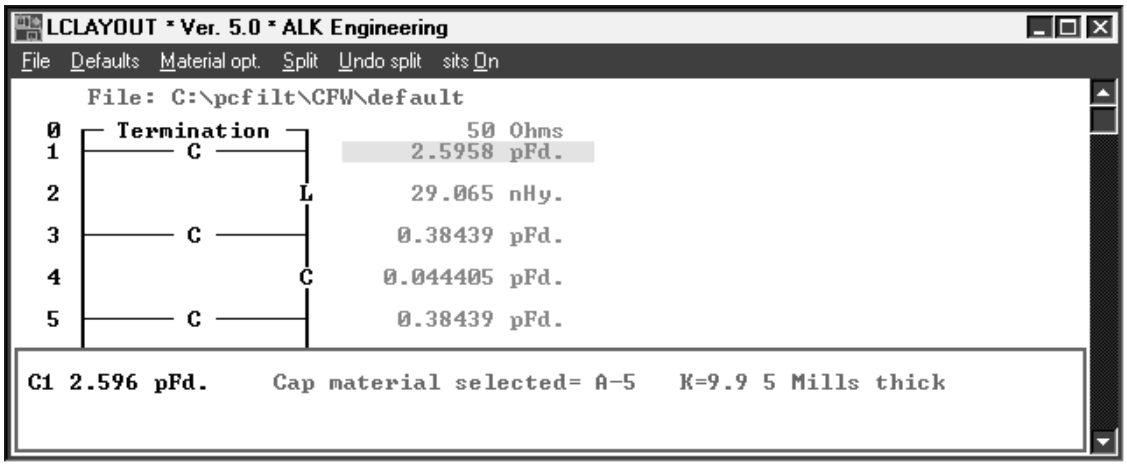
LCLAYOUT 5.0

LCLAYOUT is a program designed to be used for turning filter designs generated by PCFILT into parts lists and layout drawings for realization as miniature filters built using air wound inductors and single layer "chip" capacitors. The design files read into the program are the actual ".DZN" design files generated by PCFILT when a design is saved. S/FILSYN will also generate ".DZN" files using the >FILE command from the PASSIVE module. LCLAYOUT will attach the physical dimensions of each part to the file and write it back to disk on exit. This file may be read in again by LCLAYOUT or by PCFILT at a later time. LCLAYOUT can be used again to make changes to parts defined by it earlier. PCFILT will read in files containing part specifications but these specifications will be ignored and lost if PCFILT re-saves the file.

LCLAYOUT is intended to be used in conjunction with the Autodesk Generic CADD or AutoCAD drawing programs to produce layout drawings for final assembly of this type of filter. Because of the tremendous variety of filter topologies that PCFILT can generate, it is not practical for a computer to actually do the layout for you. The objective is to draw each part, to scale, on the drawing so that the user can manipulate each one as needed.

Installation

LCLAYOUT has no installation program and requires no hardware key. It may be installed on as may computers as you like. Simply copy all of the files into a directory (folder) of you choice from the distribution disk. Use the “Create shortcut here” feature of Windows from Windows explorer to create a desktop icon. When the program is invoked a typical display will look like this:



Loading a design file from PCFILT or S/FILSYN

The design to be realized in miniature form can be loaded into LCLAYOUT by using the File and Open options of the top menu. This will open a standard file dialog box. The program will automatically load the file DEFAULT.DZN at startup.

The display

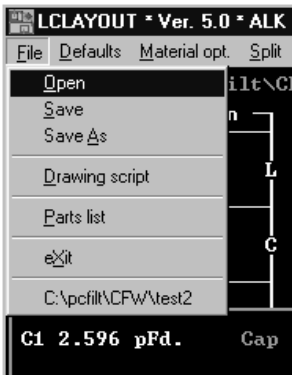
When the main window opens the schematic diagram of the filter will be displayed in the same format used by PCFILT except that a single part value will be highlighted by a blue background. The highlighted part may be moved by using the keyboard arrow keys or by clicking on the part value with the LEFT mouse button. Initially, all of the part values will be displayed in grey until they are define. After each part is defined it will brighten to white. The boxed in area at the bottom of the window displays the physical dimensions of the highlighted part once it is defined.

Picking the part to define

So that the physical dimensions of all the capacitors and inductors in a filter design can be defined in any order that the user finds convenient and so that changes can be made at a later time to any single part, a method of picking one part at random from the network to define was included. You can pick a single part for definition in several ways. The easiest way is to double-click the desired part value with your mouse or to single-click the highlighted branch value. The highlighted branch can also be selected by pressing the <Enter> key.

Top menu

*File



Open - Recall a design

Save - Save a design including all the part definitions.

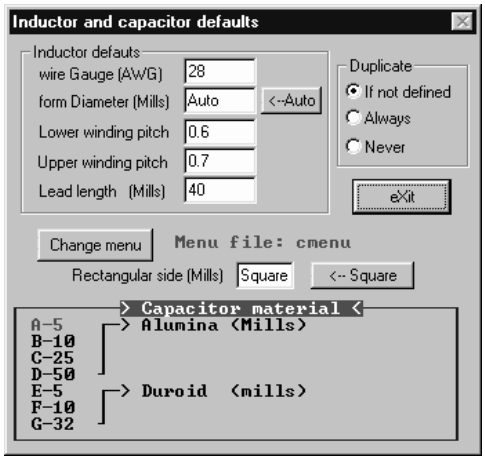
Save As - Save a design under another name.

Drawing macro - Writes a script file or "macro" that may be input by AutoCAD or Generic CAD which will draw all of the parts in the design to scale.

Parts list - This will generate a listing of every part in the design with the physical dimensions assigned to each.

Exit - Leave the program. If you have not saved the changes you have made a warning message will be displayed and the option to save and exit is provided.

*Defaults

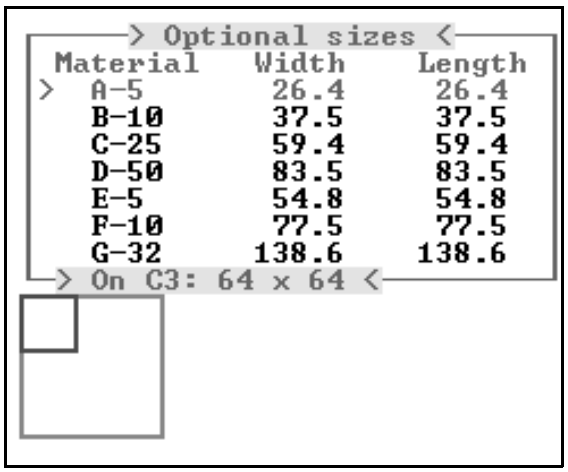


The Defaults dialog box is used to set all of the beginning parameters used to define the physical dimensions of the parts. This includes the substrate materials used for single-layer capacitors, the wire gauge used to wind the inductors and allows the user to determine how multiple parts having the same value within a design are to be defined. Most of these values can be overridden when each part is actually defined. These are simply starting values.

*sits On

Any coupling chip my be positioned on top of the shunt chip it is connected to. Selecting this menu option will allow the chip the highlighted coupling is to rest on to be specified. When this coupling chip is highlighted from then on an arrow will point to the shunt chip it is resting on. The drawing macro will draw a line between these two to identify the combination on the drawing.

*Material options



This will generate a tabulated display of all the sizes possible for the capacitor that is highlighted by the blue background. It will include sizes for each of the substrate materials in the menu file (cmenu.lcl) which are displayed on the Default dialog box. It will also list all the optional sizes for any single capacitor in the design if the RIGHT mouse button is clicked over its part value. The currently selected material is in green and the material currently selected for this chip is identified by the “>” arrow.

If a coupling chip is chosen and the chip it sits on is defined, the size of the chip it sits on is shown below the table. The background color will be blue if the highlighted coupling chip has a smaller area than the chip it sits on. If the area of the coupling chip is too large the background will be red. Square chips are assumed.

A graphic representation of a coupling chip and the chip it sits on is displayed below the table if a coupling chip is selected with the RIGHT mouse button. The coupling chip is in light blue. If the coupling chip is too large, it is shown in red. The material currently selected for this coupling is identified by the “>” arrow on the table of optional sizes.

* Split

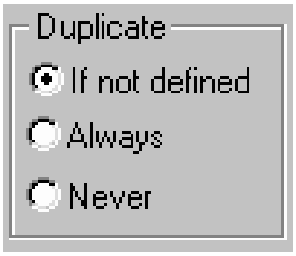
It is often necessary to split a component (usually a capacitor) into two or more parts in parallel to total the value you need or to get the physical dimensions of one that will be required to sit on top of another. This command will automatically split the selected component into two parts allowing you to specify the value of one while the other is computed.

*Undo split

This command will only be active after a part has been "Split". It will allow the "Split" to be Un-Done. This will only be possible until some other part or group of parts is changed. You can split or redefine the same group of parts several times and still be able to back up to the point before the first split on the first component. This is done by saving and recalling the entire design to a file (tmp_lcl.dzn). The file is saved when a "Split" command is given. As soon as some other component is defined, the design in the file becomes out of date and useless. The "Back up" option will then generate a warning message if you select it.

Operation

The keypad up and down arrow keys will move the highlighted part up or down. Any multiple branches will also be highlighted, in turn, as the up or down arrow key is pressed.



When the <Enter> key is pressed, the highlighted part is selected to be defined. If it is an inductor, the air wound inductor screen will come up. A capacitor will call up the chip capacitor screen.

Parts that have the same value throughout the filter may be defined as equal in size: Always, Never or only when other "equals" have not been defined. This selection is made on the Defaults dialog box.

Single layer chip capacitors

When specifying the physical size and material of chip capacitors, it is necessary to have access to several types if materials and thicknesses so that coupling capacitors may be made smaller than the shunt capacitors they will be required to sit on no matter what the actual capacity of each happens to be. Because every user has his own sources and favorite types of materials, the menu used to select them must be user definable as well as easily accessible while defining each part in the filter. The capacitor material is chosen form the Defaults dialog box. The items on this menu are defined in a disk file which is called in from disk when the program starts up. The default file name is CMENU.LCL. At any time, the name of this file can be changed allowing optional menus to be read in. These alternate menus my be made, by you, using a simple format that will be described later.

The capacitor material default menu

[Change menu] Menu file: cmenu

This button allows other menu files to be loaded. CMENU.LCL is the default menu file. The name of the file being displayed is identified. Selection can be made by pressing the key letter to the left of each material.

Rectangular side (Mills) [____] [<-Square]

If you would like to make all of the chip capacitors in the filter the same width, maybe to cut all of the chips from one long strip of substrate material, this dimension can be set here. Pressing the [Square] button will force all of the chips to be square in shape

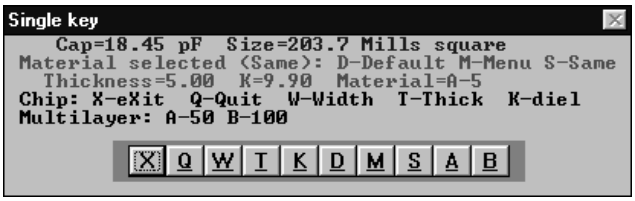
All of the menu items below these three are for different substrate materials and may be anything you define them to be in the menu file. You can make these menu files yourself according to the format described below. The default file name is CMENU.LCL. The CMENU.LCL file supplied with the program looks like this:

```
|    |<—— The left most 5 characters will appear in the parts list
|    | as your "part number".
|    |                                  |..... This area will not show ........

A-5    ┌—> Alumina (Mills)             :  9.9      .005
B-10   |                               :  9.8      .01
C-25   |                               :  9.8      .025
D-50   └                               :  9.9      .05
F-10   ┌—> Duroid  (mills)             :  2.3      .01
F-32   └                               :  2.3      .032

|                                      |   |       |
|                                      |   |       49Th character = Thickness of
|                                      |   |       (6Th tab-stop)  material (In.)
|                                      |  41st Character = Dielectric constant K
|                                      |  (5Th tab-stop)   of material.
|<—— Menu item description area ——>|
|                                     38Th character
└ Left most character is the highlighted "key" letter.
```

If you want a blank, comment or heading line to appear on your menu, simply begin the line with a dash (-) in place of a key letter, like this:

```
C-25   |                               9.8      .025
D-50   └                               9.9      .05
-        --- Duroid substrate ----                        <-- A heading line
F-10   ┌—> Duroid  (mills)             2.3      .01
F-32   └                               2.3      .032
```

When a typical capacitor is selected to be defined, the chip capacitor screen appears:



The dialog box displays the capacitor material selected on the top line and the value and physical size of the capacitor that has been “picked” for definition using this particular material on the second and third line.

*[X]-eXit [Q]-Quit

Press the <Enter> key or the [X] button to accept the dimensions shown and return to the pick-part schematic scroll function. Quit will exit without changing or defining the chip.

*[D]-Default

Selects the material selected from the defaults menu BEFORE entering the chip definition menu.

*[M]-Menu

Selects a different material from the defaults menu to use for this particular chip.

*[S]-Same

This option only is available when a chip has already be defined. It allows the same meterail

previously used to be used again. Use it to change the dimensions (aspect ratio) of the chip.

Special materials not on the defaults list.

*[W]-Width

One side of the chip can be specified. The other dimension will be calculated to get the desired capacity.

The next two commands will override the selected substrate material type and thickness defaults from the capacitor default menu and will defeat the identification of the material that would normally appear in the parts list if the <Enter> key is pressed. The identification become "<—?". These commands are best used to experiment to see what size chip capacitor would result if other materials were used.

*[T]-Thick

The material thickness may be changed with this command.

*[K]-Diel. K.

The dielectric constant may be changed with this command.

It is often impossible to make a capacitor in single layer form if its capacity is too large. In this event, a multi-layer capacitor can be chosen. These are assumed to be either 100 mills or 50 mills and cube shaped like those made by American Technical Ceramics (ATC) and other manufacturers.

Multilayer: [A]-50 [B]-100 cube

Air wound single layer inductors

The inductors in every L-C filter design must be given serious attention in order to ensure good performance. The inductors are the weakest link with respect to "Q" and insertion loss.

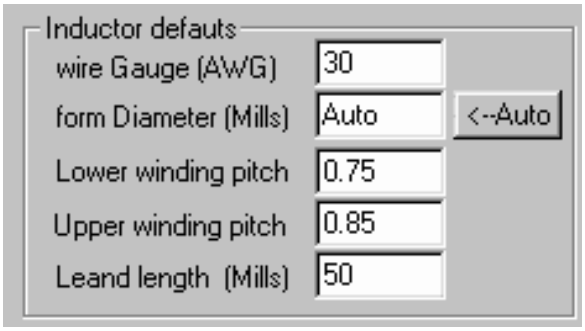
As a general rule, the bigger the physical size of an air wound single layer coil is made, the higher the "Q". The "Q" is also related to the turns spacing (turns per inch) and aspect ratio (length to diameter).

This portion of the program has been developed to design air wound inductors in such a way as to allow full control of these parameters. It is screen formatted to allow the effects on all other parameters to be seen when any one is changed.

A double iteration scheme is used. The first will find the exact number of turns required to give the required inductance with the winding data you suggest. Then the number of turns is changed to the nearest integer number and the length is adjusted to compensate. All results are then presented along with any warnings.

NOTE: The inductors designed by this program can be closewound and installed in your filter still closewound. This will cause your filter to be low in frequency when first tested. The very act of tuning the filter up to the correct frequency by spreading the turns will yield the correct turns / in that you have design for. Do not spread the coil in advance unless it is not possible to do otherwise.

Inductor defaults menu



wire Gauge (AWG) [30]
Sets the wire gauge.

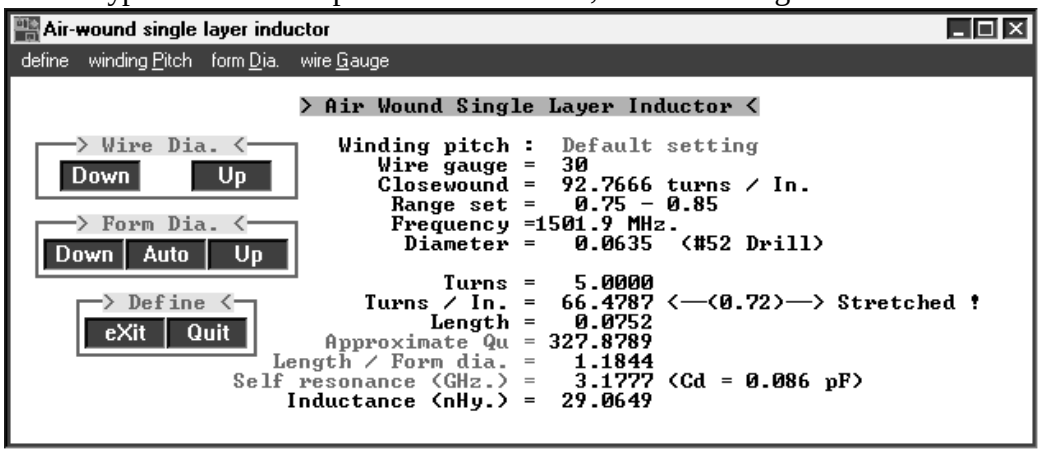
form Diameter (Mills) [Auto] [<--Auto]
Sets the diameter of the coil form or the winding mandrel. If you press the [<-Auto] button the (Auto) mode is turned on. In the Auto mode. The coil diameter is adjusted so that the length to diameter ratio is approximately 1.5 : 1.

Lower winding pitch [0.75]
Upper winding pitch [0.85]

These set the upper and lower winding pitch limits. A winding pitch factor if .75 mens 75% of closewound. If you specify 0 for either the upper or lower limit the initial range settings of .75 and .85 are reset.

Lead length (Mills) [50.0]
This sets the length of lead as drawn by the macro generated for the macro file. This is initially set to three times the wire diameter.

When a typical inductor is picked to be defined, the screen might look like this:

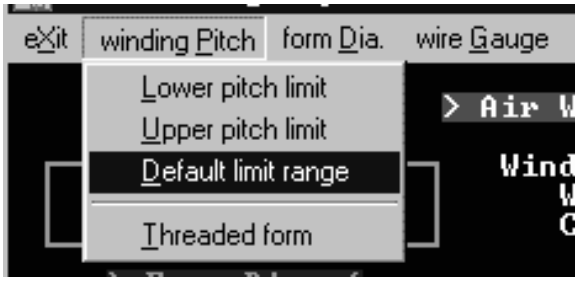


Overriding the inductor default settings

The menu options allow you to override the default settings displayed on the inductor defaults menu. This allows each individual inductor to be optimized separately.

In the example above, the default setting have caused the second iteration to "stretch" the coil longer than the .75 lower winding pitch (75%-of-closewound) limit set by the defaults. A slightly smaller diameter would allow the coil to be compressed back into range.

The menu options and buttons



*eXit

Accept the coil as displayed and return to the main window.

* winding Pitch

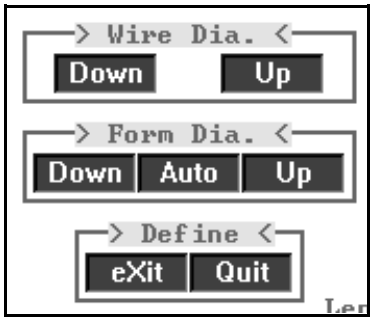
Winding pitch allows the upper and lower limits of the turns / Inch range to be adjusted or returned to the default range. The option to design for a threaded form is also provided.

* Lower and Upper pitch limit - Change Upper or Lower turns / inch limits

Allows the winding pitch, or turns per inch range to be changed. The first iteration will calculate the number of turns for a turn spacing midway between the extremes you specify. The default range is set on the inductor defaults menu. Closewound is the tightest winding possible, that is, each turn touches the one next to it. Specify the upper and lower limits as a factor of closewound. 50%-of-close wound would be specified as a winding pitch factor of .5. The turns per inch value for a closewound coil is shown for the wire gauge set, this corresponds to a winding pitch factor of 1.0.

* Form Dia.

Coil form diameter is specified in mills. The "form" can be a removable winding mandrel such as a drill bit. The form diameter is adjusted internally to give the inductance formulas used the wire center-to-center diameter needed. If you specify 0 (zero) for the diameter, the automatic diameter mode will become active. Auto diameter mode is turned off when you manually specify a form diameter. Auto mode will adjust the length to diameter of the inductor to get a 1.5 to 1 ratio before the second iteration is done to force an integer number of turns. This means that some manual adjustment to the diameter may be necessary. By pressing the [Down], [Up] and [Auto] buttons you can adjust the form diameter to standard drill bit sizes or to set the mode back to "Auto". Standard sizes are provided between numbered drill 1 to 60 and larger bits up to 1 inch in steps of 1/32 inch.



* wire Gauge - Wire Gauge or diameter

This command allows the wire diameter to be changed. Any number specified that is 1 or greater is taken to be a AWG number. The wire diameter is calculated internally. The maximum turns per inch possible (closewound) is calculated and presented. This closewound number is adjusted for the insulation thickness normally found on enamel or "poly" wire. The mouse can be used to press the [Down] or [up] buttons to step the wire gauge by 2 up or down.

The frequency displayed is that of the frequency assigned to the design file read in, or in the

case of a finite transmission zero (a notch) it is the zero frequency itself. This is the frequency at which the coil is to be used.

Inductor warning messages

There are several warning messages that will appear to help in the event a design is not reasonable. These are:

The LOWER limit on turns per inch has been exceeded, The coil has been stretched beyond the lower limit when the second iteration rounded the number of turns UP. The final winding pitch factor is displayed.

<--(.51)--> Stretched !

The UPPER limit on turns per inch has been exceeded, The coil has been compressed beyond the upper limit when the second iteration rounded the number of turns DOWN. The final winding pitch factor is displayed.

-->(.92)<-- Squashed !

If the iterations compress the winding pitch tighter than closewound, (which is impossible) this message will appear:

Tighter than closewound !

In the event the coil length is suspicious, these warnings may appear. These two should be considered less important that the unloaded Q.

Coil is short and "fat":

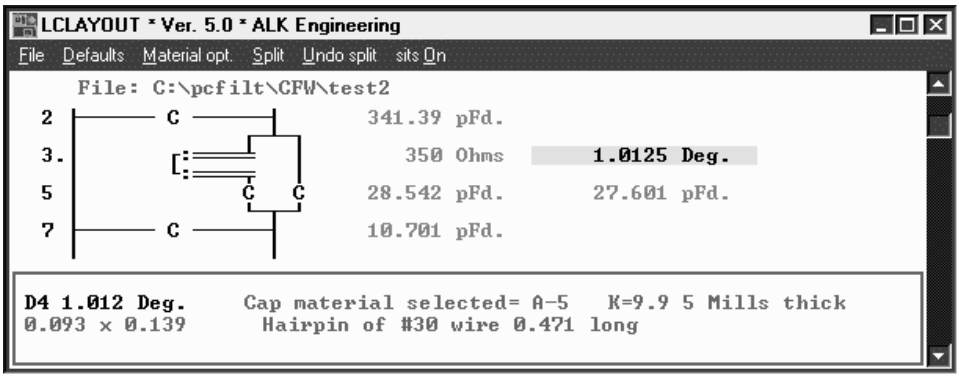
<-- Short ?

Coil is too long relative to its diameter:

<-- Long ?

Hairpin “stub” inductors.

Some inductors used in this type of filter are to small to be realized using normal coils. In this case a single loop if wire can be used. The PCFILT stub generator can be used to calculate the impedance and electrical length of a shorted open-wire transmission line stub (350 Ohms is suggested). LCLAYOUT will calculate the spacing and physical length of the wire used to form the hairpin. **IMPORTANT:** Note that the input file must have this type of inductor defined as a stub when it is read in. Make sure that all inductors that will become hairpins are in stub form before you begin. This is usually any inductance less than 10 nHy. If you recall the file into PCFILT after any parts have been converted to chips or inductors all your work will be lost when PCFILT saves the file! To change one inductor from a coil to a stub means starting over!

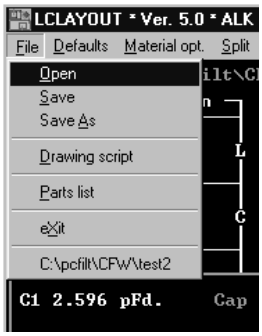


The branch to select is that of the stub **length**, not its impedance. This dialog box will appear:

The wire gauge to be used will be the same as that defined for normal inductors initially. The wire gauge used for hairpins may be changed here. The new setting will remain for all hairpin inductors.

Exiting with the results

When all of the parts have been defined and it is time to exit the program there are several options:



* Print list

At the bottom of the parts pick display, below the schematic scrolling window, the physical size and identification of each part is shown. This data can be sent to a printer or a disk file for all of the parts at once for use in making a bill of material. All of the capacitors and inductors are grouped separately. If a disk file is chosen, it will have the same file name as the design but with the extension ".6HC". This is in keeping with disk file the documentation scheme used by PCFILT. For example: DEFAULT.6HC.

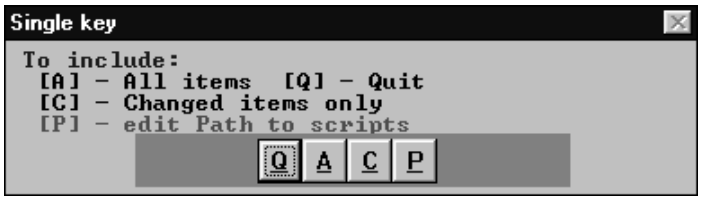
* eXit

This option will exit LCLAYOUT closing the Window. If changes were made to the parts definitions that have not been saved you are give the opportunity to save the changes. This will replace the active PCFILT ".DZN" file with the same design having all of the part definitions. It may be recalled again by LCLAYOUT to make changes to individual parts and saved again. It may also be read by PCFILT which totally ignores the part definitions that have been added by LCLAYOUT.

If PCFILT is used to change the design however, the newly saved file will be stripped of all the part definitions. You will have to start the part definition process all over again. DON'T SAVE IT from PCFILT unless you want to start all over!

* Drawing macro

This option will write a macro or "batch file" to disk that will draw each part using either the "Generic CADD" or "AutoCAD" drafting programs from Autodesk. All of the defined parts may be drawn, or just those that you have changed since reading in the design file. If the entire filter is to be drawn, the macro file generated will have the same name as the PCFILT ".DZN" file that was read in initially but will have the extension ".MCR" or ".SCR". If only the changed parts are to be drawn, the macro file name will be MOD.MCR or MOD.SCR.



After selecting the macro to draw ,the entire filter or just the modified items, this prompt appears to select which CAD package is being used:

Using AutoCAD

Each type of part is drawn to separate "layers" of the drawing. The parts that are associated with establishing the length of the filter are drawn in one row horizontally. These are the series connected inductors and capacitors connected directly to ground. Other parts such as coupling capacitors, which are normally placed on top of shunt capacitors are placed in a separate row below the first. Shunt connected inductors and "notch" section components are also drawn in the second row. Some hairpin inductors are drawn in a third row.

The parts are drawn onto specific drawing "layers" and in different colors as indicated below:

Layer#	Color	Name	Part type
0	white	--- not used ---	
1	Yellow	SHUNTS	Capacitors to ground.
2	Cyan	COUPLINGS	Series connected caps.
3	Green	COILS	All inductors.
4	magenta	JUMPERS	Jump wires are for you to draw.
5	white	HOUSING	The housing is for you to draw.

When the script file stops drawing, the current layer is left set to the JUMPERS layer since jump wires are normally the first additional items actually drawn. The first step is often to arrange the capacitors that are soldered directly to the bottom of the housing, then stack the remaining capacitors and inductors. It is assumed that the housing will be drawn last.

You may want to draw the filter housing as the first step and then move the shunt capacitors into it. In this case, you should first select the HOUSING layer manually. What you do first will depend on if the filter elements are to determine the housing size or if the filter must be built into a specific housing.