch":
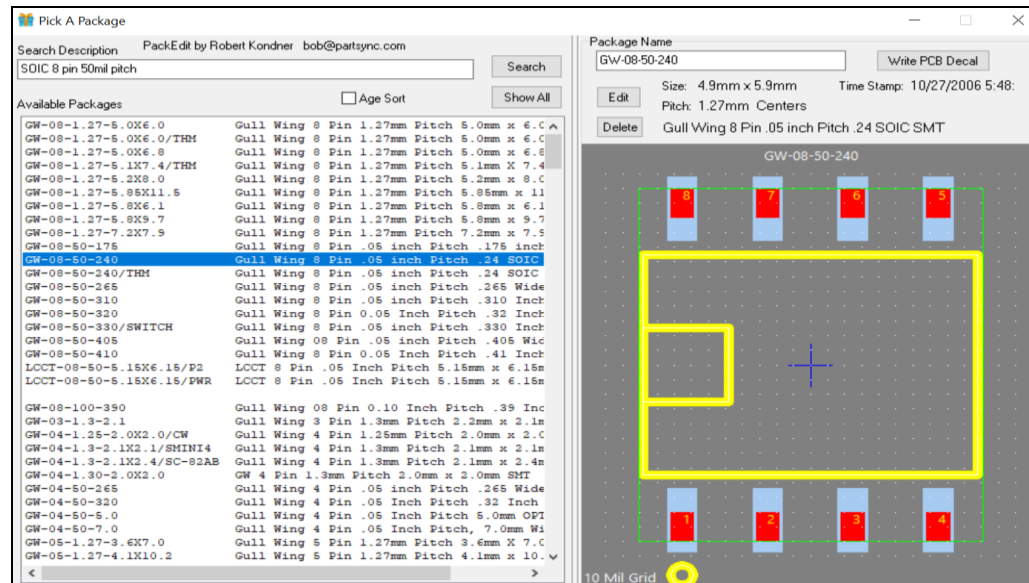


Along the righthand side, just under the Package Name field is the Edit button. Click the Edit button to enter into the Package Editor.

**Batch Mode Updates**

When "Backup All" is selected the backup cycle also executes a Batch Update process which uses .txt files to simplify changes to package names and descriptions. After backup completes software looks for two text files used to change package names and descriptions in the C:\ProgramData\IndexDesigns\PackEdit\Data directory.

- _BatchDescriptions_Edits.txt

- _BatchPackNames_Edits.txt

Each of these files consists of text arranged in two columns. The left most column is simply the name of a package in the database. The second column is either the description for the package or a package name. These files are read in line by line and changes to the second column (right column) is inserted into the database for the package names in the left column. This scheme allows users to rapidly rework package names and descriptions.

After these files have been read and processed (if they exist) two similar files are written using the database content. These "Current Data" files are located in the C:\ProgramData\IndexDesigns\PackEdit\Data directory.

- _BatchDescriptions.txt

- _BatchPackNames.txt

Renaming these files with the "_Edits" in the file name prepares for later use as update content. All package names are written to these files. The next Backup All process will read these _Edit files, update the database, delete the _Edit files and then create fresh "Current Data" files. If the user wishes to same the _Edit files they should be copied elsewhere as Batch Update deletes the _Edit files after they are read and processed.

All user edits are made to the right side column. Note that spaces, tabs or any other whits space is not allowed in package names. Descriptions can contain spaces.

**Import, Export and Exit Controls**

After selecting a package in the Pick A Package window, hitting the Edit button on the righthand side will open a new window, the Package Editor. In the figure below, the 0204 package was selected in the Pick A Package window and then opened for editing in the Package Editor. The figure shows just the upper right portion of the resulting Package Editor window. Above the package graphics, to the right of the Name and Descriptions fields, are two buttons - one for package import and one for package export. These buttons will read and write package data into a text file with a .pkg file extension. These .pkg files are simple text files which can be viewed in a text editor, but their contents should not be edited as this could corrupt the file format. The content is very specific and it is organized in a line-by-line manner. Users can transfer these files to other users or place them in a library. (Access .mdb format files can also be distributed.)
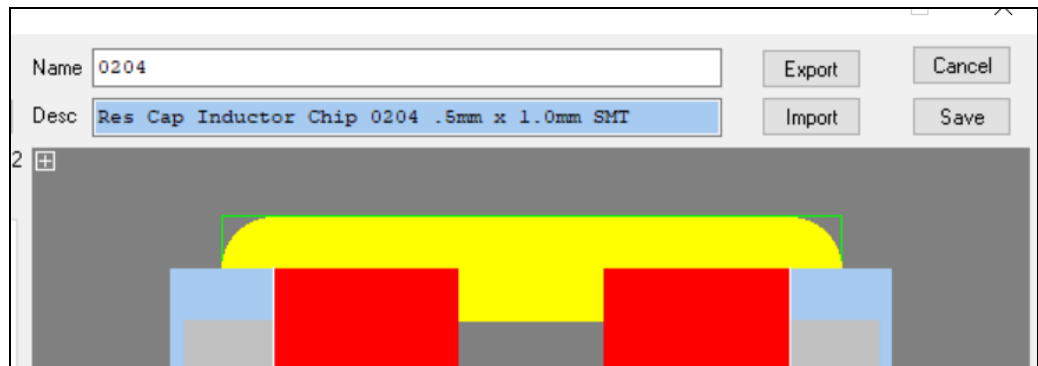


Clicking on the Cancel button will discard any edits and close the Package Editor window. All edits will be lost and there is no undo. The Save button writes the package data to two internal text formats and returns to the Pick A Package screen. The first internal text format is the full package data. This full string is used when the package is re-loaded into the editor. The second internal text format is a simple subset of the package data which is used for graphic display. This second string is placed in the .mdb file in a number of fields. These fields simplify the task of rendering graphic images in various windows. The full data is compressed and placed in a memo field within the .mdb file.

Below the main graphic image are several checkboxes to control graphic display in the Package Editor. If a box is checked, that graphic item is displayed as a solid. If un-checked, that item will be displayed as an outline. This is useful when creating complex PCB pads or when a large package become complex.
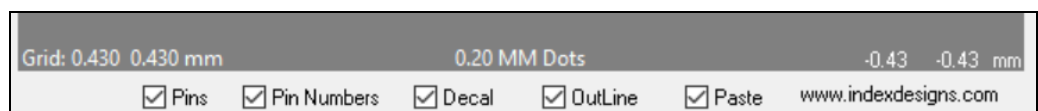
# Package Editor Tab Controls

Clicking on the Edit button in the Pick A Package screen opens the selected package in the Package Editor. A long string is extracted from the .mdb database, and numerous parameters are separated and loaded into various editor fields. The fields are grouped into several tabbed sections and a graphic image is generated from these parameters. Every time a parameter is changed, the graphic display updates in a manner similar to the operation of a spreadsheet. Tabs are provided for editing of Package, Pins, Outline, PCB Pads and Paste:

| | |
|---|---|
| Package Tab | Overall shapes, pin size and count and spacing. |
| Pins Tab | Additional details about pins |
| Outline Tab | Silkscreen iutlines and graphics. |
| PCB Pads Tab | Generation and control of automatic PCB footprints. |
| Paste Tab | Generation of solder paste stencil apertures. |

As shown in the figure below, over top of the tabs are Lead Group controls for selecting, adding and deleting "Lead Groups". A lead group is a container structure used to simplify the construction of packages with multiple types of leads. For example, a 25 pin D connector would have two lead groups. The first group, Main Group, is for the 25 signal pins. The second lead group would be for the two mounting holes. There may even be a third lead group for grounding-shell solder leads. Search for "con 25 pin D smt". Click the Edit button and open it into the Package Editor:
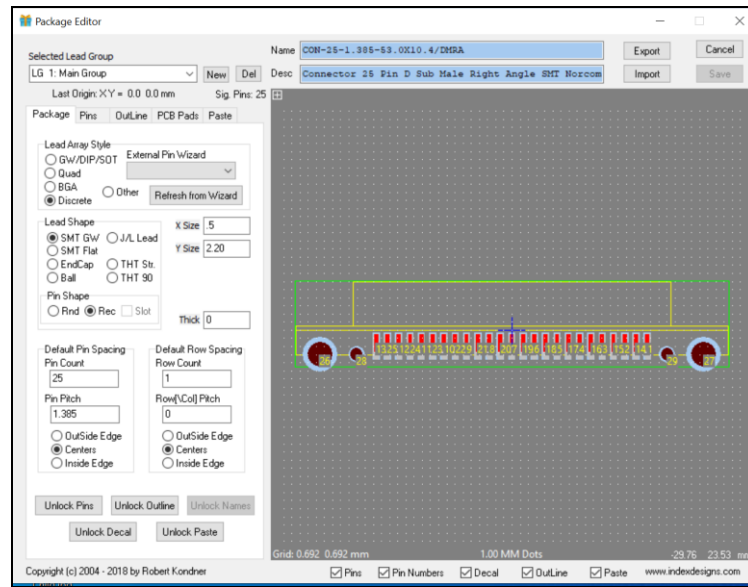
The graphical righthand side is an editable view of the selected package. Above the graphical editor window are fields for the package Name and Description. The Name field is a unique name used to store the complete package in the .mdb database. The Description field is very important as it feeds the search engine. The content of these fields should follow naming and description syntax as illustrated in other packages. The background color of these fields is light blue, which indicates that these fields have not been edited. Before a new package is saved, these fields should be edited, which will result in the fields reverting to white. The light blue serves as a reminder to edit these fields prior to saving. Imported packages can be saved without editing Name or Description if the Name is unique and the Description is complete.
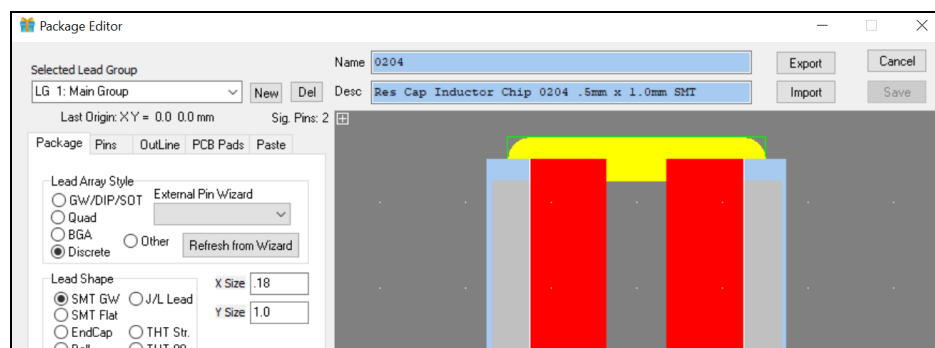


### Package Tab

Package tab parameters control overall shape and pin sizes for the selected Lead Group. Lead Array Style selects between common package styles, and a custom pin wizard allows custom code to generate default pin assignments. Changing a Lead Array Style by selecting a different radio button will delete any existing package definitions and will create a new set of defaults.

In the Lead Shape region, there are several fields:  X Size, Y Size, Lead Thickness and others. Several of these are not used as they were included for a future 3D parametric modeling interface. X Size, Y Size, THT and Pin Shape are used. These are the basic controls for defining pin shapes in the selected lead group. Fields X Size and Y Size support mil, mm and inch units. Numbers without decimal points are considered to be in mils and numbers with decimal points are considered mm. Entry of 50 or 1.27 both represent the same number in microns which is the unit used internally. Entry as 50.0mils is also supported. Simple arithmetic functions such as 100/2 or .1+.05 can also be entered.

Below the Lead Shape region is the Default Pin Spacing region, where Pin Count, Row Count and Pin Pitch are used to space pin around the package center. Typically, the Pin Count is divisible by the Row Count which results in a number of pin rows with equal pin counts. If not divisible, then the pins will default to various locations and the user must manually place the pins. These pin definitions apply to the pins in the currently selected Lead Group.

Row[\Col] Pitch is used to space rows and optionally columns. Text entry such as 240\300 will space rows at 240 mils high and columns at 300 mils wide. Radio buttons can adjust the spacing to be set for Outside, Center or Inside pin positions. This greatly simplifies pin placements when using datasheets that don't reference pin centers.

At the package tab bottom are buttons to Unlock various pin attributes. When a user manually changes a pin position, lead shape or other attribute, then that attribute becomes locked and certain automatic "graphic spreadsheet" functions become disabled. The Unlock Pins button clears these lock flags, allowing automatic

functions to again control package pin settings. Similar unlock buttons are provided for Outline, Names, Decal (i.e footprint) and Solder Paste attributes.

At the top of this Package tab, to the right of the Lead Array Style region, there is a drop-down for the name of a Pin Wizard file.  For more information on Pin Wizard operation, see the PINDATA_Sample.txt file in the directory:

C:\ProgramData\IndexDesigns\PackEdit\Data\PinWizards\

**Pins Tab**

While the Package tab controls how pins are shaped and placed on the package, the Pins Tab provides detailed information about pins themselves. As shown in the figure below, Pin Type can be Signal, Mount or Shield. Signal pins are the most common, as they carry signals. When describing connectors, we use the number of signal pins as the description's pin count. A 25-pin D connector will have 25 signal pins and 2 or 4 mounting pins. A BNC connector or a crimp terminal has 1 pin. There is logic in the PCB footprint calculator that rounds up drill sizes for through hole signal pins. A 20 mil pin might not fit in a 20 mil drilled hole due to drill and plating tolerances. Through hole signal pins get a small increment in diameter followed by a round up to a common set of standard drill sizes defined in the DrillRules.txt file. Through hole mounting hole sizes are never adjusted. They are passed into footprint data for exact drill sizes. All of this logic is controlled using the DrillRules.txt file.

Toe Positions are defined in IPC-7351 standards document. In general, the "Toe" is on the outside edge of a lead and the "Heel" is on the inside edge of a lead. The other two sides are considered "Side" positions. BGA pins default with no Toe or Heel and only Side is used. When generating PCB footprints, the internal IPC-7351 software definitions in PackEdit will use Toe, Heel and Side settings to form footprint models. The "Auto" checkbox will allow the software to define toe positions for each pin. Other check boxes will fix the toe positions for all pins in the lead group.

Many packages include a large thermal pad in the package center. If the Large Center Pin region's X Size and Y Size fields are non-zero, a center thermal pad will be generated. If a "Quad" package was selected, the "Pin 1 Positions" region will show radio buttons for selecting the location of pin 1. Quad packages typically have pin 1 positions in 1 of 3 locations.

Pin Name Style, Pin Number Order and Pin Column are used to control the automatic generation of pin number (numeric and/or alpha) and how they are placed around the package. Worse case, the user can manually define pin names and positions using the keyboard and mouse.

There are occasions when the pin connection points are not located at the pin center. Pin Connection Offsets can be defined for both top and bottom side pins. The Side Assignment region controls the placement side for surface mount pins. Placement is controlled for odd, even or for ½ of the pins. This last option is used with edge connector fingers where ½ are located on the top side and the remaining are on the bottom. Odd and Even are also used for edge connector fingers.

All the settings shown in the Pins Tab are the default parameters for all the pins in a lead group. With the Pins Tab selected a double-click on a pin in the graphics area allows these parameters to be changed for a single pin. A simple click on a pin allows a pin to be picked up, and a second click will drop the pin in a new position. If a pin is picked in error, pressing ESC will abort the pickup.

Various options will be active or inactive, depending on the type of package selected in the Package Tab.

**Outline Tab**

Outlines represent the silkscreen artwork on a PCB Footprint. Several parameters are defined for controlling automatic generation while most of the actual content is created using the keyboard and mouse. Automatic outline generation of outline height and width are created from parameters applied to either pin outside edges, inside edges, centers or the origin. Body shape can be round or rectangular depending on package shape. The figure below displays the contents of the Outline tab.

Line segments, corners and pin 1 dots can be moved in the graphics area using mouse clicks. Grid XY values and Origin should be set for the graphics area before moving items. A right-click provides options for objects to be moved or changed. Lines can be picked at either endpoints or in the center. Arcs are created by selecting two end points, followed by a pull of the arc.

Before entering graphical outline data, verify that the origin, grid and line width settings are appropriate. A right-click, followed by selecting Insert Outline on the pop-up, will start the entry of a line sequence. Additional clicks will add corners. A double-click will end the line drawing process. An additional click will start a new line sequence. When finished, press ESC to exit entering lines.

**PCB Pads Tab**

The PCB Pads tab is used to control the generation of footprints. Pin sizes and shapes are merged with IPC-7351 settings to form the footprint pad patterns (i.e. copper, shown in light blue graphic shapes). The lead geometry for the pin is shown in red. That is, a red rectangle in the graphics window, sitting on a light blue rectangle, represents the location of where the lead will touch down on the footprint's pad. IPC settings for various lead types can be selected in this PCB Pads tab, along with IPC Max, Medium and Min settings. Sliders for Toe, Heal and Side simplify adjustment of the footprint's land (i.e. pad) patterns in a WYSIWYG manner. Optionally, a user can define T, H and S settings using text entry fields. A quick calculation identifies routing distance between pads for most common package styles.

Through hole annular ring dimensions are calculated for big, medium and small holes. Small holes are defined as holes smaller than the Medium. Pin 1 Square is used to force pin 1 of a through hole component to a square shape for assistance during board assembly. A package courtyard is provided and the outline can be included in computing courtyard size. If unchecked, only pin outside edges are used. Pin 1 dots are not used when computing courtyard size. Additional offsets can be entered for Courtyard X and Y.

**Paste Tab**

Custom solder paste apertures are defined in the Paste tab. The current implementation of PCB footprint logic ignores this data as most PCB footprint editor tools include their own pastemask definition logic. Most assembly houses have their own logic for the generation of paste apertures from PCB copper images. For example, many assembly fabrication houses prefer that customers supply pastemask geometry (for openings in a pastemask stencil) to be 1:1 with the copper pads. If your board requires footprints to have pastemask openings that are other than 1:1 to the copper pads, you would modify the pastemask geometry in your PCB tool's footprint editor. PackEdit simply creates 1:1 (pastemask opening to copper pad) geometry when Write PCB Decal is used to create a footprint from a package model.

The Paste tab provides for user setup of gasketing size as well as minimum size and space rules. Aperture shapes can be set to As Pin, Rounded, Rectangular, Home Plate or None.

Again, these paste settings are ignored in current PCB footprint export logic.

# Mouse & Graphic Editing

**Overview**

Controls in the various screen tabs set up the default parameters for lead groups in a package. In many cases, these work fine but there are occasions where Pins, PCB Footprint or Paste settings need to be changed for individual pins. Mouse clicks and double-clicks are used to modify Pins, Footprints and Paste sizes and shapes along with other parameters. Selecting the Pins tab will enable edits to Pin data. Likewise, selecting the PCB Pads or Paste tabs will result in editing of footprint pads and paste mask openings.

In the graphics area, a single click will "pick up" a shape, and another click will drop the shape. The ESC key will return the shape to the original position and end the selection. A control-click will show pin and shape parameters is the "Pin and Pad Edit form. This form is enabled by checking the Pin/Pad Details checkbox in the lower right of the Package Editor screen.

When a pin is edited, that pin is no longer under complete control of the tab default parameters. The color of a pin with a manual edit is changed to note the manual changes. Pins change from red and blue (top and bottom pins) to magenta and cyan. Manually changed pins are said to be locked. Locks can be removed using Unlock buttons on the Package Tab or in the Pin & Pad Edit form.

A right-click provides several options for pin and outline editing and for adding and deleting pins. Outline content can be added or changed, and the active edge (top, bottom, left, right or center) for moving a shape can be selected. Outline segments can be selected at corners or at line centers. Several text commands exist that do not use mouse functions. These are called "Modeless Text Commands". See keyboard entry in later chapters.

Graphic display is controlled using keyboard keys. Zoom In is accomplished with the Page-Up key. Zoom Out is accomplished with the Page-Down key. The Insert key will pan. In each case, the mouse pointer defines the new graphic center. Pressing the Home key resets the pan and zoom to show the entire package centered in the graphic area.

There are two grid settings. The "Grid" is used when dragging and dropping, and it is not visible. The setting of this grid is shown in the lower left of the graphic. The lower right shows the current X Y position of the mouse. The second grid setting is the "Grid Dot" setting which is shown on screen as small white dots. The value of the Grid Dots spacing is shown in the graphic window's lower middle.

In the graphic window's upper left is a "Drag By" indicator. A small white box and small cross show the edge of the shape that will be used to control pickup and moves. The cross is either located on the top, bottom, left, right or center of the box. This displays how a shape will be picked up and dropped. When moving shapes, it is best to first set the grid and origin using modeless commands. The Drag By indicator can be set using modeless commands or the via the right-click menu.

### Pin Edits

The most common edit is to change a pin position. After setting grid, origin and Drag By parameters, a simple click (with Pins Tab selected) on a pin will pick it up, and a second click will drop the pin. The pin edge used for pickup and dropping (dropping is always on grid) is shown in the upper left Drag By indicator (small box with a small cross.)

Pins can be added or deleted after a pin is selected, using the mouse. Select the pin to be deleted (the pin becomes attached to mouse cursor) and press the Delete key. A black hole will be shown where the pin was previously located. A deleted black pin can be selected again, and an additional Delete key press will restore the pin. This function is useful when adjusting the pin pattern of a large BGA and a required pin is accidentally deleted.

To add an additional pin, first select an existing pin that is the correct size and shape. Move this attached pin to the required position and do a mouse right-click. In the pop-up menu, select Add Pin. A new pin is added, and the pin that had been selected is returned to its original position. With the new pin generated, it can be selected and dragged to the required position.

Adding pins should be done after the grid spacing and origin have been set. These are described in the keyboard command chapter. Many other pin parameters can be changed using the Pin & Pad Edit form.

Another common edit is creating a pin shape using a polygon. Right-click (Pins tab) to bring up the pop-up menu and click on insert polygon. Next, left-click on the pin that is to become the polygon shape. The cursor changes, and the starting point of the polygon can be defined with another left-click. Additional polygon line segments can be created, and the Backspace key can be used to delete incorrect previous points. A double-click will finishes the polygon. PCB decal and paste shapes will follow the pin shape as 1:1 and can be changed later.

Another common edit is creating a footprint pad shape using a polygon. Right-click (Pin Decal tab) to bring up the pop-up menu and click on insert polygon. Next, left click on a PCB decal shape that is to become a polygon shape. The cursor changes and the startng point of the polygon can be defined with another left-click.

Additional polygon points can be created with left-clicks while the Backspace key will delete previous points. A double-click will finish the polygon.

### Re-Number Pins

A right-click on the graphics window offers a Re-Number Pin option. Selecting this option immediately changes the mouse cursor and enters a new pin number of 1 to the left of the XY location display. See Below.



Clicking on a pin will change that pin's number to 1. If a different number is required, simply type that number on the keyboard and the displayed number will be changed. When the required number is set, clicking on a pin will assign that number to the pin and the next sequential number is displayed. Additional pins can be numbered sequentially by simply clicking on additional pins. Pressing ESC will terminate the renumber process.

### Add / Delete Corners, Circles & Arcs

When editing Outline graphics (the Outline tab must have been selected.) a right-click option is available to add a corner to a line segment. When a line segment center is selected, and graphic editing is active, a right-click followed by an Insert Corner click will insert a corner into the line segment. That corner can then be dropped on the graphic screen. Corners are deleted by selecting an Outline corner with the mouse, and pressing the Delete key. Selecting the center of a line segment and pressing Delete will delete the entire segment along with any attached segments.

Circles are added using a right-click and Add Circle. The cursor changes shape, and the first click sets the circle's center position. Moving the mouse sets the circle diameter and the second click completes the circle. Circles are selected by clicking the circle center. Selected circles can be moved or deleted by pressing the Delete key.

Arcs are drawn by using a right-click then Insert Arc. The mouse cursor changes shape. Then the first click sets the arc start point. A second click selects the arc end point and arc drawing starts. As the mouse moves, the arc radius and direction is drawn. A third click ends the arc. Arcs are selected by clicking the arc center. A selected arc can be dropped with a mouse click or deleted using the Delete key.

### Drag By Control

Drag By refers to the ability to move pins and shapes by center or edges. The letters C, T, B, L and R are used to select Center, Top, Bottom, Left and Right edges. Center is the default if no letter is specified. Options in the right-click menu allow

the pad and shape pick up location to be defined. A small graphic box and cross in the graphic upper left shows the current pickup option.

Dragging pins and shapes by edges assists with package creation, as many device datasheets dimension pins by their edge lengths A significant amount of sketching and arithmetic is avoided by simply defining origins and placing edges on the grid. The figure below shows the Drag By Center indicator:

# Pin & Pad Editing

Settings that define pins and pads originate from the controls shown in the left hand side of the Package Editor. These are the default settings for a Lead Group and multiple Lead Groups can be defined. All pins and pads in a Lead Group receive the same defaults when the Lead Group is defined. It is very common for one or two pins or pads to require different setting and this is done using the Pin & Pad editing form. This form is enabled using the Pin/Pad Details Checkbox. Once this form is displayed it should be moved to a screen position next to the Package Edit form. The following image show the Pin/Pad Details box checked and the Pin & Pad Edit for to the right of the Package Editor form. Notice the circled checkbox at the screen bottom.
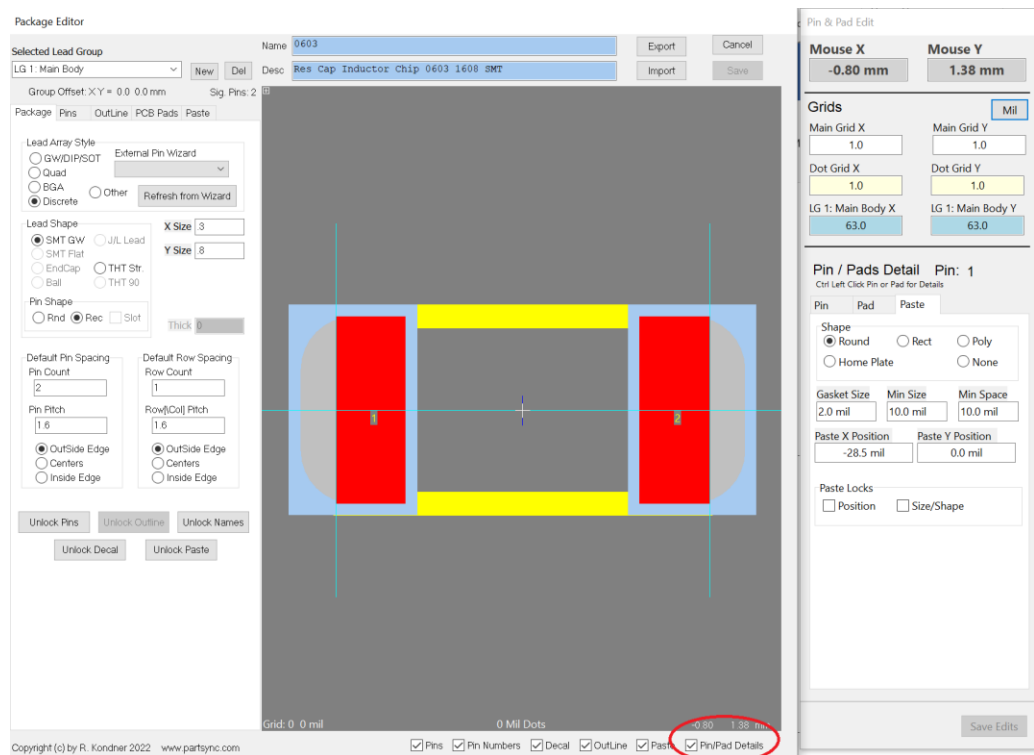


**Edit Form Layout**

The top section of the Pin & Pad Edit form shows current mouse position, grid spacings and a clickable indicator button for controlling displayed units. Clicking this button changes the displayed units from mils to mm and back. Editing of the grid X Y fields is not supported. Use the Modeless commands to change the Grid

and Dot Grid settings. The Lead Group Grid is defined by settings in the left Package Tab when a Lead Group is defined or changed. In the above image the light blue vertical line positions show that the pin outside edges were defined as alignment points.

Below the Grids section is a display of the selected pin and 3 sets of tabs used to control pin, pad and paste settings. In the above image pin 1 is identified as the selected pin. Control left click on a pin to select a different pin. Within each tab are several radio buttons, check boxes and text fields. Clicking a radio button or check box will immediately apply the change to the package and then lock that parameter which overrides the default lead group settings. This locking is shown as checks in the lock checkboxes. Clicking on a checkbox will remove the locked conditions and lead group defaults will again control the selected pin or pad.

### Text Field Editing

Text fields are used for the display and entry of numeric values. When a pin is selected current package parameters are used to populate the text fields in units as defined by the Unit control button. As a user edits a text field the color changes from white to red to green. White indicates the text field content matches the current package content. Red indicates a syntax error in the text field content. Green indicates the content is valid and is different from the current package content. When one or more fields displays in green the Save Edits button at form lower right is enabled.  Clicking Save Edits will save the green field contents into the package and set the appropriate lock flags.

Text field provide special editing options to simplify data entry. Unit selection can be defined for either mils or mm by including either "mm" or "mil" in the entry. If neither is entered then the presence of a decimal point controls units. No decimal point indicates mils while the presence of a decimal point indicates mm. Text fields also support some very simple operation using + - * / characters. These operators should include space characters on both the left and right.

| Pin X Position | Pin Y Position |
|---|---|
| -25.6 mil | 0.0 + 5 mil |

| Pin X Size | Pin Y Size |
|---|---|
| 11.8 * 25.4 mm | 31 mil |

| X Connection Offset | Y Connection Offset |
|---|---|
| rlk mil | 0.0 mil |

# Modeless Text Commands

Text commands for various functions are accepted from the keyboard when there is no other control in focus. To terminate any existing control that might have focus, press the ESC key once or twice. Examples of controls that might be in focus are X Size or other text entry fields. If such a field is in focus, then keyboard text would feed into that field and not the command processor.

Text commands available are:

| | |
|---|---|
| G | Grid defines mouse drop points. |
| GD | Grid Dot defines white graphic dots. |
| O<br>OPn<br>OR | Origin. Sets Origin for grids and controls the Origin push stack. Origin is specified using XY values or pin numbers. OPn sets origin at pin n. OR resets origin for Lead Group > 1. |
| M | Move pin. |
| S | Search to X Y or pin during drag of pin or outline. |
| Pn[T B L R C] | Pin select by top, bottom, left, right or center. Arrow keys will then be used to move pin on the grid. |
| L | Lead group select. |
| A | Align lead group using pin number. |

When the first key of a command is pressed on the keyboard, a User Command window will display, showing help for various command options. As each command starts with different letters, help for that commend is shown. Parameters can be entered, and pressing Enter will end the entry and the command will execute.
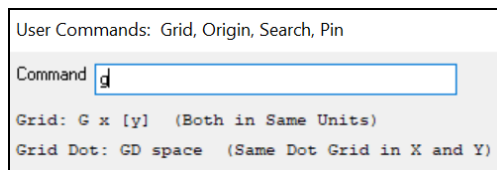
### Grid and Grid Dot

Two grids exist in the package editor. Grid Dots are not functional, and they only show as small white dots on the graphic screen for visual reference. The Grid serves as a grid used for mouse placement of pins and graphic shapes. A command such a "G 25" will force dropping of items on a grid of 25 mils in both X and Y. Using "G 1.0 .5" will set the drop grid to 1mm in the X and .5mm in the Y. If a single numeric value is entered, it is used for both X and Y. Setting different X Y numbers simplifies entering grids for most components.

Both grid and grid dots are always centered on the package origin. Manual placement of pins is best done by first setting the origin to the placement reference point, and then setting the grid, followed by moves of pins. When an origin is set, the previous setting is pushed onto a stack. Using an Origin command without any parameters will pop the previous origin off the stack. This makes the process of changing origin, moving pads, and then restoring the origin very simple.

Grid Dot is similar except there is no Y value. The small white dots are always drawn with identical X and Y spacing. Command "GD 50" places dots on 50 mil centers. "GD 1.27" also places dots on the same 50 mil (1.27mm) centers. The grid is always aligned to the package origin.

As an example, consider a move of pin 1 to the left 1.0mm from where it is now:

1. O P1
2. G 1.0 1.0
3. &lt;Click on Pin1, move, click to drop pin 1&gt;
4. O

The above sequence first sets the origin to the current Pin 1. Next, the Grid is set to 1.0mm in both X and Y. Then Pin 1 can be selected using the mouse, and it will be moved in units of 1.0mm in both X and Y. Another mouse click will drop pin 1 at the required position. The Grid Dot is very useful when dropping pins when using fine Grid definitions. The final Origin command, O, will restore the Origin to the initial location.

### Origin

Setting the origin (0,0) of a package to different locations is very useful when manually positioning Pins and PCB Pads. New origin points can be defined using X Y values, pin numbers, and pin edges. Both the Grid (drop grid for mouse drops) and the Grid Dot display are referenced to the package origin. If pins 3 and 4 need to be moved to locations around pin 1, then the origin should be set to pin 1. Command "OP1T" will set the origin to pin 1 top edge. After setting the Grid, pins 3 and 4 can easily be moved around the pin 1 top.

A thin blue cross is placed at the package origin. When a PCB footprint is generated, this point becomes the package origin. If package data is used to drive Pick and Place assembly robots, the origin should be adjusted close to the center of

mass for large components. For Lead Groups > 1, a small white cursor is displayed at the origin for the lead group.

When a new origin is set, the previous origin is pushed onto a stack. Once pins have been moved, an Origin command with no values, O, will pop the previous value off the stack. The stack is 10 levels deep, which allows complex origin movements while preserving the starting origin locations. For example:

1.   O P1T
2.   O

The above example moves the origin to the top of pin 1 and then back. The X Y value entered can contain simple arithmetic expressions.

Origins operate slightly differently for Lead Groups > 1. As origin settings are entered for a Lead Group > 1 that lead group is moved over top of the main lead group (Lead Group 1). When a PCB footprint is generated, all the lead groups are flattened, leaving a single set of merged pads. The origin for the base group (blue cross) remains, and is used for the footprint center.
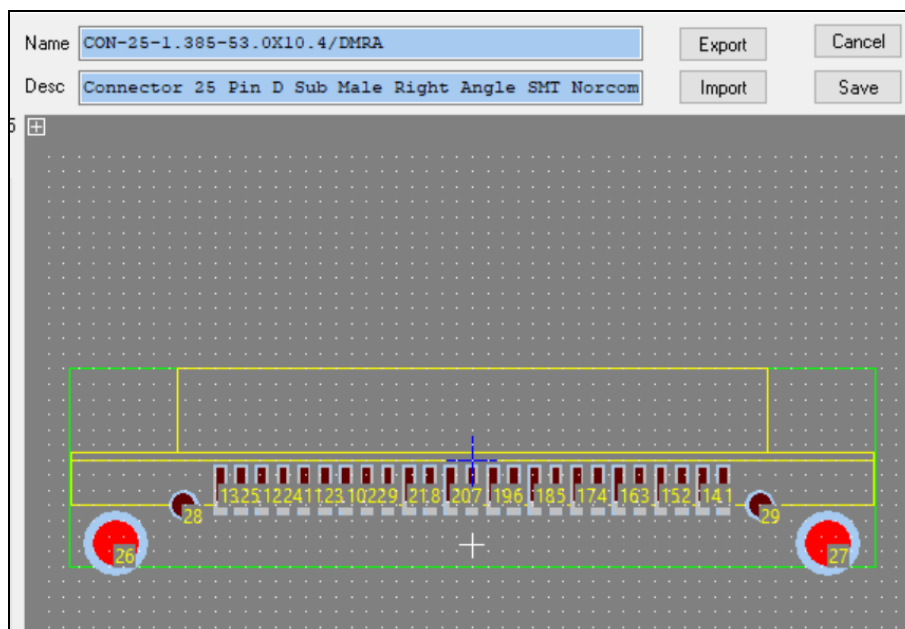


The ground lug lead group has been moved down 100 mils (O 0 -100) showing both the main group origin (blue cross) and the lug lead group origin (white cross).

Each lead group has its own stack. When the main group origin is changed, all leads in all groups are offset. When the origin for other groups is changed, only pins for that group are moved. The user can move back and forth between groups, and use the "O" command to pop back previous origins. Note that the origin stacks are not saved in exported package data. Origin stacks only remain in existence during the editing process.

### Move Pin

The "M" command is used to move a pin to a specific X Y location. Pins can be moved by name (Pin Center) to either an X Y value or to another pin center. Edge moves are not supported. Example:

1.  M P1 0 -100          Move pin 1 down 0,100  in mils.
2.  M P2 P3              Move P2 to P3 position.

When a pin is placed using the M command, the pin is dropped exactly where specified, and the Grid setting is ignored. If X Y values are specified, they are relative to the origin of the main lead group (blue cursor). If a pin is specified, the pin is moved to the XY of that pin.

### Search

When graphically editing pins and shapes using the mouse, there are times when placement at a specific X Y location is required. To do this, the mouse must be disabled and a "Drop Click" is done using the keyboard space bar. As an option, the keyboard arrow keys can be used to move the shape. All movements will be on the current Grid.

To use the search command, either a pin, shape or outline must first be selected. Either top, bottom left, right or center can be selected for pins. Outline allows selection at endpoints, corners or line centers. With the object attached to the mouse, the Search command can be entered, and the object will move to the defined position. Mouse movement will be disabled, and arrow keys will move the object in increments of the Grid settings, but not onto Grid locations. Finally, the object is dropped at the location shown using a left mouse click or the keyboard space bar.

### Pin Select

Pin Select allows a pin to be selected and moved using the keyboard. Arrow keys will move the pin by the Grid values and mouse moves are disabled. A left-click or keyboard space bar press will drop the pin at the shown position. While you can move a pin from any lead group, the PCB pad and pastemask geometry will not be updated until the lead group is selected.

When positioning pins manually, plan on how to use the origin and grid functions before starting the move. Once the pin is in the proper position, a final Origin command with no parameters, O,  can be used to restore the original origin location.

### Lead Group Select

A Lead Group can be selected using the drop down at the screen top left or through the "L" text command. Lead groups are numbered from 1 to n, with 1 being the main group. Examples:

L2              Select Lead Group 2,
L1              Select Lead Group 1, the Main Group.

The selected group will be shown in red (top side) and blue (bottom side) colors with other groups shown in dim colors. Pins of either colors can be selected using the mouse and moved to new locations. If a pin from another group is moved, the PCB pad might remain at the previous location. Selecting that other group will cause the PCB pad to return to the pin. Someday we might fix that.

### Align Lead Group

A Lead Group can be aligned to an X Y location or another pin, The command "A P4 P1" will align (move) P4 to the P1 position. The first pin (pin 4 in this example) must be in a lead group > 1 and the lead group containing pin 4 must be selected. If an X Y value is passed, it is in reference to the Grid of lead group 1, the main group. For Example:

```
L 2              Select Lead Group 2,
A P4T P1B        Align P4 Top (in Lead Group 2) to P1 Bottom
O                Pop old origin from origin stack.

Note: P4 must be in Lead Group 2.
```

All pins in the selected lead group will be moved.