Advanced Circuits
PCB ARTIST LAYOUT SOFTWARE

PCB Artist **User Tips Guide**

PCB Artist Layout Software

This page is not intended to replace the Design Tutorial in PCB Artist, but to clarify important items for users. The Design Tutorial is found in the PCB Artist application itself. It is accessible from the main menu under Help>Design Tutorial.

Technical support is available at layouthelp@4pcb.com

Component Creation and Library Manager
Back to the Menu

Not finding the part you are looking for

The most effective search method is using this method:
1. From the main menu, go to File>Libraries.
2. Click on the Component Tab.
3. Use the Find button in the central row of buttons.
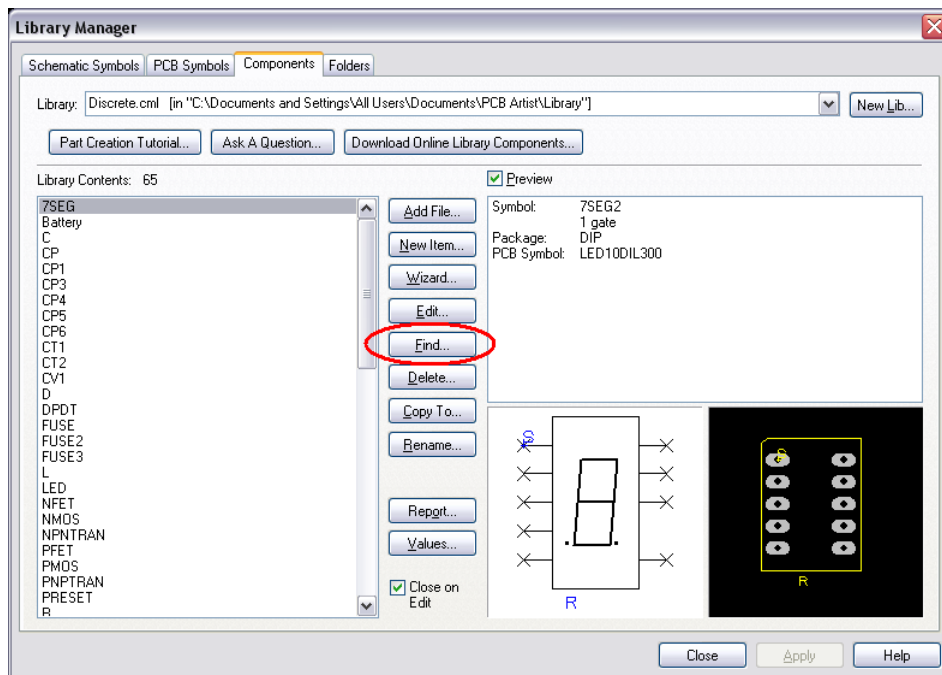
4. Next to the Name field, Change the drop down menu from Is Exactly to Contains.
   o It can also be very effective to search by pin number, unless the pin count is very small.
5. If you cannot find the part you are looking for in 5 minutes, you should just create it.
   o Almost any part can be created in 5-15 minutes in PCB Artist, especially with the PCB footprint wizard.
   o That new part can then be saved into a library you create and is organized in a way that best suits your preferences.

   PCB Artist has an online library available. This may be found: Here

Creating Components

There are millions of parts in circulation; it is not possible to pre-package a library with all parts in it. There is a downloadable tutorial available for creating parts.
- This link will download that PDF tutorial.
- Procedure for getting maximum benefit from the Part Creation Tutorial:
  1. The tutorial builds a part step-by-step. Build that part in PCB Artist along with the tutorial.

2. Refer to the overview of the PCB Symbol Wizard and Manual Part Creation in this document.
3. Note down any questions you have in the process.
4. Email the questions to layouthelp@4pcb.com when you have completed the tutorial.

Overview of the Schematic Symbol Wizard

1. Schematic Symbol Creation
   a. Click on the Schematic Symbols tab in the upper left of the Library Manager.
   b. Click on the New Lib button in the upper right. This will create a new Library for your part. Always put edited or created symbols of any kind in a library you created or you risk losing them in the future.
   c. Click the Wizard button in the center column.
      i. Click Next on the 1st page
      ii. Choose the Units and make the Precision 4 if you use Inches, Precision 3 if you use mm, and 1 if you use mils. Click Next.
      iii. Choose one of the schematic shape symbols. In the upper right, change the Origin radio button to Pin1. Click Next
      iv. On the Styles page, leave all at the defaults and click Next.
      v. On the Pins page, distribute the pins on each side of the symbol as you wish. Leave the Distance between Pins, Width across Symbol, and Length Of Pin Leg all at the defaults. Click Next.
      vi. On the Finish page, Name the symbol. Click on the Save Symbol to Library checkbox and change the dropdown to the library you created above. Click off the Edit Symbol New checkbox. Click Finish.

Overview of the PCB Symbol Wizard

1. Look up the data sheet of the part you need to create from the distributor or manufacturer.
   a. Warning: If the data sheet shows a PCB footprint (which is rare), you should create to that footprint exactly as specified by the manufacturer.
   b. Some types of specified footprints are easier to manually create (see below for manual part creation), than to create with the wizard.
   c. Parts with no specified PCB footprint are usually easier to create in the PCB Symbol wizard.
2. Go to the Library Manager, which is on the top icon bar. It is the fifth icon from the left that resembles a book.
3. Click on the PCB Symbols Tab.
4. Click the Find button in the middle column. Search for the exact name you wish to call the part you will create. No two parts in the library can have the exact same name. Confirm the exact name is not used anywhere else.
   - **IMPORTANT** - All parts you make or edit must be placed in a library of your own creation. They should never be placed in a library that came pre-packaged in PCB Artist. **When upgrading PCB Artist, all of the pre-packaged libraries will be overwritten and new or modified parts residing in them**. That directory can be backed up in advance of the upgrade, but the best practice to prevent data loss is

to create your own libraries to store your new parts and also parts you have modified.

5. Click the New Lib Button on the upper right. This will create a new library and you will be asked to name it.
6. Click the Wizard button in the center column of buttons.
7. In the wizard Start page, click the Next button.
8. In the Technology page of the wizard:
    a. You can choose to use a Technology file. Technology files save many types of settings from previous parts, such as Styles settings, Units settings, etc.
    b. Units: This can be independently set from the schematic or layout units that you will use in your design. This should be set to the same units that the part's data sheet.
    c. Unit Precision: This controls how many decimal places are shown after the whole number of any displayed or entered numbers.
        i. For example, if the Precision is set to zero and the Units are inches, a .250" hole would show as size zero.
        ii. If the Units are set to millimeters and the Precision is set to 1, the user can enter a .25 grid to in Settings>Grids but the system will change it to .3.
    d. Click the Next button.
9. On the Type page there are several packages to choose from. Choose SOIC for this example. Click the Next Tab.
10. On the Pads page you specify the dimensions of the part from off of the datasheet of the manufacturer.
    a. A representation of the part is constructed on the right. It adjusts as you change values.
    b. Notice that if you change the number of pads, the symbol changes on the right to match.
    c. The gold and gray colored rectangle in the bottom center of the page:
        i. Viewed from looking down upon it.
        ii. The gold is the pcb surface mount land.
        iii. The gray is the actual lead as it will sit on the surface mount land.
        iv. The "H" dimension controls the heel, which in PCB Artist is defined as the sides and heel.
        v. The "T" dimension controls the toe.
        vi. The "PW" and "PL" dimensions represent the actual size of the surface mount.
    d. Click Next to continue.
11. The Silkscreen Shape page is for specifying the details of the silkscreen. Click the Next Page.
12. The Placement Outline page is for specification of the part outline that will be placed on the Document layer. That layer can serve as an assembly drawing. Click the Next button.
13. The Finish page is where you save the part and specify its library. Make certain you are saving to a new library of your own creation.

Overview of the Component Symbol Wizard
1. Creating the Component. This will join the schematic and PCB Parts.
   d. In the Library Manager, click the Component Symbols tab.
   e. Click on the New Lib button in the upper right. This will create a new Library for your part.
   f. Click the Wizard button in the central column.
      i. Click Next on the Start Page.
      ii. Choose Normal Component and click Next.
      iii. Details page:
         1. Component Name will be the name of the symbol in the library.
         2. Package is the type of package, you can type one in if you do not like the ones listed in the drop down menu. These names do not have any particular attributes that impact the part of its function.
         3. Default Reference is the name the reference designator that will show on the silkscreen in the PCB layout.
         4. Component Pins is as the name suggests. In selecting the part, it will only show PCB Symbols with this pad count in a particular library.
         5. Number of Gates. For this type of part, just leave it at the default 1. Click next.
      iv. Schematic Symbols page, first choose the library the Schematic Symbol was saved to, then select that part on the left. Make sure the Preview button is clicked on. Click Next.
      v. PCB Symbol page works the same as the Schematic Symbols page. Make sure the Preview button is clicked on. Click Next.
      vi. Assign Pins page, click the Assign 1 to 1 button. Click Next.
      vii. Finish Page; choose the Component Symbol library you had created above. Make sure the Edit Symbol Now button is checked if you want to assign logic names to the pins, if you do not want that you must un-check this checkbox. Click Finish and the Component Symbol Editor will launch.
2. Manual Component Symbol editing.
   a. The Gates column will be with only one gate that is as is should be.
   b. The SCH Symbol Name column should be left as is.
   c. The SCH Symbol Terminal Name is where you can enter the logic names of the pins of the part. You do not have to enter them, but if you do not, they will not show on your schematic. On a BGA, it will likely be too crowded to display them.
   d. The next three columns are for pin association. They should have the exact same number in each row.
   e. The Net Class Name column should be empty.
   f. Go to File Save. Exit the part.
3. Making the logic names show in the schematic.
   a. Select the specific component, right-click and go to Properties.
   b. Click on the Pin Names and Pin Numbers checkboxes to see those items.
   c. Click OK to exit and save your change.
   d. The logic names of a pad can be seen in the Status Bar in the lower left frame of the application without displaying them in the PCB Design.

Manually create a connector or part (simple)

1. Look up the data sheet of the part.
    a. **Warning:** If the data sheet shows a PCB footprint (which is rare), you should adhere to that exact footprint in regards to hole sizes and any surface mount land patterns. Thru-hole pads should use a pad that has no less than .014" annular ring.
2. Most Data sheets use relative dimensions to define the part. We must manually convert all of these to absolute dimensions from of a point we can easily reference.

3. Go to the Library Manager, which is on the top icon bar. It is the fifth icon from the left that resembles a book.
4. Click on the PCB Symbols Tab.
5. Click the New Lib Button on the upper right.
    a. All parts you make or edit must be placed in a library you have created.
    b. When upgrading PCB Artist, prepackaged libraries will be overwritten (unless they are backed up in advance). To prevent data loss, the best practice is to make new libraries.
6. Click on the New Item button. This opens an editing screen similar to the Edit PCB screen.
7. Making your pad and hole sizes.
    a. Go to Settings>Units and set the units to the type used in the part drawing. In this example, use Inches with Precision set to no less than 3.
    b. Go to Settings>Styles.
    c. Click on the Pads Tab.
    d. Making the Component Pads Style:
        i. Click on the Add Style button
        ii. Give the pad a name. This is its Style Name, which you should note down. For this example, call this Pad1.
        iii. If no hole size is suggested, it is recommended that you make the hole .010" larger than the pin diameter. Since the diagonal of the square is given, we will use that dimension as the diameter. In this example, the four outer pins .060", so the hole size needs to be .070".

    iv. Make the pad (Width) no less than .014" (.36mm) over the hole size. In this example, the hole is .070" and the pad is going to be no less than .084".

    v. Click on OK to add the Style.

e. Creating the center hole style

    i. Click on the Add Style button

    ii. Give the pad a name. This is its Style Name, which you should note down. For this example, call this Pad2.

    iii. For this example, the pin size is .050", so the hole size should be .060".

    iv. Make the pad width at least .074" for this pin.

    v. Click OK to add the style.

f. Adding the four pads.

    i. Go to Add>Pad, but before clicking to add, type the <S> key to select the style, (Pad1), then select OK.

    ii. Then, right click and select Add Multiple Items. Since the four pads are symmetrically spaced apart, we can input the proper X and Y distances from the centers of the pad. For this example they are spaced .200" from center to center per the fabrication drawing.
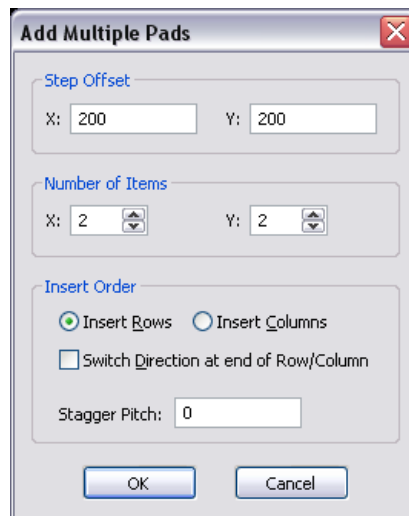
    iii. Place the pads in the design space. Type the <Esc> key to exit out of the Add Pad command. You can zoom in or out to adjust for a proper view of the pad.
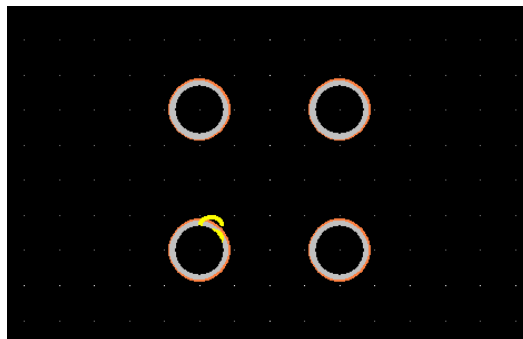
g. Setting up the grid to add the center pad.
   i. Right click on the lower left pad and select Origins>Set System Origin at Item.
   ii. Now, we need to set the grid up so that it will be easy to place the center pad in the correct place.
      1. Go to Settings>Grids.
      2. Set up the grid so that it is at 50mil intervals and click OK.
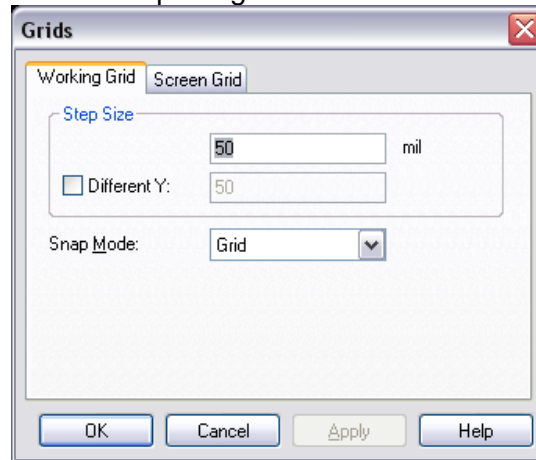
h. Adding the center pad.
   i. Since the grid is setup so that the center pad ill be placed directly in the center of the four outer pads, go to Add>Pad.
   ii. Before placing, type the <S> key and choose the center pad style (Pad2). Then place it in the center of the four pads and type the <Esc> key so that you will not continue to add any more pads.
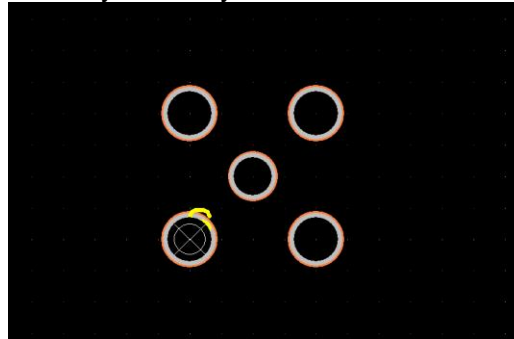
8. Adding silkscreen outline.
   a. Go to Add>Rectangle (or any other shape). Type the <L> key and select the top silkscreen layer to create a silkscreen outline for your part.
9. Save the PCB Symbol to the Library you have created for it and click OK to save and exit.
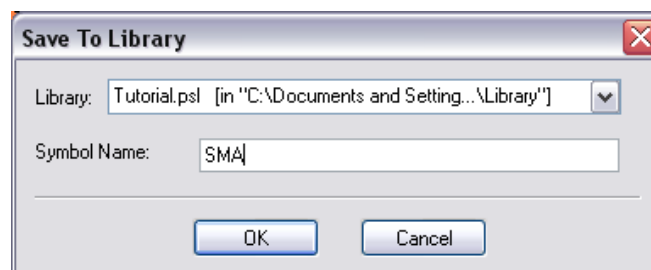
Manually create a connector or part (complex)

There are no connector templates in the PCB wizard. This is the process to create one. There are many other types of parts that are so different in footprint variation that they cannot have a common template in the PCB symbol wizard. These must be created manually.

1. Look up the data sheet of the part.
    a. **Warning:** If the data sheet shows a PCB footprint (which is rare), you should adhere to that exact footprint in regards to hole sizes and any surface mount land patterns. Thru-hole pads should use a pad that has no less than .014" annular ring.
2. Most Data sheets use relative dimensions to define the part. We must manually convert all of these to absolute dimensions from of a point we can easily reference.
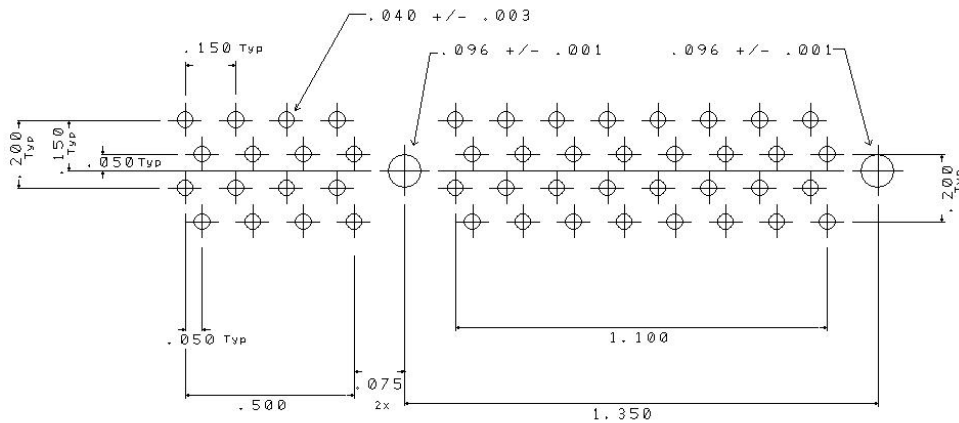
   a. Print out the data sheet, it is recommended to blow up the dimensioned area. Use a pencil to make the conversions and note these absolute dimensions on the print-out as seen in the example below (the notes are in red).
   b. Choose an easily accessible point on the part as the origin from which all absolute dimensions will reference. In the graphic example below, the indicated hole will serve as the origin and all manually converted absolute dimensions referencing it are in red.
   c. You must always determine the x and y location of each mounting hole and also the lower left component hole of each grid of component holes.
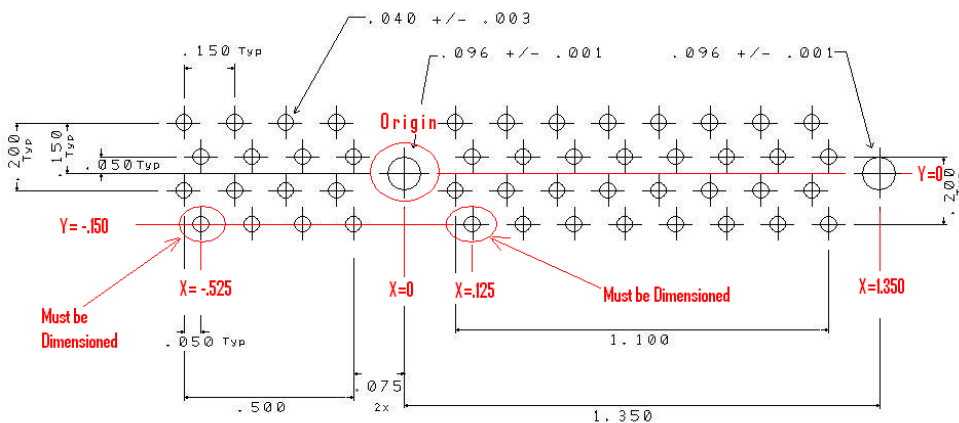
3. Go to the Library Manager, which is on the top icon bar. It is the fifth icon from the left that resembles a book.
4. Click on the PCB Symbols Tab.
5. Click the New Lib Button on the upper right.
   - All parts you make or edit must be placed in a library you have created.
   - When upgrading PCB Artist, prepackaged libraries will be overwritten (unless they are backed up in advance). To prevent data loss, the best practice is to make new libraries.
6. Click on the New Item button. This opens an editing screen similar to the Edit PCB screen.
7. Making your pad and hole sizes.
   a. Go to Settings>Units and set the units to the type used in the part drawing. In this example, use Inches with Precision set to no less than 3.
   b. Go to Settings>Styles.
   c. Click on the Pads Tab.
   d. Making the Component Pads Style:
      i. Click on the Add Style button
      ii. Give the pad a name. This is its Style Name, which you should note down. For this example, call this CompPad1.
   iii. Make Drill the finished hole size suggested by the manufacturers drawing. In this example .040"
      iv. Make the pad (Width) no less than .014" (.36mm) over the hole size. In this example, the hole is .040" and the pad is going to be no less than .054".
      v. Click on OK to add the Style.
   e. Creating the Mounting Holes Style:
      i. Click on the Add Style button
      ii. Give the mounting hole a name. This is its Style Name, which you should note down. For this example, call this MountingHole1.
      iii. Make Drill the finished hole size suggested by the manufacturers drawing. In this example .096"
      iv. Make the pad (Width) the same size as the mounting hole, .096" in this example. The pad exists as a presence on Top Copper for other features such as Pour Copper to avoid. In manufacturing, the pad will be removed.
      v. Click off the Plated checkbox.
      vi. Click OK to save and exit the dialog.
      vii. A warning will say that the drill completely removes the pad.
      viii. Click OK to save and exit the Styles dialog
8. Place the Mounting holes.
   a. Placing the Origin.
      i. Because the datum is a mounting hole, we must place them first. The origin position defaults to the first pad placed.
      ii. Click Add Pad, drag the cursor to the work area, but do not add it yet.
      iii. Click the <s> hotkey to choose the Style of pad to add. Choose the mounting hole you had made from the list.
      iv. Add the pad.

     v. Change to Select mode by clicking the white cursor arrow far right icon on the top icon bar. The icon looks like the cursor.

     vi. Select the pad, right-click on it, and choose Origins>Set System Origin At Item (NOT "At Cursor") from the right-click context menu.

       1. All dimensions displayed or entered in the design are now based off the center of this pad.

  b. Placing the second mounting hole.

     i. Click Add Pad.

     ii. The Style will be the same as the last one used, so it's not necessary to change it.

     iii. Place it anywhere.

     iv. Change to Select mode by clicking the far right icon of the top icon bar. The icon looks like the cursor.

     v. Select the pad that was just placed, right-click over it and go to Properties.

     vi. In Properties, Type its destination X and Y axis coordinates in the Position fields. In this example, those coordinates are X=1.35, Y=0.

     vii. Click OK. The mounting hole will move to those target coordinates.

9. Placing the grid of component pads.

  a. Go to Settings>Grids. Click on the Working Grid Tab.

  b. Change the first textbox of the Step Size section to the X axis coordinate of the lowest component pad in the far left. In this example, that is .525" (the true coordinate is -.525", but the grid system does not accept negative numbers and a negative number is not necessary).

  c. Click on the Different Y checkbox. In that blank type in the Y axis coordinate of the lowest component pad in the far left. In this example, it is .150" (the true coordinate is -.150", but the grid system does not accept negative numbers and a negative number is not necessary).

  d. Click OK to accept the changes and exit the Grid dialog.
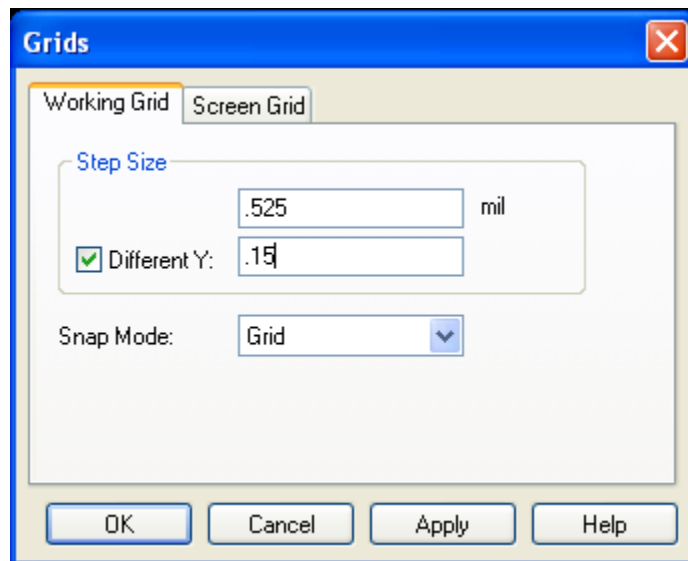
e. Click on Add Pad. Drag your cursor to the work area, but do not place any pads.
f. Right Click and choose Add Multiple from the right-click context menu.
g. The Add Multiple dialog window:
   i. In the Number of Items section, we must set how many pads we are adding in the X and Y.
      1. In the X, we are setting 4 component pads (for this purpose, ignore the stagger on every other row). Set the X to 4
      2. In the Y we are also using 4 in the Y axis. Set the Y to 4.
      3. Many options on this screen that were grayed out are now editable.
   ii. The Step Offset Section.
      1. The X is the center-to-center distance between pads in the X axis. In this example, enter .150" in the X.
      2. The Y in this example is .100". Ignore the shift offset for now. In this example enter .100" in the Y.
   iii. The Insert Order Section.
      1. In this case, make sure the toggle is set to Insert Row. This is due to the fact that every other row is staggered. (If changed to Insert Column, then every other column would be staggered.)
      2. Stagger Pitch is to establish how much the 2nd row is staggered from the 1st. If set to Insert Row, this is create a stagger (or indent) an X axis dimension. If set to Insert Column, it would be a stagger in the Y axis dimension. In this case the stagger is -.050" (the negative number is allowed and necessary). The negative number means the 2nd row will start further to the left than the 1st row. If the drawing showed the second row starting to the right, it would be a positive number (.050"). Enter -.050".
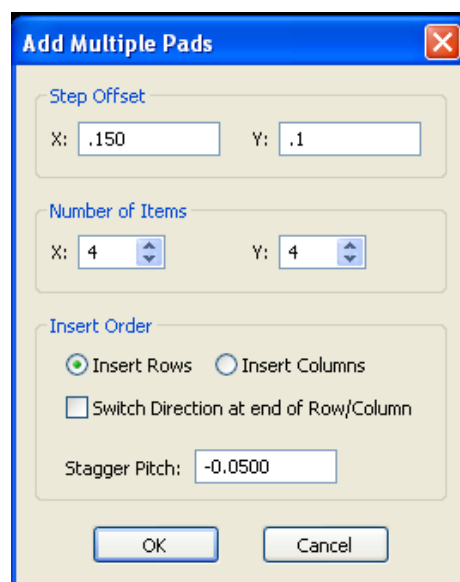
         iv.   Click Ok to save your settings and exit that screen.
- h. Click the <S> hotkey and choose the appropriate Style for this pad. In this example, use CompPad1.
- i. Now move your cursor until it is at the correct position. Use the Status Bar coordinates in the lowest left frame of PCB Artist to ensure you are at the correct grid coordinates. Then click once to add that grid of component pads.
- j. Add the second grid of component pads using the same process as described above, with these exceptions:
  - i. Go to Settings>Grids and change the grid X to .125" and leave the y as offset at .15.
  - ii. Change the number of items in X from 4 to 8.
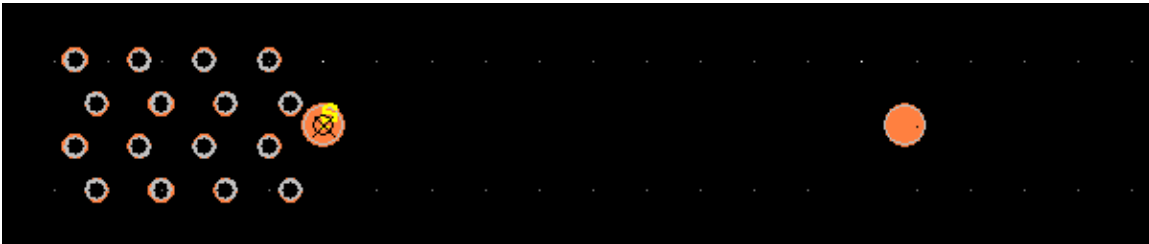
- k. Confirm the Dimensions are correct.
  - i. Find each dimensioned pad, right click on it and choose Properties from the right-click context menu. Confirm that the coordinates displayed in Properties are correct.
  - ii. Use Origins>Set System Origin At Item to change your origin to confirm that the part data sheets original relative coordinates are correct.
  - iii. **Warning:** Confirm every relative coordinate on the data sheet.
    1. This verifies:
       - a. That the conversion to relative coordinates was done correctly.
       - b. That nothing was incorrectly typed in.
       - c. That no assumptions or oversights occurred during the entire process.
    2. **Be aware** that occasionally a manufacturer's data sheet is incorrect, and the numbers do not add up. At this point the part manufacturer would have to be contacted for the correct dimensions.
10. Legend Edits.
    - a. Go to Settings>Grids and change your grid to a .050" grid. Click off the Different Y checkbox and click OK to save the changes and exit the dialog.
    - b. Click on Add Shape Rectangle
    - c. Hit the <L> hotkey and change your layer to the Top Silkscreen (if it isn't already on the Top Silkscreen).
    - d. Use Add>Shape Rectangle to add the legend outline of the part by clicking in the lower left to start the rectangle and double-clicking on the upper right corner.

11. Go to Add>Reference Origin and Click on any open area. The "+R" is a place holder for the part's reference designator. The actual reference designator text "Conn" would be applied when editing the Component.
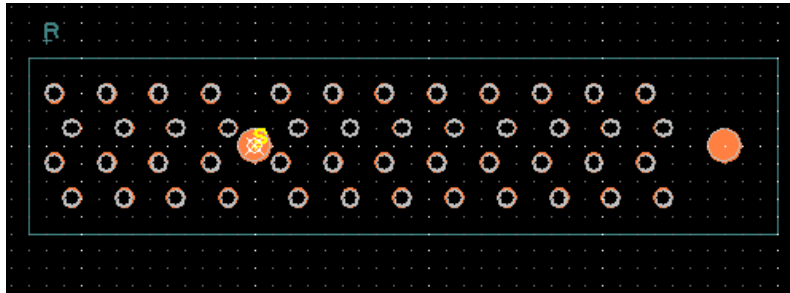
12. Adding the Placement Outline.
    a. The Placement Outline is a box, often a bit bigger than the Legend outline.
    b. Add it in the same manner as the Legend.
    c. This outline will go on the Document1 layer (or the Document 2 layer in some cases)
    d. The Document layers are usable for Assembly Drawing layers, and the outline can serve that purpose, or can be used for many other purposes.
13. Changing the Part Datum
    a. Find Pin 1 on the part Data sheet. In this example, it is the pin in the upper left.
    b. Select that corresponding pad on your part, right click on it and choose Origins>Set System Origin At Item. That pad is now the 0,0 center.
    c. Select the "+S" symbol that is the Symbol Origin. You many need to use the <n> hotkey to cycle through the different objects at that location until it is selected.
    d. Right click and choose Properties from the right-click context menu.
    e. Change its Position to 0,0. The Symbol Origin must always be on Pin 1.
        i. It is recommended to always put the System Origin on pin 1 so that the pads on the PCB Grid. This make it much easier to route tracks to the pads.
        ii. Some believe that placing the System Origin at the middle of the PCB Symbol helps CPL file (component position file) generation. The System Origin has no affect on the determining the center of components. Placing the System Origin at the center will often cause all pads of the component to not be on the PCB grid.
14. Renumber Pins
    a. Pins are automatically numbered as they are added, including mounting holes. So when we placed our mounting hole origin for this part, it became pin 1. We must change this.
        i. To View the Pin Names more clearly, go to View>Colors. Click on the Other Items tab of the Colors Dialog box in the upper left. Click on the Pin Names Checkbox.
    b. Go to Edit>Renumber Pins.

c. Click OK in the Renumber Pins dialog box to start numbering pins.
     i. Notice that in the Status Bar on the lower right application frame is now in Renumber Pins mode.
     ii. You will also notice in the Status Bar the "Next Number" box indicates the number 1. This is the number that next pad you click on will be changed to.
d. Click on pin 1.
e. Click on each pin, in sequential order, until all the pins are numbered.
f. Lastly click on each mounting hole, they also must have a pin number.
g. Hit the <esc> key to exit Renumber Pins mode.
15. Save the PCB Symbol to the Library you have created for it.
a. Make certain that you are using a unique name that is not used anywhere else in the library.
b. You must add this new symbol to a Component Symbol to add it to any design, regardless of whether it is being added to a schematic or a PCB design.
16. Component Creation
a. Any part that contains mounting holes cannot be created with the Component Symbol wizard.
b. Use the New Item button in the Library Manager's Component tab to create any components with mounting holes.

Creating Gold Fingers and Edge connectors
Gold edge connectors have to be created as a component, similar to a connector. This is a three step process. First make the schematic symbol, then the PCB Symbol, then join the two and associate their pins in creating a Component. On the instructions below, we assuming you're familiar with Part Creation Tutorial and the PCB Artist Tips & Tools web page.
1. Create a Schematic Symbol with all the pins for the top and bottom of the connector.
2. Create the PCB Symbol manually. As far as creating the PCB symbol is concerned, you first have to determine the dimensions of the fingers and the pitch from center to center. The basic steps are touched on in the FAQ document under the topic "Manually Create a Connector or other part", here are the differences:
   • Instead of making drilled pads, these will be surface mounts.
   • Use the Add>Add SMD tool and use the Add Multiple feature to add the contact pads, very similarly to how its done in the FAQ.
   • Then highlight all of them, Edit>Copy to copy them, press the <L> key to change your working to the bottom side, then Edit>Paste to paste the second row on the back side.
   • Pay special attention to the Renumber Pads section. The pads are numbered by default in the order they are added, so you will likely have to change them.
3. You will then have to create your component. The FAQ topic "Overview of the Component Wizard" will show you how to do that and also how to map the pins. The Part Creation Tutorial also has some information on this.

The following is an explanation of how the gold fingers are defined per the PCB Configuration (Settings>PCB Configuration>Board Requirements):
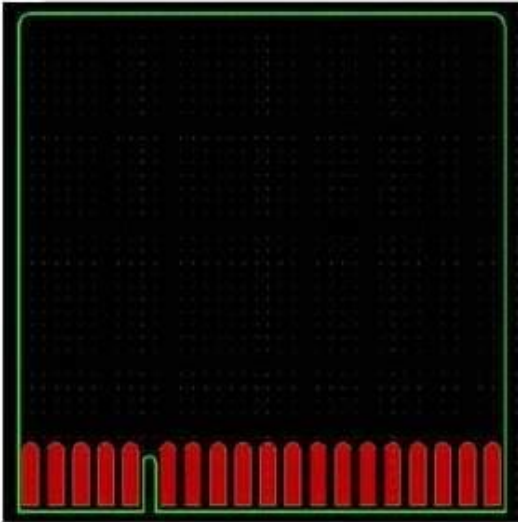
1 edge connector   2 edges connector
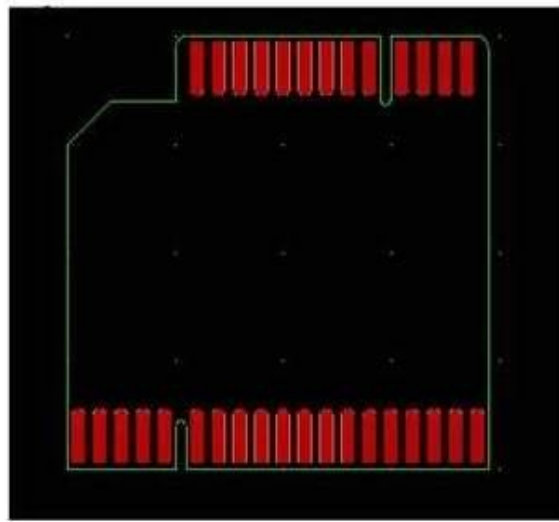
Creating a Circular Part/Circular Placement

For Circular placement of pads within a component, one will have to take several extra steps to set up the component. This will assume that you have gone through the component creation tutorial found under Help>Tutorials.

This footprint will have to be created manually in the library. This example is for a "Tube Socket" part, but the concepts found here may be used for any circular parts placement.

In the Library Manager, on the PCB Symbols tab click "New Item". One will be presented with a blank screen. Go to Settings>Styles. On the pad styles tab add a style. Make it the size of the drill hole indicated by the manufacturer, and for the pad width add an additional .014" or .3556mm (or more) for the annular ring.
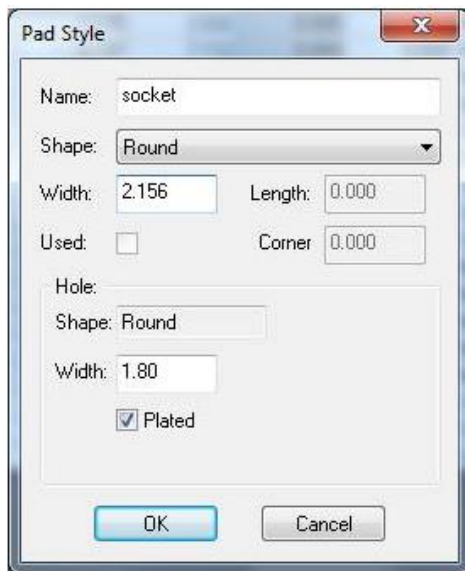
Set the system origin then center a circle on it. Use Add>Shape>Round for the reference line. The symbol origin was also added to the systems origin.
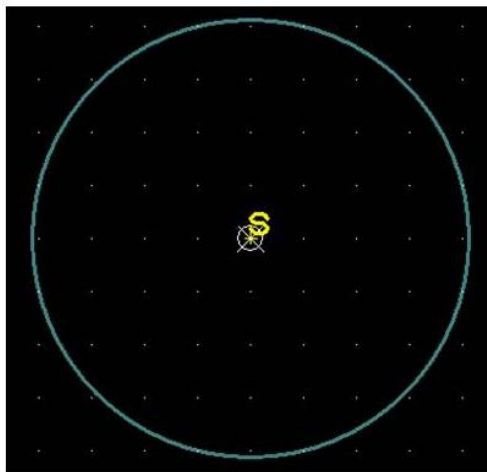
Add a pad (Add>Pad) or use the add pad icon. Hit the "S" Hotkey to change the style.

Place the first pad by coordinate. This can be done by right clicking and selecting the option, or by hitting the "=" key

Place the first 2 pads, one directly above 10.5mm on the y and below -10.5mm. Put a pad in the center also.
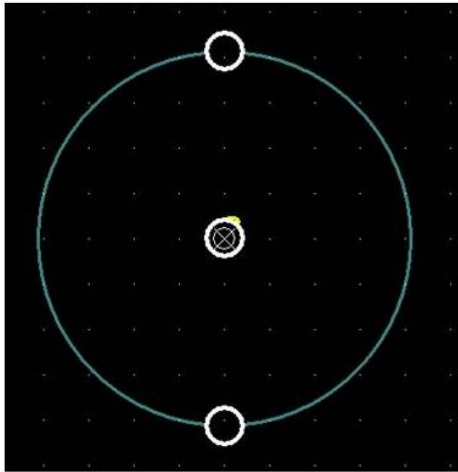
Select the outer pads first and then the center pad. Right click and select Rotate by, and put in the 36 degree value.

Add two more pads as above, but rotate these 72 degrees, using the center pad again as an anchor (remember to select the center pad last). Rotate in increments of 36 (36,72,108,144)

When done it should look like this:
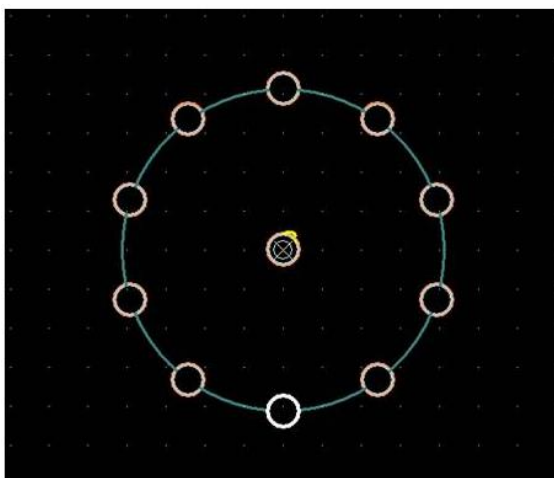
Remove the center pad, and the extra pad on the right. Also, right click on each pad and select Properties. Here one may change the pins for pin mapping.
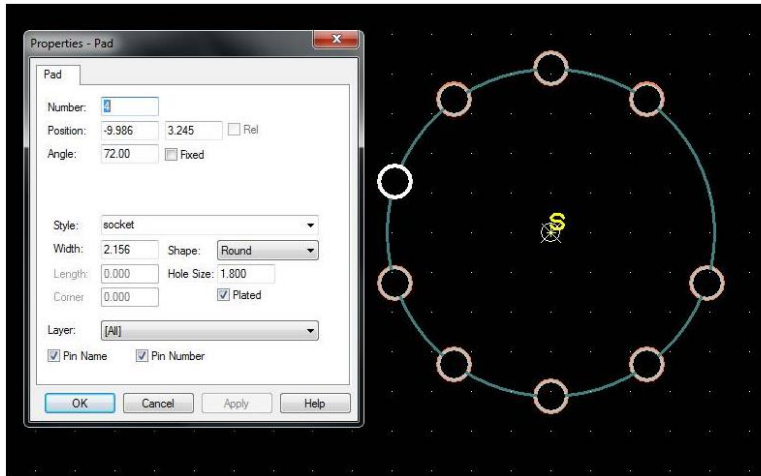
Once finished save the foot print off in your own personal library.

All components should be created top side. One may place a component on the bottom side by pressing the "F" hotkey when drawing out their PCB.

Another possible option would be to create this component with the CAN footprint within the PCB Symbols Wizard. One will be given the option to edit the footprint at the end to remove extra pads, renumber pads, and change the silk outlines.

Verifying Component Footprints
Often when users have created a footprint they may need to verify it.
The first step would be to review the data sheet for the footprint.
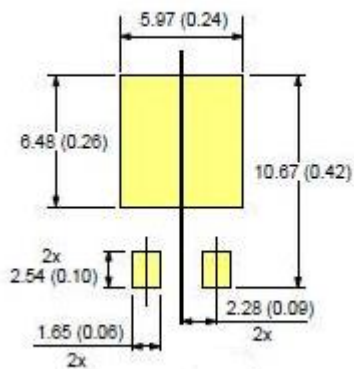
The first thing to do is check the pad size. Place the part onto a blank PCB. Turn off the soldermask layers (these are clearances for your pads, not actually your copper pads). To turn off the soldermask layers go to the interactive bar (F9) and uncheck them on the layers tab. Highlight a pad and right click, select properties. The properties of the pad will appear:
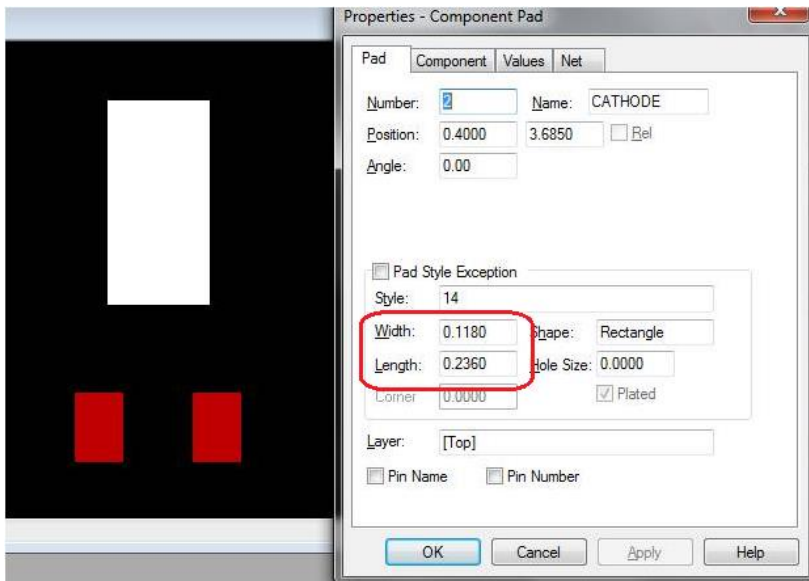
The largest pad is .118 x .236". The documentation shows this pad should be .24" x .26". The largest pad size is incorrect.

The smaller pads are .06" x .1" in the documentation, and .055" x .08" in the drawing (also undersized).

Next, use the measurement tool . This also appears in the menus under Tools>Measure.

The smaller pads should appear under the larger pad with a .06" gap between them. Note in the picture below the measuring tool is on the Gap Tab:
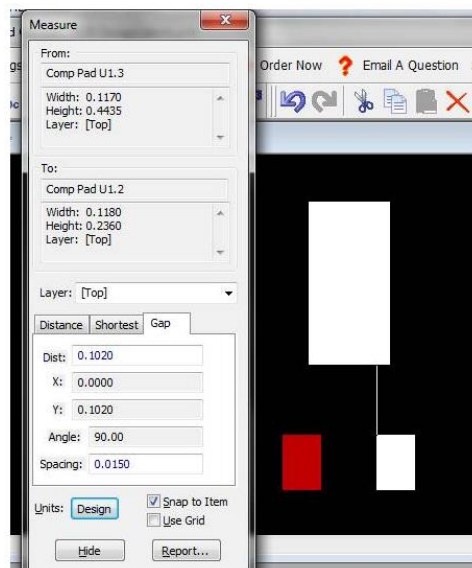
The above shows the pads are .102" apart not .06".

Next lest select the spacing for the two smaller pads. These can be measured center to center. It shows in the diagram both pads are .09" from the center, so center to center measurement should be .18" apart. These show .136" apart. Note the measurement tab is on Distance, which will show center to center spacing.
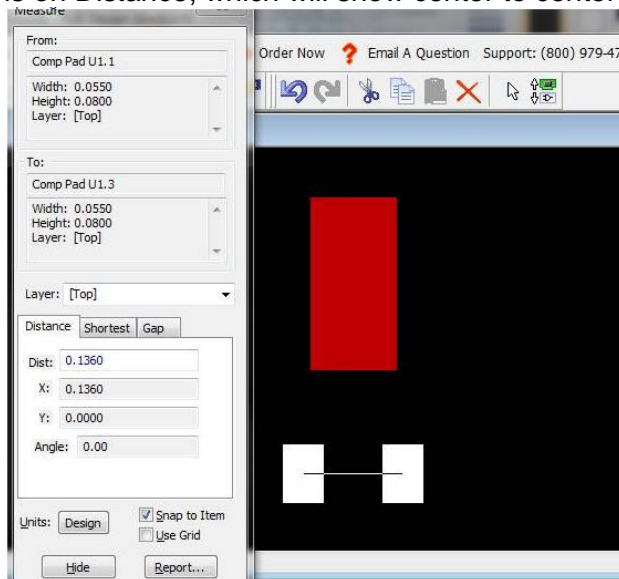
The final test, once the part is corrected is to print the part out at 100% size and place a sample part on it. It's a good way to see if there are any glaring issues with the footprint.

Unusual soldermask clearances

This is an explanation of what may be needed if a soldermask clearance needs to be manually created do to a unique shape.

Here is an example of a unique soldermask:
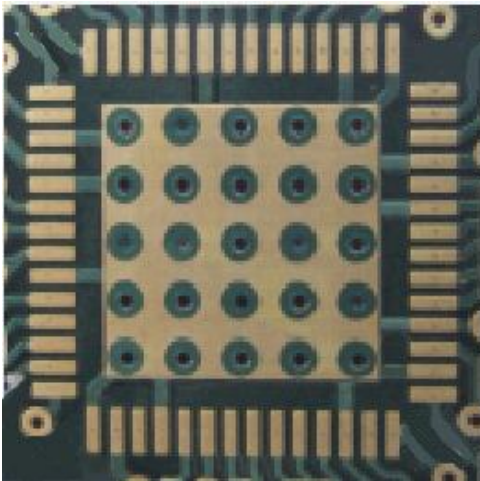
This layout is difficult to achieve in the software as the vias will share the soldermask clearance of the thermal pad below them.

As shapes only can appear solid with no holes missing, one will have to work their way around this. This is a basic example of what one can do. We are going to create a clearance out of several clearances to allow the pads to be covered. To do this one will have to manipulate the soldermask outlines for the desired affect by double clicking, breaking and moving the lines around. At the end one may use the Arc option when right clicking on a line. Bare in mind, the central pad and the thermals will have to be added AFTER the part has been placed on the board. There is not a way around this as the central pad created will have full soldermask relief if part of the actual component.

In this case, start with the thermal pad only appearing on the Top Copper. This will remove the soldermask clearance. For the vias use Add>Via. To set the soldermask correctly for these vias, one needs go to Settings>PCB Configuration>Layers. Check the Cover Vias box. This will work for only vias, not regular pads being used as vias. If done correctly this set up will not have any soldermask relief (right now). In the following examples copper is indicated in red, Vias in yellow and soldermask relief in blue-gray.
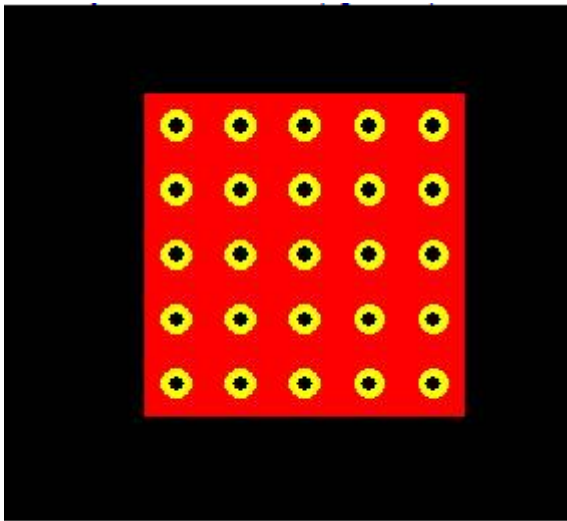
Now go to Add>Shape>closed shape. Carefully trace on the Top soldermask a shape that looks like this:
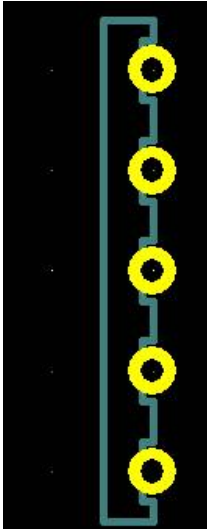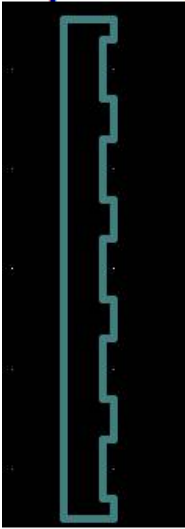
The same with just the vias present:

Right click on a straight edge, and select Arc. I used 140 for this example arc.
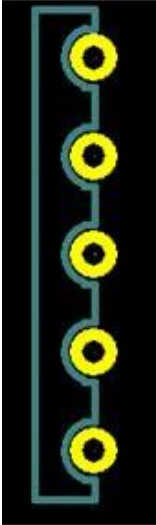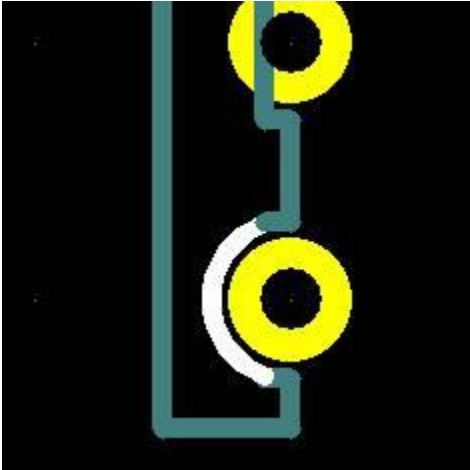
Repeated:

Right click on it and select properties. Click the filled box. Copy this outline out 5 times.
The last time rotate it 180 degrees.
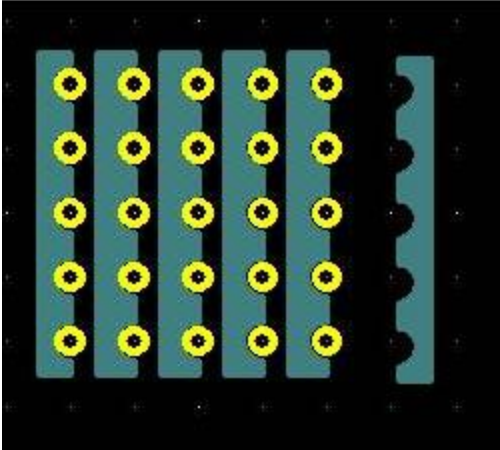
Copy the other side back over 4 more times and manually place the clearance.
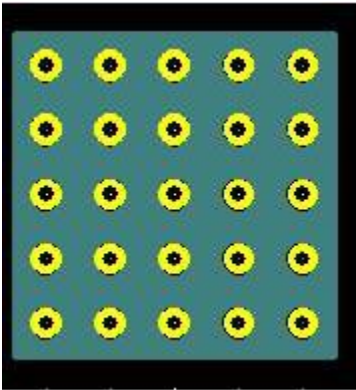
The above is the complete soldermask clearance.

What it looks like with the copper:
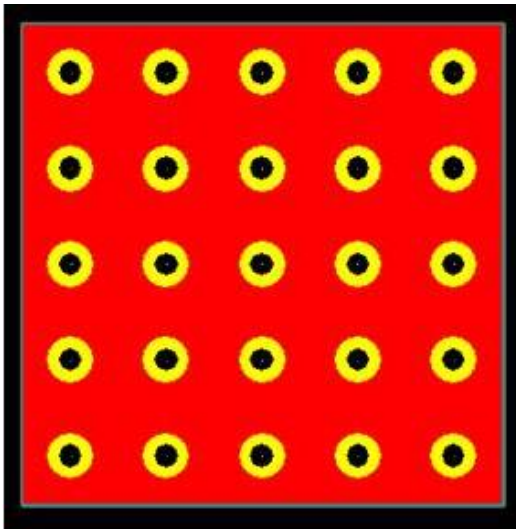
Verify the soldermask relief using the interactive bar:
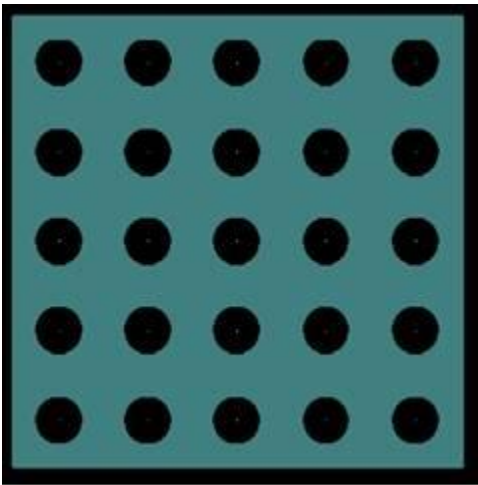
Finished hole size

Always research the manufacture's part and use what they recommend.

- In the event that the manufacturer provides no information and all other information sources are exhausted, the hole should be make at least .010" over the thickness of the lead. This is intended as a rule of thumb. The burden of research and verification that the rule applies to a particular design scenario is upon the designer.

Pad size

The minimum requirement is a pad that is .014" over the hole size. So for a .020" hole, make at least .034". For vias, the minimum required pad is .01" over the hole size.

Mounting holes

You can make them in two different ways:

1. The Quick way:
   a. Use Add pad to place the pads.
   b. Use <Ctrl> to select all of the pads to be changed.
   c. Right-click on any one of the pads and select Properties from the right-click context menu.
   d. Change the Drill to the required finished hole size.
   e. Change the Width (which is the pad width) to the same size as the drill.
   f. Click off the Plated checkbox.
   g. A warning will say the drill removes the pad, click OK.
   h. This method will not give a Style name to these holes and will make it difficult to group select the all of them by the Style name. This technically puts them into the "Unnamed" Style group with many other shapes making them tedious to have to select by group in the future.
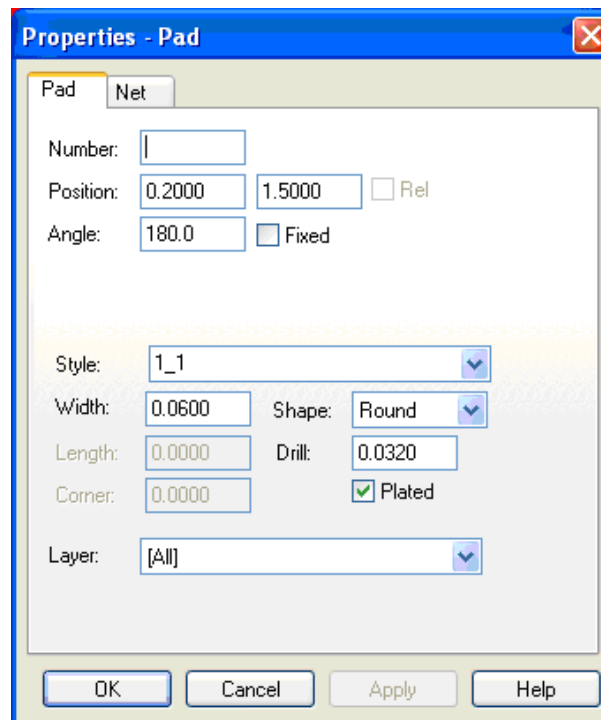
2. The Best Practices way:
   a. Go to Settings>Styles.
   b. Click on the Pads tab.
   c. Click the Add Style button.
   d. Enter a Name by which the new style will be referenced.
   e. Enter the Drill Size.
   f. Enter the Width (which refers to the pad width).
   g. Click off the Plated checkbox.
   h. Click the OK button to save and exit.
   i. An error message will say the drill removes the pad, click OK.
   j. Click Add Pad and drag your cursor over the work area, but do not yet add the pad.
   k. Click the <S> hotkey (or right-click and choose Change Style).
   l. In the dialog box that appears, choose the name of the style you created.
   m. Add the Pad(s).
   n. The advantage to creating a style is:
      1. You only need to go to Settings>Styles to edit all the holes in the group at one time in the future.
      2. You have the ability to select and display all items using the Style in the Goto tab of the Interaction Bar.
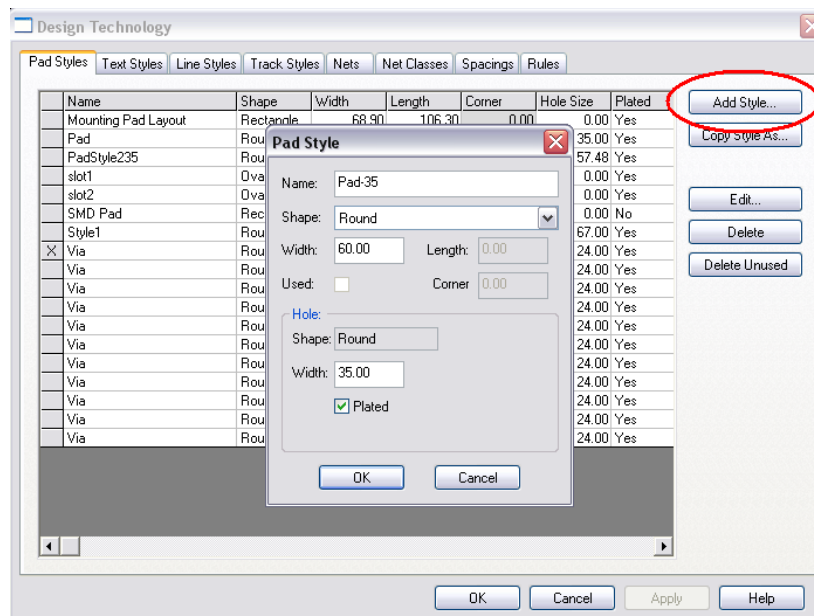
Make sure the mounting hole pad width is the same size as the through hole.

Place pads dimensionally
1. Print out your data sheet for the part.
2. Data sheets usually have relative dimensions. We need to manually create absolute dimensions to an origin which can be easily referenced on the design. Manually convert all dimensions to their absolute dimensions from that origin. You can place a pad to serve as the origin and delete it later if necessary (but this will require the use of Edit>Renumber Pins to correct).
3. Right click on the datum and click Origins>Set System Origin At Item. The displayed coordinates and grid will now be snapped to that object.
4. Go to Settings>Styles of the objects that need to be placed dimensionally.
5. Click Add Pad, drag the cursor over the work area, and hit the <s> to choose the Style to be added. Place the objects anywhere in the work area.
6. Select each object, right-click and choose Properties from the right-click context menu. Type in their absolute coordinates. When you click OK to save your edit and exit Properties, the object will move to those coordinates.
7. When all items are placed, change your origin as necessary to confirm all relative and absolute coordinates.
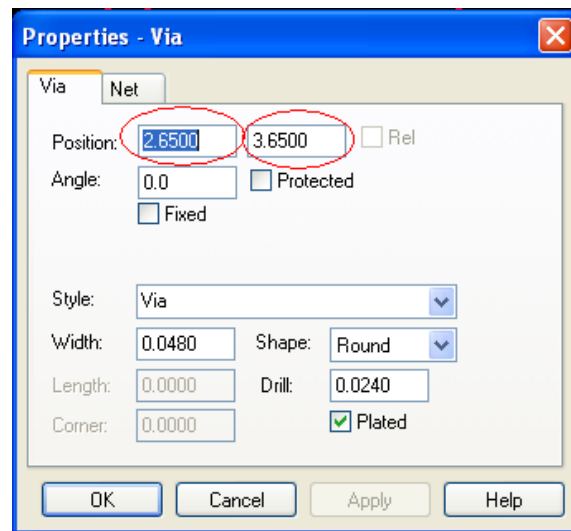
Soldermask clearances

Here is now the soldermask layer works:

- Thru-hole pads are placed on layer [All] automatically. This means they have pads on all layers when added to or translated into a PCB. This includes an automatically generated soldermask clearance of exactly the needed size.
- If you use the PCB Footprint wizard, it will automatically place surface mount devices on the [Top] layer. The [Top] layer automatically creates a clearance for the surface mount device on the Top Copper layer, the Top Soldermask layer, and the Top Solder Paste layers.
- If you place a surface mount component on a PCB and use Flip to change it to the bottom side, all [Top] values automatically become [Bottom] and all other relationships to layers are treated accordingly.
- If you need a soldermask clearance that does not correspond to a specific pad, such as a large soldermask clearance over the whole component, use Add Shape. Use the <L> hotkey to change your working layer to the Top Soldermask. Draw the polygon and then select any segment of it, right-click and choose Properties from the right-click context menu. In Properties, click on the Filled checkbox.
  - Use the Layers tab of the Interaction Bar to display only the Top Solder Mask layer(s) to view the shapes.
- Be aware that all objects on the soldermask are clearances in the soldermask and everything that is black in the soldermask layer will have the green soldermask covering.
- **WARNING:** If you create a contact land with copper pour it will NOT have a soldermask clearance. Copper pour is not the correct tool to make contacts with. A solder mask clearance would have to be manually created using Add Shape Polygon to create the cleared area on the appropriate soldermask layer and then go into the polygon's Properties to click on the Filled checkbox.
- **WARNING:** If you Add Pad and then use Properties to make it a surface mount, it will be associated to the [All] layer, where all thru-hole pads live, and will be present on every layer. Surface mounts must be on layer [Top] or [Bottom]. The netlist will also believe the surface mounts have an electrical connection to the other surface mount counter parts on the other layers, were none exists, and will display the rats nests and autoroute accordingly.

Non Plated Cut outs in a Component Footprint
If a component requires non plated cut outs within a component footprint, they will need to be placed after the component is placed on the board. For reference one may draw the component information out on the silk or documentation layer, but the same information will need to be drawn on the Board layer.

Updating Components
Often a component will need to be changed or updated. Even if the component is updated just by the footprint, it should be updated on the schematic and the PCB file to make sure the boards are consistent. The function to update components is found in Tools>Update Components>Browse. Although the option to update all components is available, it is suggested to only update the one component selected through the browse list.

Packages
Package in component generation relates to the type of chipset and shape the component part is. When creating a part, by default the package is "User".
The following is a list of a few packages and possible footprints (details and drawings are approximate, and are just examples):

DIP: Dual Inline Package	**SOIC**: Small Outline Integrate Circuit
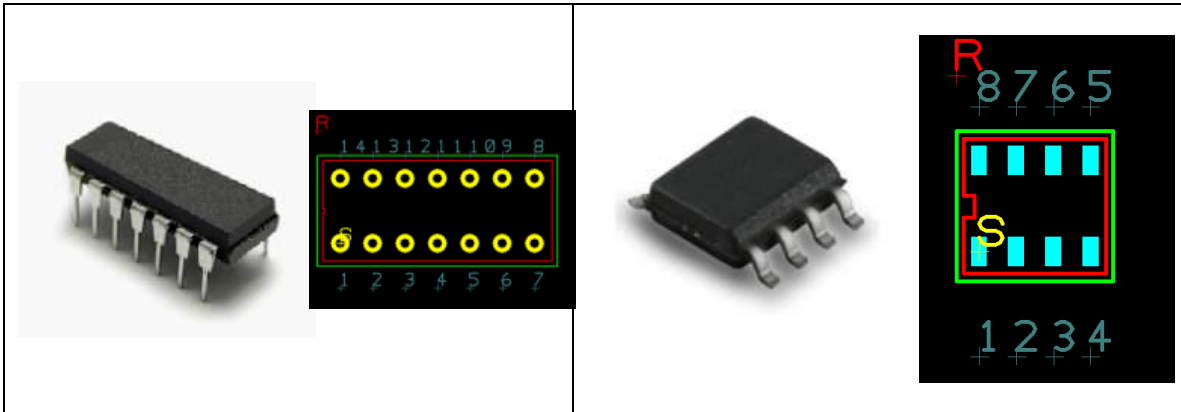
SOJ: Small Outline J-lead	**SON**: Small Outline No-lead
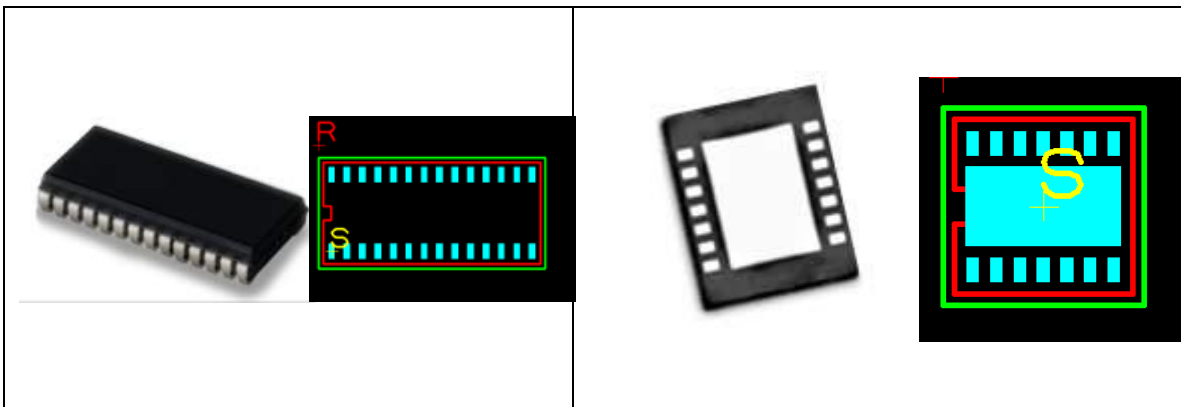
QFP: Quad Flat Pack	**QFN**: Quad Flat No-leads
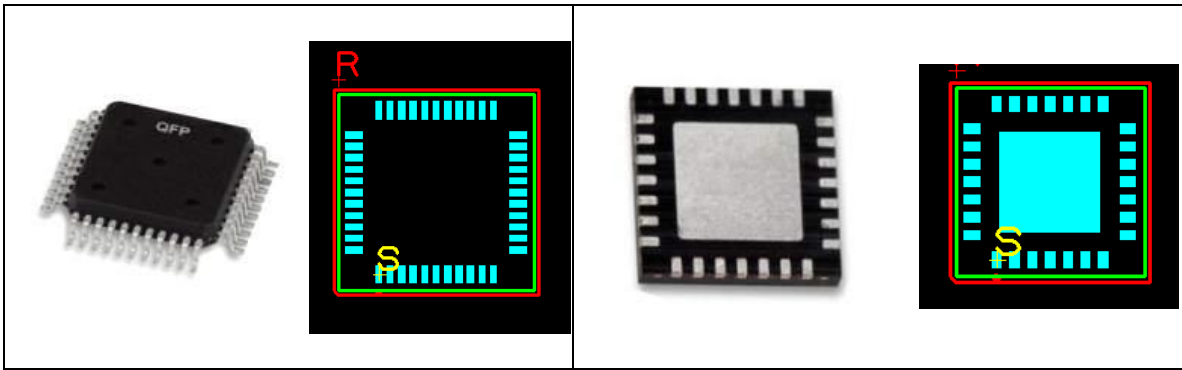
PGA: Pin Grid Array

BGA: Ball Grid Array
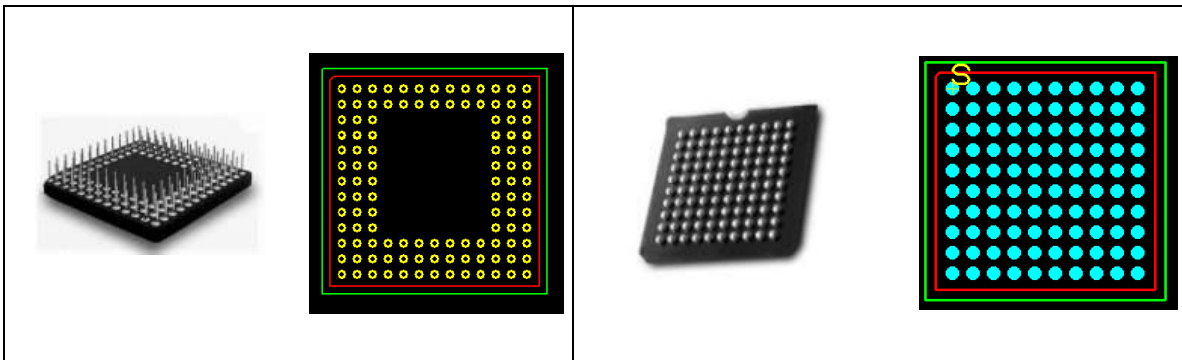
CAN:

Axial: (through pin resistors, etc)
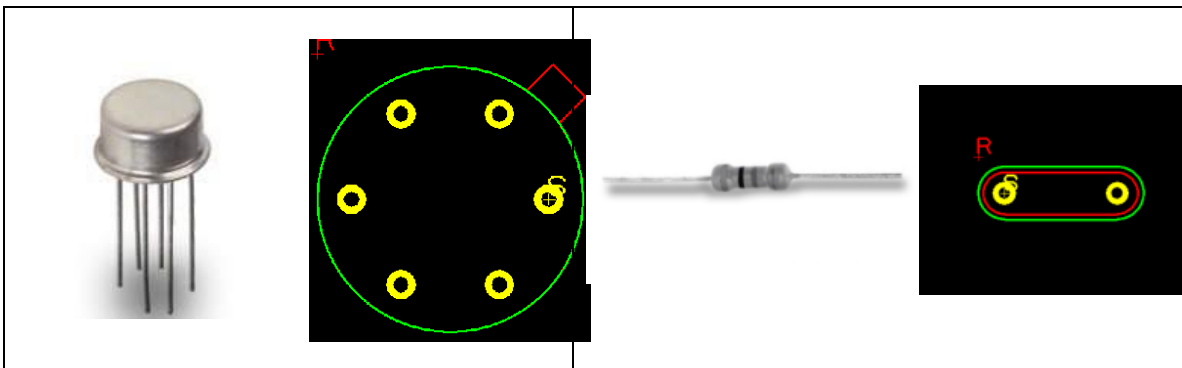
SIP: Single In-line Pin

SOD: Small Outline Diode
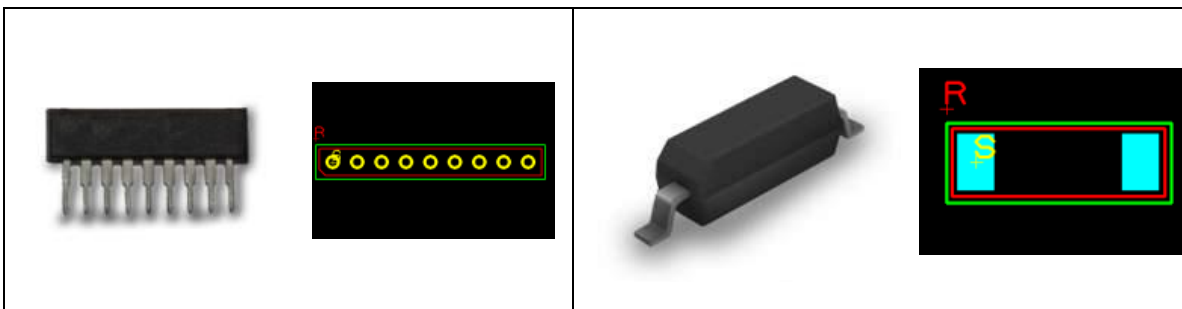
SOT: Small Outline Transistor

Simple Connector
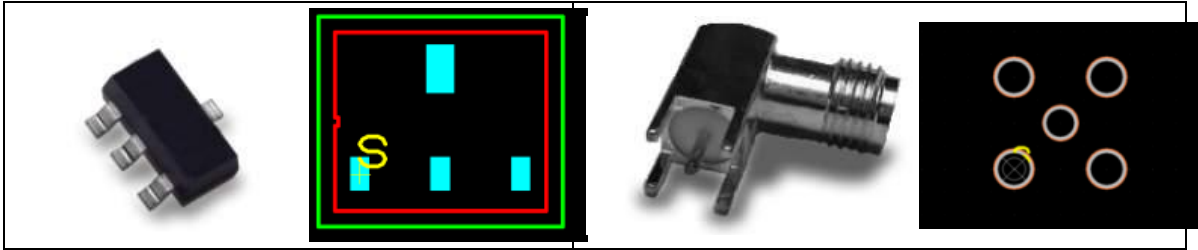
Extracting components from the .pcb and .sch files

One can extract the components from a .pcb file and .sch file in the Library Manager. To do so, go to the Library Manager and select a tab. For the schematic symbol, you will need the schematic. For the PCB symbol one will need the PCB File. For a complete component (PCB And Schematic symbols both) you will need both the schematic and PCB file found within the same place.

1. Start with the Schematic Tab in the Library Manager, and create a new library for the components that will be extracted. Next, click on the Add File button. Navigate to the schematic file, select it and press OK.
2. This will extract all the schematic symbols into the newly created library. Do the same for the PCB symbols with the PCB file.
3. The components tab will also extract from the .pcb file. If the schematic and PCB symbols already exist in the libraries, extracting the components will then be complete.

Adding a schematic symbol to a PCB only Component

There is an option to add a schematic symbol to an already PCB only component by using the Add menu option in the Component editor.

- Open the Library Manager (Ctrl +L), and go to the Components tab.
- Select a PCB only component, and click the Edit option.
- One may select Add>Gate option to add the Schematic symbol. Locate the gate in the library pop-up. The user will need to make sure the pin mapping is done correctly, using the upper table found on the Edit window.
- Once done, save the component in the library.
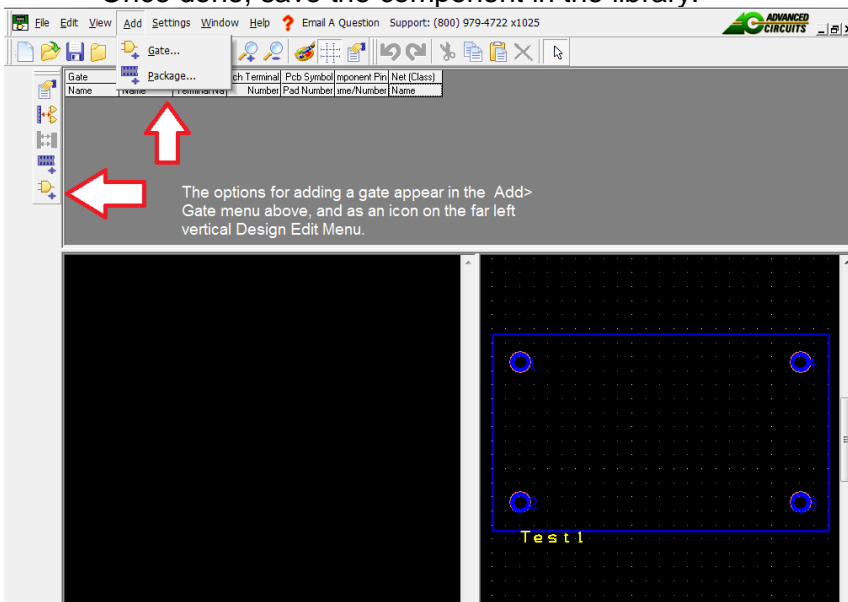
Adding a PCB symbol to a Schematic symbol only Component
One may add a PCB symbol to an already Schematic only component by using the Add menu option in the Component editor.

- Open Library Manager (Ctrl +L), go to the Components tab.
- Select a Schematic only component, and click the Edit button.
- One may select Add>Package option to add the symbol.   Select the PCB symbol in the library pop up. The user will need to make sure the pin mapping is done correctly, using the upper table found on the Edit window.
- Once done, save the component in the library.
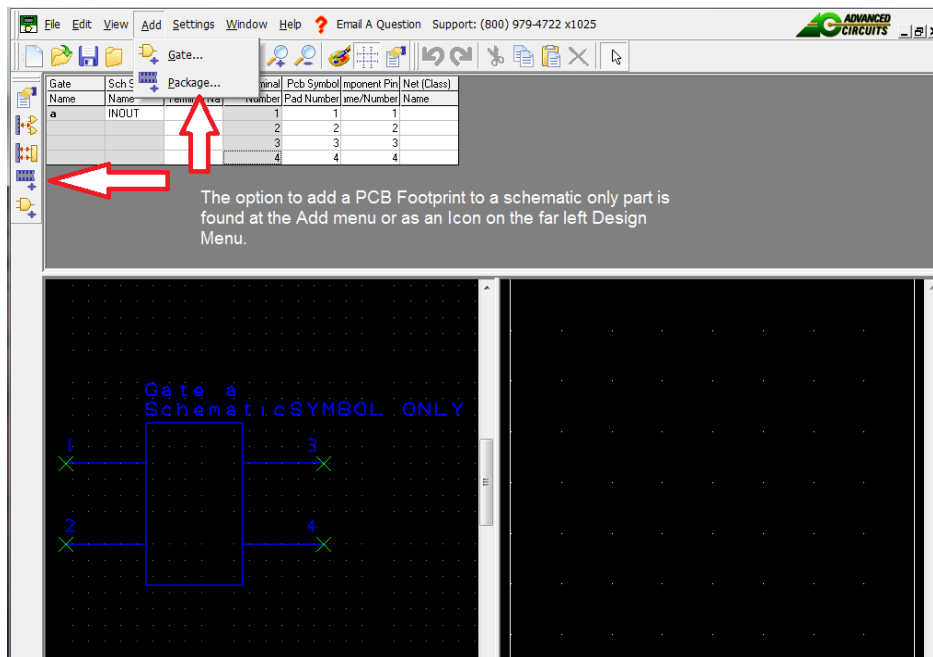
Adding Multiple Different Gates
Users are given the option to have multiple and different gates added to a component. This option is different than drawing a gate that has multiple shapes in it for the schematic symbol. When gates are added in this way, each gate can be moved to different areas of a schematic, and even to another schematic page. One may add a separate different gate to a component use the Add>Gate icon while editing the component. This is the same icon one would press to add a schematic symbol to a PCB only component. Further modifications may be made in the Component Editor by going to Edit>Gates. This will then allow one to remove a specific symbol or add other symbols.

Adding Multiple Different Footprint Options for a Component.
One may add variations of footprints to the same component. To do this one must be in the component editor.

Select the Add Package symbol on the far left Design Menu.
This can be done several times to the same component.

When adding this component through the Add Component tab on the Interaction Bar, one will be presented with a drop down below the parts list. This drop down will only be available when a component has multiple packages available to it.
Packages may be added or removed from a component in the Component Editor by going to Edit>Packages.

Library Export to .palib
Once a component has been created, a user is able to export it to a .palib file. This file will retain all the information in regards to the component and can be installed onto another computer with PCB Artist. The file acts the same as the downloaded .palib files from the online library. One can "Drag and Drop" the files onto the PCB Artist Icon to install them, or use the Add File button in the Library Manager.

The option to output a .palib file appears on the Components Tab of the Library manager as the Export button.
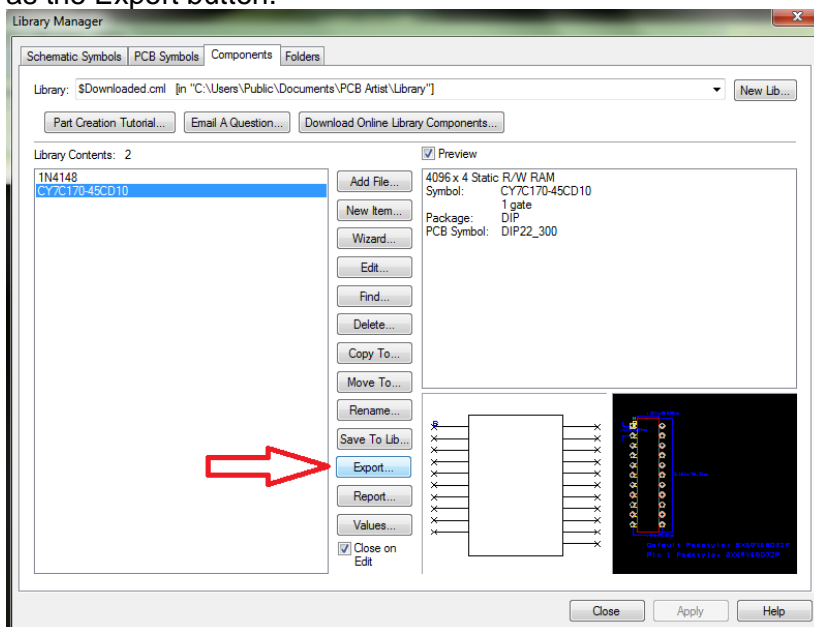
Schematic Creation

Multipage schematic
1. Go to File>New and choose a Project.
2. Then go to File>New and choose a new Schematic. Check the box at the bottom of the dialog to Add To Open Project.
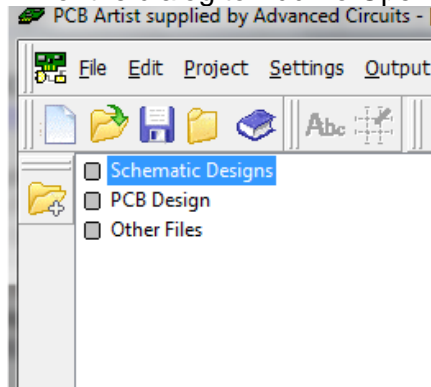
3. Page frames for the schematic can be found in the Schema library.
4. Off page references between schematic pages are made simply by the nets having the same name set in Settings>Nets.
    a. This does not apply to generically named nets (the "N0000" named nets). Generic nets are local to the page they are on only.
    b. In Settings Nets, if a GND net is created and it exists on another page, they will connect whether or not a connection or off page reference is drawn.
    c. In the Implied Connection instructions (the topic directly below) also apply to off page references between Project sheets. However, the hanging connection will turn red indicating it is unconnected. It will connect due to the net associations, despite appearing unconnected. To make this clearer and to remove the hanging connection highlight, use the To and From symbols of the Schema library. Those symbols do not have any special attributes; the net name itself is all that is needed to make the connection.
5. Splitting component gates is best covered in Help section of PCB Artist.   Go to Help>Contents and type into the search blank the "project gates". Click on the topic "Splitting Components Across Schematic Designs" for complete details.

Implied connections
1. Draw a connection off of a pin. Drag it a few grid points and double-click to terminate it there. The connection will turn red to indicate an un-terminated connection.
2. Select the connection, right-click and choose Change Net from the right-click context menu. Choose the net that you want it joined to.
3. The color of the connection should change to black to indicate the connection is no longer un-terminated.
4. Right click on the connection again and choose Display Net Name to verify the process was successful at a glance.
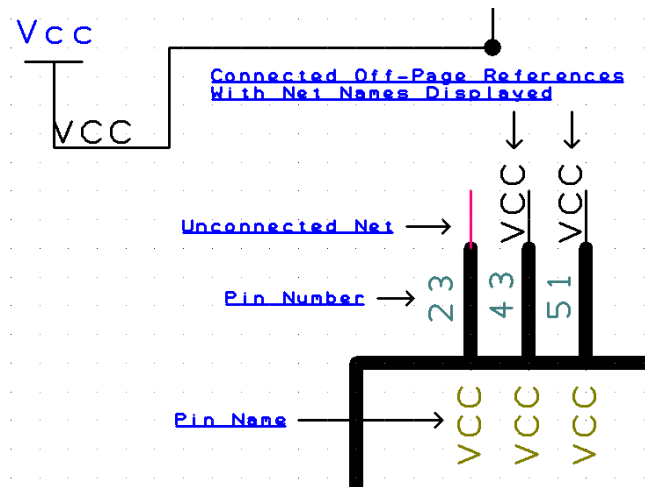
Power and ground symbols
They are located in the Schema component library.

Changing values of components
Use either a window selection or the <Ctrl> button (in the same way you can use it to select specific files in Windows Explorer) to select all the specific components of that type to have the value changed. Then right click on any one of them and choose Properties. In the Properties window, click the Values tab. Select the value in the list and use the Edit button to change it. All the components will now have the new value.

Symbol doesn't show on Add Component list
Schematic and PCB Symbols cannot be added to a board directly. They must be first joined into a Component Symbol. Only Component Symbols can be added to a PCB or Schematic.

Fetch Component
In the PCB and Schematic context menu one may fetch a component to the current cursor position by right clicking in a blank area and selecting Fetch component. The hotkey for this is Shift+F.
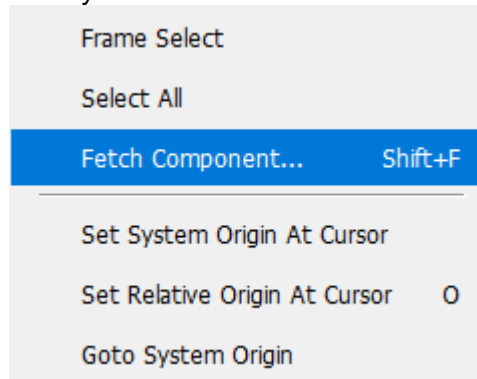
This will list any components that are in the Component bin and found on the drawing. By selecting it from the list, the component will instantly appear at the cursor.

Fetch Component to cursor ✕

[input field]
OK
Cancel

C1

☑ Show Fixed Components
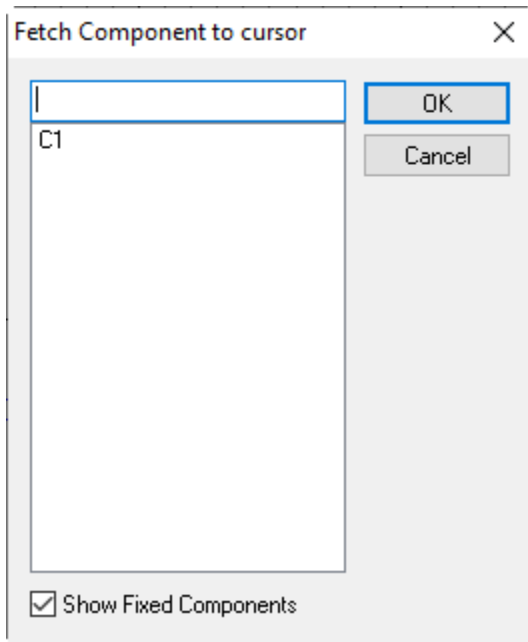
Displaying pin logic names

This is actually done at the component level.
1. Go to the Library Manager
2. Click on the Component tab.
3. Go to the correct library and select the part. Click the Edit button
4. Enter the logic name in the Terminal Name column (third column from the left).
   On the schematic part, the new names will show at the N# locations.

5. Save and exit.
6. Open the schematic and then go to Tools>Update Component.
7. If the logic names are not displayed, right click on the component and choose Properties. Click on the check box for Pin Names.

8. If the Pin Names do not show, go to View>Colors. Click on the Other Items tab and click on the Pin Names checkbox if it is checked off. Click OK to save your changes and exit the Colors dialog box.

Auto Rename Components
Under Tools>Auto Rename Components, is an option to allow one to Rename the components across a schematic automatically. Options are as follows:

Forward Changes from schematic to PCB
Often when changes are needed on existing board files it is best to do them to the schematic first, then forward those changes to the pcb drawing. This allows the schematic and pcb drawing to remain consistent, and for the designer to run a consistency check to make sure both designs are still the same. To forward changes from the schematic go to Tools>Schematic⇔PCB>Forward Design changes.   This will create a report that will state all the changes that has been done to the pcb file. Forwarding Design changes will unroute any nets that no longer match the schematic,

remove parts that are no longer in the design, add parts, and change net names .
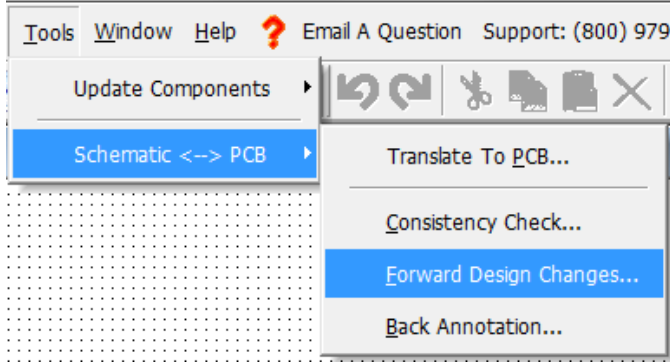
Spice Simulation
Spice Simulation is covered in the Help>Context Help menu under Spice.
Schematic Busses
In a schematic, one may add a bus through the Add>Bus menu.
One can create a closed bus by adding nets through the properties of the bus.
Adding connections to a bus without assigning nets to it will make an open bus, and any net may be assigned to it.

Schematic to DXF Printing
PCB Artist can output the schematic as a DXF format print. The option appears on the top menu, Output>DXF.
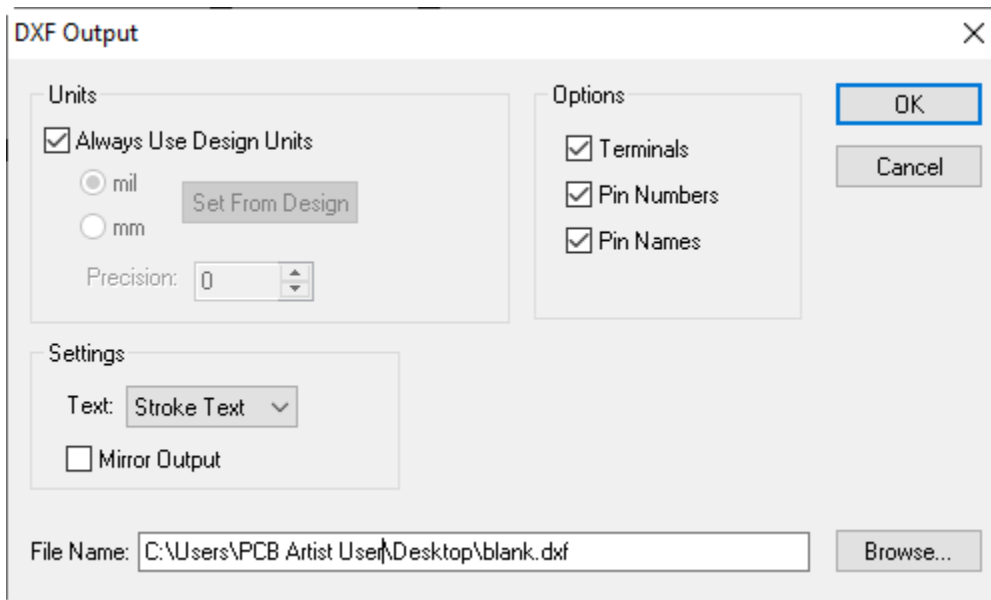
The New PCB Wizard

Change the board size
You can delete it and completely redraw it in the PCB, but it must be created and finalized before placing an order.

Unit Precision
Unit Precision is found in Settings>Units. Precision is how many decimal places any displayed coordinates round up to. If you have the Units set to Inches and the precision is set to zero, then the displayed size of a .250" hole will be 0". Set Unit Precision appropriately for the units you have selected. Recommended Precision for Inches is Precision 4, for Millimeters is Precision 3 and for Mils is Precision 1.

Board Template
When you create PCB Designs, you can save as a template. That will save the board with all its settings, components, board outline, and even tracks and copper pours. You can then use that saved template to start a new design with all those features built in for future designs. It can be used to save as little as unit settings or as much as whole designs. Be aware that if you save a template for a design, the entire design is saved, not just select features.

Layers
It is easy to add layers after the New PCB Wizard, but when eliminating layers, you can only eliminate the last inner layers. So if you create a 10 layer board and find out that you only need 6, when you go to change the layer count you can only eliminate layers 9, 8, 7, and 6 (the Bottom Copper will become the new layer 6). This also changes the possible plane layers to 2, 3, 5, and 6, which may be a major conflict of some of those layers are already routed signal layers. So it is recommended to start out with a small number of layers and then once the layout has progressed to a point that the need for more layers is unavoidable, add the layers then from Settings>PCB Configuration.

Split and mixed inner planes
The plane layers are intended as pure planes. They should never be edited under any circumstance, even to draw a single track. On the layers page, leave them as signal layers and then when editing the PCB, use Pour Copper to draw split and mixed planes.

Minimum track width and space
The default is 10 mils, you can change it in the design, but you will have to go to Settings>PCB Wizard when you are in the PCB design and change this value on the Board Parameters page. The system will not allow you to submit the order if this number is not correct. It will prompt you if this value is different.

V-score and Tab rout
Basic Design Requirement orders cannot have a v-score or tab route, due to overwhelming request of customers seeking the lowest possible cost. This includes orders with arrays. To get those options, select Expanded Design Requirement on the Design Requirement Page of this wizard.

Special Requirements text box
It's intended for any requirements that do not already have an option in the wizard and also for extra details of items you can select. Some Examples:
If the board is a flex design and requires a thinner final thickness not listed on the board thickness drop-down menu of the Board Parameters page (but was verified as possible for your design, through PCB Artist Technical Support), this would be the place to specify it. Special billing or shipping information can be noted here as well any special instructions. The board has countersinks, use this space to specify the counter sinks and to differentiate which holes are the countersinks. You can return to this page from Settings>PCB Configuration once you are in the design.

Part number and revision
We must have a way to reference the part for future orders. The software requires both of those blank be filled out before order placement.

Arrays
See the Array section **here**.

Change options after New PCB Wizard
Some options ran in the PCB Wizard can be changed long after PCB Creation. This can be done from the main menu click on Settings>PCB Configuration to change some values.
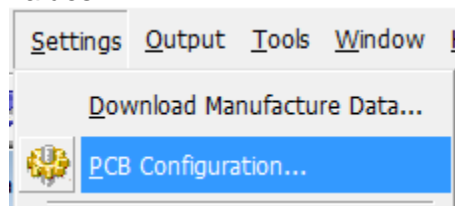
- **Warning: Reducing the number of layers in this manner can eliminate nets.**
  If layers must be eliminated, go to Output>Net Completion Report to get a report of the nets that are now incomplete. Then from the main menu, go to Tools>Optimize Nets to make the un-routed net connections appear. They can be manually or auto-routed from that point. After they are routed, it would be wise to go to Output>Net Completion again to confirm nothing was lost.

PCB Design Edit

View Power plane layers
Go to View>Powerplane>Show to view pure plane layers. Mixed and Split planes are considered signal layers (one uses pour copper to make the planes), so those layers are viewed normally as any signal layer.

Selecting a particular item
Use the "N" hotkey to cycle through objects at that location until the object you want is selected.

Remove net highlight
This can be done by going to View>Highlight Net>Remove Highlight. If the highlight is from selecting a net on the Interactive bar, go back to the net selection area (Interactive Bar>Go to Tab>Nets in the drop down). Right click under the list, or in an unselectable area and select "Remove Highlight". The other option is View>Goto>Remove Highlight.

Cross Probe Function
In PCB Artist version 1.5 a Cross Probe function has been introduced. This allows users to select nets and pins on the schematic to highlight the same nets on the PCB file. Users are also able to highlight the nets, tracks or pads on a PCB Design to highlight the same on the schematic.

This function may be accessed under Edit>Cross Probe, or by the icon .

Displaying layers
Bring up the Interaction Bar with <F9> and Click on the Layers tab at the very bottom of the Interaction Bar. Right-click over the list of layers and choose the All Layers Off option. Then click on just the layers you want to view.
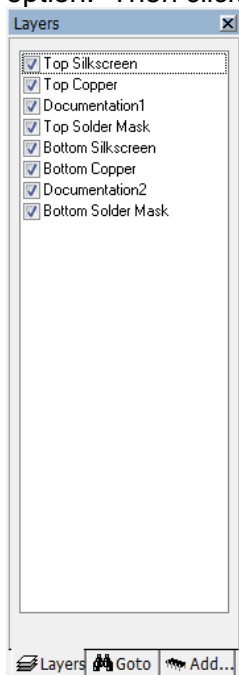
Properties of an object display size as zero
Go to Settings>Units and change the Unit Precision to a larger number. Precision is how many decimal places any displayed coordinates round up to. If you have the Units set to Inches and the precision is set to zero, then the displayed size of a .250" hole will be 0". Set Unit Precision appropriately for the units you have selected. Recommended Precision for Inches is Precision 4, for Millimeters is Precision 4 and for Mils is Precision 1.

Select all the segments of a trace at once
Push and hold down the <shift> key, then select the trace. You can also select multiple whole traces by pushing and holding down the <ctrl> and <shift> keys.

Pad and Feature Colors
On the default color settings:
- Grey pads are present on all layers.
  - In Properties, they show as being on layer [All].
  - Surface mounts should not be grey. If they are, the component needs to be edited to change them from layer [All] to layer [Top] (NOT to be confused with layer Top Copper! An object on layer [Top] automatically has a soldermask clearance and solderpaste clearance generated. It will also flip all those to the [Bottom] layer if the Flip option is used on the component.
- Dark Red pads (using the default color settings) are on Layer [Top] IF they are a slightly darker red than the Top Copper traces. If they are the same shade of red as the traces, then they are not on the [Top] layer, they are on Top Copper and will have no soldermask or solderpaste clearance. They must be changed to layer [Top] in Properties (see previous bullet point).
- Dark Teal pads (using the default color settings) are on layer [Bottom]. They are slightly darker than the traces on the Bottom Copper Layer. The have all the other properties of objects on layer [Top], described above.
- Yellow pads are vias created by the application when using the autorouter, moving a track segment to another layer, or other processes where the application places the hole automatically. They are present on layer [All], but are not associated with a component. Most commonly these are vias.

The following information relates to what layers the feature will appear on, function, or error.

PCB Color Coding:

Gray: Appears on all layers, and the soldermask layers. In the properties drop down it will show as [All].

Red: Appears on the top copper only, and not the soldermask. If this is a pad, in the properties drop down it will show as [Top Copper].	
Dark Red: Appears on the top copper and the top soldermask. In the Properties drop down, it will show as [Top].	
Cyan/Teal: Appears on the bottom layer with no soldermask clearance. If a pad, this will show as [Bottom Copper] in the properties drop down.	
Dark Cyan/Teal: Bottom copper with soldermask clearance. In the properties drop down, this will show as [Bottom].	
Yellow Pads: These represent vias. Vias appear on all layers.	
Bright Green: Board layer. The board layer is the outline/route of the board.	
Dark Green: Documentation 1 layer	
Dark Blue: Documentation 2 layer	
Slate Blue: Top Silk Screen	

Purple: Bottom Silk Screen	
Orange/Coral: Top soldermask layer	
Light Gray: Bottom Soldermask layer	
Function colors:	
Thin yellow Lines: Thin yellow lines indicate disconnected nets that may need to be drawn. These are commonly known as "Rats Nests".	
Dark Blue Tracks/Pads: This may indicate that a net has been highlighted.	
White: This is an actively selected object.	
Track with a Pink Inner Line: This indicates a disconnected track	
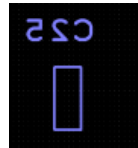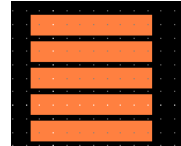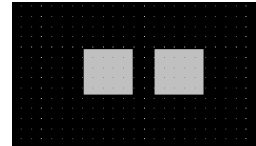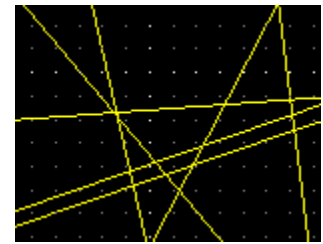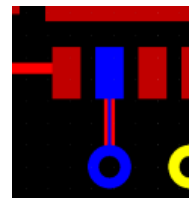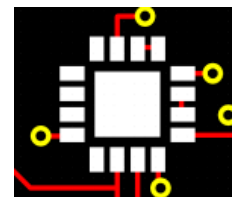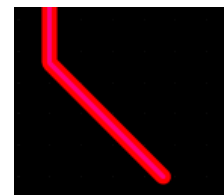
Green Highlight: Non net highlight. One may turn off the highlight by going to View>Goto>Remove Highlight	
Schematic Color coding:	
Orange: Selected Connection	
Cyan: Net Selected Connection Highlight	
Black: Un-highlighted Connection/Schematic Symbol	
Pink: This indicates an incomplete connection.	
Dark Cyan/Teal: Pin number	
Olive: Pin name, sometimes will show as a N# as a designator	
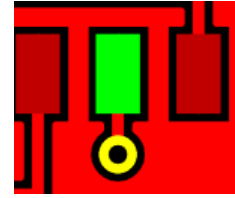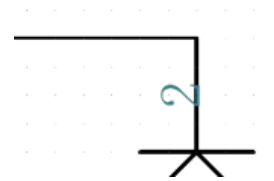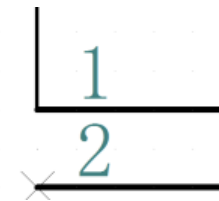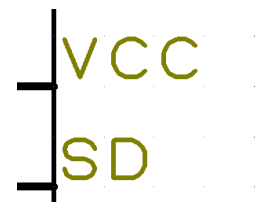
Blue: Text, Symbol Text, and Value Position

U6
1N914
AFL

Drawing on a different layer
Here is one way to do it.
1. Click on Add Track
2. Right-click and choose Track Layer OR hit the <L> hot key.
3. Click the box that says Apply To All Segments if want the whole track on the layer you are switching to.
4. Select the layer you wish to draw on in the dialog box that appears. It will automatically default to a layer other than the one you are on, so just hit return to accept it if you are on a double-sided board and you will be on the Bottom Side automatically.

Draw at odd angles
1. Click on the icon of the type of draw you wish to make (such as Add Track, Add Board Polygon, Add Shape Polygon, Add Track, Add Copper Pour Area Polygon, etc.).
2. Click to add the draw and move your cursor to the work area.
3. Before starting the draw, right click to open the context menu.
4. Select Segment Mode>Free
5. Now lines can be added at any angle.
6. If moving shape segments, use the Free Rubber-banding option from the right-click context menu to allow the attached segments to take any angle.

Moving component to the other side of the board
Select the component, right-click on it and choose Flip. This may also be done by selecting the component and using the "F" hotkey. It's best not to have track connections to the component before placing it on the opposing side.

Corners with radius or chamfer
If you double click on 90 degree rout corner, it will take the corner into Miter or Fillet mode. In the Status bar in the lower left frame of the application, the phrase Edit Miter (or Fillet) Mode will appear. When you drag the cursor, it will increase the Fillet or Chamfer at each grid point. Go to Settings>Grids in advance of the edit to set the grid to the correct value. Right click while in miter (or fillet) mode to toggle between miter and fillet.

Make draws with 45 degree angles
1. Click on the icon of the type of draw you wish to make (such as Add Board Polygon, Add Shape Polygon, Add Track, Add Copper Pour Polygon, etc.).
2. Click to add the start of the shape.
3. Right Click to open the Context Menu while you are still in draw mode.
4. Select Segment Mode>Miter.
5. You can now draw at angles and the system will automatically put all junctions at 45 degrees.

6. It may help to change your draw mode to Free. See the question above regarding drawing at odd angles.

How to remove track segments
Often a track has to be partially redrawn if clicking and dragging the track is not sufficient. To remove one segment of a track:
1. Select the track, and right click on it.
2. Select Net>Unroute Track Segments

The highlighted track segment will disappear. PCB Artist will consider both sides of the track gap and all included pads and copper features to be still part of the same net. This will be indicated by a yellow line by default, commonly known as a "Rats Nest".

Change a pad (or track) size universally
Here is one of several ways to do this:
1. First determine the pad style used from the properties box.
2. Go to Settings>Styles.
3. The Styles window will open, click on the Pads tab.
4. The names of each pad style are listed. A new style can be created or an existing style can be edited from this window.
5. Be careful to be sure that the change will not affect more pads than you intended to affect.
   - Various combinations of selections and added styles can be used to accomplish different goals in selectively changing styles.
   - The Interaction Bar's Goto tab can be used as a powerful tool when set to styles. Right clicking on the name of the style and choose Select All Find Items is another way to select all the items and to see them all selected before editing them either with Properties or through Settings>Styles as described above.

Allowing Duplicate Tracks
Initially PCB Artist is set up to not allow multiple tracks to connect the same two points on a board. This can be bypassed by going to Settings>Preference>Interaction tab, and putting a Check in the box beside "Allow Duplicate Tracks".
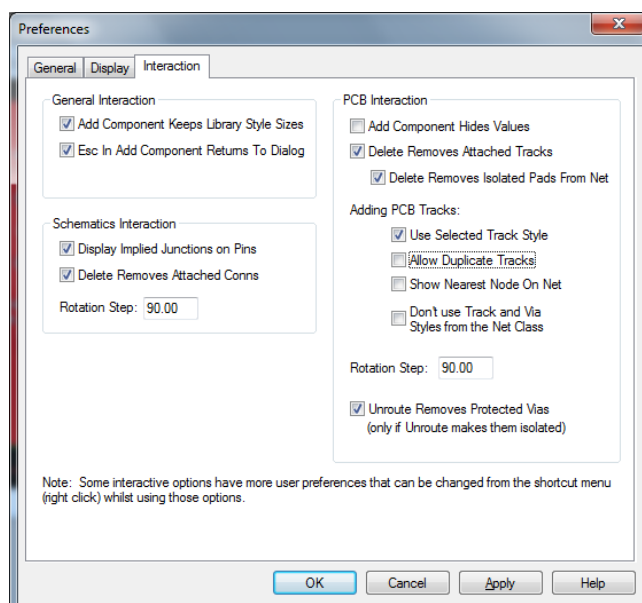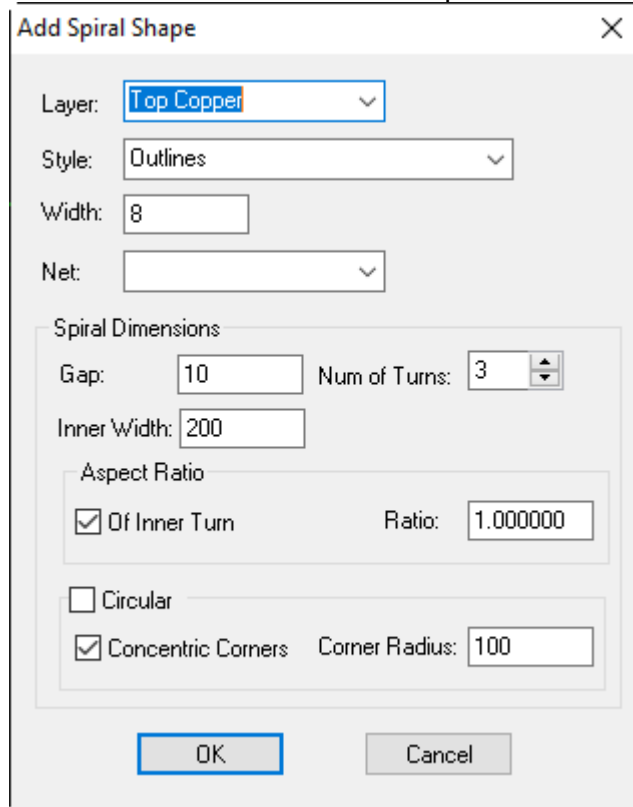
Mounting holes
That answer is above in the Component Creation section found here: Mounting holes

Adding a Coil or Spiral
One may add an evening generated spiral using the Add>Spiral Function inside of PCB Artist. This can be done as a shape or track with the following options:

Add Dimensions
In version 3.1 of PCB Artist, users are able to add vertical, horizontal, straight dimension to their drawings for reference. This is done through the Add>Dimension menu option, or by using the Icons on the PCB design menu:

Plated Slots

Here are the procedures for making plated slots, and slots for components.

1. Non-plated Slots may be made using the Add Board polygon. Any Board outline within the main board outline will be treated as a hole. The minimum width of any slot is .031" non plated, .026" plated on a regular Expanded board.

2. Plated Slots for PCB and PCB Symbols.
   - This is a new feature for version 1.1.2 or later. If you have PCB Artist 1.1, go to Help>Check for Updates to download that patch. If you version is older than 1.1, download the latest version from www.pcbartist.com.
   1. Go to Settings>PCB Configuration in the design (or if you are making a component symbol, just make sure this step is done before adding the component or the plated slot will not appear in the component).
      a. On the first page of that wizard, set the design to Expanded Service.
      b. On the Board Parameters page, click on the checkbox for Plated Slots.
      c. Go to the Production Page of the wizard and click the Finish button.
   2. Go to Settings>Styles
      a. Create an Oval pad style that is the exact dimensions of the finished plated slot.
         - The Smallest width possible .026" for an Expanded board, .02" for a Premium Expanded.
         - Make the Drill Hole size zero.
         - Give it a name, such as Slot
      b. Create an Oval pad style that will be the pad of the plated slot.
         - The width and length of the pad should be .014" over the slot dimensions at the minimum. (For example, a .030"x.150" slot must have a pad no smaller than .044"x.164".)
         - Slot pads Slots should always be on layer [All], which they will be on by default when built in this method.
         - Make the Drill zero.
         - Keep the Plated box checked.
         - Give the style a name, such as SlotLand.

   3. Click Add Pad. Move your cursor to the work area, but no not add anything yet.
   4. Click the <s> hotkey and choose the style of the slot created in step a above (not the slot pad style created in step b).
   5. Place the pad at its correct final dimensional location.
      c. **Warning** – once the slot is changed from a pad to a draw, using copy and paste or move features will no longer be grabbing the object from center, but from the center point of the lines in the oval.
         - This means the pad must be placed in the required position before conversion to a slot polygon.
         - This also means that multiple slots placed with copy and paste should be completed at this point and not after conversion to the slot polygon.
   6. Select the oval pad.

7. Right click on the Selected Oval Pad and choose Change Shape Type. In the drop down menu, choose Shape.
   - **Warning** – Do not change origins or grids from this step until the slot land is added.
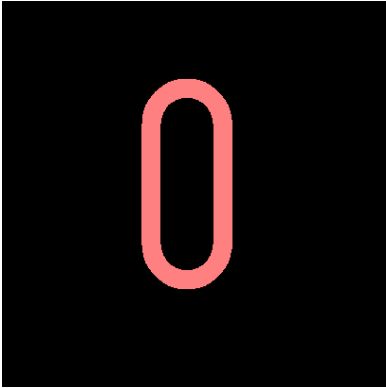
8. Click Add Pad. Move your cursor to the work area, but no not add anything yet.
9. Click the <s> hotkey and choose the style of the slot pad created in step 1-b above (not the slot style created in step a).
10. Add the pad right exactly over the center of the slot.
11. The Plated slot will be viewable on layer Plated Slot of a PCB Symbol and the Plated Slot layer in PCB Artist.
    a. **Warning** – If a component with a plated slot is placed before the option has been activated in Settings>PCB Config>Board Parameters, the plated slots information that will show on the plated slots layer will be lost!
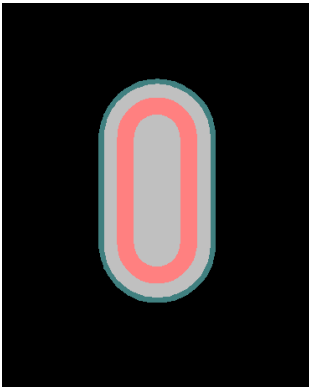
Reminder: Make sure the Plated slot layer is activated in PCB Configuration before placing any components with plated slots in them.

Creating a complex board Shape
The first approach is if you already have a complex board shape created in .dxf format.
Then all that is needed is to import the .dxf onto the Board layer. See **"**How to Import a
.Dxf" for instructions.

If this is not an option, then starting with a general board shape, then modifying it will be
your best option. Some of the most complex board outlines may only be 5 or 6
continuous lines:
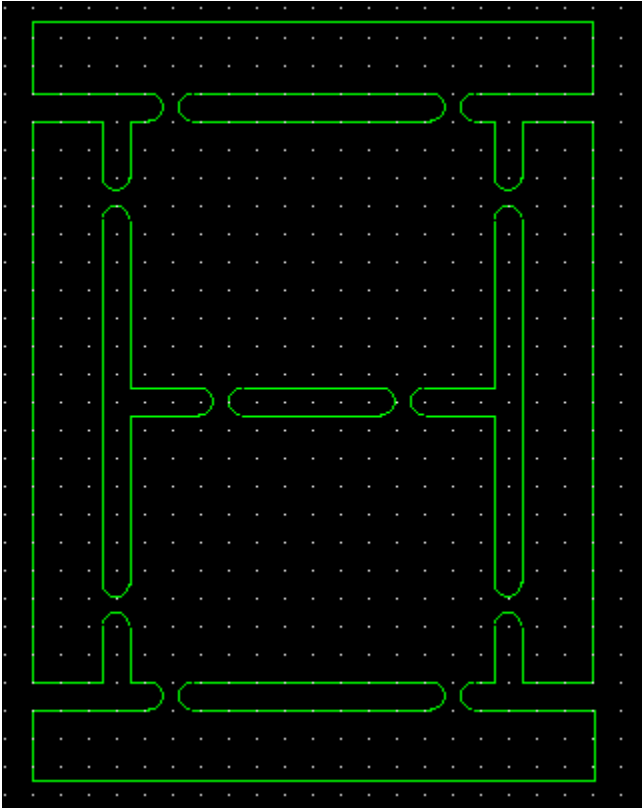
The above outline started with a large rectangle Shape from Add>Board>Rectangle. By
double clicking the board outline to modify segment (also an option available with right
clicking and selecting modify segment), we were able to pull and drag our outline to a
closer shape of the desired board.   To deactivate the edit, double click again. To reset
the edit while editing, one may hit escape and all changed features in the current edit will
snap back to the previous shape. It's good practice to have the grid (and Grid Snap) set
to a desirable distance for snapping to. The Grid snap and Grid functions are available
in Settings>Grids (or Ctrl+G). Segments may be changed into Arcs by right clicking on a
line segment and selecting Arc. There will be various options there for different Arc
degrees, and the ability to also Flip the Arc.

Complex Round Outlines.

Outlines are required to be closed. To create a circle that has notches missing takes planning.

There are two approaches to this. First one can use a .dxf of the complex shape from another software program, then import it as outlined in this FAQ here. The other option would be to draw the complex shape within PCB Artist. This follows some directions found in this FAQ here.

First as a guideline draw the outline shape on a documentation layer. This will be reference for the outline that will be built out of a polygon option single closed line. It depends on how the board has been set up. Adding an extra notch into a board outline with an extra shape can be misinterpreted by CAM, and the notch may not be created.

This example starts with a Polygon shape:
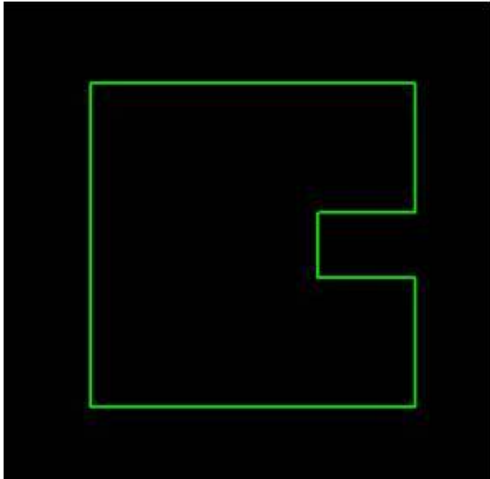
Then by double click on the corners and drag them inward to create a miter.
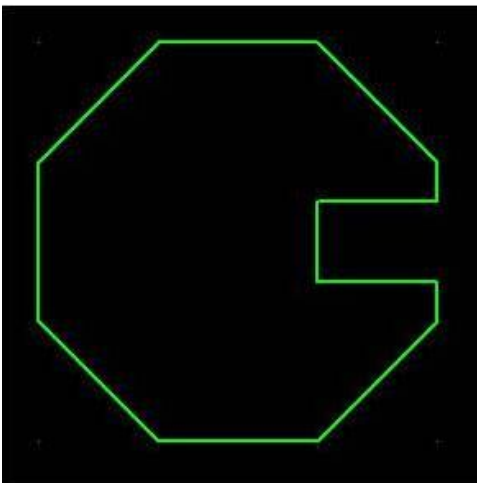
From here one may right click on specific segments and select Arc. For the above I selected a generic 45 degrees. When an arc appears backwards, one may right click on it again, select Arc, and select Flip Arc.
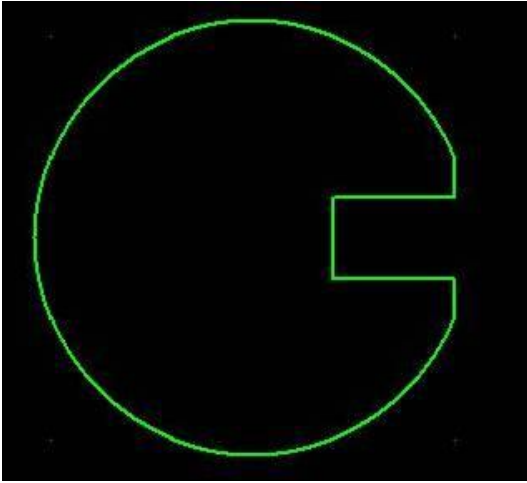
This is approximate above and may take practice and tweaking to get it to be a perfect shape.
By using the Arc function under the free option, one can heavily modify a segment with an Arc.
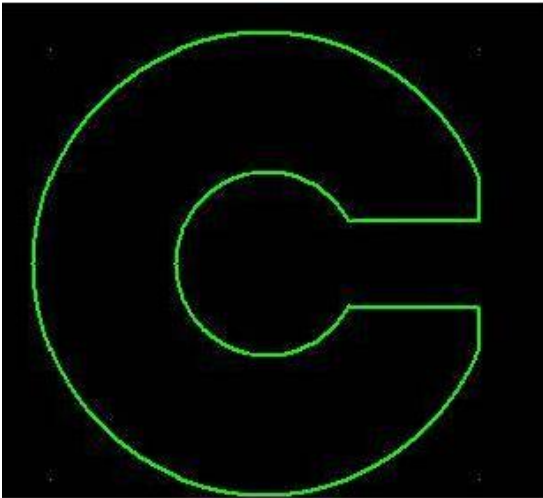
With straight lines, one may double click and break the line a drag the segment of the outline around. The outlines are easily deformed.

Creating a Quarter Circle/ Curved board shape
It is possible to create a circle segment inside the software, but it takes a bit of pre-work to get it to run correctly.

First, set the grid snap to the thickness of the ring you are making (for this example we are making the grid and grid snap .25". (settings>grids)

Go to Add>Shape. Add a circle. To create a circle from the center, right click while making a circle and select "Define from Center".
To change the layer the circle is being made, hit the "L" hotkey. We will use the documentation layer for this. Right click again, and select diameter, set the diameter to 16.
Do this a second time with a circle that is .5" less (15.5")

Then create an 8" by 8" board outline. Delete the old outline or modify it. To add an outline go to Add>board.

At this point is should look something like this:
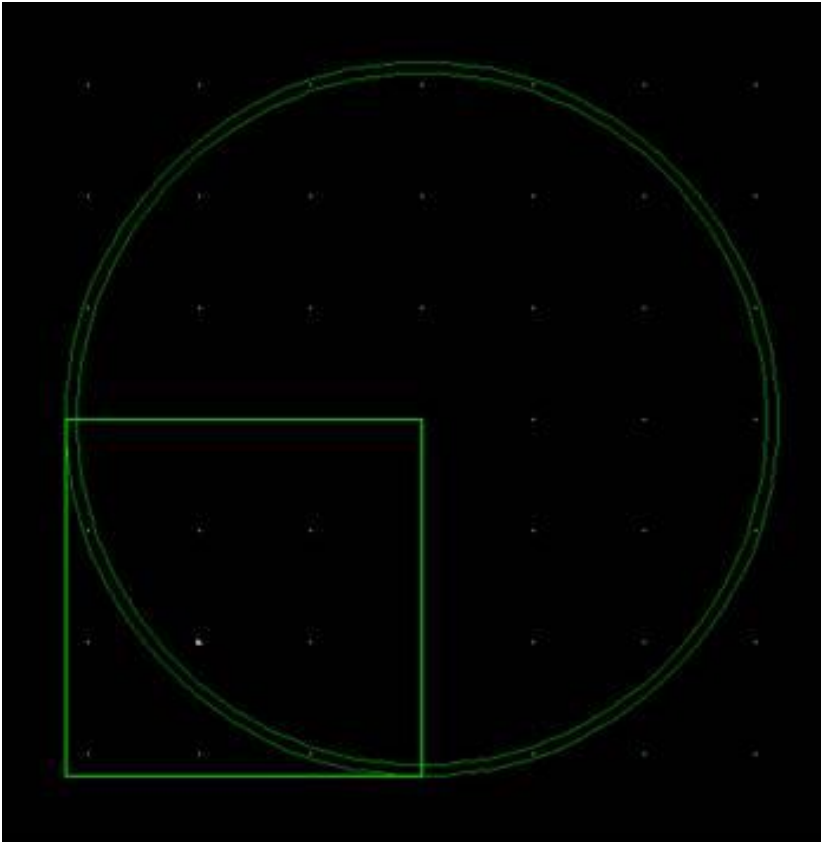
Double click on the upper right corner. This will activate the "miter" function. Drag in the corner until the line reaches the inner circle:
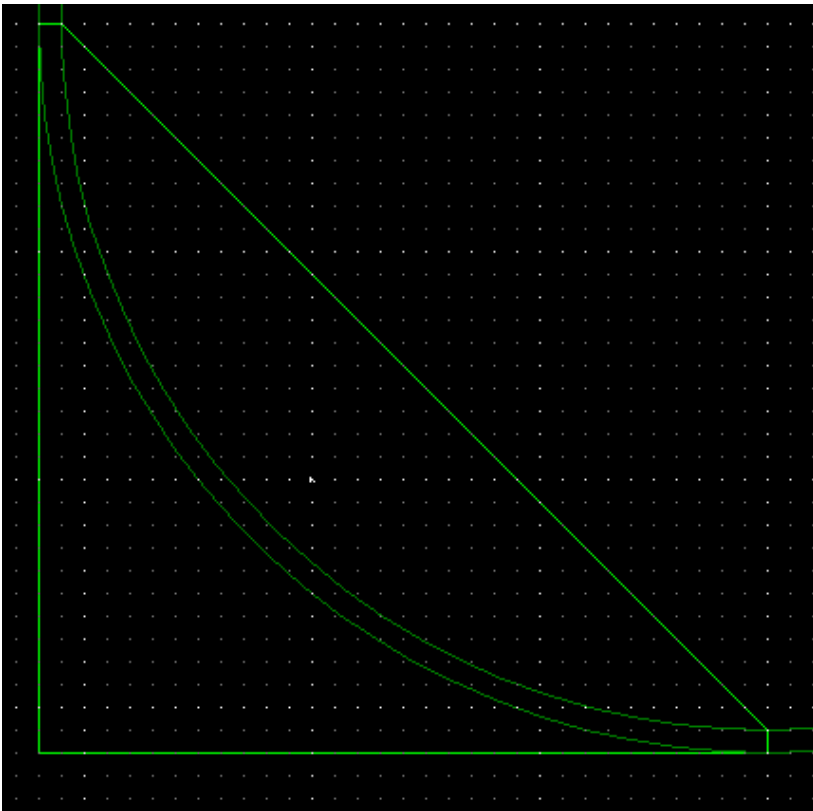
Do the same for the lower side:
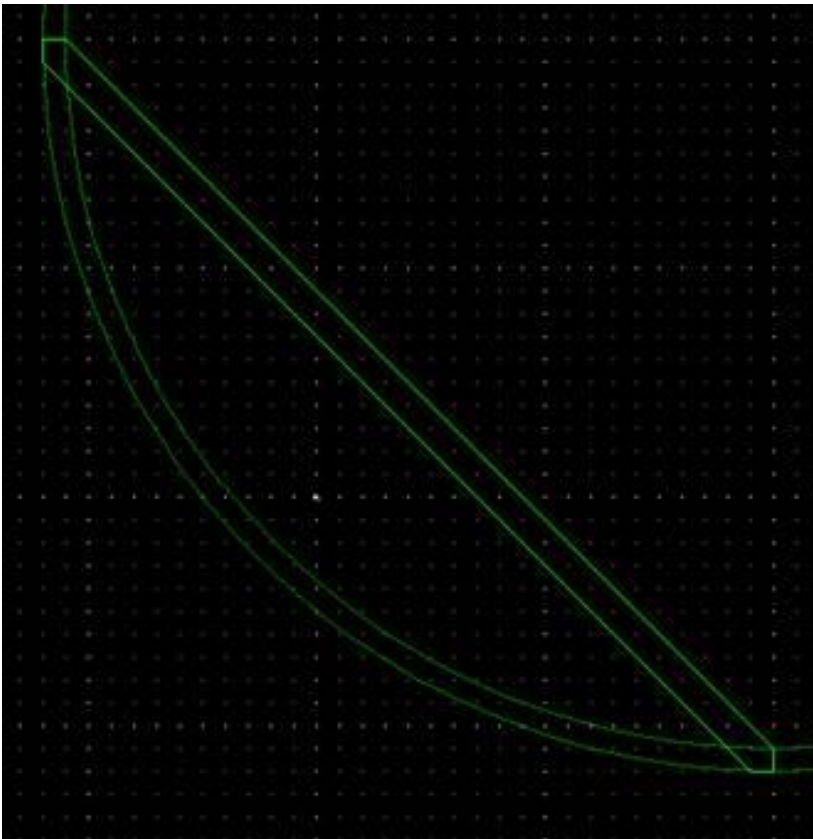
Right click on the mitered corner and select ARC. Select the free option. At this point one can drag the arc to match the outline on the documentation layer:
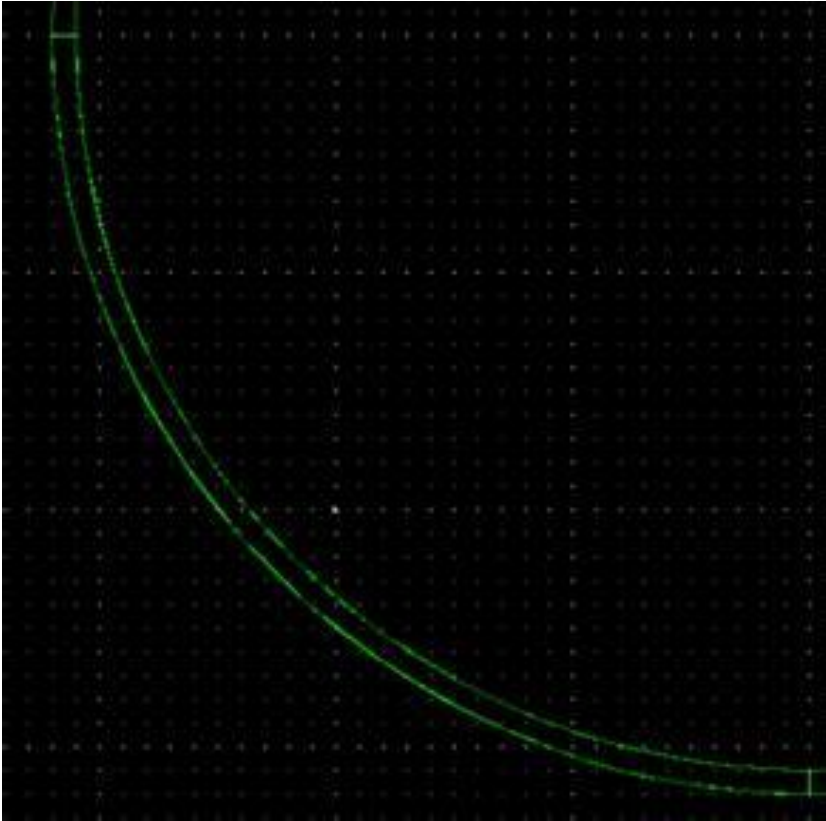
Without the documentation layer:
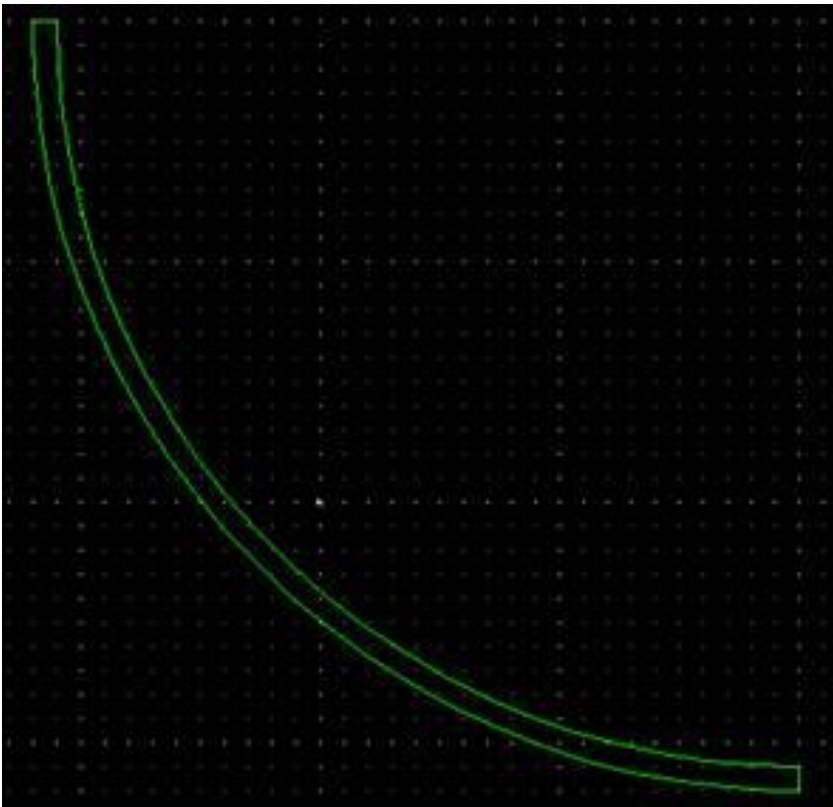
Keep in mind the above set up above is considered a 8" x 8" board.
After the board has been laid out, one can select all objects with a frame select (with all layers activated) and rotate the board 45 degrees. This would give the lowest profile:
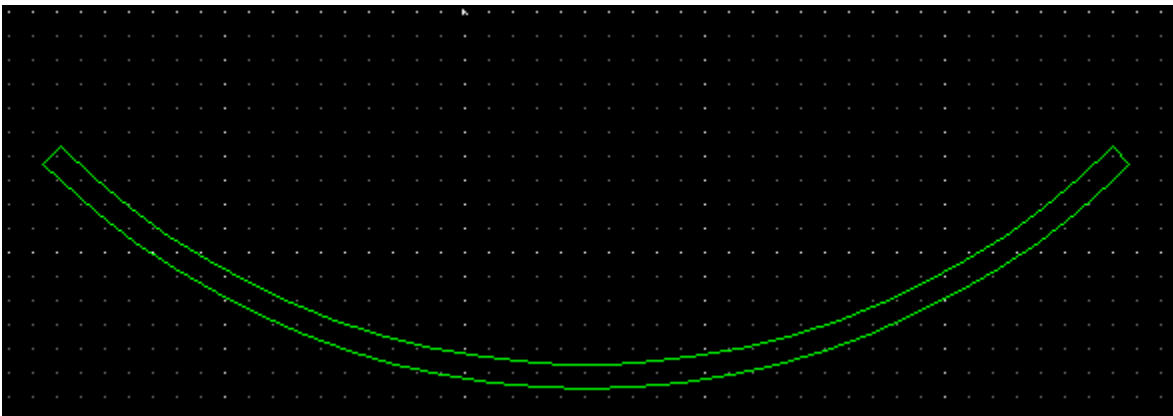
The above boards would be roughly 3.2" x 6.2" on a Panel.

Non Plated Cut outs
Non plated cut outs are created on the board layer with the board outline. One may go to Add>Board and select a shape. The smallest non plated cut out dimension available is .031" on a Basic or Normal Expanded spec board. For a premium Expanded spec board, the smallest non plate cut out dimension is .02".

Editing the soldermask
That answer is above in the Component Creation section.
- Be aware that all objects on the soldermask are clearances in the soldermask and everything that is black in the soldermask will have the green soldermask film covering.

Via pads
Make the via pad style 010" larger than the drill size. So for a .018" via hole, make the pad style width .028". Make certain that you are using the Via style in Settings>Styles or the Design Rule Checks will not know they are intended as vias and will flag them as annular ring errors.

Blind and Buried Vias in PCB Artist.
PCB Artist currently does not easily facilitate Blind and Buried vias. It outputs only one drill file. Drill holes appear through the board completely, and the pads associated with them will also appear on all layers normally. Even if the drill hole appears on a few layers rather than through the board, the pads often appear still on all layers.

However, with notes and the use of a document layer, one may illustrate what drill holes appear on what layers.

An Example would be:
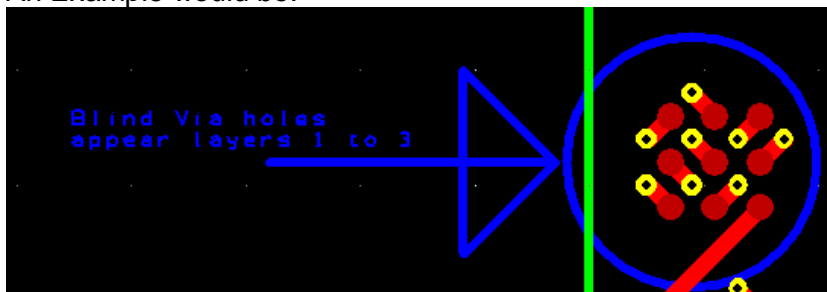
Adding personal logos and symbols to the Silk Screen or Copper layer
To add personal logos into PCB Artist, one option is create the logo in a font editor as a new Font. There are many font editors available online. Once the logo has been created in the editor and connected to a particular keyboard key, save the file as a True Type Font with a distinct name.   Install the font onto your system, but keep it handy as it will need to be sent to Advance Circuits to allow processing of your PCB Files. To place the logo into PCB Artist, one will place it as a text using the newly created font with the logo. Be sure to send PCB Artist the true type font file. Make sure the true type font's name is very specific to avoid possibly overwriting the fonts in our database and the possibility of your own personal font from being overwritten.

The next option would be to import the image as a DXF file. There are many Raster to Vector image converters available online if the Logo is in Bitmap format. Otherwise one may draw the logo entirely in Vector format, save it as a DXF file and import it.

Directions for DXF import are found **here**.

Long delay when moving components on the PCB
The software is also capable of performing a dynamic optimize during component move. When moving a component, right-click and select the Place option from the context menu. Then move the component again and right-click and choose Dynamic Optimize. This will remain a default choice for all parts moved on all future designs, until deactivated.   To deactivate it, one may do the same option as above, and uncheck Dynamic Optimize.

Creating a new Net Class
The most common net classes in PCB Artist is signal, power and ground. These classes reflect not only the track types and widths, but will reflect the type of vias added the connections. Net classes may be used to control the defaults for track widths. To create a new net class one may go to the Settings> Net Classes and click the "Add" button. In the Net Class pop up, there will be options to set the minimum and maximum widths of the tracks for this class, and options on how the vias will be set up. Once created the new net class can be assigned to a net, and the net will reflect the properties of that new class type. More one Net Classes will be found in Help>Context Help, under Net Classes.

Custom Colors for Nets and Net Classes

Options for adjusting custom colors for nets or net classes may be set in the colors menu (hotkey: C). This will allow users to display any net or Net class with a specific color indicator (often a colored centerline highlight).
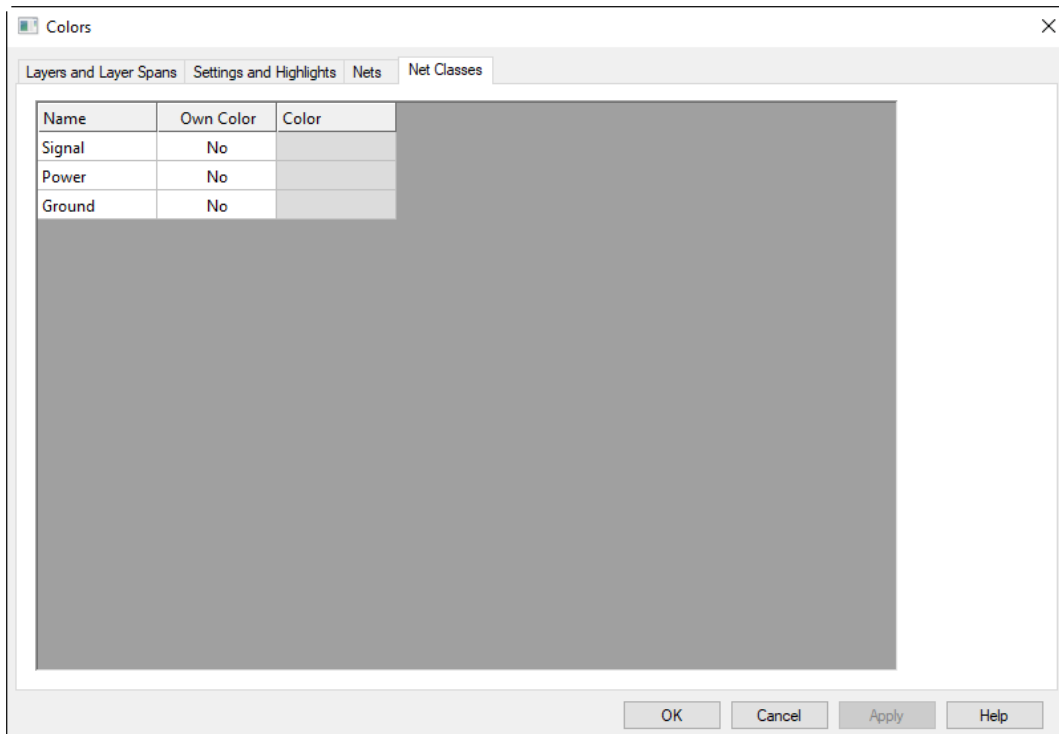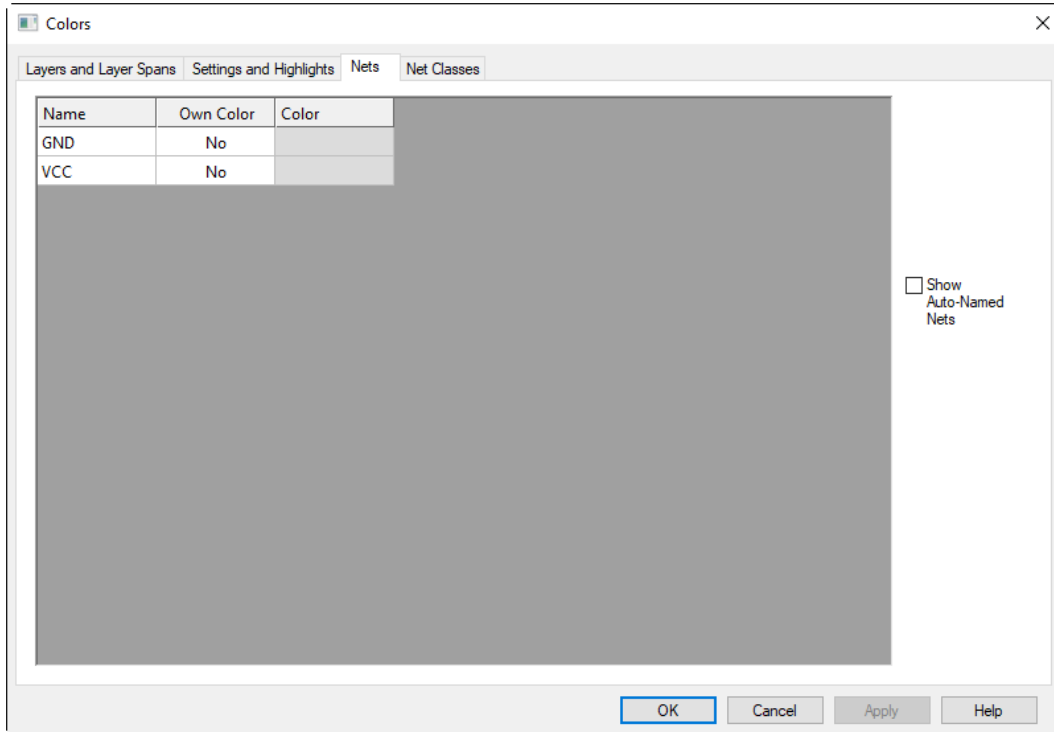
Disconnecting Combined Nets

If two different nets are combined on the pcb file, the user will receive an error while connecting.
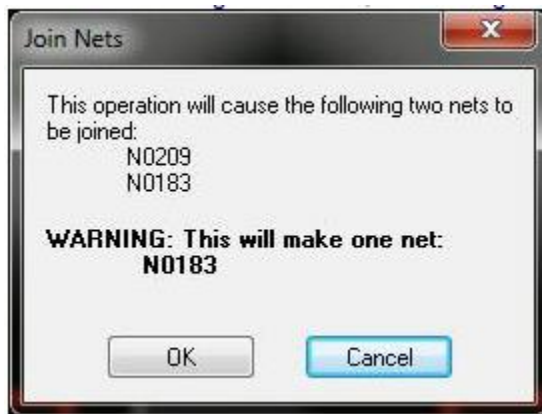
This will assign the two nets the same net name, combining them. Once this has been done, it can be difficult to separate these nets and all features between them into to two separate nets again.

The first and easiest option is only available to users who started with a schematic in PCB Artist and converted it to PCB. The option to forward design changes from the schematic will fix any netlist changes, intentional or not. If tracks have been created with the incorrect net name, it is possible the changes will remove tracks, but correct the net connections. This is why it is suggested one should always do changes on the schematic first and forward changes. The option to forward design changes appears at Tools>Schematic⇔PCB>Forward Design Changes.
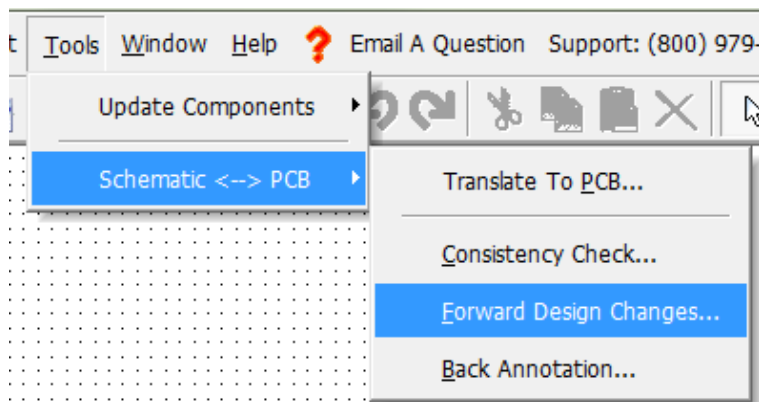
An ECO from an OrCAD Netlist will do the same thing, if the board had initially been started from a netlist.

If a user did not start with a schematic, they will have a number of tasks to separate objects that are now on the same net.

First, one must unroute any tracks and copper pours that are connecting to the pads that are not on the correct net. For tracks right click on the track and select Net>Unroute

Track Path. For copper pours, right click on the pour and select Unpour copper. One will be given an option when repouring the copper the change the net the pour appears on.

Once disconnected, the pads can then be removed from their incorrect net. Right click on an unconnected pad and select Remove From Net.
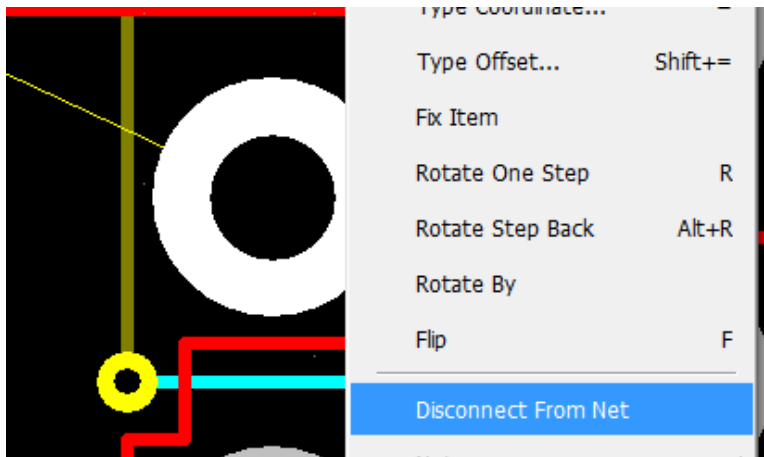
Do this for every pad that will be in the separated net. Proceed now to right click then on the unconnected pads again, and select Add to Net.
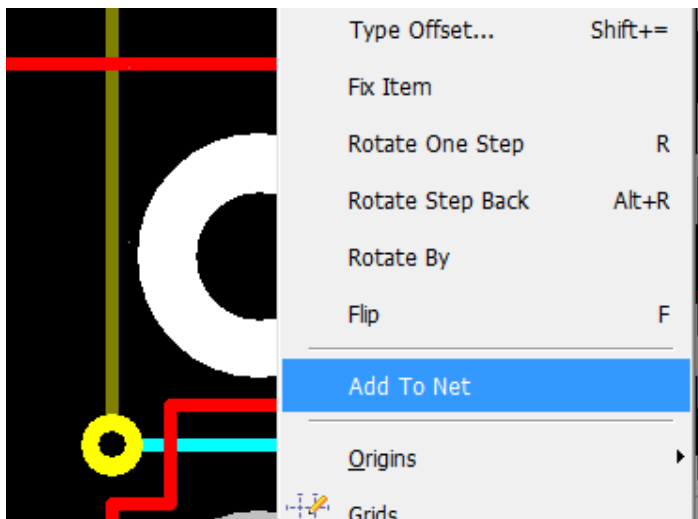
Add the pads to the correct net. Connection "Rats nests" will begin to appear on the board that should state the correct net.

Because the pads are now on a different net, the routes generated between them will also now be on that different net.

Pour Copper area
1. Draw the bounding box polygon Using Add>Copper Pour Area.
   - You can draw right over the rout and the poured copper it will clip itself back from the rout by the exact amount required (which is set in Settings>Spacings).
   - The copper pour will clip itself back from all objects in the area that are not on the same net automatically (the spacing used is set in Settings>Spacings).
   - In Settings>Spacings, the Shapes row governs the distance the copper is clipped back from the copper pour. The entire Shapes row must always be set to no less than .010".
2. Click on the Selection Icon to exit draw mode (from the top icon bar, the last icon on the right that looks like your cursor).
3. Click on any one segment of the bounding box you just added.
4. Right-click and choose Pour Copper from the right-click context menu.
5. In the dialog box, choose the net it should be on. Leave that text field blank to have the copper pour unconnected from a net.
6. Click on the Thermal Pads checkbox to automatically add thermal pad connections.
7. Click on the checkbox to remove Isolated Areas at the bottom of that dialog window. Isolated areas will cause rats nests to show those areas as unconnected. Isolated areas will behave as an antenna on the finished board. Another option is to use the Highlight checkbox and connect those isolated islands with vias to their proper net.
8. **IMPORTANT** - The copper pour is applied once to the area it fills. If changes are made after the copper pour (such adding vias, mounting holes, or moving tracks) the copper pour will NOT adjust to this new change. To have the copper pour adjust to those changes, simply select the bounding box of the copper our, right click and click Pour Copper again. The Pour Copper dialog box will appear with the exact same settings of the previous pour. Click OK and the pour will be adjusted considering the changes made to the area.
9. **IMPORTANT** – The copper pour bounding box is not actually a copper object on the board, it will not intersect traces or end up in the actual board. Do not delete the bounding box.

Filling copper pour area
The edit is not yet complete. Here is how to complete it:
1. Select any line or arc of the copper pour boundary that you have drawn on the copper pour area and right click.
2. Select properties from the right click menu.
3. Select the Pour Copper Option
4. In the next window, select the nets the copper pour will be connected to.
5. **IMPORTANT** – once the net is selected, several checkbox options in the widow become active. To have thermal connections, the checkbox for Thermal Pads MUST be checked in this dialog box.

Copper pour outline box
The bounding box of the copper pour remains after the copper pour and does appear to intersect all traces it crosses. However it is not really present and is gone when the data is output. **WARNING:** Do not delete the copper pour's bounding box in case you need to re-pour the copper again at a later stage.

Adding items in an existing copper pour
Select one segment of the bounding box, right-click and pour copper again. It will re-examine the connected tracks, holes, routs, etc and re-pour to clear back from them. Some notes on this:
- All previous settings of the previous copper pour will be preserved in the dialog box. So if those do not need to be changed, just click the OK button to re-pour.
- New isolated areas may be created by these new edits. If the Remove Isolated Areas check box is checked, these new isolated areas will be removed automatically.
- If you DO NOT re-pour the copper, all those new edits will cause shorts to each of those individual nets.

Copper pour around the rout edge
You can draw the bounding box right over the rout edge and it the copper pour will automatically clip itself back from the rout to the exact amount necessary.

Thermal connections on plane layers
In the New PCB Wizard, either no net names were assigned to the planes or they were not spelled exactly the same as the net name (proper case is required as well).
1. To fix this, go to Settings>PCB Configuration in the PCB file and go to the Layers screen.
2. Use the drop down menus for assigning the planes to the correct nets to ensure that nothing is misspelled or has a different case.
3. Go to View>Powerplane>Show and view each plane to confirm they now have thermal connections.

Improved Thermal Connection Options
Pad Properties dialog has more control over thermal connections on power planes and copper pours.
Copper pour areas can now override this thermal information.
Pour Dialog now shows what thermal parameters will be used when pouring a selected area.
1. Copper pour areas in previous designs will be converted to use the equivalent new rules.
2. If a thermal pad has a track on it on the area layer it will now be a valid replacement for a thermal spoke.
3. More options for pads Thermal connections are available on all pads and vias through the pad's properties (a right click option).
This was available on vias and pads within a footprint, but is now also available on component pads and free pads in a PCB design.

See PCB Artist help pages for more details.

Mixed and split planes
On outer layers, just use Pour Copper. On inners, DO NOT make the intended inner layers plane layers, keep them as signal layers. This can be modified while in the design from Settings>PCB Configuration on the Layers page of that wizard. You can then create split and mixed planes using Pour Copper just as it would be done on the outers. Never try to draw on a true plane layer, not even a single trace.

PCB Artist Import/Export
Currently, PCB Artist may import .dxf files for mechanical functions (like outlines and references) and OrCAD EDIF format netlists in place of using a schematic. PCB Artist is able to export .dxf format, and allow printing to .pdf format (See Create a .pdf File). One will be able to export all layers but the copper circuit, in .dxf format. Gerber files are generated by request of the customer on boards that have been ordered and processed by the Advanced Circuits CAM department.

How to import a DXF format file
One may import a dxf format file into PCB Artist, using the File>Import function. A pop up will give options for import.
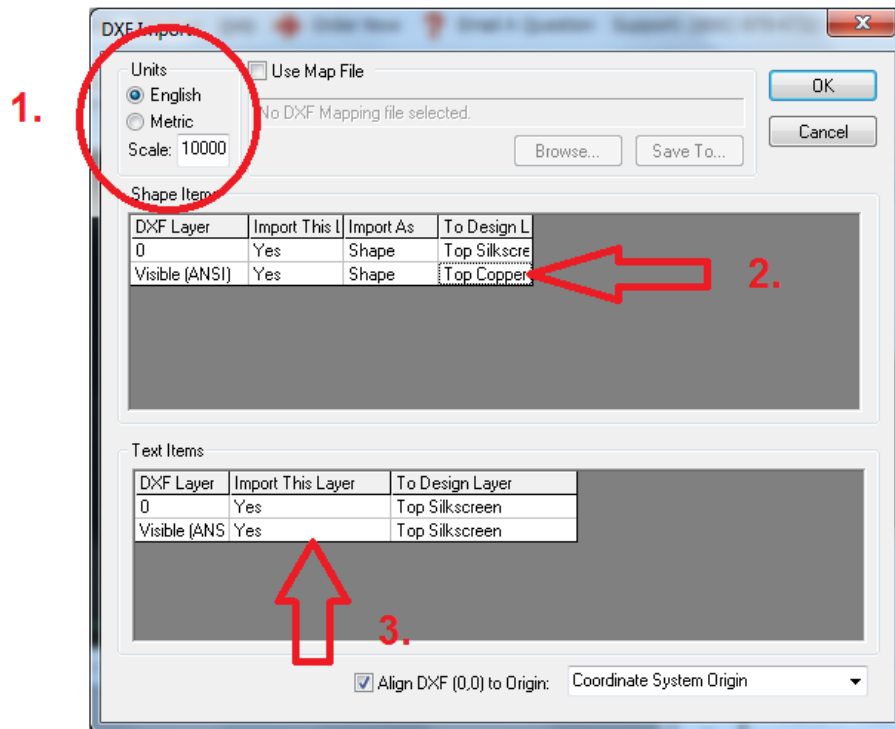
1. This will control scale. Often when a dxf is imported and the software does not show the scale may be far off.
2. Double clicking here will show a drop down for what layer the information will appear. The options of shape affect how the information will appear in PCB Artist. It is not recommended to import tracks and pads from DXF to any copper layer. It's best to use the import DXF function for mechanical reasons such as board outlines. To import an outline the Import as will need to be change to 'Board".
3. It is not recommended importing text. Text information and fonts do not carry over well from DXF, though the option is still there. If one was to double click under "Import This Layer", they will have an option to change the import from yes to no.

Import OrCAD EDIF Netlists
Instead of using a schematic generated in PCB Artist. There is an option to use an OrCAD EDIF format netlist instead. The option to do this is through File>Import. Before doing this one must have all the components already build into PCB Artist. One must also be aware of all the component names found within PCB Artist and the OrCAD drawing. When importing the netlist into a fresh PCB Drawing, The designer will need to "Edit Map". This pop up will initially be all blank values. These value are manually input to create an OrCAD.map file that PCB Artist will use as reference to match the netlist in parts and net connections. More about Importing OrCAD EDIF netlist may be found in Help>Context Help under OrCAD Import.

ECO From Netlist
To update any changes done to an OrCAD EDIF netlist into PCB Artist one must use the ECO From Netlist function. This is available in File>ECO From Netlist, and requires the updated OrCAD EDIF netlist. Like Importing an OrCAD EDIF netlist, if any new parts have been added, the parts will also need to be creating in PCB Artist, and the OrCAD.map file will need to be updated. More about ECO from Netlist may be found in Help>Context Help under ECO from Netlist.

Importing From Eagle
Native Eagle Schematic (.sch) and PCB (.brd) XML format files can be loaded into PCB Artist by either using a drag and drop from the Browser onto the PCB Artist product or by using Open from the File menu. They can also be used in the Add File option within the Library Manager.
Native Eagle Library (.lbr) XML files can be read directly into the libraries using the Add File mechanism in the Library Manager.
This does not include downloaded single parts, as they are scripts to build the part in the Eagle software. Once the part is built and saved into an Eagle library, the library file can be imported into PCB Artist.

Alternatively, PCB Artist is able to import PCB designs, schematic designs, component, pcb symbol, and schematic symbol libraries from Eagle versions 4.01 to 6.x. This requires having the Eagle software on hand as the file formats will be ran through the ULP function.

See PCB Artist Eagle Import help page for more detail.

The Auto Router

Auto route with fine pitch components

There are a few different factors that constrain the autorouter with specific design rules. The key to successfully autorouting tighter pitched parts is in reducing the various rules the autorouter must obey:

1. Power and Ground tracks: Failing to route is often due to the trace width size being too large, especially power and ground tracks. The default power and ground sizes are quite large and can perhaps be much smaller, depending on the power requirements of the board.
   A. The track size used by the autorouter is determined by the Net Class assigned to a net from Settings>Nets. Power Class and Ground Class will be routed with the Power Nom track style and Signal Class nets (the default class to any net) are routed with the Signal Nom track style.
   B. To change the tracks sizes used, go to Settings>Styles from the main menu. Click on the Tracks Tab.
   C. From their click the Tracks tab.
   D. Click on the Power Nom track style. The default size is 50 mils, which will not be able to connect to a .5mm pitch SMD.
   E. The trace width can be changed there for the entire board including traces not yet added to the board.
   F. **IMPORTANT:** Please be sure to properly calculate trace widths that will accommodate the power requirements of the PCB. This link will take you to the 4pcb.com trace width calculator. If the track size is made too small for the board's power requirements, the track may become a fuse.
   G. The autorouter will always use the Power Nom track style to draw the power traces unless the Minimum Width checkbox on the Autorouter checkbox is checked. In that case it uses Power Min. All other track styles are ignored by the autorouter.
   H. Style settings can be saved (from a blank PCB file) from File>Save As Template if the sizes changed will frequently be used in other designs. (It is best to save templates from blank designs, so as not to accidentally save other features into the template, such as the board outline).

2. Signal Traces:
   A. To change the traces sizes used, go to Settings>Styles from the main menu.
   B. From their click the Tracks tab.
   C. Click on the Signal Nom trace style.
      o Track width can be set at small as 7 mil and be usable with Basic Design Requirement.
      o With Expanded Design Requirement, the traces can be as small as 5 mil.
      o If a signal track width is changed from the default for either Design Requirement type, go to Settings>PCB Configuration. In the

Board Parameters page of that wizard, change the Min Track Width/Gap to match the new smallest track on your design.

D. The trace width can be changed there for the entire board including traces not yet added to the board.

E. The autorouter uses the Signal Nom style from Settings>Styles in the Tracks tab to draw all signal tracks. If the Minimum Width checkbox is checked in the Autorouter dialog, it will instead use the Signal Min track style.

F. From their click the Tracks tab.

G. Style settings may be saved (from a blank PCB file) from File>Save As Template if the sizes changed will frequently be used in other designs.

3. Another factor in autorouting is the spacing requirements that the autorouter is constrained to follow.

A. If you go to Settings>Spacings, change the pad to track spacing.
   o One may set the spacing to as low as 7 mils with Basic Design Requirement.
   o One may set the spacing to as low as 5 mils with Expanded Design Requirement.
   o One may set the spacing to as low as 3 mils with a Premium Expanded, this is still marked as an Expanded in PCB Configuration.
   o If a spacing is changed from the default, for either Design Requirement type, go to Settings>PCB Configuration. In the Board Parameters page of that wizard, change the Min Track Width/Gap to match the new smallest gap on your design (unless you already changed if for the track width, and it is the same number).

B. You can change whole rows or the entire grid by changing one cell, then clicking out of it, then right clicking over the cell you changed and choosing Apply to Row or one of the other options.
   1. You can set the spacing to as low as 5 mils with Expanded Design Requirement.
   2. If you Apply to All, you need to change the Board row back to 20 and Apply it to the whole row. The minimum allowable is 10 mils for a tab rout or rectangular rout and 15 mil for v-scored designs.

C. Autorouter Grid:
   1. In the Autorouter Dialog, a change to the Track Grid Size can free the autorouter to find new ways to connect.
   2. The grid should be set to a derivative of primary grid. (For example, a 25 mil grid will not rout to a .5mm part. A 12.5 mil or 6.25 mil grid will allow the autorouter the freedom to do its job.)

D. Change Via Styles:
   1. Got to Settings>Styles in the Pads tab.
   2. Change the Via style to as low as .015" hole (with .025" pad) for Basic Design Requirement, as low as .008" (with .018" pad) for Expanded Design Requirement.

3. If using Expanded Design Requirement and using a via smaller than .015 you must go to Settings>PCB Configuration and on the Board Parameters page of that wizard change the Minimum Hole drop-down to .010" (if you are using a pad smaller than .015" but larger or equal to .010") or .008" if using a size smaller than .010".

Keep out area

Keep out areas are created through the Add>Copper Pour Area menu option. After the Copper pour area outline is created, right click on the outline and go to properties. In the properties check the "keep out box". The keep out area will stop any copper pour from crossing into the keep out area. The keep out also will work on power plane layers (if the keep out is drawn on said layer). This does not show however in the View>powerplane option view.
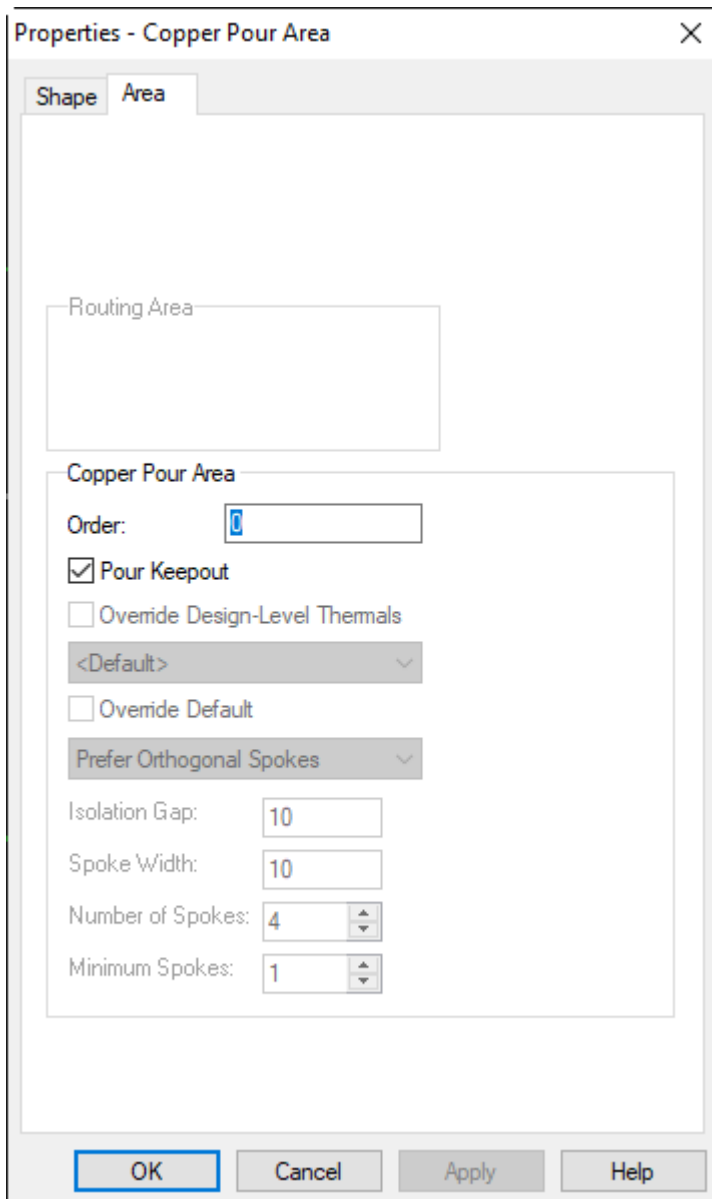
Track Routing Areas

Under Add>Track Routing Area. Track routing areas are outlines that can be used to dictate to the software's autorouter where the tracks and/or vias are to appear. After placing an Track Routing area outline, right click on the outline and select properties. In the properties dialog are options to allow or hinder tracks and vias within that enclosed area. Track routing areas may be set to all layers, or a specific layer within its properties.

Design Rule Checks

Consistency check
Go to Tools>Schematic←>PCB use Consistency Check to verify the design when compared to your schematic. This will output a report to review any inconsistencies between your Schematic and PCB files. Both files will need to be named the same other than extensions, and be placed in the same folder to have PCB Artist recognize the files being connected.

Reviewing design rule check errors
Here are some effective ways:
1. Reviewing a small number of errors:
   A. Select the error message itself.
   B. Use the <n> hotkey to cycle between objects at that location until you have the error message selected.
   C. Right-click and choose Properties from the context menu.
   D. There should be a short message giving more information about the error.
2. Reviewing a large number of errors:
   A. Bring up the Interaction Bar <F9>.
   B. At the bottom of the Interaction Bar, click on the Goto tab.
   C. At the top of the Interaction bar, change the drop-down menu to Error.
   D. This will organize the errors by type and by layer. Click on the individual coordinate list to zoom to that specific error.

Deleting my design rule checks
Go to Tools>Design Rule Check. In the Design Rule Check window, click the Delete Errors button.
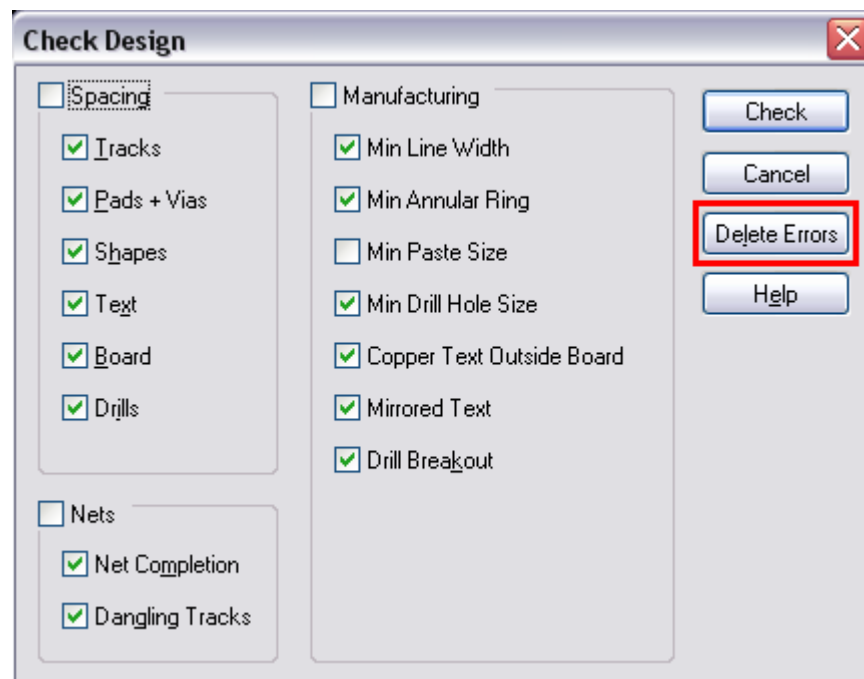
Design Rule Check errors
- General Types of Errors:
  - Some design rule errors are per design and cannot be eliminated. The application is simply looking to bring something questionable to your attention.
  - Some errors are not un-manufacturable, but violate previously set design rules. For example, a design may have a 9 mil spacing error when the spacing is set for 10. The instance can either be edited to a 10 mil space or the Settings>Spacings screen can be edited to allow 9 mil spacing for that type of error.
  - Some errors are un-manufacturable or would produce a definite violation of IPC specifications.
- The DRC Spacing checks are run according how Settings>Spacings is set. The grayed-out values cannot be changed. Other values can be changed, but may affect the pricing of the order.

Design Technology

Pad Styles | Text Styles | Line Styles | Track Styles | Nets | Net Classes | Spacings | Rules

	Tracks	Pads	Vias	Shapes	Text
Tracks	10.00				
Pads	10.00	10.00			
Vias	10.00	10.00	10.00		
Shapes	10.00	10.00	10.00	10.00	
Text	10.00	10.00	10.00	10.00	10.00
Board	20.00	20.00	20.00	20.00	20.00

OK | Cancel | Apply | Help
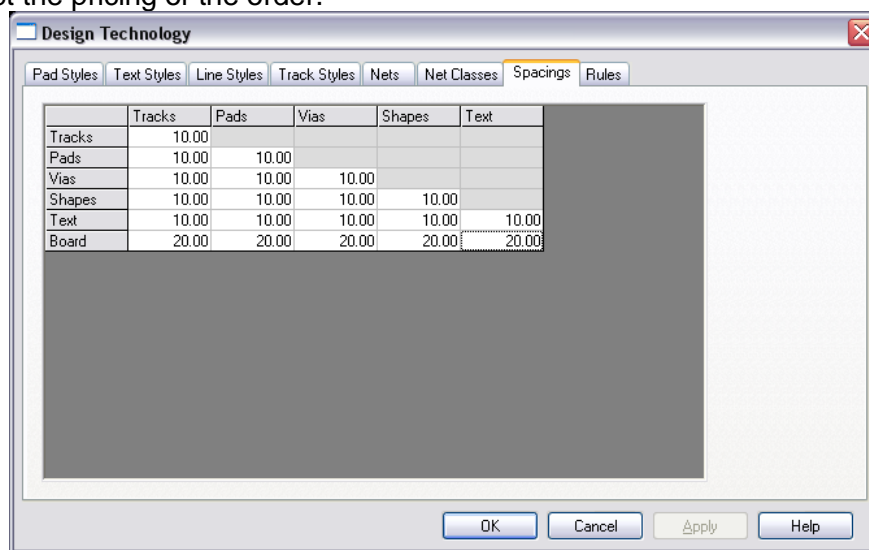
  - All Spacings Design Rule Checks are measuring the distance from copper to copper with the exception of Board spacing and drill spacing.
    - It is important to remember that Text and Shapes are on an a copper layer are going to be copper and will have all the electrical properties of a trace or a copper pour.
    - Basic Design Requirement orders cannot have a track width or space below 7 mils. Expanded Design Requirement orders cannot have a track width or space below 5 mils.
    - Spacing or Track width changes require the user to go to Settings>PCB Configuration and change the Minimum Track Width/Gap settings for the design on the Board Parameter page of that wizard. The application will not allow the order until that information is updated.
    - The Shapes row governs the distance the copper is clipped back from the copper pour. The entire Shapes row must always be set to .010" at the minimum.
    - Board spacing is for the distance between the board rout line to copper features.

- Board to copper spacings should all be set at .020" as the default in Settings>Spacings.
- For a routed or tab routed part, the settings could be set to .015" at the minimum
- For a scored part, the minimum allowable spacing from copper is .020" or the designer risks exposed copper.
  - Drill spacing is to discover if drill holes are overlapping. This can happen when another hole is placed inside a much bigger hole by accident and cannot be easily viewable.
  - Most often this is seen when a via is accidentally placed inside another pad.
    - Click within the pad where you see the error and press <N> to cycle through objects present at the point the cursor was clicked on.
    - If the via is not totally within the hole, the yellow pad of the via may be visible.
- Net Design Rules check for incomplete nets and also Dangling Traces.
  - Intentionally dangling tracks will produce an error message that will have to be ignored. Simply note the number of errors so when the order is submitted, you can see if the number of error has increased, indicating a new problem that must be investigated before ordering.
    - Add a Track to fix the error, if the connection is confirmed to be needed.
    - Use Tools>Optimize Nets (or the hotkey <Ctrl+D>) to have the software reconsider the netlist. Sometimes it will discover the connection to the net exists in a different path than it was expecting, elsewhere in the board.
    - If there was no schematic created, there is no way to verify the integrity of the design. A mistaken connection in the design can only be manually confirmed (if that).
    - Overlapping copper pour areas can cause this. The system is arranged so that overlapping polygons should never be necessary.
    - Isolated areas of a pour copper can cause this error. It is always best to re-pour the pour copper and click on the Remove Islands option. Isolated islands of copper behave as antennas.
    - Add Shape objects and filled polygons should only be used in areas too small for an effective copper pour. It is preferred to use copper pour wherever possible.
    - It is recommended to always ensure that the schematic is diligently preserved as the point of all changes (using Tools>PCB←>Schematic>Consistency check to forward differences from the schematic to the PCB), to avoid the most serious of these types of errors, even on the most simple board.
- Manufacturing Design Rules cover many different areas of board design.
  - Minimum Line Width checks for tracks, draws and text using a smaller line .005"
  - Minimum annular ring check for:

- .007" annular ring for component pads, minimum. That translates to .014" over the hole size.
- .005" annular ring for via pads, minimum. That translates into .010" over the hole size.
- Annular Ring Errors on mounting holes – The Plating checkbox in Properties has been left on. If it is clicked off, the errors will not continue to show.
  - Minimum Drill checks for holes smaller than .008".
  - Text outside board test is to flag text that is outside of the PCB and may be an indication of an unplaced component.
  - Mirrored text check is to ensure that all bottom side text is wrong-reading. Right-reading text on the bottom side will be wrong reading after manufacture of the part.
  - Minimum solder paste test should be turned off and ignored. Solder stencil services vendors should have their own DFM tests that should perform for their customer depending on many factors that are unknowable.
- If there is doubt about the correct way to proceed, email your .pcb file and a brief description of your situation to layouthelp@4pcb.com.

Arrays

What is an array?

An array is putting multiple pieces of the same part number into frame to make small boards more manufacturable for fabrication and also to reduce assembly costs for small boards. To configure your design to be in an array, go to Settings>PCB Configuration and go to the Production Page of that wizard. Click on the Array Checkbox in the middle of the page to open the array options for editing. These are the various aspects of arrays:

1. The array size must be large enough to encompass all the individual pieces, the spacing between those parts, and the border material that holds them together.
2. The Borders are used for assembly operations to clamp onto securely without overlapping onto the board and interfering with assembly.
3. Tooling holes are also used for assembly operations to pin to. This is a more secure and precise way to hole onto the board than clamping to the borders alone.
4. Fiducials are used as optical targets for automated assembly. They are used to determine precise board alignment.
5. Scoring is a "v" grove cut into the top and bottom surfaces of an array configuration of multiple PCB's or a PCB with removable rails or frame. The cut depth is set such that the material left is the lesser of .018" or 1/3 of the finished thickness. When this process is used the printed circuit boards are typically set up side by side and end to end with no gap between the edges. This results in the v-grove cutting into the area of the boards, so a .020" clearance between the edge of the boards and any conductors is required.
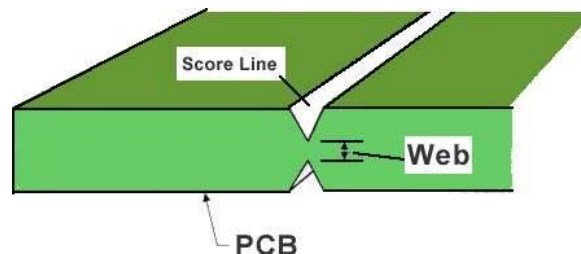
   a. When placing an order that requires scoring please indicate this requirement by setting going to Settings>PCB Configuration.
   b. On the first page of that wizard, make the board Expanded Design Requirement.
   c. On the Board Parameters page, click on the V-Score checkbox on the right.
6. Tab Routing for Arrays uses the router bit that cuts the board from the production panel to cut slots between the individual images, leaving behind small "tabs" of unrouted material to keep the array border and individual connected as one.
   a. When placing an order that requires scoring please indicate this requirement by setting going to Settings>PCB Configuration.
   b. On the first page of that wizard, make the board Expanded Design Requirement.
   c. On the Board Parameters page, click on the Tab Rout checkbox on the right.

An example of Tab-routing:
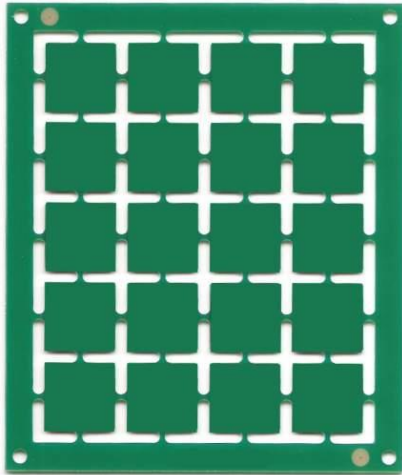
Making an array

Go to Settings>PCB Configuration. Go to the Production Page of that wizard. Click on the Array Checkbox in the middle of the page.

From here there are two options available for your array set up. The first is an array matrix. This allow a user to simply state the number boards across and up the array will contain. This then calculates the size of the array, based on the number of x and Y boards, but also the type of separation (tab-routing or scoring) of the boards. Checking the box for rails and inputting a value there will also increase the size of the arrayed board set up.

The other option allows for direct values to be input. This is shown below on how to set this up.

1. The optimum array size for efficient panel usage and lowest possible production cost is 10.8" x 7.8". The goal is to come as close to that array size as possible without going over.
2. Here is an example of a PCB that is 3.5" x 2.2 to set up in an **tab routed** array:
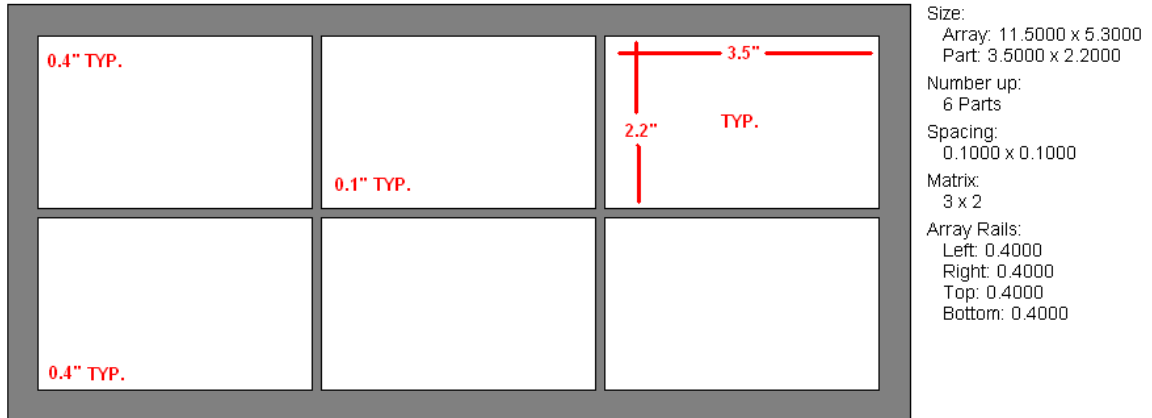
- To Determine the Array Width for PCB Artist, simply add up the x-axis dimensions. For this board, the array width must be no less than 11.5". Simply add up .4" + 3.5" + .1" + 3.5" + .1" + 3.5" + .4".
- The formula for the Array Width is the same, but for the Y axis dimensions. This array width can be no less than 5.3". That is .4" + 2.2" + .1" + 2.2" + .4".
- Borders to use:
    - The .4" border around the board is the minimum recommended for by assembly companies. .5" is often preferred by assemblers.
    - A .2" border is the minimum manufacturable and cannot be changed, other than to increase it. Assembly will be very difficult as only .1" will remain of material. Tooling holes and fiducials cannot be added to this size.
    - To Add Tooling Holes and/or Fiducials to the array, you must have a minimum of .4" border per side to the array.
- **Warning:** PCB Artist currently allows a minimum of a .1" minimum border to accommodate scored arrays and will allow an order of a tab rout with a .1" border. However, this is not possible to produce and will cause the board to go on hold pending customer authorization to increase to a .25" border (.2" at the minimum) and approval of any associated changes to the cost of the order.
3. Here is an example of a 3.5" x 2.2" board in a **v-scored array**:
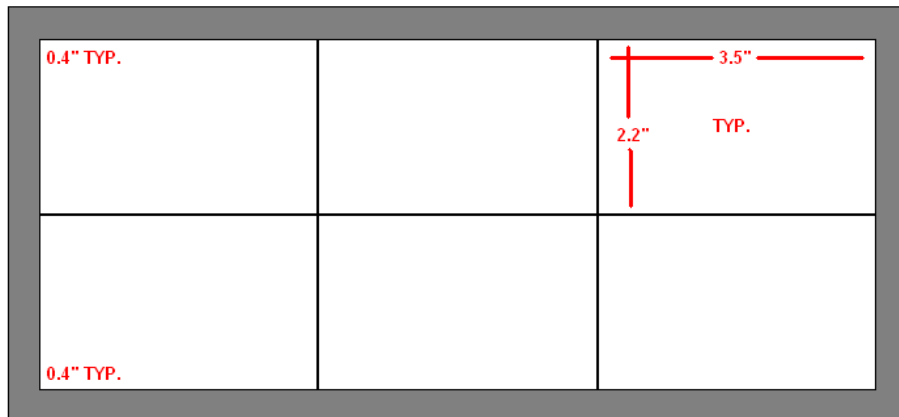
Size:
  Array: 11.3000 x 5.2000
  Part: 3.5000 x 2.2000
Number up:
  6 Parts
Spacing:
  0.0000 x 0.0000
Matrix:
  3 x 2
Array Rails:
  Left: 0.4000
  Right: 0.4000
  Top: 0.4000
  Bottom: 0.4000

- The array width on this example can be no less than 11.3". That is .4" + 3.5" + 3.5" + 3.5" + .4".
- The array height on this example can be no less than 5.2". That is .4" + 2.2" + 2.2" + .4".
- There is no space between scored parts.
- The .1" border is the minimum allowed cannot be changed, other than to make it larger.

4. Using the PCB Artist Array Dialog:
   - This example is using the tab rout.
   - Note that the Array Width and Array Height have the word "mils" at the end of each field, indicating this board is set to mils, and the units entered must be in mils, as seen in entering 11.5" as 11500.
   - Array Width is 11.5" (or 11,500 mils) as calculated above.
   - Array Height is 5.3" (or 5,300 mils) as calculated above.
   - Array up is the total number of parts in the array. In this case 6.
   - Note that the Quantity ordered is 18 parts. 18 individual parts which is 3 arrays.
     o The Quantity is for the total number of part numbers, NOT the number of arrays. Arrays are never counted, only the individual parts which are contained in the arrays.
     o The Quantity MUST be a multiple of the number of boards in the array. It would give an error if I made the quantity 17 or 19. However, it would accept 6, 12, 18, 24, etc.
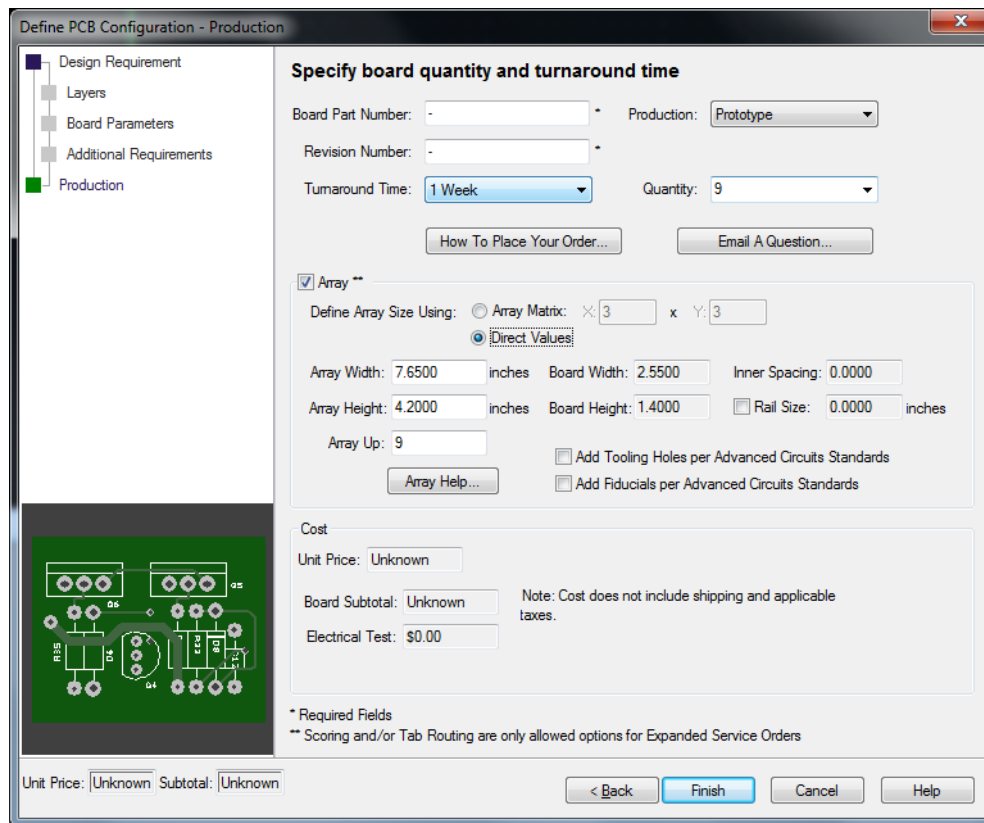
- **IMPORTANT** – Basic Design Requirement orders are heavily discounted and do not include v-scoring or tab routing. To get v-scoring or tab routing of an array, you must go to Settings>PCB Configuration and set the board to Expanded Design Requirement. Then go to the Board Parameters page of that wizard and click on the checkbox for either v-score or tab routing (or both).

Copy and paste arrays

It is possible but not recommended.

- The schematic will now not be able to verify the design
- The silkscreen reference designators auto-number, meaning that R1 on one part will not be R23 on the second part pasted into the array and r41 on the next part pasted in. This will reflect on the BOM, but basically very problematic in production assembly.
- It is possible that in selecting the part across several layers that something may be missed and not copied, which a schematic comparison will now be unable to catch.

Order Process Questions

Placing an order
This link takes you to instructions to the ordering a Basic Design Requirement order.
This link leads to instructions to place an Expanded Design Requirement order.

Directions for ordering a Student Special board may be found here: Student Specials

Student orders follow the same specs as 33 and 66 specials. You may order less than 4 boards on a student order.

33/66 Specials and Barebones Boards in PCB Artist:
All requirements are similar to the specials listed on their respective pages.
- Directions for ordering a Barebones Board may be found here:
  http://www.4pcb.com/BarebonesOrderProcess.pdf
- Directions for ordering a 33 Special board may be found here:
  http://www.4pcb.com/33-66EachOrderProcess.pdf

Because of the nature of promotion codes the pricing for them will appear on the web ordering page after you've uploaded the files.

$33 and $66 specials with multiple circuits (Multi-part boards) are subjective to a one time per order 50$ fee due to set up. This multipart drawing is set up by the customer only, not by Advanced Circuits CAM (they will not array it for you in this case). If this is a case, the customer will need to approve the "step and repeat" charge by noting it in Settings>PCB Configuration>Additional Requirements, in the Special Requirements box. $33, $66, and barebones specials do not have internal routing available, either for cut outs, scoring, or Tab-route.

Promotion Codes
The location of a promotion code being applied to ones order will appear differently in the software, depending on the spec of the board.

Basic spec boards will only have Barebones, 33 Special, 66 Special, and Student Special promotions available to them. These are placed here:

Student specials will have other steps to take to ensure the correct promotion will activate. Be sure to review their respective directions (Links are in this Order Processing section).

Other promotion codes are placed when ordering Expanded Spec boards through PCB Artist. The option will appear on step "3 of 5" of the ordering process, just after uploading files.

If the user wishes to have the promotion added to a Standard spec board, they will have to contact a sales representative to have the board quoted Standard spec.

FreeDFM
PCB Artist does not interact with the Advanced Circuits Freedfm Service. Go to the Order Processing section for how to submit your board order. The Design Rule Check Section covers how to view any possible errors with the design that may reflect on the manufacturability of the board.

Errors when ordering
The submit order process automatically runs the Design Rule Checks as part of Submit Order to prevent the possibility of not running them before ordering and also to catch accidental edits that the customer may not have been aware of.
- If the Design Rule Checks have not been run, it is critical to check them before submitting the order.
- If the checks have been run and the number of errors is verified as per design, click to ignore the errors and continue the order process.

Other Errors when ordering:
- Missing Revision or File Name or Number

  In Settings>PCB Configuration on the Production section there are two values that will need to be filled out. The Board Part Number and Revision are required to order. Once done, click finish and submit the board again.

Error

The "Board Part Number" cannot be left blank when submitting an order.

You can enter a value using the "Production" page of the "PCB Configuration" wizard on the "Settings" menu.

OK
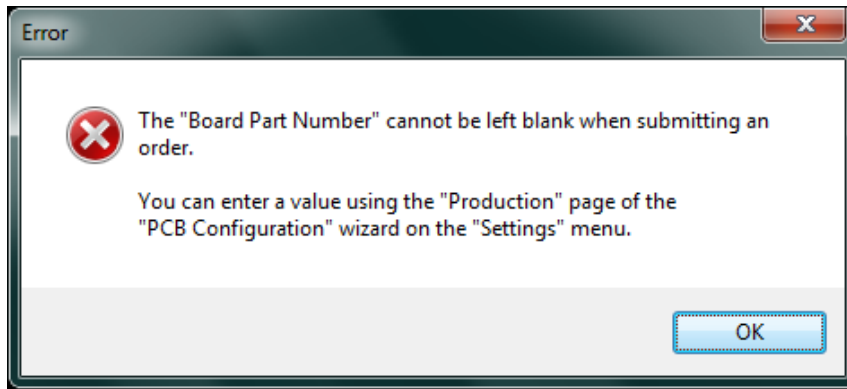
PCB Artist information missing
Step 1 of 5

In order to complete your order you will need to close this window and return to PCB Artist to correct the following:

Error: revision is missing.

Please Close this window to correct the issue in PCB Artist and then resubmit

- Items on an Automatic Generated Power Plane.

Power planes (an option within PCB Configuration) are drawn in negative. Any feature added to that layer will actually remove copper when the boards are produced. To view a power plane, go here.
If a feature needs to be added to the layer, then the layer cannot be an automatically generated power plane. The option will need to be turned off in PCB Configuration. After the features are added, a copper pour may be used on the same net as the original power plane.

PCBArtist

The PCB design has items on pure plane layer 'GND Plane'.

Plane Layers created in the New PCB Wizard Layers Page or from Settings>PCB Configuration on the Layers page cannot have any drawn objects on them at all. Powerplanes can be viewed from View>Powerplanes>Show. To create Mixed Planes or Split Planes, go to Settings>PCB Configuration and on the Layers Page change the plane layer to a signal layer. The layer should then be made a plane using Add>Pour Copper.

Do you wish to proceed with your order?

Yes   No

The .fab file
The .fab file is a copy of the .pcb file you were working in when you hit the Place Order or Get Quote button in the Submit Order process. It also serves as a "frozen" version of the exact file that was ordered. If your order has issues and you need to send an edited

file, do your edit in the .pcb file and email the edited .pcb file to us. The .fab file will not be up to date and should only be used when ordering online.

Missing Bottom Silkscreen on a Order
By default PCB Artist will not order a bottom silk layer. This must be activated by checkbox in Settings>PCB Configuration>Layers Section. If the bottom silk option is not checked then it will not output, nor be ordered.

Where do I get a quote?
Where one gets a quote through PCB Artist will depend on the type of board that is being ordered. A basic spec board will have the pricing without shipping or tax in Settings>PCB Configuration, on the lower left. Basic spec boards have a discount for them as they are ordered online and are considered a "Hands Off" board. The full price will be available when ordering the board, after uploading the files, and selecting shipping options.

Expanded spec boards use the custom spec pricing matrix found online. To get a quote on an expanded spec board, one may go to Output>Submit Order. Because it is an expanded spec board, the option "Get Quote" will be available, but not the Place Order option. Make sure that you have no web browser open. If you have an online account, list the e-mail and password, then click the "Get Quote" button.   If you do not have an online account yet with Advanced Circuits, then please go here to sign up for one before trying to order or get a quote: https://www.my4pcb.com/Net35/Registration.aspx

ITAR Required Boards
If International Traffic in Arms Regulations (ITAR) is required on an order, one can note it in Settings>PCB Configuration>Additional Requirements section in the Special Requirements box. If ordering through a sales representative, one will need to state to sales this is an ITAR board. During the online order process you will be asked if ITAR is required, in order to complete the processing.

 "International Traffic in Arms Regulations" (ITAR) govern the export of weapons and weapons-related technology from the United States. If your board design data is ITAR-controlled, please inform us of that when you place your order. For detailed information on ITAR, go to http://pmddtc.state.gov/.

Installation

OCX control error

This is a known issue with some revisions of a Windows file. The fix depends on your operating system. It is worth checking the date and version of the MSFLXGRD.OCX file on your system. There are several different varieties of this OCX file in the field, some of them quite different in size and date but showing the same version number.

- Windows 7 and Vista 64:
    1. Right click on Start/Accessories/Command Prompt and select Run As Administrator.
    2. Type: CD C:\Windows\Syswow64 <enter>
    3. Type: regsvr32 msflxgrd.ocx <enter>
    4. Type: exit <enter>
    5. Right click on the PCB Artist program icon and select Run As Administrator.
    6. From the PCB Artist main menu, go to File>Libraries
    7. Go to the Components Tab, select any component from the left-hand list, and click the Edit button from the central list of buttons.
    8. Exit without saving and Close the PCB Artist program.
    9. Restart the PCB Artist normally.
        - The key point here is that the "Run As Administrator" option is only available from a menu or shortcut, so you need to launch the command prompt itself in this mode by finding it on the Start menu.
- Windows Vista 32:
    1. Right click on Start/Accessories/Command Prompt and select Run As Administrator.
    2. Type: CD C:\Windows\System32 <enter>
    3. Type: regsvr32 msflxgrd.ocx <enter>
    4. Type: exit <enter>
    5. Right click on the PCB Artist program icon and select Run As Administrator.
    6. From the PCB Artist main menu, go to File>Libraries
    7. Go to the Components Tab, select any component from the left-hand list, and click the Edit button from the central list of buttons.
    8. Exit without saving and Close the PCB Artist program.
    9. Restart the PCB Artist normally.
        - The key point here is that the "Run As Administrator" option is only available from a menu or shortcut, so you need to launch the command prompt itself in this mode by finding it on the Start menu.
- Windows XP/2000/ME/98/95:
    1. From the Start menu, choose Run
    2. Type CMD (or on Windows 95/98/ME, type COMMAND), and click OK.
    3. Change directory to the folder containing your Windows system by typing:
        CD \WINDOWS\SYSTEM32 <enter>
        - On some systems your Windows files may be somewhere different, such as \WINNT.

- Use the Windows Search feature to locate the file if necessary.
4. Type: REGSVR32 MSFLXGRD.OCX <enter>.
5. You should get a message telling you that the control has been registered.
6. If you get an error message when running regsvr32, perhaps suggesting that although the Flexgrid control is present but cannot be registered.
   - Download the file from this link:
   - The file is compressed, unzip it and put it the unzipped file in CD \WINDOWS\SYSTEM32.
   - Repeat the process starting at step 1, above.

Permissions accessing a file during installation

This is a very old Windows XP issue that has come up many times over the years.

- ❖ **WARNING:** Advanced Circuits recommends seeking the aid of or retaining the services of an experienced IT specialist to deal with this Windows issue. Customers performing this edit do so at their own risk. It is possible to cause irreparable harm to the computer and lose all files on the computer by performing any edits to the registry.
- ❖ **Before Editing:**
    1. **Backup** – This is the process for backing up the registry prior to editing it.
        A. Go to Run and type: Regedt32 <enter>
        B. Click on File>Export
        C. Save the file to a directory and make a note of its location.
    2. **Restore** - If the computer needs to be restored to its previous settings before the edit:
        A. Run Regedt32 as was done in step a, above.
        B. Click on File>Import
        C. Locate the file from Step c above.
    3. If the computer will not boot up:
        A. Press F8 while it is trying to boot.
        B. Choose the option to restore the last known good configuration.
- ❖ **The Edit:**
    1. Run regedt32.exe (It has to be regedt32 **NOT** regedit).
    2. Select the window HKEY_LOCAL_MACHINE ON LOCAL MACHINE.
    3. Select the SOFTWARE key.
    4. Click SECURITY (or PERMISSIONS) from the menu bar (under the "Edit" menu).
    5. Click PERMISSIONS.
    6. From the Security section, highlight USERS.
    7. Check the option FULL CONTROL.
    8. Click APPLY.
    9. Click the ADVANCED button.
    10. Highlight USERS from the Permission Entries section.
    11. Click VIEW/EDIT.
    12. From the permissions section, uncheck the DELETE option from the Allow column.
    13. Click OK.
    14. Click APPLY before continuing to the next step.
    15. Check the option RESET PERMISSIONS ON ALL CHILD OBJECTS AND ENABLE PROPAGATION OF INHERITABLE PERMISSIONS.

16. Click APPLY again.
17. Click YES at the security dialog warning box.
18. Click OK to the message about registry editor not being able to set permissions on some keys.
19. Click OK back at the access control settings for the SOFTWARE key.
20. Click OK back at the permissions for SOFTWARE dialog.

Proxy firewall installation problem
Please send an email describing the issue to layouthelp@4pcb.com.

Directory of the design
There is a link to that directory in Start Menu>All Programs>Advanced Circuits>PCB Artist Data.

- In Windows XP and Windows 2000 the directory path is: C:\Documents and Settings\All Users\Documents\PCB Artist. This directory structure allows PCB Artist files in that directory to be stored in a way that allows all users of that computer to open and access the file.
- In Windows 98 the directory path is: C:\Program Files\Advanced Circuits\PCB Artist.
- To make files accessible to only the user, in the save dialog box navigate to My Documents and save the files there. The files cannot be shared with all users from that directory.
- PCB Artist will remember the last directory it saved a file to and will default to continue to save there as is typical in many Windows applications.

Other operating systems
At this time only Windows systems from Windows Vista to Windows 10 are supported.

- Some users are successfully using PCB Artist on the following operating systems with emulation software:
  - o Macintosh using Parallels
  - o Macintosh using Virtual PC
  - o Ubuntu Linux using Wine

Downloading PCB Artist through www.Download.com
Download.com (CNet) has included a mini downloader that will need to be installed before the actual download of PCB Artist. More information on the installer may be found here: CNet Installer FAQ. This installer is an extra step in security for CNet. The downloader will also have options for other software applications to be installed, that are not required for PCB Artist to be downloaded and installed. This extra software is not supported by PCB Artist, or Advanced Circuits, so the user will download them at their own discretion. Some security software may flag the CNet downloader, and not allow downloading. One will have to review their security if this is an issue. Contact PCB Artist support at Layouthelp@4pcb.com if you are still unable to download from this location.

Assembly

Optimizing the design for Advanced Circuits Assembly Services
If you right click on a component and choose Properties from the right-click context menu. Then click on the Values tab. From there you can enter values for the Manufacturer, Manufacturer Part Number, Distributor, and Distributor Part Number.

- If Advanced Circuits is providing a turnkey service of ordering the parts for an order, this information will tremendously reduce the amount of time spent ordering the parts. This would eliminate a large amount of redundant communication with the customer.
- If the customer is providing the kit for the assembly, these populated values will make it easier for the customer to have a comprehensive list of exactly what to order.
- PCB Artist will create files that help to optimize the assembly process. These files are embedded in the board design and only available for Advanced Circuits Assembly.
- Do not put via holes inside surface mount lands, unless there is a critical electrical reason.
- Always click on the Thermals Pads checkbox in the copper pour dialog when pouring copper. Via holes can be directly connected to the plane and is a separate checkbox option.

Getting a quote for board assembly
Contact your region's sales representative for information regarding board assembly.

Creating a Bill of Materials (BOM) and/or Component Positions Report
Assembly reports may be found under Tools>Reports. They are available in text or Excel formats.   It is up to the board designer to review and correct these reports for parts information and placement.

Adding fields to the BOM
One will be able to add new fields by creating them as Values first on the component.
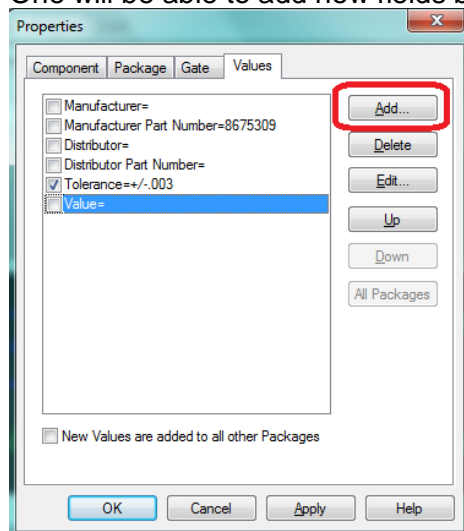
Under Output>Reports, it is best to copy the BOM report then edit it. In the new bill of materials report options select the edit button. In the next window select edit again. It is here that one may add the value, label it and have it appear on the BOM. Select Value in the Field and the title of the Value in the Caption. Because this is a Value, the Value area will no longer be grayed out. In the Values area select add, then select the new value from the list. These values will only appear if the components on the board already have the listed value within their definitions.
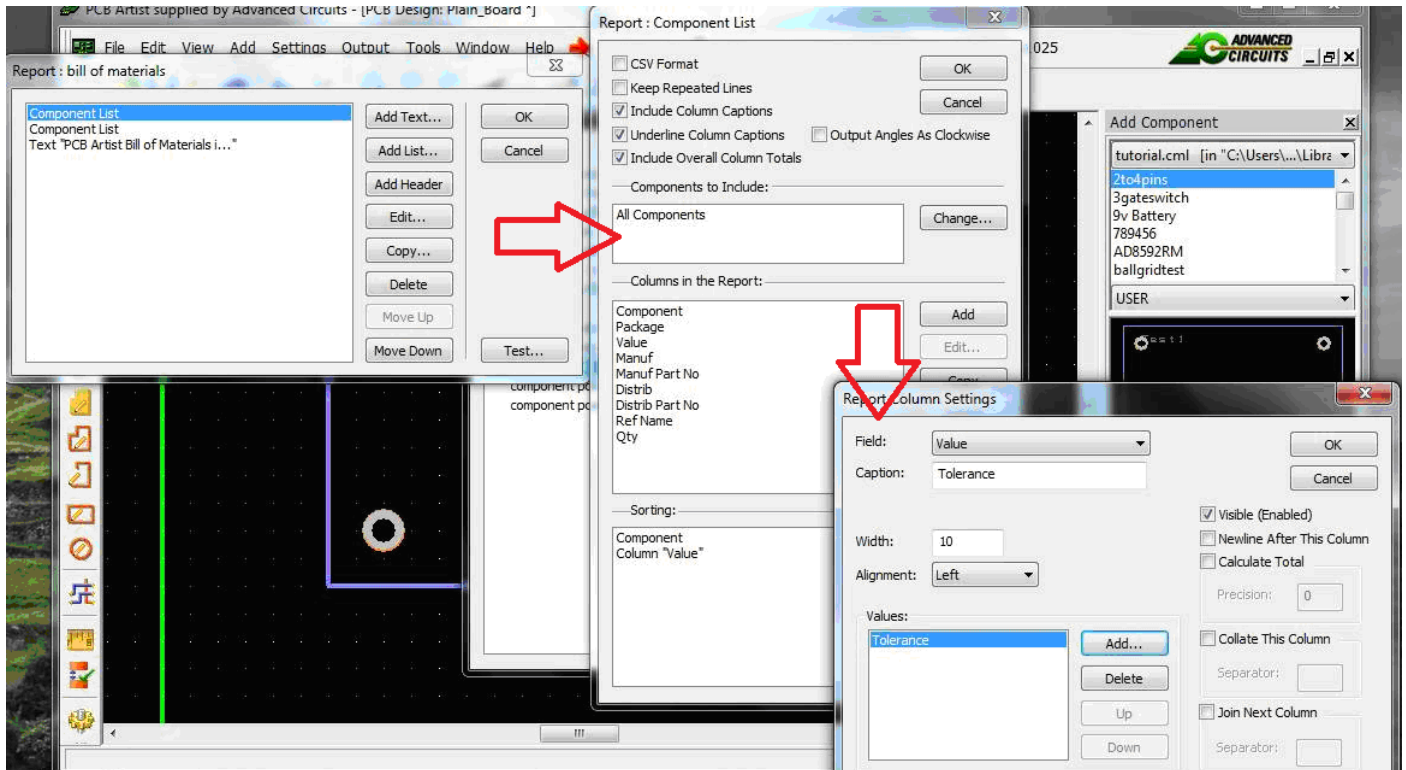
Best Practices

- **The file with the Tilde "~" in the front of it is the security backup.** This is not your actual working file. The interval of how often it gets backed up is from Settings>PCB Configuration. If you set that to zero, it will not perform a backup, which is not recommended.
- **Do not use Delete to remove tracks for redrawing.**
  - Delete will remove the track and also its rats nest connection. Delete is intended to completely remove the connection from the netlist.
  - To Remove the track, but not the connection:
    - Select a segment of the track
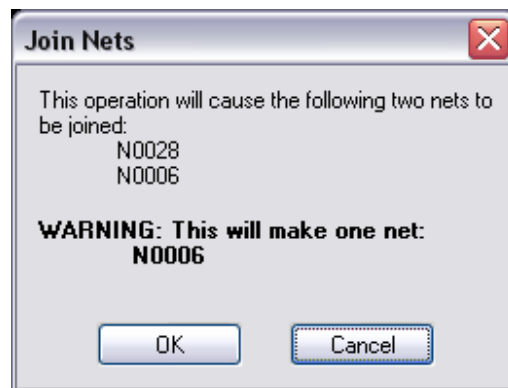   - Right click on it and choose Nets>Unroute Track Segments (or Unroute Track Path) from the right-click context menu. The track will be replaced with an unrouted rats nest connection.
- **It is not recommended to use Undo to remove the results of the autorouter.** Use Tools>Unrout Nets>All Nets.
- **Be careful when joining nets.**
  - A warning will appear such as this:

  -

  - Be sure that the new net created by both is the correct one. This depends upon which net you were drawing with. Cancel and redraw your connect from the other net to make the resultant net the desired one.
  - Once the net is joined, it cannot be split. The smaller net must be deleted and redrawn.
- **Avoid overlapping copper pours where possible.**
  - Overlapping copper pours confuse the netlist.
  - If click down on the wheel of a wheel mouse, it will pan across the screen while dragging different types of draws including polygons. This makes drawing overlapping polygons unnecessary.
- **Do not use copper pours for contact lands.**
  - They will not have a soldermask clearance. They will be covered by soldermask, which is very hard to scrape off of the contact.
  - Options:
    - Reconstruct them as surface mounts.
    - Create a clearance of a filled rectangle or polygon on the soldermask layer.

- **If your surface mount is colored gray, it requires a crucial edit.**
  - o Go into the Properties of the object and change them from layer [All] to layer [Top].
  - o If the surface mount is part of a component, the edit must be done in the PCB Symbol editor.
  - o This assumes that the default color settings are being used.
- **Make sure your power and ground nets are properly defined in Settings>Nets as Power or Ground in the Class Column.**
  - o Power and ground traces usually need to be thicker than signal traces. All nets are classified as signal until the designer changes them to a Power or Ground Class.
- **Always create a new library for any parts you create or modify.**
  - o If a future version of PCB Artist needs a complete installation, any existing pre-packaged libraries are automatically overwritten and all parts will be lost.
  - o All part creation documentation in this document describes the process of making a new library for your part.
- **In Settings>Spacings, never change the Board row to less than .020" or the Shape row to less than .010".**
  - o Those settings are not only for Design Rule Checks, but also provide rules for how copper pours are created.
- **Do not create parts with the same name as any other part in any other library.**
  - o The system acquires parts in a hierarchy of libraries that are searched when symbols are updated in the PCB or Schematic. If the other part with same name in the other library is located first, it will now be associated to that symbol.
- **Do not add Vias inside Surface Mount Lands.**
  - ▪ Many designers add vias inside surface mount lands as a way of conserving space in a design. This creates greater difficulty and lower quality solder joints in assembly. This practice is not recommended unless absolutely necessary for electrical purposes.
- **Always use Thermal Pads in Copper Pours for component holes.**
  - ▪ Regardless of heat sink needs, always use thermal pads in copper pours for component pads. Thermals pads are not necessary on vias. Directly connecting lands to the planes will cause assembly problems are poor quality solder joints.


- **Preventing accidental edits.**
  - o Making layers or object types unselectable is possible from View>Colors. Go to the Layered Items tab.
    - ▪ The Selectable row locks objects by changing the cell from Yes to No with a double click of the mouse button.

- The Selectable column locks objects for specific layers by double clicking the cell from Yes to No.
- Individual objects can be locked from editing by selecting that object, right-clicking on it, and choosing Fix Item. This will prompt a dialog box asking for confirmation if an edit is attempted on a specific object.

- **Create a pdf file.**
  - One may create a pdf file by printing to a pdf format. This can be done by finding a free pdf generator program online and installing it. The option to create a pdf will be listed as a printing option beside the other printers under File>Print Setup>Printer drop down.

- **Do not create features with a 0 (Zero) dimension**
  - Rounded rectangular pads with a 0 radius for the corners, tracks with a 0 mil width, and various other features that have a zero dimension do not process well outside of PCB Artist, and will cause issues with the production of the board. It's best to avoid using 0 in any dimension that may require a positive value.

- **Order and Submit Boards through PCB Artist**
  - Advanced Circuits has numerous websites for ordering and uploading files. With PCB Artist, boards are ordered and submitted through the software itself. These are accessible through the options Output>Submit now, and the Order Now option on the Menu. More may be found about ordering through PCB Artist **Here** in this document.

- **Missing PCB Design Menu**
  - Sometimes the PCB Design Menu will be missing. To retrieve the menu, right click on the gray area located right of the Cursor Icon. This will show a pop up of all available menus. Check the PCB Design option in the pop up. If the PCB Design Menu is already checked. Uncheck the option, and recheck it.

- **Don't Ignore Pop up Boxes**
  - As easy as it is to ignore a box that has popped up, one really shouldn't. Many of these pop ups are warnings that will directly affect your design, and may never appear again if it regards a particular feature being placed. Carefully read the error, and make a decision if you should proceed with what is causing the pop up to appear.

-
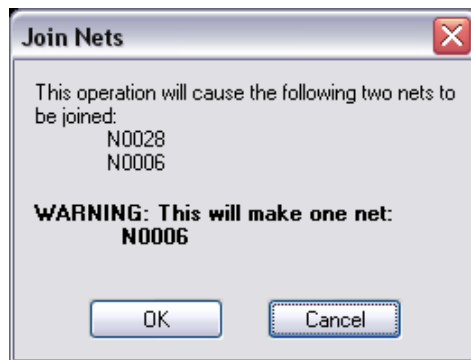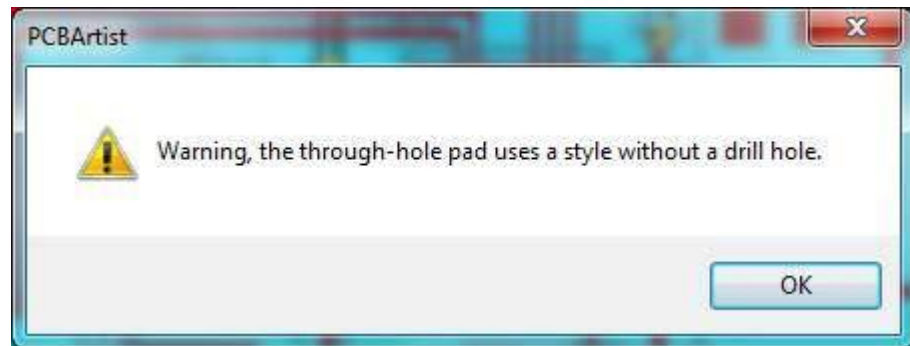
Where to find a list of Hotkeys in PCB Artist.
Hotkey reports may be produced from the Report link under Help>Shortcut Keys.

PCB Artist Hotkey Keyboard Map

PCB Artist Software
Leading the PCB Industry in Quality & Innovation
ADVANCED CIRCUITS

Cancel — esc
Add Connection or Track — F1
Help
Add Pad — F4
Refresh — F5
Add Text — F6
Add Component — F8
Interaction Bar — F9
Libraries — F11
Edit Segment First
Edit Segment Previous

Zoom — 1 2 3 4 5 6 7 8 9 0
Type Coordinate — =
Delete Segment — Backspace

Swap Layer — Q
View All
Zoom At Cursor
Refresh
Colors (menu)
Add Board
Select Next
Segment Mode Next
Change Layer
Pan

Place Horizontally
Rotate one Step
Go to Next Error
Place
Change Style
Deselect
Flip
Add Text
Place Vertically
Toggle Grid
Highlight Selected Net
Unzoom at Cursor
Invert Shape
Set Relative Origin at Cursor
Swap Connections

Alt+ Commands:		Shift+ Commands:		Ctrl+ Commands:	
Alt+Backspace	Undo	Shift+1	View Side - Top Only	Ctrl+1	Switch Top On/Off
Alt+Enter	Properties	Shift+2	View Side - Inner Only	Ctrl+2	Switch Inner On/Off
Alt+F10	Check Design	Shift+3	View Side - Bottom Only	Ctrl+3	Switch Bottom On/Off
Alt+F11	Fix Design	Shift+=	Type Offset	Ctrl+C	Copy
Alt+I	Support Info	Shift+A	Add Corner	Ctrl+D	Optimize All Nets
Alt+R	Rotate Step Back	Shift+C	Change Cursor	Ctrl+F	Frame View
		Shift+D	Delete Corner	Ctrl+F11	Preferences
		Shift+Delete	Cut	Ctrl+G	Change Grid Snap
		Shift+E	Goto Previous Error	Ctrl+Insert	Copy
		Shift+F1	Design Technology	Ctrl+L	Libraries
		Shift+F4	Add SMD Pad	Ctrl+N	New
		Shift+Insert	Paste	Ctrl+O	Open
		Shift+L	Next Layer	Ctrl+P	Print
		Shift+O	Goto Relative Origin	Ctrl+S	Save
		Shift+R	Switch Rel/Abs Coords	Ctrl+U	Unroute Track Segments
		Shift+X	Cross Probe Mode	Ctrl+V	Paste
		Shift+Z	Reset Delta Coords	Ctrl+X	Cut
				Ctrl+Y	Redo
				Ctrl+Z	Undo

If you have a new question, please send it to Layouthelp@4pcb.com

Advanced Circuits, PCB Artist - 2020