COPYRIGHT NOTICE

WARNING

You may make a single copy of the software for backup purposes. However, you are warned that the software contains an encrypted serialization system. Any given copy of the software is therefore traceable to the master disk supplied with your licence.

PROTEUS also contains special code that will prevent more that one copy using a particular licence key on a network at any given time. Therefore, you must purchase a licence key for each copy that you want to run simultaneously.

DISCLAIMER

No warranties of any kind are made with respect to the contents of this software package, nor its fitness for any particular purpose. Neither Labcenter Electronics Ltd nor any of its employees or sub-contractors shall be liable for errors in the software, component libraries, simulator models or documentation, or for any direct, indirect or consequential damages or financial losses arising from the use of the package.

Users are particularly reminded that successful simulation of a design with the PROSPICE simulator does not prove conclusively that it will work when manufactured. It is always best to make a one off prototype before having large numbers of boards produced.

Manufacturers' SPICE models included with PROSPICE are supplied on an 'as-is' basis and neither Labcenter nor their originators make any warranty whatsoever as to their accuracy or functionality

TABLE OF CONTENTS

INTRODUCTION **1**

OVERVIEW OF THE LAYOUT EDITOR **2**

- The Main Window 2
- The Control Bar 3
- Navigation 5
- Visual Aids to Design 5

Display Options **6**

- Overview 6
- Graphics Mode 7
- Opacity 7
- Auto-Pan Animation 8
- Highlight Animation 8
- Multi-Sampling (Anti-Aliasing) 9
- Displayed Layers Dialogue 9

COMPONENT PLACEMENT **10**

- Overview 10
- Components and Packages 10
- The Board Edge 11
- Work area, Co-ordinates and Snap 13
- Placing Components and the Ratsnest 18
- Mounting Holes and Pad Styles 23

DESIGN RULES AND NET CLASSES **27**

- Design Rules 27
- Net Classes 28
- Keepout Areas 31

ROUTING THE BOARD **34**

- Placing a Route Manually 34
- Panning and Zooming 36
- Placing Anchors 37
- Getting Stuck / Re-routing while routing 37
- Placing Vias 37
- Object Hugging 37

Abandoning a Route 38
Making Connections 38
Deleting a Route 38
Editing a Route 39
Layer Pairs & Manual Routing 40
Basic Auto-Routing 44
The Selection Filter 46
Advanced Auto-Routing 48
POWER PLANES AND SLOTS 52
Placing Power Planes 52
Nesting and Islands 53
Slots 55
3D VISUALISATION 58
Basic Navigation 58
Bareboard View and Height Clearances 59
Custom Views 59
BOARD OUTPUT OPTIONS 61
Printing 61
Output for Manufacture 62
APPENDIX: CREATING NEW PACKAGES 65
Drawing the Footprint 65
Packaging the Footprint 72
3D Visualisation 74

GETTING STARTED GUIDE

INTRODUCTION

The purpose of this getting started guide is to familiarize you as quickly as possible with the main features of ARES to the point that you can use the package for real work. Users with modest computer literacy should find it possible to learn the package and produce their first board within a day or two.

It is extremely important to realise that this is **not** intended to be a comprehensive resource. The online manual, accessed from the Help Menu in ARES, is a full reference manual and contains far more detailed discussion on all of the topics covered in this guide. It follows that this should be the first port of call should you need clarification of techniques or functionality.

This tutorial is a continuation of the ISIS Getting Started Guide and looks at how to get from a schematic drawing through to a completed PCB layout, including:

- Basic techniques for placement and routing.
- Configuration of Design Rules and Net Classes.
- Board Auto-routing.
- 3D Board Visualisation.
- Hard copy generation and manufacturing output.
- Library part creation.

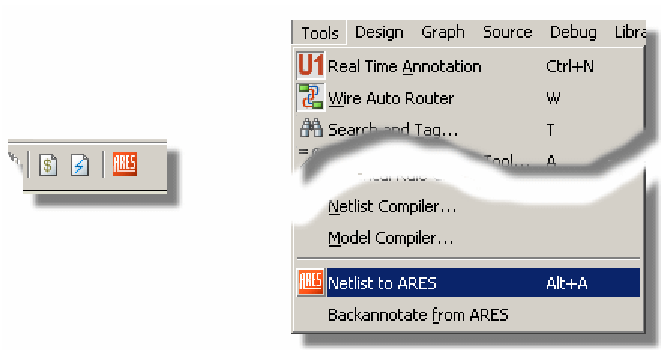
There is a lot of common functionality across the applications and particularly with regard the user interface (zooming, selecting, editing etc.) This material is covered in some detail in the accompanying ISIS Getting Started Guide and a working knowledge of basic operation is assumed in this document.

We do urge you to work right the way through the tutorial exercises as many things are pointed out that if missed will result in much wasted time in the long run. Also, having worked through the tutorial and thus got a basic grasp of the concepts behind the package you will find it much easier to absorb the material presented in the reference chapters.

Finally, note that throughout this tutorial (and the documentation as a whole) reference is made to keyboard shortcuts as a method of executing specific commands. The shortcuts specified are the default or system keyboard accelerators as provided when the software is shipped to you. Please be aware that if you have configured your own keyboard accelerators (System Menu – Set Keyboard Mapping) the shortcuts mentioned may not be valid.

OVERVIEW OF THE LAYOUT EDITOR

We shall assume at this point that you have installed the package, and that the current directory is some convenient work area on your hard disk. This tutorial is a direct continuation of the ISIS Getting Started Guide so we will start by loading the completed schematic. This can be found in the `..\Samples\Tutorials\` directory of your Proteus installation and is called `dsPIC33_REC.dsn`. This circuit was designed and completed in the ISIS tutorial so all we need do is transfer the connectivity information (netlist) through to ARES for PCB Layout. We can do this via the *Netlist to ARES* command on the Tools Menu or via the ARES icon at the top right of the ISIS application.

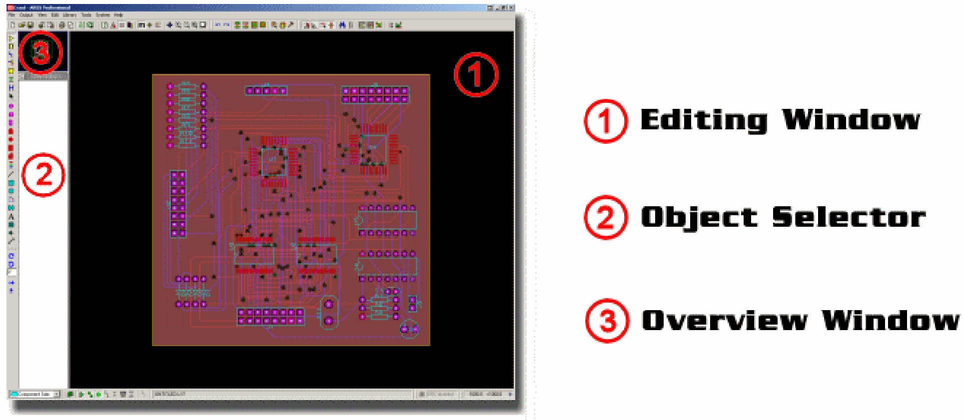


After invoking this command a copy of ARES will open and present you with some options from which to create the new layout. Since we are starting from scratch in this tutorial we can select the 'blank' option which will then leave us with an empty layout.

Templates and Technology Data re-use in general is beyond the scope of this tutorial. However, creating and saving templates and technology data is a valuable time-saver and is well worth learning about. Please see the chapter on Templates & Technology in the *Reference manual*.

The Main Window

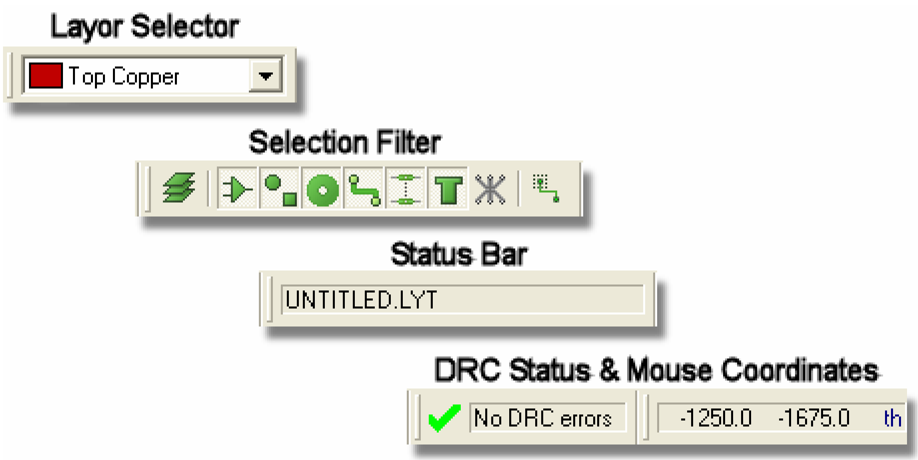
The largest area of the screen is called the *Editing Window*, and it acts as a window on the drawing - this is where you will place and track the board. The smaller area at the top left of the screen is called the *Overview Window*. In normal use the *Overview Window* displays, as its name suggests, an overview of the entire drawing - the blue box shows the edge of the current sheet and the green box the area of the sheet currently displayed in the *Editing Window*. However, when a new object is selected from the *Object Selector* the Overview Window is used to preview the selected object - this is discussed later.



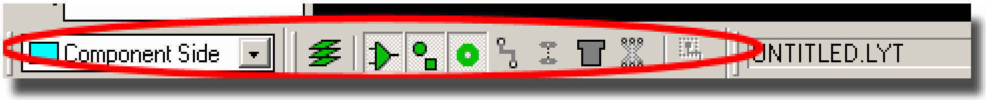
An Overview of the ARES Layout Editor

The Control Bar

The control bar at the bottom of the application is different from what we have seen in the ISIS application and essentially splits into four sections:

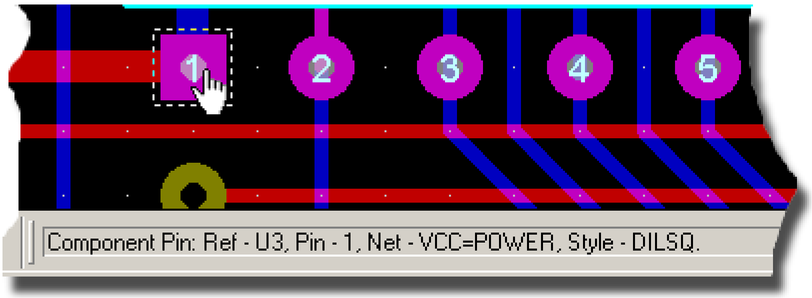


At the left hand side is the *Selection Filter* which allows you to configure both the layers and the objects that will be selected in the current operating mode. Typically, the default rules will suffice and this serves simply as a convenient override where you may wish finer granularity in selection at a given time. The *Layer Selector* combo box also determines the current layer or layer set and also applies to the placement of PCB objects.



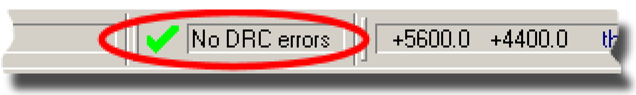
Selection Filter in the ARES Layout Editor.

In the middle is the *Status Bar* which provides textual 'hints' on the object currently under the mouse. This is particularly useful when you hover a mouse over a pad for example, as it will inform you which net the pad is on.



Status Bar in the ARES Layout Editor.

Towards the right hand side is the live *Design Rule Checker*. This will report any physical design rule violations that occur while the board is being designed. A left click on this will launch a dialogue detailing the violations with the further option of zooming in to examine a particular error.



Design Rule Indicator in the ARES Layout Editor.

At the far right hand side is is the co-ordinate display which reads out the position of the cursor when appropriate. These reflect not the exact position of the pointer but the location to which it has been snapped. Default snapping options are selectable from the *View* menu (or via keys CTRL-F1 and F2 -F4) and the snap values can be configured from the *Grids* command on the *Technology* menu.

Co-ordinate Display Indicator in the ARES Layout Editor.

The co-ordinates can be in imperial or metric units as set by the *Metric* (default key mapping 'M') command. You can also set a false origin using the *Origin* command (default key mapping 'O') in which case the co-ordinates change colour from black to magenta.

The dot grid on the *Editing Window* can be toggled on and off using the *Grid* command, or via it's keyboard shortcut (by default this is 'G'). The spacing of the dots normally reflects the current snap setting, except when zoomed out. In this case, the dot spacing is set to a suitable multiple of the snap spacing.

ARES can be set to display an X cursor at the position to which it has snapped the pointer through the *X-Cursor* command, default key mapping is 'X'.

We'll become familiar with all of these items and use them regularly as we work through placement and routing of the board.

Navigation

Navigation of the layout (middle mouse zoom, keyboard shortcuts, etc.), is identical to the ISIS schematic capture package. Refer to the ISIS Getting Started Guide or the ISIS reference manual for more information.

Visual Aids to Design

As in ISIS the ARES package will use visual effects to help you understand what is happening during board layout. There are two principle techniques:

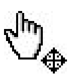
ARES will 'twitch' an object when that object is under the mouse and the selection filter enables selection of that object type. This serves to identify when an object is 'hot'.

ARES provides dynamic cursors which change to identify what a left click will do at any given time (place an object, select an object, move an object, etc.). A list of cursor types is shown below:


Standard Cursor.


Placement Cursor – left click will place an object according to mode.


Selection Cursor – a left click will select the object under the mouse.


Movement Cursor – a left drag will move the object(s) selected.

Throughout this tutorial – and indeed whenever you are working in ARES – you should make use of these visual indicators to as an aid to understanding the software.

ARES and ISIS have a multiple 'undo' and 'redo' system via the shortcut keys `CTRL+Z` and `CTRL+Y` respectively. If you make mistakes throughout the tutorial remember that you can unwind as many times as you need to!

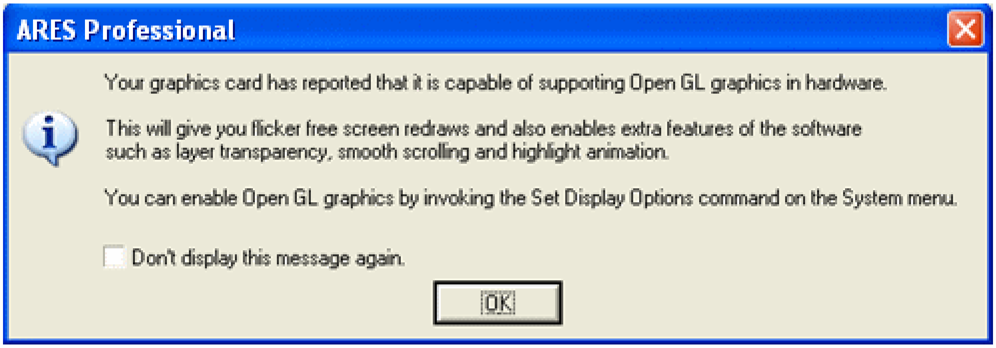
Display Options

Overview

ARES is capable of harnessing the power of your computers graphics card to speed up operation and provide true layer transparency on the layout. However, as not all machines have sufficient graphics cards the software is also capable of using Windows to perform display and graphical operations. The three modes of operation are called:

- Traditional Windows GDI Mode.
- Double Buffered Windows GDI Mode.
- OpenGL Hardware Accelerated Mode.
- Direct 2D Graphics Mode.

When you open ARES for the first time it will query your computers graphics card and inform you if your machine is capable of running in OpenGL mode or Direct2D mode.

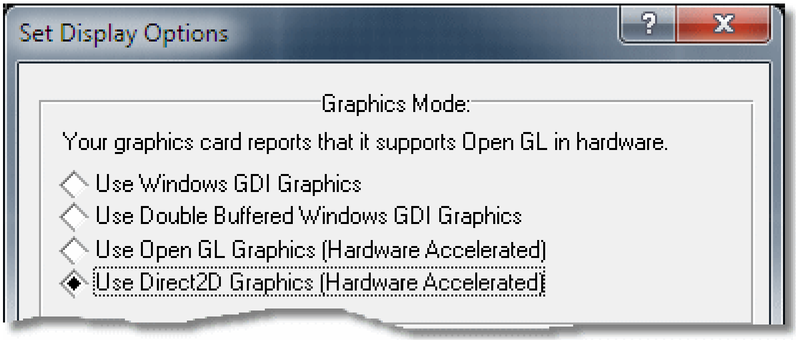


If your graphics card is not powerful enough to support hardware acceleration then ARES will simply default to traditional Windows GDI mode. In this situation your display options are reduced to either the traditional or double-buffered Windows GDI modes. While still using Windows to render, double buffered mode redraws the screen onto an invisible buffer and then copies the screen onto the application display when complete. This avoids ‘flicker’ during redraw and may be preferable for some users.

Regardless of which modes are available, all configuration of the screen display takes place from the *Set Display Options* on the *System* menu. Some options are specific to particular hardware acceleration modes and will not be available when other modes are selected.

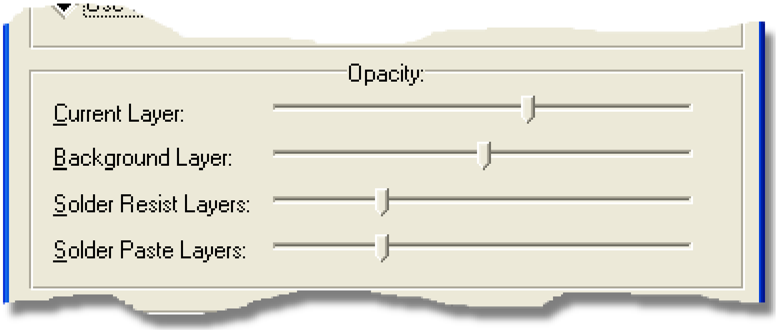
Graphics Mode

The first section of the dialogue reports on whether your graphics card will support OpenGL or Direct2D hardware acceleration and, if so, allows you to switch from either of the Windows GDI modes into an accelerated graphics mode.



Opacity

When hardware accelerated graphics are selected the *Opacity* section of the dialogue allows you to configure the transparency of various layers on the board.

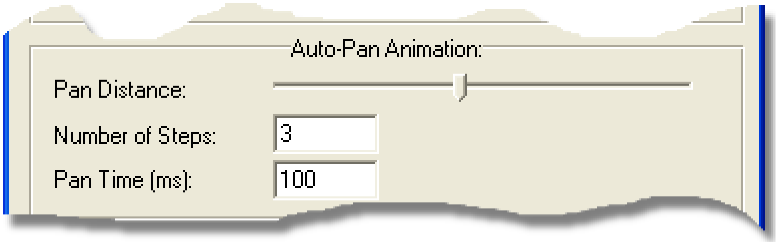


In these modes, ARES places special significance on the current layer, giving it a higher opacity than other copper layers on the layout. This means that objects on the layer you are working are clearly visible and that objects on the other layers of the board are dimmed. You can control the relative opacity of both the current layer and the background layer(s) using the sliders on this section of the dialogue form. For example, if you wish to disable transparency altogether you could set the background layer slider to maximum.

A new feature is the ability to view the solder resist and paste mask around pads and vias on the layout. When enabled, you can adjust the opacity of these layers using the appropriate slider controls.

Auto-Pan Animation

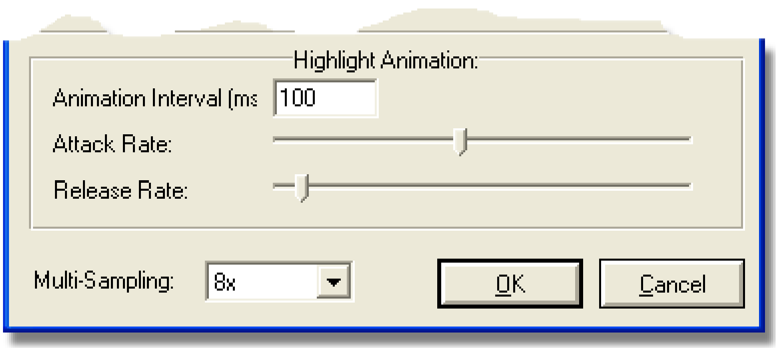
In ARES, holding down the shift key and bumping the mouse against the edge of the *Editing Window* allows you to pan the screen. Similarly, dragging an object against the edge of the *Editing Window* will pan the screen in the direction of the drag. This navigation feature is called auto-pan and is controlled via the options in this section of the dialogue form.



You can adjust the distance moved on auto-pan (pan distance), the number of steps taken to cover this distance (number of steps) and the speed of the animation (pan time). All of these options are available regardless of which mode you are working in.

Highlight Animation

When working in one of the hardware accelerated modes objects under the mouse will smoothly increase their intensity to inform you that they are selectable. The highlight animation options on this dialogue allows you to control the smoothness and speed of this effect.



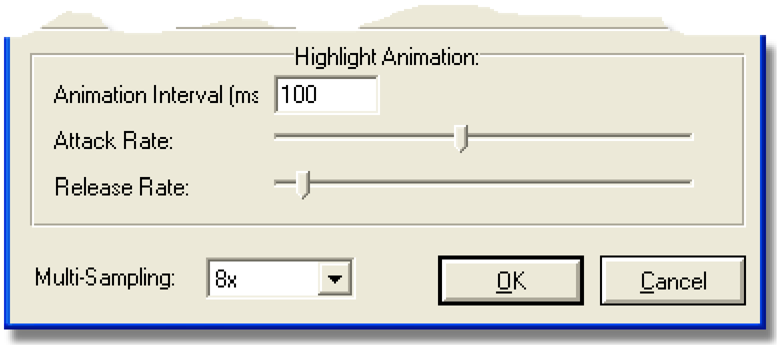
The animation interval controls the frame rate of the animation and therefore the smoothness of the fade-in/fade-out effect. Normally, the default value for this is suitable.

The attack rate specifies the speed at which an object will go from ‘fully off’ to ‘fully on’ whilst the release rate allows you to adjust the speed at which an object will return from ‘fully on’ to ‘fully off’.

These options are not available in the standard Windows GDI modes.

Multi-Sampling (Anti-Aliasing)

Multi-Sampling is a method used by graphics cards to reduce anti-aliasing effects when graphics are displayed at different zoom levels. It is particularly applicable to text but impacts on all graphics on the layout.



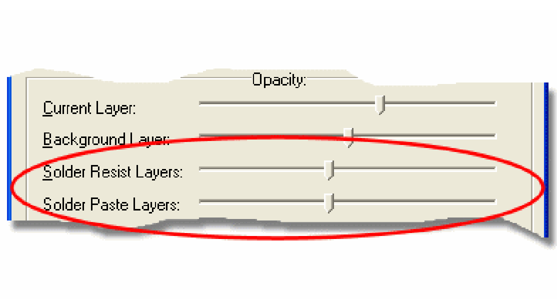
When working in OpenGL mode you can specify the level of multi-sampling you want to use. The higher the level of multi-sampling the better the resulting display but the more GPU resources are consumed. If you select a level of multi-sampling which is not supported by your graphics hardware the software will reset the level to the closest one which your card can handle. For normal operation in ARES a multi-sampling level of 4x is quite sufficient.

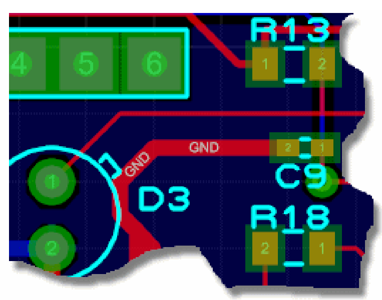
Displayed Layers Dialogue

The displayed layers dialogue allows you to control the visibility and colour of the various layers on the board. You can launch this from the *View* menu in ARES or by clicking the mouse on the status bar at the bottom of the application.

You can change the colour of a layer by clicking on the colour beside the layer and control the visibility of the layer via the checkbox beside the layer (where appropriate). All changes are updated live on the layout.

If you are working in a hardware accelerated mode, the *Resist/Solder Paste Display* options allow you turn on full display of these layers on the board, showing the resist and paste coverage around pads and vias. When enabled, you can change the intensity of these layers by switching to the Thru-View settings tab and adjusting the appropriate slider controls.





COMPONENT PLACEMENT

Overview

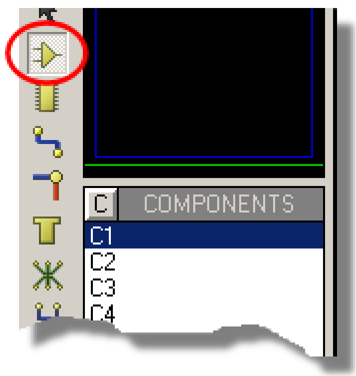
Since we have netlisted the project from our schematic design we have already supplied the ARES software with much of the information it needs to simplify the layout process. In particular, we have specified which footprints are associated with each schematic symbol and ARES can therefore pre-select these for us ready for placement. This brings us to an important distinction in the software; the difference between a component and a package.

Components and Packages

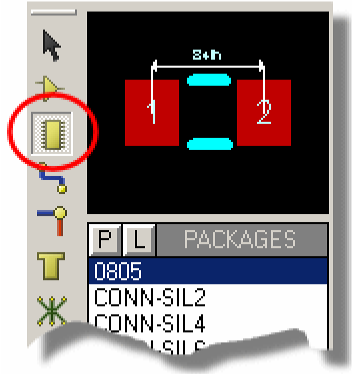
- A *component* is an instance of a footprint that has been netlisted through from the schematic.
- A *package* is a physical footprint that exists in the ARES Libraries.

Selecting component mode will access footprints which have been specified as relating to parts in ISIS and carry connectivity information whereas selecting package mode will access 'unbound' physical instances of a footprint. When working with a layout which is driven from a schematic we therefore exclusively use component mode.

The *Component Mode* icon is second from the top directly underneath the *Selection* icon. Clicking on this will display a list of items in the *Object Selector* which correspond directly to the parts in ISIS that we used to create the schematic.



The *Package Mode* icon is directly underneath the *Component mode* icon and clicking on this will show us the physical footprints corresponding to the components in the layout.



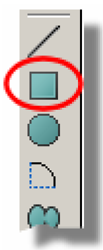
When placing, routing and laying out a PCB with a netlist (such as with this project) we will be working with components

The Board Edge

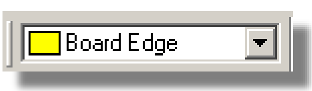
Before we can place the components on the board we need to define what shape and size the board will be. For our project we need only a simple rectangular board edge but we do want to control the dimensions of the board (`115mm` wide and `40mm` high).

The first thing to point out here is that ARES will operate in either imperial or metric units and you can switch between modes either by toggling the *Metric* command on the *View* menu or via the `'M'` keyboard shortcut. You may need to switch units for the placement of the board edge and also elsewhere in the documentation.

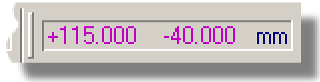
To start placing the board edge, select the 2D Rectangle icon from the left hand side of the application.



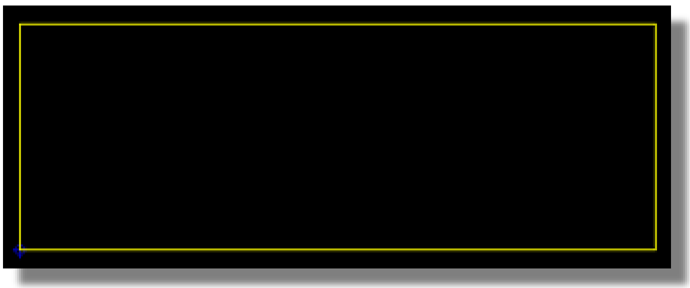
Next, change the *Layer Selector* to the *Board Edge* Layer.



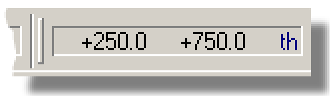
Move the mouse to the approximate starting point (e.g. top left) for the board edge. Now, hold the mouse still and press the 'O' key on the keyboard to set the origin of the co-ordinate system to the point under the mouse (switch to metric if you are currently in imperial units). This will be reflected in the co-ordinate display at the bottom right of the application. Left click to start placement and drag in the normal way. The co-ordinate display at the bottom will show you the dimensions of the board edge as you drag.



Press the `'M'` key on the keyboard to toggle between metric and imperial units as required. Once you have the desired size of board edge click left again to commit placement. Don't worry about where in the *Editing Window* you have placed the board edge – we will move it to the center of the world area shortly.

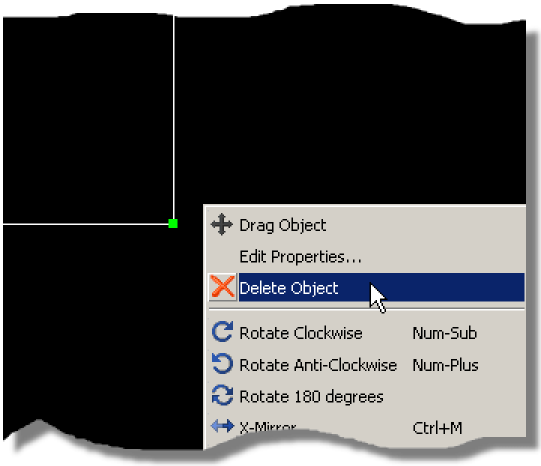


Finally, restore the co-ordinate system to it's global origin by pressing the 'O' button on the keyboard again. The colour of the co-ordinates will change from Magenta to black to indicate that the global origin is now in use.



Placing a board edge should always be the first task as the software needs to know the boundaries of the board in order for example, to know the limits of where the autorouter can function or the size of a power plane.

If you need to delete the board edge or resize it you can do so via the context menu options and drag handles that will appear when you right click over the board edge graphic.

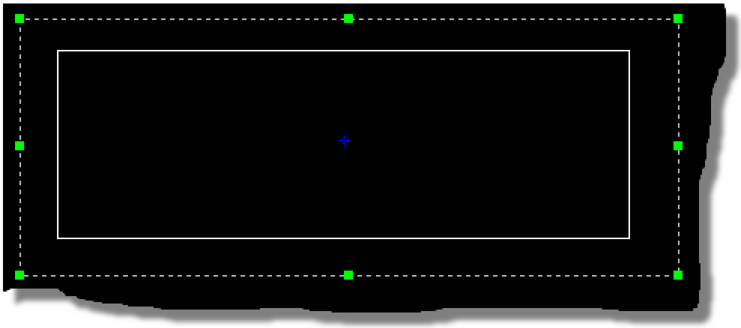


You can create a board edge of almost any shape using the 2D graphics path tool and using the CTRL key during placement of curved segments. This is discussed in the reference manual.

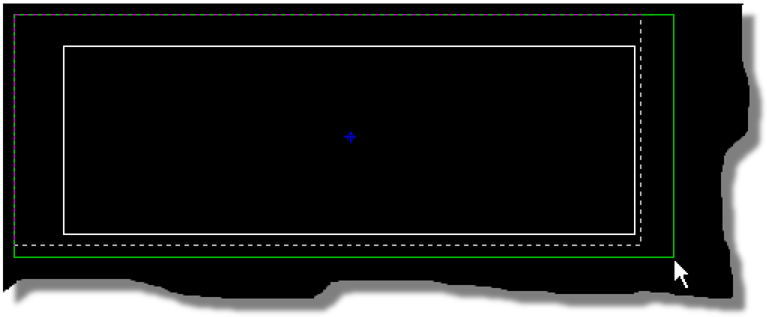
Work area, Co-ordinates and Snap

It is likely that your board edge covers a small portion of the *Editing Window,* which is less than ideal as all of the work will be taking place inside this area. We can of course zoom into the area (point the mouse and roll the middle mouse button or use the F6 key) but the default zoom level is designed to show the entire *Work area*. The *Work area* is the area inside the dark blue box at the edges of the Editing Window. Let's tidy this up a bit before we move on.

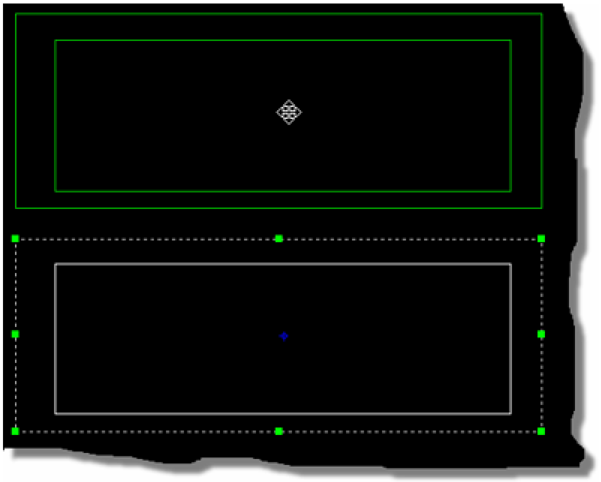
Start by switching into *Selection Mode* and then left click and drag a selection box around the board edge.



If you get it wrong and don't completely cover the board edge, use the green handles to resize the selection box until it completely encompasses the board edge.

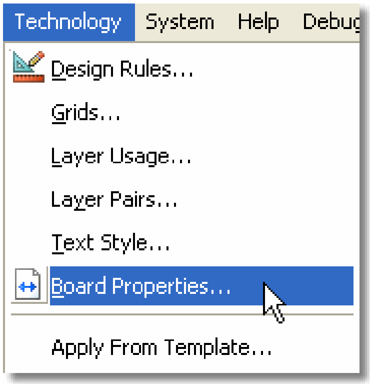


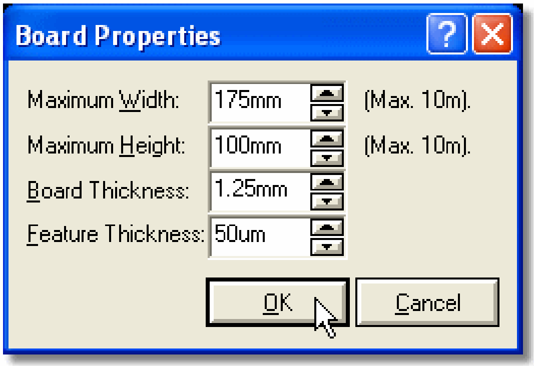
Now place the mouse inside the selection box, depress the left mouse button and drag into the center of the work area, releasing the mouse button to commit the placement.



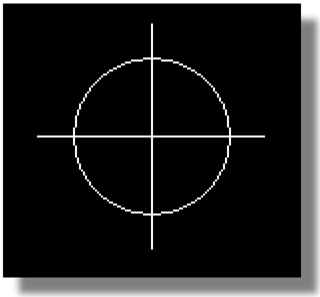
This is a **very** important technique, allowing you to easily block select and then perform actions on (move, delete, etc.) groups of objects.

Having centered our board edge in the Work area we can shrink the work area to a suitable size. From the *Technology Menu,* select the *Board Properties* command and then set to something like `175mm` by `100mm`, such that the resulting work area forms a reasonably boundary to the board.



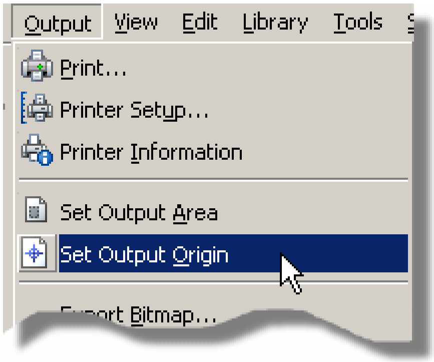


We saw briefly when placing the board edge that we can specify a temporary origin and use that to place items of a given size (or to place items a specific distance away from our origin point for example). We can however, also specify the location of the *Output Origin*. This is the default origin used whenever we do not explicitly set a temporary origin and is shown on the layout as a small blue marker.

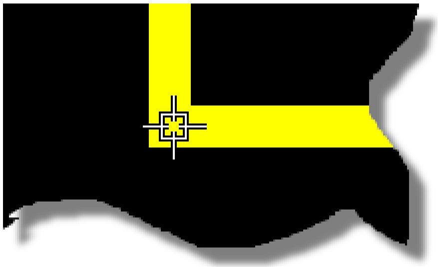


It is generally useful to move this to a corner of the board and particularly so if we have mechanical constraints that we need to consider during placement (e.g. mounting hole locations). For our design, we'll move the origin to the bottom left corner of the board edge.

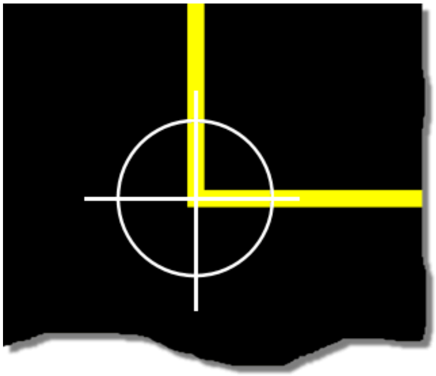
1) From the *Output* menu choose the *Set Output Origin* command.



2) Move the mouse towards the bottom left of the board edge. Roll the middle mouse button as required to zoom in for more accurate placement.



3) Left click the mouse to commit placement.



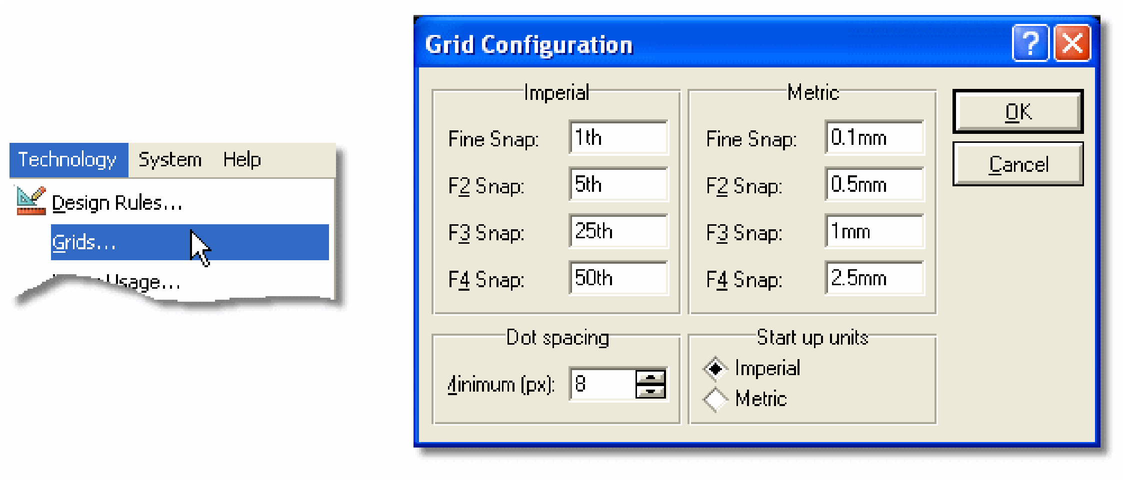
The co-ordinate display will now provide values relative to the position at the bottom left of the board by default, and relative to any specified temporary origin when set.

Remember that you can also toggle metric or imperial units via the command on the *View* menu or the `'M'` keyboard shortcut.

A useful trick once we have specified our origin is to use the *Goto-XY* command on the *View Menu* to move the mouse to the exact location we want to place position sensitive parts.

ARES has a default grid and will snap objects onto the grid, making it easy to form connections and control board layout. There are four hotkeyed snap settings for both metric and imperial units, changeable from the View Menu or by keyboard shortcuts. It follows that changing the snap setting to a lower unit will allow finer granularity, whereas raising the snap setting will make it easier to select objects at higher zoom levels. It is not advisable to change the snap settings regularly through a design but rather to choose the highest appropriate setting for the board you are working on.

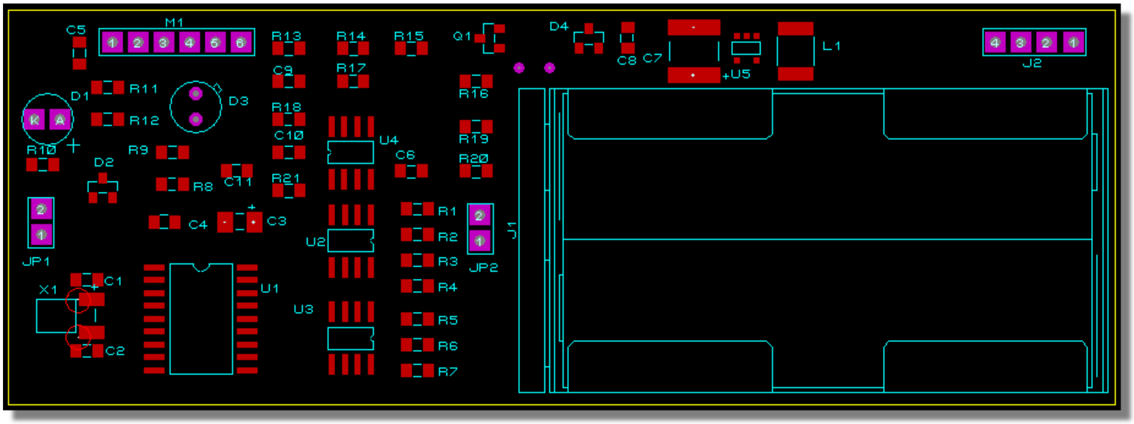
You can change the four default snap settings via the Grids command on the *Technology Menu*



The grid display itself can be toggled between off, dots and lines via the 'G' key on your keyboard, whilst the colour of the grid can be configured from the Displayed Layers Dialogue form (View Menu).

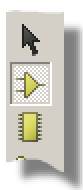
Placing Components and the Ratsnest

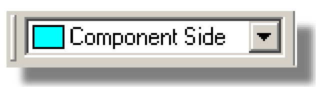
Now that we've handled all the basics we can finally start to get our components on the board. The following screenshot shows a fully placed board and we can use this to get our approximate positioning.



Placing a component in ARES is very similar to that in ISIS.

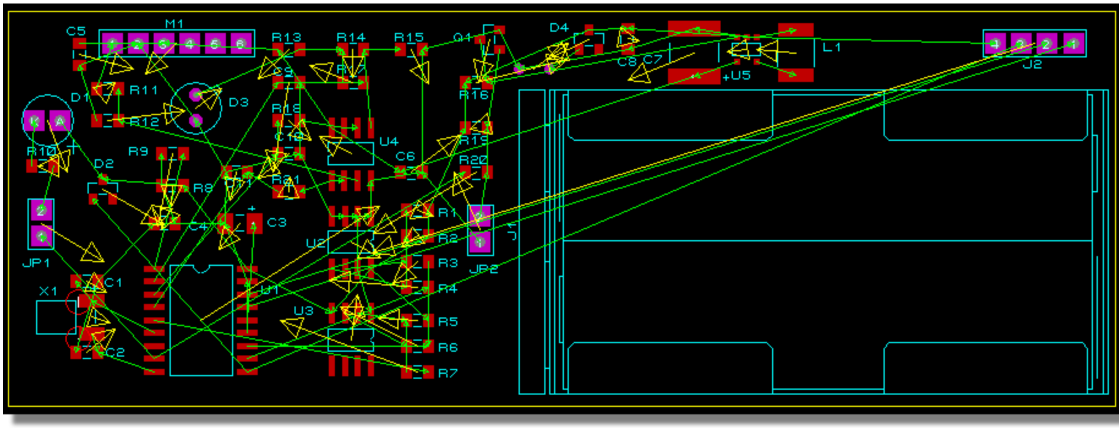
Start by selecting *Component Mode* from the left hand side icon set and then ensure that the layer selector is set to *Component Side* – we won't be placing any solder side components in this project.



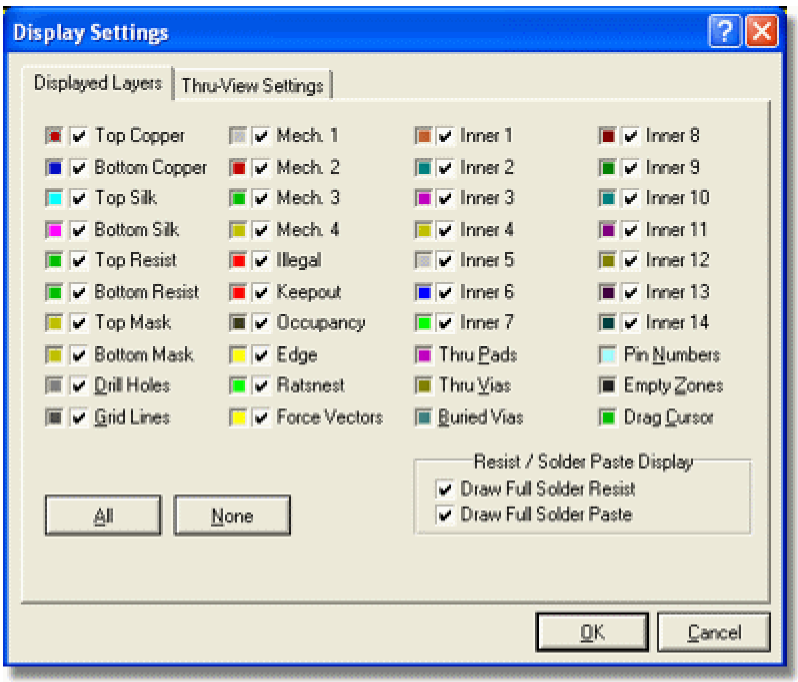


Let's get the AA battery holder down first on the right hand side of the board. Select J1 from the *Object Selector*, position the part using the previous screenshot as a guide and then left click again to commit placement. Note that once we have placed the part it is removed from the *Object Selector* and we can continue by placing the J2 connector above and towards the right of the battery.

You should notice both during placement and afterwards that there are green 'elastic' lines between the two components and also a yellow arrow from each component. The green lines are ratsnest lines and indicate connections that have to be made between the two parts, whilst the yellow arrows (named 'force vectors') indicate a preferred position for the part to minimize ratsnest distance. The force vectors are provided as guidance only and are based solely on logic to reduce ratsnest lines. Since we will be using the earlier screenshot to dictate positioning we can turn them off.



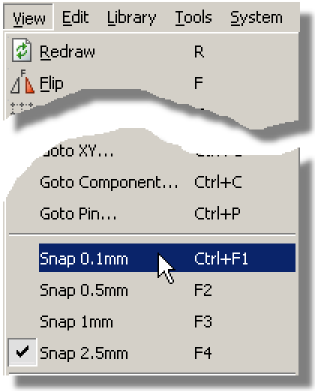
From the *View Menu*, select the *Layers* command. The resulting dialogue form shows all the layers in ARES with colour and visibility configuration options. All we need do here is deselect the checkbox for the 'Vectors' layer and exit the dialog.



You can also launch this dialogue form by clicking the mouse on the status reporting bar at the bottom of the application.

It is important to remember that this dialog form controls visibility only; to control whether objects on a layer were selectable/editable we would use the *Selection Filter* which is discussed in more detail later in the documentation.

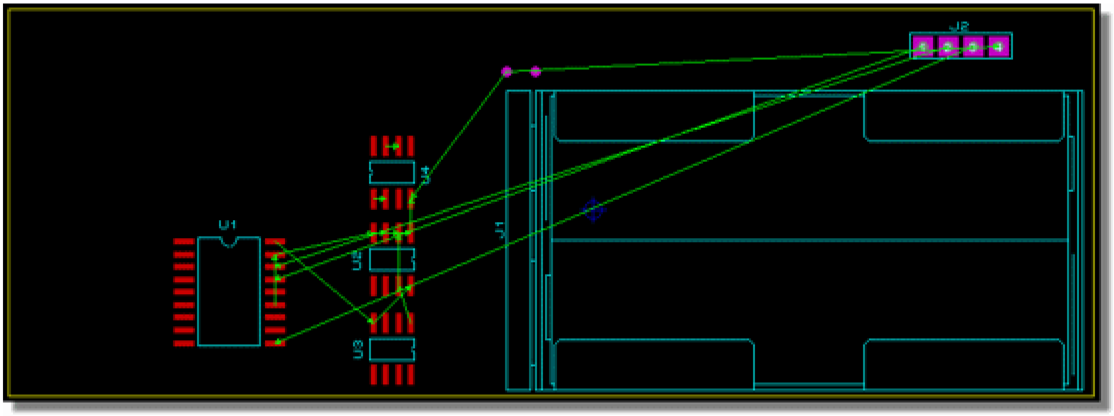
You may find that you need more control over the positioning in order to move connector J2 into position. Remember that objects are linked to a snap grid so all we need do is reduce the snap grid via the options on the *View Menu*, for example to 1mm snap.



If you are working in imperial units you can either change the snap to 25^{th} or use the 'M' shortcut to switch to metric units.

Now that we have the granularity we need, simply right click on the part to select it and then drag it into the desired position. We can move the connector closer to the board edge if we move the part label underneath the part. Again, the process is identical to the one we covered in the ISIS tutorial – remember this time to right click on the label itself and not the component body.

If we now consider the other principle components we can go ahead and place U1 (dsPIC33), U2 (I^2C memory), U3 (temp/humidity sensor) and U4 (dual op-amp) in exactly the same way, such that we end up with something like the following screenshot.

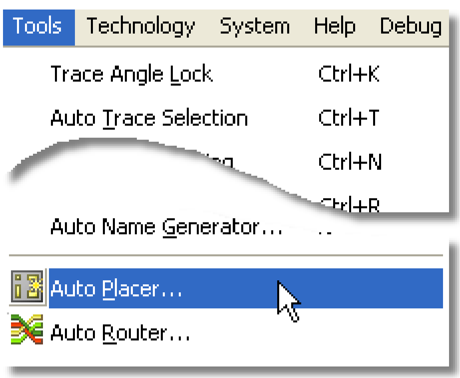


Note that the S08 footprints used for the U2, U3 and U4 components have been rotated appropriately to reduce ratsnest length. This is best done at placement time as the ratsnest display will update live to give visual guidance.

Start by placing U1 in the normal way and then start placement of U2 by left clicking once on the *Editing Window*. Now use the '+' and '-' keys on your numeric keypad to rotate the component as you move it into position, left clicking the mouse again to commit placement as before. Repeat the process for U3 and U4, moving or rotating parts after placement (right click and then context menu optionjs) until your board approximates the previous screenshot.

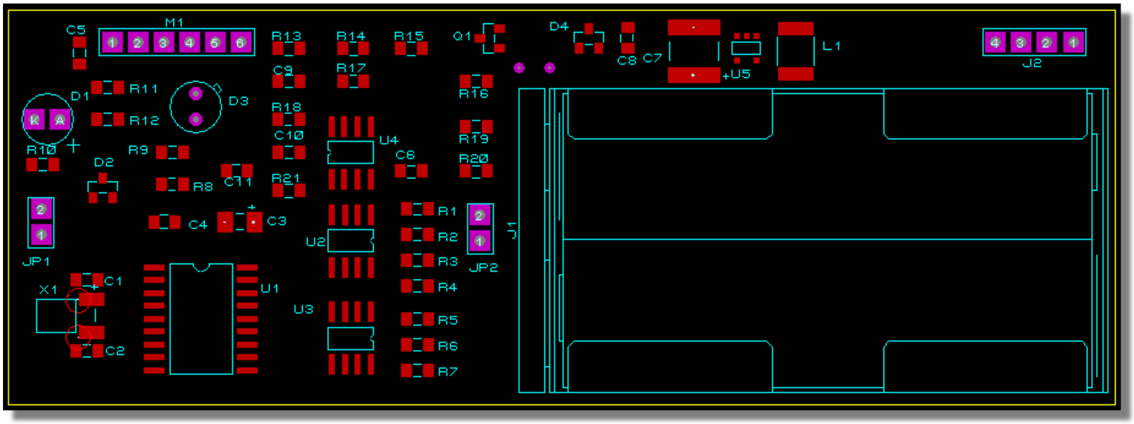
Generally speaking you have two options for placement of the board. You can either manually place the components in their positions or, if you have the Advanced Features Set (Proteus PCB Design Level 2, Level 2+ or Level 3), you can use the Autoplacement feature to get all the components on the board and then move them into the desired locations. In either case, you may find it useful to temporarily disable the ratsnest lines during placement; remember that this is controlled from the *Layers* dialogue form as discussed earlier.

The autoplacer can be invoked from the *Tools Menu* in ARES and for our purposes all the default options will suffice.



For more information on the Auto-placer and the available options please refer to the reference manual (Help Menu – ARES Help).

Whichever route you follow the task is to lay out all the components on the layout, using the following layout as a reference. Try to leave some space at the bottom of the board so that we can run traces down and along from connector J2



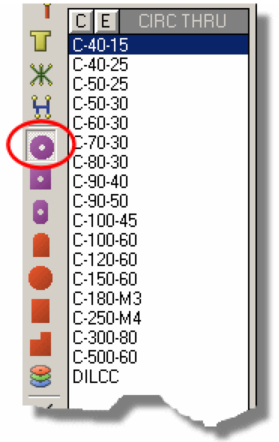
The following points are usef[ul to bear in mind as you continue placement:

- The middle mouse will zoom you in and out as you place components (as will keyboard shortcuts F6 and F7)
- Right clicking on a component once placed will present a context menu option, allowing you to move, rotate or delete the part.
- Changing the snap settings to a finer grid will allow more accurate positioning at the expense of more precision being required to select the part.
- If you place a part in an illegal position (e.g. over another part) you will get one or more design rule violations. For now, simply move the part to a legal position – we will look more closely at design rules in the next chapter.

Once you feel comfortable with placing and moving parts, feel free to move on to the next section. We will load a board with all the parts placed before we discuss design rules and routing.

Mounting Holes and Pad Styles

Before we move on to look at connectivity considerations we should complete the physical layout by placing mounting holes for the board. In our case we want to use pad with a `3mm` hole and a diameter of around `0.18in` and to position them conveniently for mini-locking PCB supports. The first thing we need to do is switch into circular through hole pad mode and scan for a suitable pre-supplied pad in the *Object Selector*.

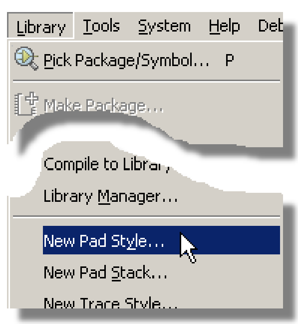


The nomenclature of pads in ARES is designed for easy reading and tends to follow the following format:

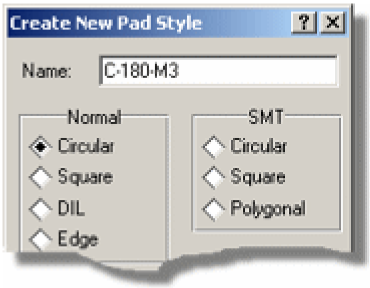
`<PAD TYPE> - <DIAMETER/SIZE> - <HOLE>`

Units are in imperial unless prefixed with an `'M'` so for example a `C-40-15` is a circular pad with a `40th` diameter and a `15th` hole and a `C-200-M3` is a circular pad with a `0.2in` diameter and a `3mm` hole. Our spec requires a pad with a `0.18in` diameter and a `3mm` hole, which we can see does not exist in the pre-supplied set. We therefore need to create the pad as follows:

1) From the *Library Menu* select the *New Pad Style* command.

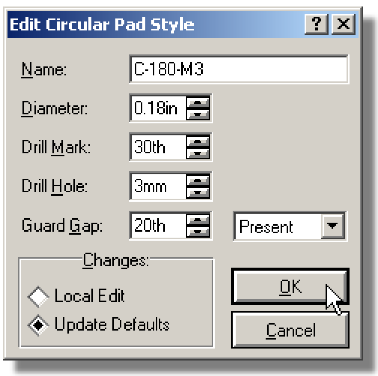


2) Enter a name for the pad; we recommend you follow the standard naming convention (i.e. `C-180-M3`).



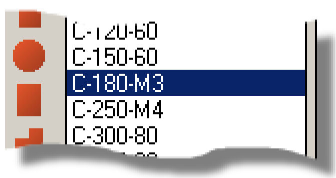
3) Specify the pad type – in our case we want Circular through hole pad.

4) The diameter of the pad is `180th` or `0.18in.` The drill mark is the size of the mark output in a drill plot; 30th would be fine. The drill hole needs to be `3mm` and the guard gap should be enlarged to `20th` The guard gap is the amount by which the pad diameter is expanded on the resist plot.



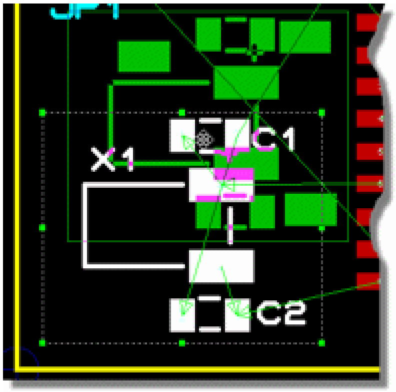
5) At the bottom of the dialogue form we have the choice whether to make this pad style permanent for future designs (Update Defaults option) or local only to this design (Local Edit option). Unless there is a particular reason for a local edit only we recommend that you leave this on the default setting.

6) When you exit the dialogue form you should see that the new pad style is available for placement from the *Object Selector*.



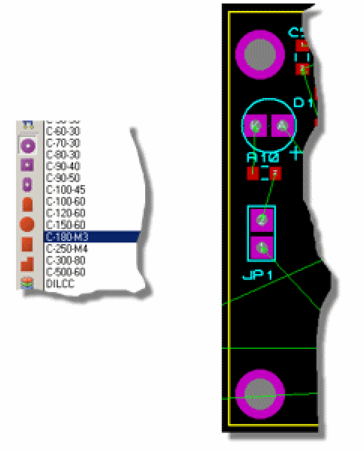
When entering specific values you may find it easier to simply type in the values that you want rather than using the up and down arrows beside the controls.

We are going to want to place two mounting holes at the top and bottom left of the board and then a third at a specific location (for mini-locking PCB supports). You may therefore have to move some circuitry out of the way to make room; for example, the crystal block at the bottom or the pressure tranducer at the top. As we've seen earlier we can easily do this by entering selection mode, drawing a selection box around the circuitry and then dragging to a new position. This is shown below with the crystal circuitry.



Remember that you can resize the selection box to include or exclude items using the drag handles if you don't get the right size the first time.

Once you've cleared some space, go to the circular PTH pad icon again, select the C-180-M3 pad style and place two at the top and the bottom left of the board.

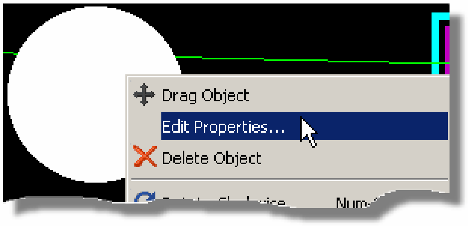


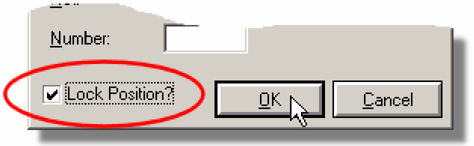
First two mounting holes at the top left and bottom left of the board.

We want to position the third mounting hole above the battery and in a reasonably accurate position. Specifically, we want to place the hole `35mm` up from the bottom of the board and `87.5mm` along from the left. We've already set our global origin to the bottom left of the board so this is set to be the reference point for the co-ordinate system. All we need do therefore is move the mouse up and right from the bottom of the board edge until the co-ordinate display reads correctly (you may need to switch into Metric units using the `'M'` shortcut key toggle)

If required, we can move the DC/DC converter circuitry as before, then switch back to pad mode and place the final mounting hole at the correct co-ordinates.

With position sensitive components it is often useful to ensure that they cannot be nudged or moved inadvertently after placement. You can lock any object in position by moving the mouse over the object until it is encircled by a dashed line, right clicking on the object and selecting Edit Properties from the resulting context menu. The *Lock Position* checkbox can then be selected to prevent movement or deletion of the part.

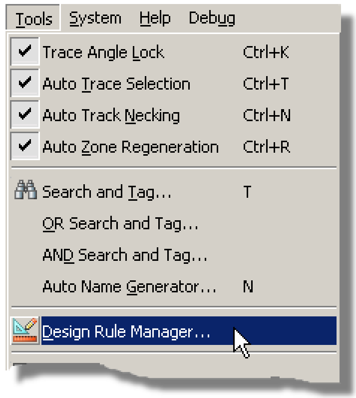




We have now completed the physical layout of the board. If you decided not to follow the full layout process you can load a version of the board in its current form from the `..\Samples\Tutorials\` directory of your Proteus installation (`dsPIC33_REC_UNROUTED.lyt`)

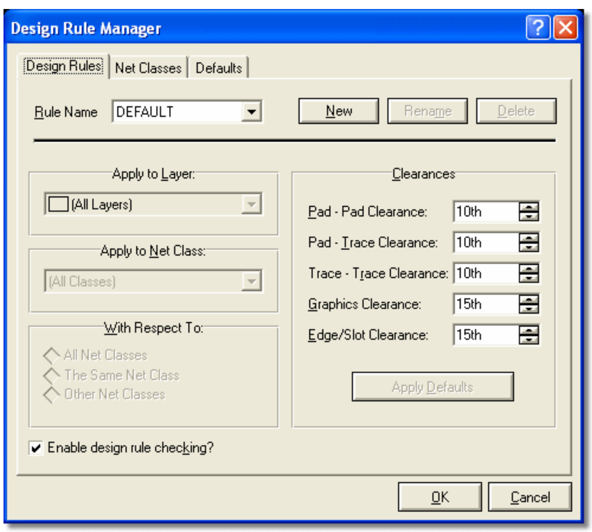
DESIGN RULES AND NET CLASSES

Now that we have a placed board we need to configure the software to obey any design constraints or electrical considerations relevant to the layout. We can do this largely from a single dialogue form called the Design Rule Manager. Start by launching this dialogue form now from the *Technology Menu* in ARES.



Design Rules

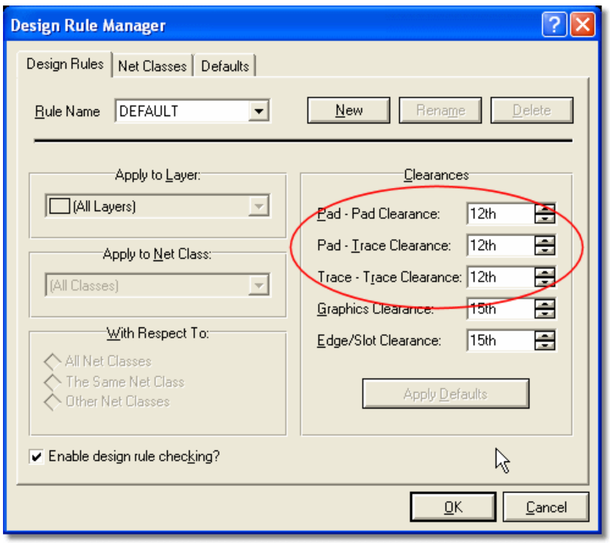
The first tab of this dialogue form allows us to configure constraints and minimum clearances for the layout. We have a DEFAULT rule loaded which must apply to all layers and all net classes, providing a set of clearances between objects equivalent to manufacturing guidelines.



This rule and these clearances are created automatically for each layout in order to provide a minimum set of constraints for a board. You can change the values of the default clearances applied to new layouts via the *Default Rules* command on the *Technology Menu.*

The first thing we need to decide is whether a single rule is suitable for all layers and all traces on the layout. It is possible to create as many new rules as required and we can limit their effect to a given layer and/or a given set of connections (net class). A great deal of information on this is available in the reference manual but for our purposes we can manage fine with configuration of the existing rule.

Given that this equipment is intended to work out-doors we'll need to increase the clearances between pads and traces to improve the insulation against condensing moisture. A 20% increase should be sufficient so we need to change the *Pad-Pad*, *Pad-Trace* and *Trace-Trace* clearances from `10th` to `12th`.



The graphics and edge clearances are fine at the default values and, as don't need to create additional rules we can move on to the net classes tab of the dialogue form.

Net Classes

This is the place where we configure trace and via styles and control which layers we route on when auto-routing the board. The selector at the top allows us to switch between different net classes and configure each separately.

Let's start with the `POWER` Net class, which should be the default selection. As we discussed in the ISIS Getting Started Guide any nets that include a power or ground terminal are automatically assigned to the `POWER` net class, unless manually overridden.

We'll set the track size to 25th, not so much for current considerations but to reduce track impedance (we'll also place a single low impedance ground plane later to help with this). In ARES nomenclature this corresponds to track style `T25`.

The neck style – if configured – specifies the track style for necking on the current net class. We won't need to worry about this and can therefore leave it at it's current setting.

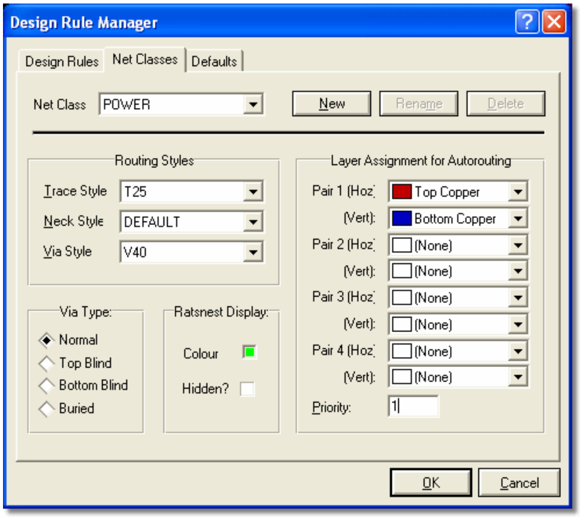
Given that we don't really have current constraints the choice of via's is a trade off between the higher manufacturing costs of smaller via's and the decreased routing quality of large via's. For the current this circuit is dealing with and for a standard 1.6mm FR-4 support plated with 35um copper, vias with the standard `0.4mm` hole are a good compromise. For the power tracks we need a suitable annular size and `40th` is a reasonable choice. We can therefore set the via style to be `V40`.

If you are not sure of the characteristics of a particular style, you can view them by selecting the appropriate mode icon (via, track, pad, etc.), highlighting the style in question in the *Object Selector,* and then clicking on the 'E' button at the top of the *Object Selector.*

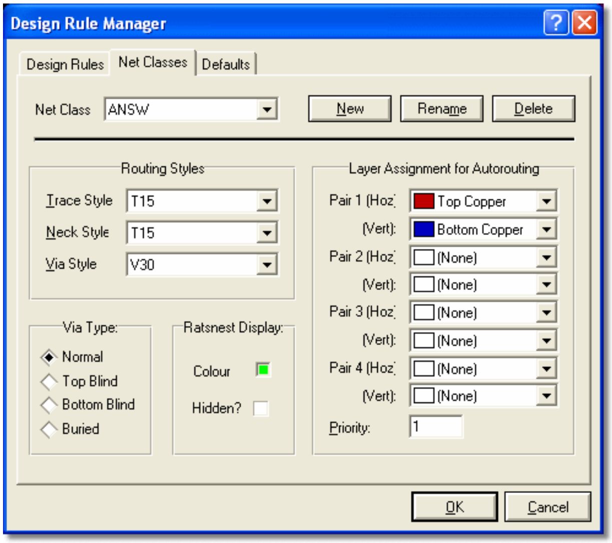
The options at the bottom of the dialogue form allow us to change the via type (clearly not relevant on a two layer board) and also to change the colour or visibility of the ratsnest lines. The latter can be useful if we are manually routing and wish to quickly distinguish between power and signal connections.

The layer assignment pairs on the right hand side tell the autorouter which layers to route on for multi-layer boards. Again, for our two layer board, there is nothing to configure here.

We should now have finished setting up the `POWER` net class – your dialogue form should now be configured as per the following screenshot.



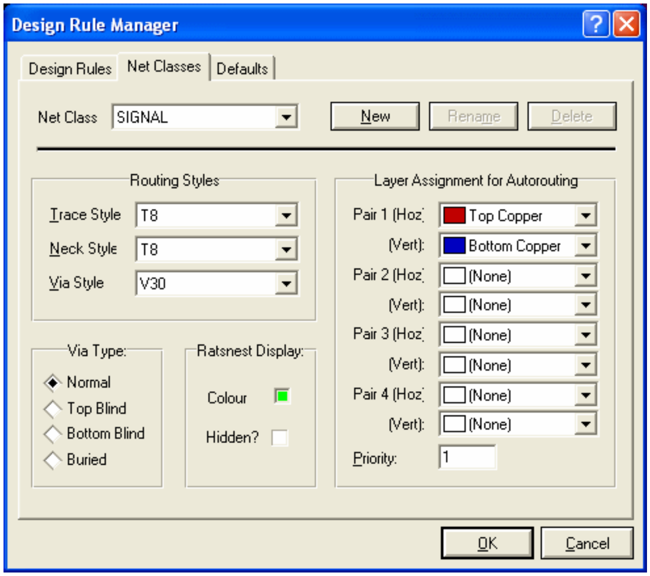
Lets move on to the next net class in the selector; the `ANSW` class. You may remember that we specifically named this net class in the ISIS Getting Started Guide in order that the 5V switched power supply for the analog circuitry (the output of the DC/DC converter) could be handled separately in ARES. What we want for these connections is a track size larger than the standard `SIGNAL` net class but smaller than the `POWER` net class, so lets change the Trace Style to `T15` (15th track). For consistency, the neck style can also be `T15` and we can use a smaller via (with the same `0.4mm` hole) by selecting the `V30` Via Style.



If you don’t have a V30 via style you can create one easily by going to the Library Menu, selecting *New Via Style* and filling out appropriately.

Creating your own net classes in ISIS is extremely simple and provides you with the flexibility you need both with regard to trace and via configuration but also with design rule constraints.

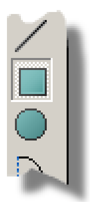
The final net class we have is the standard `SIGNAL` net class that encompasses all non-power and unspecified connections. We’ve deviated from the widely used “8/10 Rules” (8th trace, 10th clearance) in order to compensate for outdoor usage but the standard 8th track width (style `T8`) is fine. We can also set the neck style to 8th and keep our standard `V30` via style for these connections.



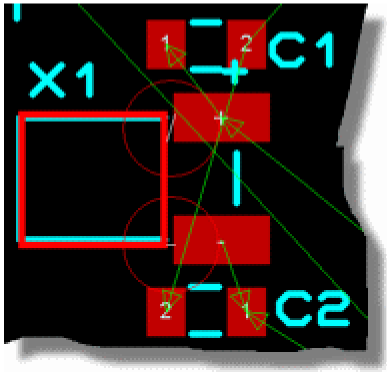
Having finished configuration, exit the dialogue form to return to the layout.

Keepout Areas

We can also introduce tracking constraints by limiting areas in which tracks can be placed. A good example of this is the crystal towards the bottom left of the layout where we do not want tracks under this area. To form a keepout area, start by selecting the 2D graphics icon and changing the layer selector to be keepout.

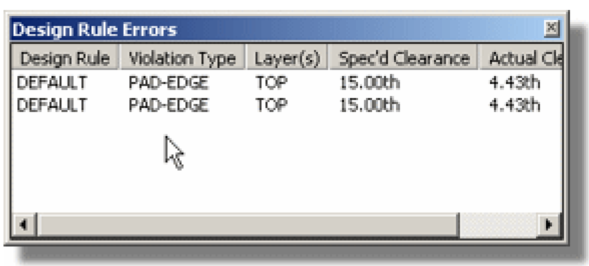


Next, place a small box around the silkscreen of the crystal in exactly the same way as we did for the board edge (left click to start placement, drag out the area, left click again to commit placement).



Unless you are very skilful you will be presented with a box indicating that design rules have been detected.

Checking the box on the dialogue form will prevent it from appearing in the future. If you left click the mouse on the DRC section of the status bar a small window will appear providing information on the errors. You should see that they are of type `PAD-EDGE` as the pads of the crystal are closer to the keepout graphic than the `15th` specified in the Design Rules.



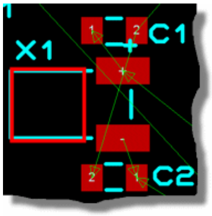
We have two options here:

1) Ignore the DRC errors. The keepout graphic will not impact on connectivity.

2) Move the graphic to a legal distance from the pads. The easiest way to do this is to first change the snap level down (View Menu), right click on the graphic and select the *Drag*

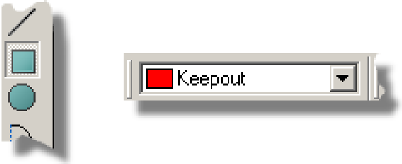
Object context menu option, and then change the snap level back when you are finished.

When you are finished your keepout area should look something like the following screenshot.

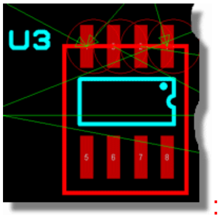


We have a similar problem around the temperature/humidity sensor (U3). We are going to want a slot machined on the board to cut the thermal path and reduce the measurement errors of the sensor (we want to measure environment temperature, not the one conducted by the PCB). We therefore need to ensure that no tracks are placed in the area we are going to slot through:

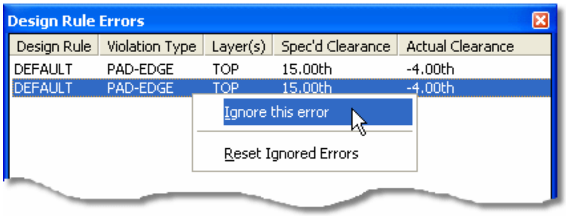
1) Select the 2D Graphics rectangle icon and then change the layer to be KEEPOUT.



2) Place a rectangle around the bottom half of the IC as shown below.



This time we have no option but to accept the DRC errors temporarily. This is not a problem as the keepout area is not output for manufacture and can be removed following the routing of the board. We will come back to this when we discuss adding the cutout slot later on in the



Right click on the error and select Ignore this error

ROUTING THE BOARD

Having configured the board constraints we can now move on to actually making connections and routing the board.

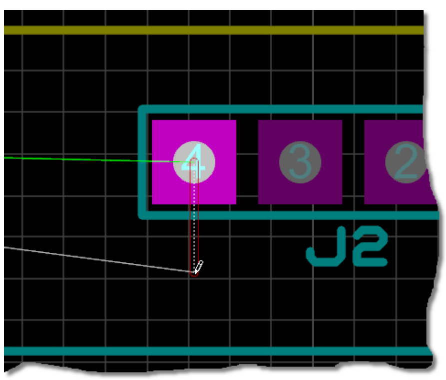
Placing a Route Manually

Let's start by manually placing some tracks on the board. Typically, we would manually lay out connections where a specific path is desired for the track or where we need greater control over track position. On our layout we want to make sure that our connections from connector J2 follow a sensible path around the board so we can start here and route them manually.

Start by selecting track mode at the left hand side of the *Object Selector* and changing the layer selector to be on *Top Copper*.

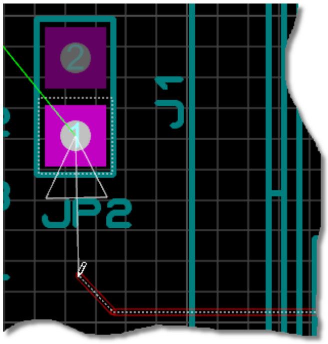
If we look at pad 4 of the `J2` connector we can see that the closest rastnest line is directing us across to the `GND` pad on the step converter. This is not ideal as we would have to navigate the mounting hole and then track into the small SMT pad. Instead, we will route this down and across the bottom of the board to a more convenient ground connection.

ARES features a sophisticated 'follow me' routing algorithm for manual routing in which the route being placed will follow the path of the mouse as best it can while obeying all of the design rules for the board. You start track placement by left clicking the mouse on pad 4 and then move the mouse downwards.

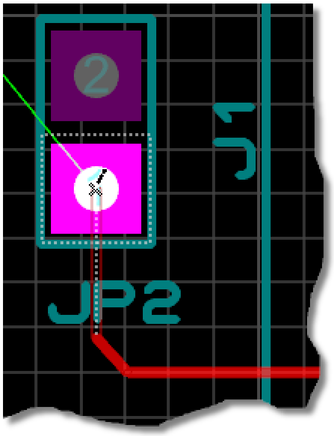


You should see that the closest legal destination for the track is now highlighted in white. This will update as we move the mouse and since we are not going to route to this destination we can ignore it for now. When the mouse approaches the bottom of the board, left click once and then change direction to the left. As we move further to the left the ratsnest guide will change to show us that we can terminate the track on pin 1 of jumper JP2.

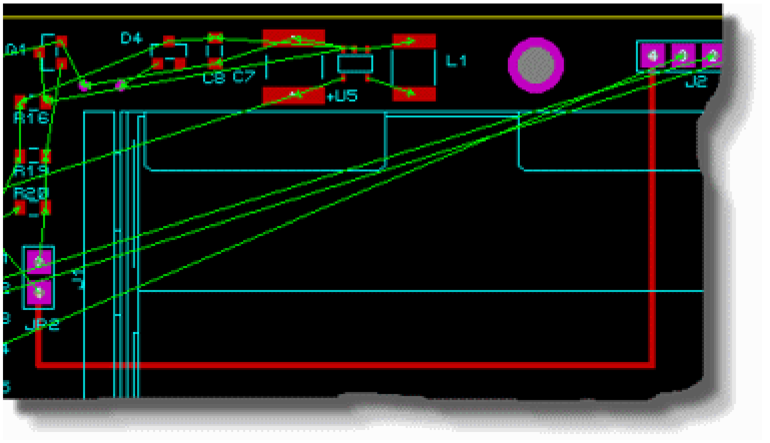
You will see that as we move further to the left the ratsnest guide will change to show us that we can terminate the track on pin 1 of jumper JP2.



Once we are underneath the jumper we can left click once again to change direction and move upwards towards the destination pin.



Finally, we can clicking left over the pin to finish placing the route. This will both commit the route and remove the ratsnest line corresponding to this connection.



Note that we did not need to select the width of track to route with as we configured this in the previous chapter. ARES recognizes that we are routing a track on the `GND` net, applied the rules for the `POWER` net class and selected the specified `25th` track style for us.

Manual routing is probably the most common action you will perform with the software and it is vital that you understand how it works. The basic rules of operation are:

- Left click on pad, track or zone border to begin routing from that object.
- Left click at any point during routing to commit the route up to the mouse position (we call this anchoring).
- Right click to terminate the route at it's last committed / anchored point.
- ESCAPE key to abandon route entry completely.
- SPACEBAR to float a via on the end of the route and left click to then place the floating via.
- Double left click to drop a via at the current mouse position.
- Move the mouse backwards over existing tracking to ‘rub-out’.

We strongly recommend experimenting with manual routing for a while on this board until you are comfortable with how it works. The following is a summary of how to perform common actions; try these while routing any of the remaining connections.

Panning and Zooming

While placing a track you can use the middle mouse wheel (or F6 & F7 keys) to zoom in and out during routing. Panning will happen seamlessly when the mouse is at the edge of the Editing window during routing.

Placing Anchors

The follow me routing algorithm will move the track being placed according the way you move the mouse. If you want a track to follow a particular path then you need to help by left clicking whenever you change direction. This places an anchor or commits the route up to the mouse pointer such that the follow me router will not change it. You will see this happening as the outline track becomes solid.

Getting Stuck / Re-routing while routing

Since the manual routing system obeys the design rules for the board you don't need to worry about clearances while placing routes. You can however route to a place where you are blocked (the routing icon will change to a no-go sign at this point). Often you just place a via and continue on but sometimes it is better to rewind and try another path across the current layer.

Moving the mouse backwards over the route being placed will rub out that portion of track so to rewind and change direction just move the mouse back to the last good point - anything you had placed from that point onwards will be removed.

On densely packed boards in particular, speed of mouse movement is an issue. Remember that you are guiding the placement so moving slowly through tight spaces will work far better than ripping the mouse from source to destination.

Placing Vias

If you double click during placement you will place a via at the point the mouse is at and can then continue routing on the associated layer. If you press the spacebar you will float a via on the end of the mouse and can then position the via manually before placing with a left click. Using the spacebar has the advantage of snapping to legal objects (e.g. via under SMT).

In either case the placement of the via is also design rule aware and ARES will not let you drop a via in an illegal spot. If you use the floating via method (with the spacebar) ARES will attempt move the via to the nearest legal spot. This can be extremely useful on busy boards or when you want to butt a via tightly against another object.

Layers used for via's are defined in the Layers Pairs dialogue on the Technology Menu. Examples of routing with vias and discussion on layer pairs follows further in the tutorial.

Object Hugging

Since the design rules are all live during placement it is quite easy to hug a track to another track or to wrap a track around an object. If the mouse is over an object where the route being placed cannot travel then the route will follow the mouse path as closely as it legally can, essentially hugging the barrier object.

Abandoning a Route

- If you want to abandon the route at the last committed (solid) segment then right click the mouse.

- If you want to abandon the route at the mouse pointer then left click to commit up to mouse pointer and then right click to terminate the track.

- If you want to abandon the route completely then hit the ESCAPE key.

Making Connections

If you are connecting directly to a pad then a left click of the mouse will complete the connection and terminate route placement.

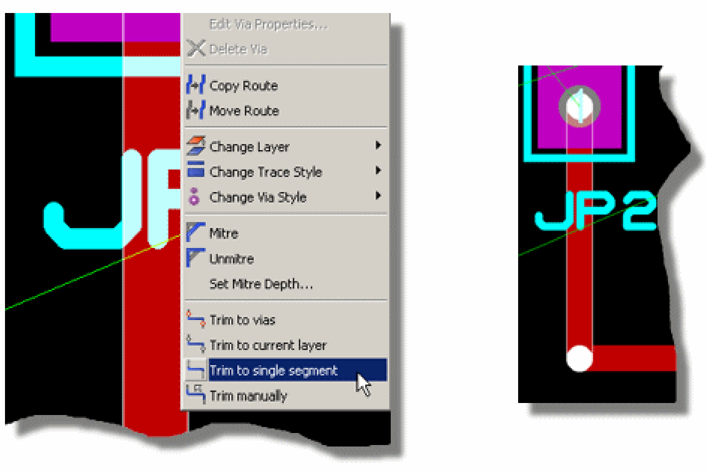
If you are connecting to a track then left click will commit the placement and right click will terminate the route forming the connection. The same procedure applies if you are connecting to a zone except that you must connect to the zone border.

Several short support movies showing different routing techniques are available to all professional users on our support forums (http://support.labcenter.co.uk). You will need to register and be activated in order to see these member forums.

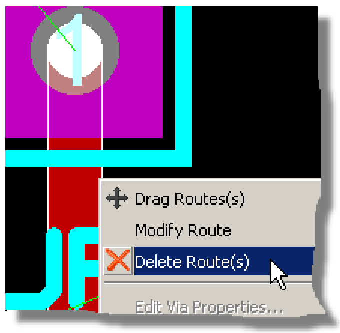
Deleting a Route

We can delete either the full route or a specific part of the route if we are not happy with the placement. Let's assume that we were not happy with our placement of the last piece of track (upwards to pad `1` of `JP2`).

Start by right clicking on the track over this segment of track. This will highlight the entire track and the *Delete Route* option near the top would then remove the complete trace from the board. However, we have far more control if we use the options at the bottom of the context menu. In this particular case, select the *Trim to Single Segment* option; this will change the selected area of track to the segment we have clicked on.



Next, right click on the highlighted segment and select the *Delete Route* command; this will delete only this segment of the route.



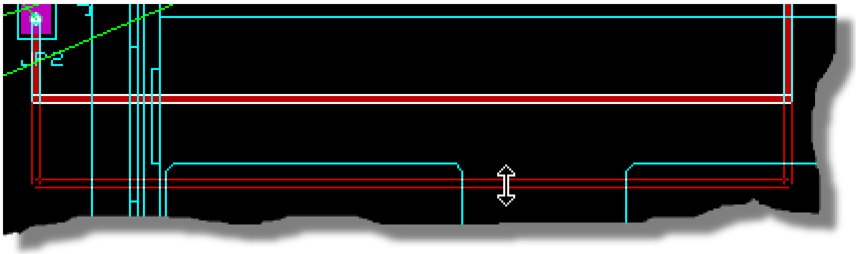
Finally, zoom in, change the snap settings if need be and replace the route from the existing track to the terminating pad.

If things have gone more badly wrong you can simply delete the entire route and start again; what you are aiming for is something like the following screenshot:

Editing a Route

It is often the case that routes need to be nudged or moved into position after placement and we definitely don't want to be deleting and replacing routes or parts of routes all the time. To take an example, lets move our track lower down the board a little, towards the bottom board edge.

Start by right clicking on the horizontal section (segment) of the track; this is the section we want to move. Next, select *Drag Route* from the resulting context menu, move the mouse down to 'pull' the track into the desired location and then left click again to commit placement.



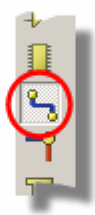
Moving tracks works as we've seen on a track segment. Using the *Trim Manually* option on the context menu for a track allows you to define your own segment and you therefore have complete flexibility with altering track topology.

Layer Pairs & Manual Routing

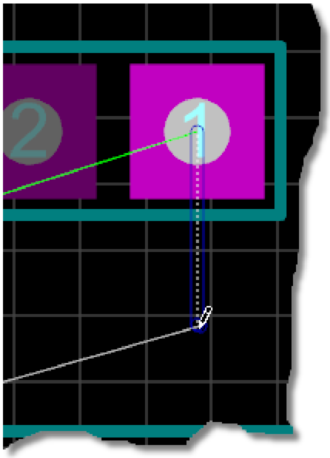
The track we have placed is of course only on top copper, whereas we often want to via up and down through the board when routing. ARES handles this with a concept called *Layer Pairs.* This means that every layer on the board has an associated layer so that via destinations are known during placement. For a two layer board this is obvious with top copper being associated with bottom copper and vice versa but with multi-layer boards configuring the layer pairs (*Technology – Set Layer Pairs*) can be an important step.

In our case, the default assignments are correct and no action is required. Let's place a couple more routes manually to see how this works. We'll look at connecting pins 1 and 2 of the J2 connector which are the transmission lines from the USART on the dsPIC processor.

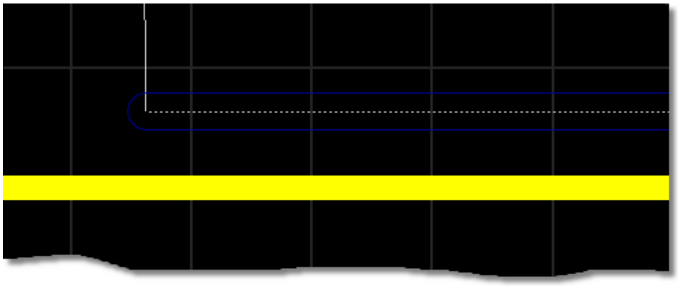
We'll start with pin 1 and route from the pin on the bottom copper layer. Make sure that you have trace mode selected and then hit the space bar on the keyboard, noticing that this will toggle the layer between the two layers associated as a layer pair. If you have ended up on top copper simply hit the space bar again to set the layer to bottom copper.



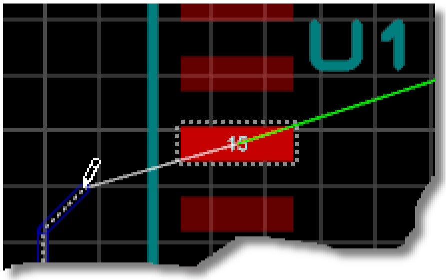
The processor is some distance from the connector and the easiest path would seem to be down and along the bottom of the board. Since this is a long connection you will find it useful to use the zoom in and out techniques to give you the desired view as you are placing the track. Begin placing the track by left clicking on pin 1 on the connector and then moving the mouse downwards



When we get near the bottom of the board, left click the mouse to place anchors and guide the mouse to the left. To maximise space on the board you can route across the bottom of the board with the mouse over the board edge graphic - this will hug the track to the absolute edge of the board leaving only the Edge Clearance between the two.



Once you get near the U1 IC bring the track up just past the right hand pads of the processor footprint.



We now need to place a via before we can connect to the SMT pad on top copper. As described above you can either double click the mouse at the point you wish to place the via or you can press the spacebar to float a via and then guide the placement point with the mouse before left clicking to commit. The latter method has the advantage of allowing you to butt the via as closely as possible against the pad thereby minimising the stub track length on top copper. (Image 1below).

Left click again and move the mouse up until you are level with the destination pad (if you can't get level adjust the snap settings and/or the zoom level). We now need to switch to top copper in order to make the connection; this is done by double clicking the mouse. ARES will place a via (of the style we specified in the previous chapter), the layer selector will switch to top copper and you can simply move the mouse the mouse over the destination pad and left click again to complete the connection. (Image 2 below).

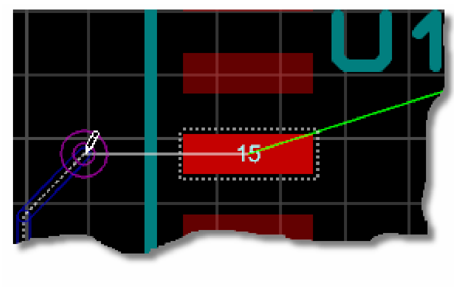


Image 1.

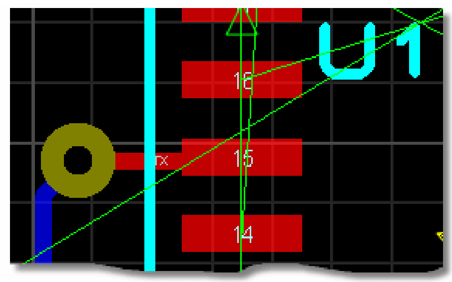
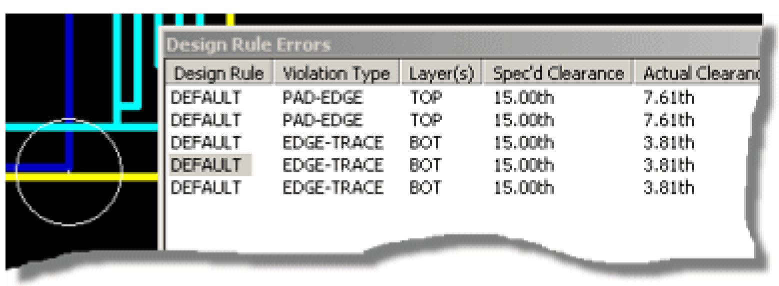


Image 2.

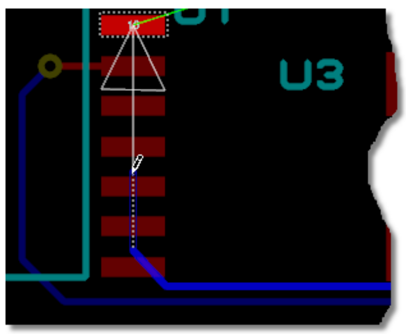
If you have placed the track too far down you may find that the status bar indicates that you have DRC errors. Left click on the design rule display on the status bar to launch the information window and then double click on one of the errors to zoom the layout to the offending area.



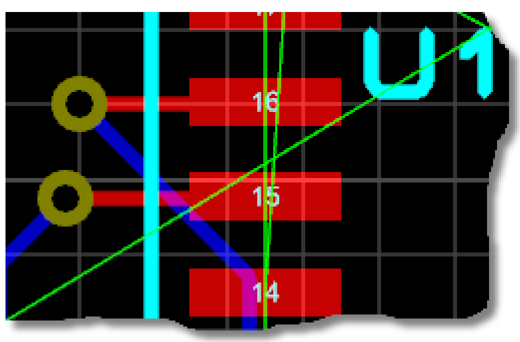
These errors will be because the distance between the board edge and the track is less than we specified in the *Design Rule Manager* in the previous chapter. As we've seen previously we can move the track by right clicking on the horizontal segment of the track, selecting Drag Route from the context menu and then moving the track upwards slightly. Once we've moved the track we should see that the DRC display returns to reporting no errors.

Quite a lot has been covered in making these connections and, unless you are fairly skilful, there has probably been some finger trouble along the way. We'll continue with the other connections from the J2 connector to help familiarize ourselves with the techniques we've introduced.

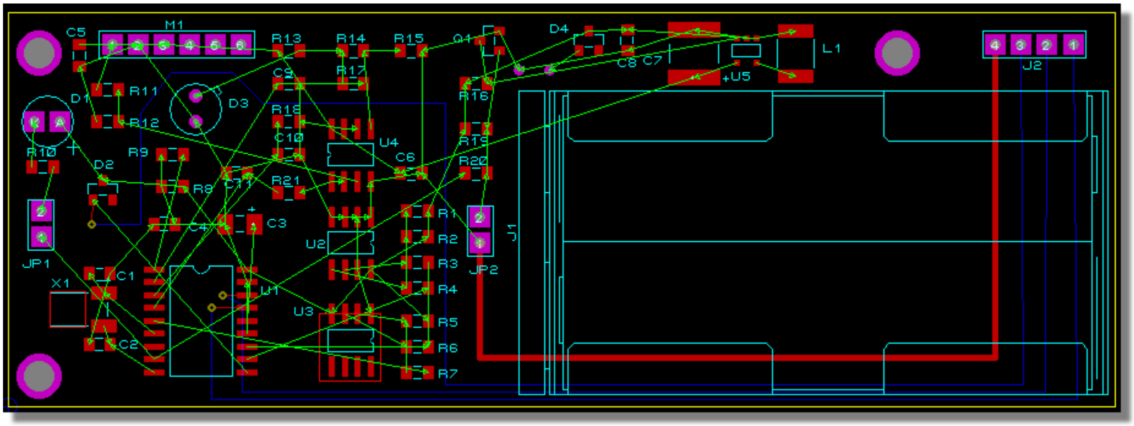
Pin 2 of the connector should follow an almost identical path to pin 1 and again we can start placement on bottom copper (check the layer selector before you start placement; the space bar will toggle the layers). We obviously need to avoid crossing the track we have just placed by routing horizontally just above it.



In order to make the connection, we can then route vertically underneath the pads and make a short 45 degree track before double clicking to place the via and complete the connection



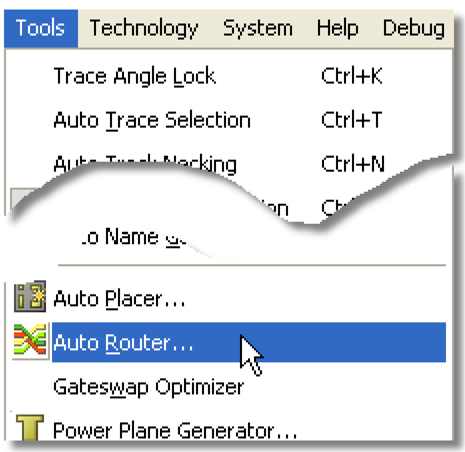
Of course there are many ways we can route the board and personal preference also plays a part. Feel free to experiment with the other connection on the connector, working around the battery holder, using the ratsnest to find a legal destination and then completing the route. If you get design rule violations you will need to either move the track or perhaps delete a segment (the trim options on the context menu) and redraw. The following screenshot is an example of what you might have when you are finished.



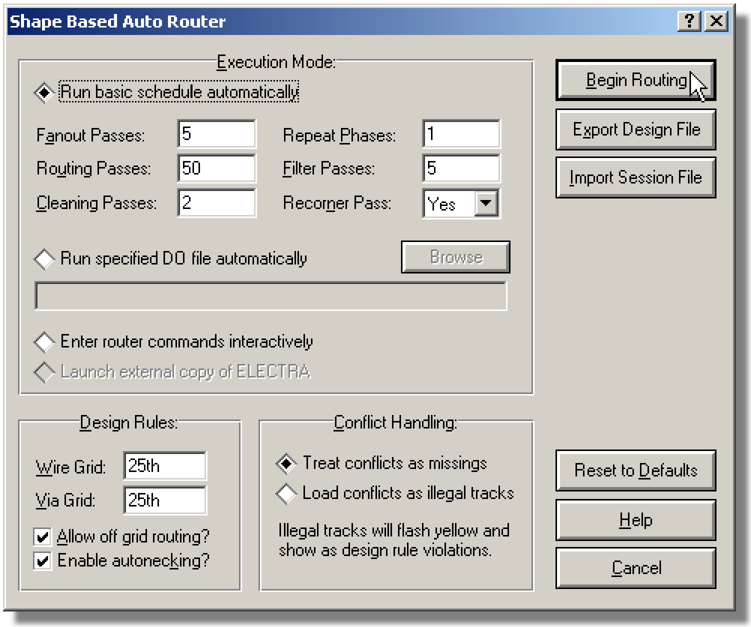
Basic Auto-Routing

From this point on we'll use the auto-routing engine to complete placement of the routes. As with the manual tracking the auto-router will obey all the design rules that we configured earlier.

Start by invoking the auto-router from the Tools Menu in ARES or from the icon at the top of the application.

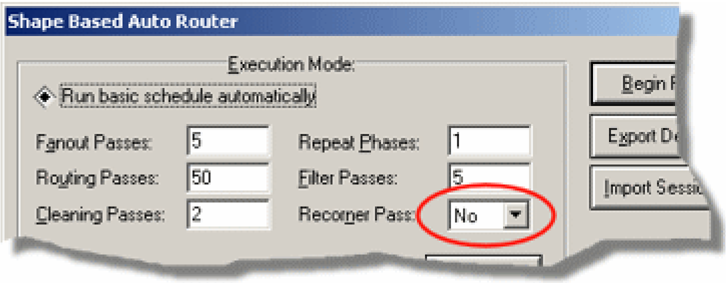


The resulting dialogue form is reasonably complex but all fields have context sensitive help associated with them. You can also find additional information about the various type of routing pass in the reference manual. For our purposes (and indeed for most boards of small/medium complexity) the defaults are more than adequate. We'll start by running in 'fully automated mode' which should be selected by default so simply hit the begin routing button to complete the remaining connections.



Everything will happen quite quickly from this point onwards but you should see routing progress on the status bar as the engine works towards completion. Once the board is complete there are two immediate points of note :

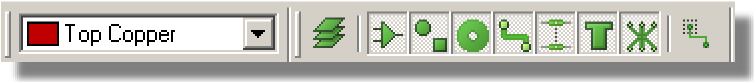
- The auto-router has preserved those tracks that we placed manually and has not ripped and replaced them whilst working on the rest of the board.
- Once the autorouter has completed a final pass is made to corner the tracks. If you prefer this not to happen you can remove the *Recorner Pass* option on the dialogue form before invoking the router.



The Selection Filter

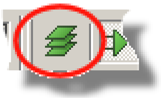
Now that we have a completed board it's worth spending a little time looking at the techniques for selecting various object types on different layers.

ARES uses the *Selection filter* at the bottom left of the application window to determine which objects are available for selection at any given time.



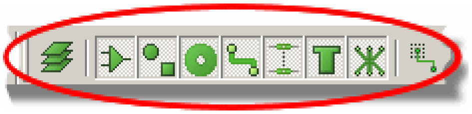
The left most button will determine whether the layer selector is active:

- When this is toggled off the selection applies to all layers on the board.

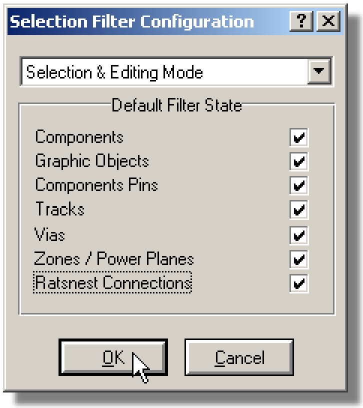


- When toggled on the selection applies only to the layer specified in the layer selector

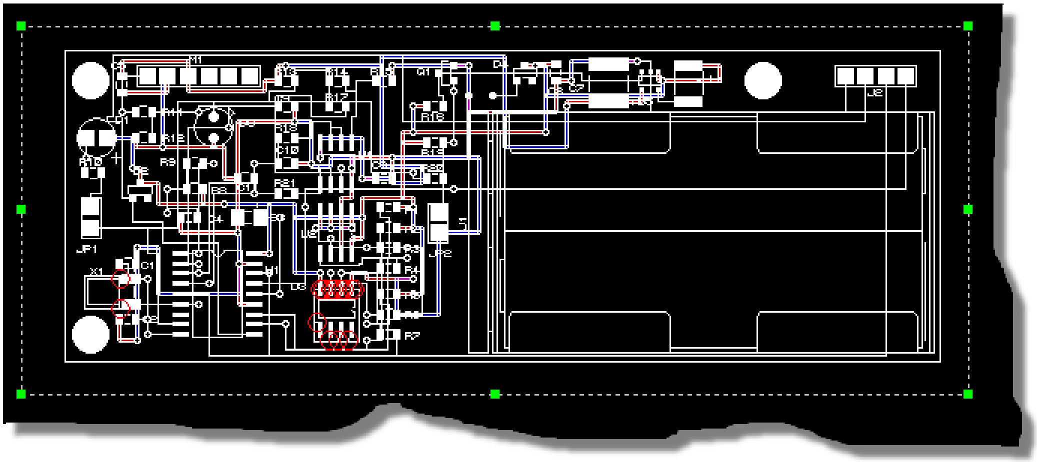
The other buttons represent different object types (tracks, components, graphics, etc.) and determine whether those object types are selectable. Hover the mouse over an icon for a tooltip description.



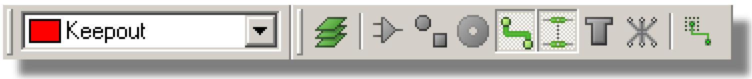
If you switch between the different modes of operation (for example from selection mode to track mode to component mode) you will see that the selectable object types will change according to the mode you are in. Whilst these are good defaults for normal operation you can change at any time simply by toggling on or off the layer switch or object type that you do or do not want to be selectable. If you find yourself changing selectable items regularly you can also change the defaults via the *System Menu – Set Selection Filter* command.



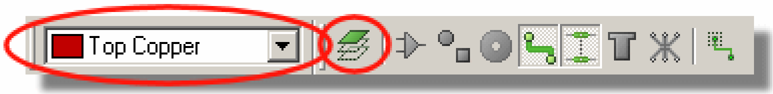
Let's take a practical example and delete all the traces on the top of the board, except the one we have manually placed. Start by entering *Selection Mode* and left dragging a selection board around the entire board.



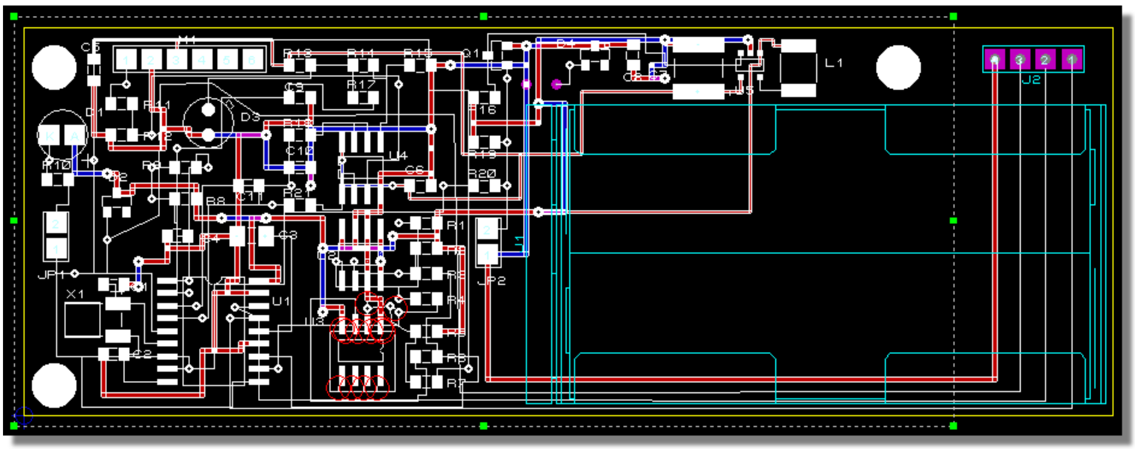
Next, deselect those objects that we do not want to delete, namely everything apart from tracks and via's. The tagged items will update automatically to provide visual confirmation of what is selected.



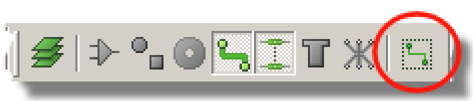
Now change the layer to be *Top Copper* on the layer selector and toggle the layer button such that the selection filter applies only to the current layer.



Use the green drag handle at the right of the tagbox to move the selection filter in until it does not encompass the J2 connector.



Finally, use the icon at the far right of the selection filter to deselect items that are only partially inside the tagbox; in our case this will deselect the track we manually placed from J2, pin 4.



We can now simply hit the delete button on the keyboard or right click inside the tagbox and select *Block Delete icon* to remove all the top copper tracks.

Bear in mind that the selection filter controls what objects are available for selection at any given time. If you ever find that you can't select an item the first thing to check is whether that object type is enabled for the mode that you are in. Alternatively, simply switch to *Selection Mode* where all items are selectable.

Advanced Auto-Routing

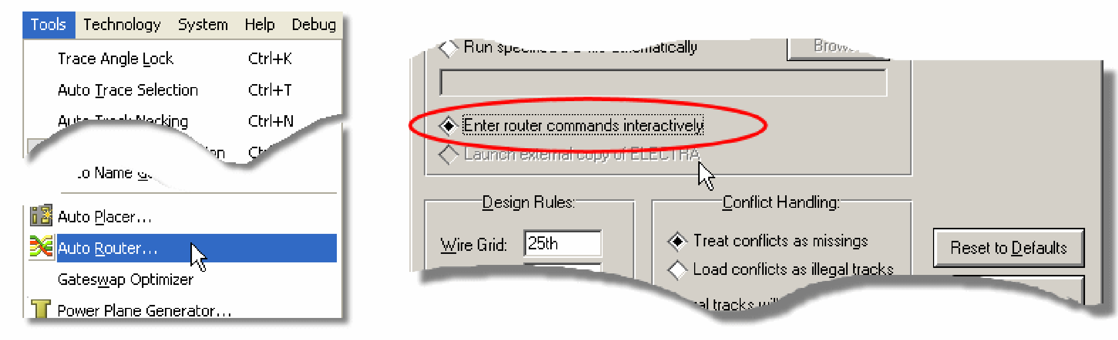
Given that we've ripped a lot of partials and removed via's as well we've actually made a bit of a mess here. Fortunately, we can fix the situation by simply re-invoking the auto-router and re-placing all our tracks. The router has it's own clean up phase so it will tidy up all the redundant partial tracks for us as it completes the new routing.

If you have a standard features version of Proteus (PCB Starter Kit, Level 1 or Level 1+) simply run the autorouter as we discussed in the previous section to re-route the board.

For those using the Advanced Feature Set, we'll use this section to show in brief some of the additional features available. These fall into two main categories:

- The ability to route only a specified area or set of connections.
- The ability to control the routing script; i.e. to determine which routing commands are executed and in which order.

Start by invoking the autorouter dialogue form (*Tools Menu)*, switching to *Interactive* mode and then selecting the *Begin Routing* button.



A command window will open at the bottom of the *Editing Window*, which will allow us to direct the routing progress interactively. ARES provides a rich command set to control routing, including such things as bend radius for mitering tracks and fanout length and direction from SMD pads. This is fully documented in the reference manual so we will concentrate on some basic practical examples in this tutorial.

There are a couple of important points worth emphasizing at this stage:

- Any commands entered will action on a set of tagged connections **or** on the whole board if nothing is highlighted.
- Changing the circuit or switching modes will automatically exit the routing interface.

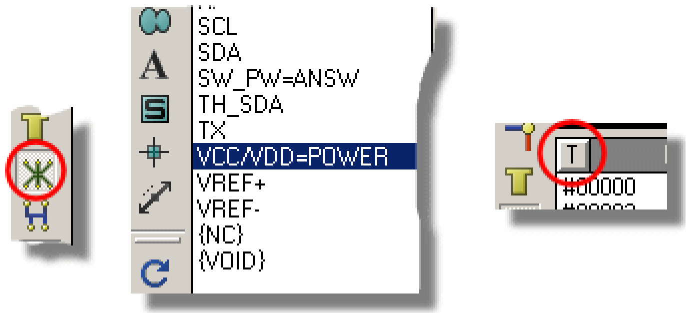
Essentially, what this means is that we can dictate what connections are routed by controlling which items are highlighted.

First of all let's clean up the loose tracking on bottom copper resulting from our previous rip of the top copper tracks. The basic syntax of most commands is:

`<command> <number of passes>`

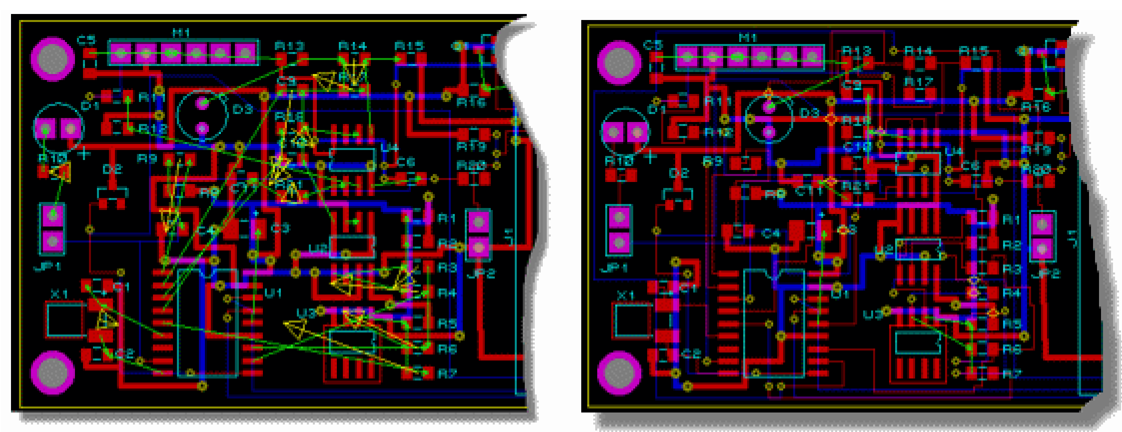
so we can start by typing `clean 2` to clean up the superfluous tracking (hit the `ENTER` button on your keyboard to action the command).

Next, lets assume that we want to route all the VCC connections. Select the `VCC/VDD=POWER` net in the *Object Selector* and click on the small `'T'` button above the Object Selector to highlight all the connections on that net. Finally type in `'route 5'` to route these connections.



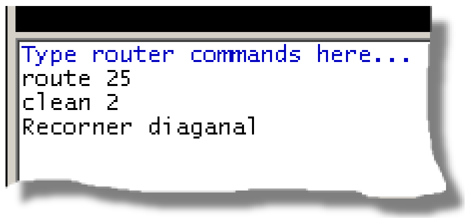
Clicking anywhere in the *Editing Window* will deselect the currently tagged items. When the focus is in the *Editing Window* you can also use the middle mouse (or keyboard shortcuts) to zoom in and out in the normal way.

Similarly, we can highlight connections on an area of the board and route them independent of the rest of the board. For example, right depress the mouse and drag a selection box around the left half of the board and then type `'route 10'` to complete open connections in that area.



You may notice that this has left a couple of ratsnest lines or has not completed all of the connections. We could try to resolve this by issuing `clean`, `filter` and more `route` commands, but for our purposes it's easier to simply revert back to a 'whole-board' scenario and then completing routing. You can do this by left clicking on an empty area of the *Editing Window* to clear the selection. Typing `route 25`, followed by `clean 2` should see the board completed.

Finally, we can reduce trace length by typing `recorner diagonal` as a finishing touch.



Do note that there is much more flexibility in the command set than that shown in this tutorial, both in the number of commands and in the parameters that control the command actions. See the reference manual(Help Menu – AREA Help) for more information.

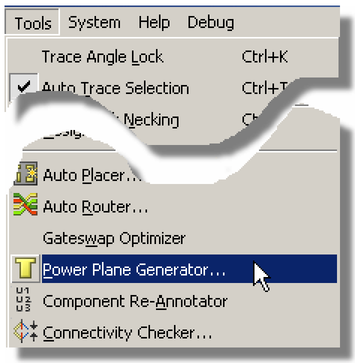
You can exit auto-routing at any time via the ESCAPE button on your keyboard.

POWER PLANES AND SLOTS

Placing Power Planes

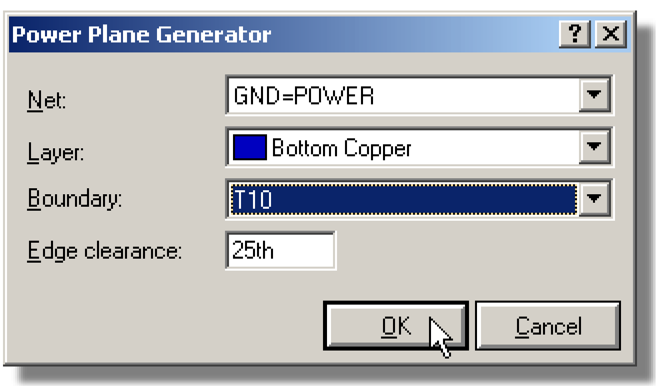
Now that we have the board routed we are nearing completion of the actual layout phase. However, to minimize track impedance we are going to place a ground plane covering the entire board.

This is actually the easiest type of power plane to place and is available in all levels of the professional software. Start by invoking the power plane generator command from the *Tools* Menu.

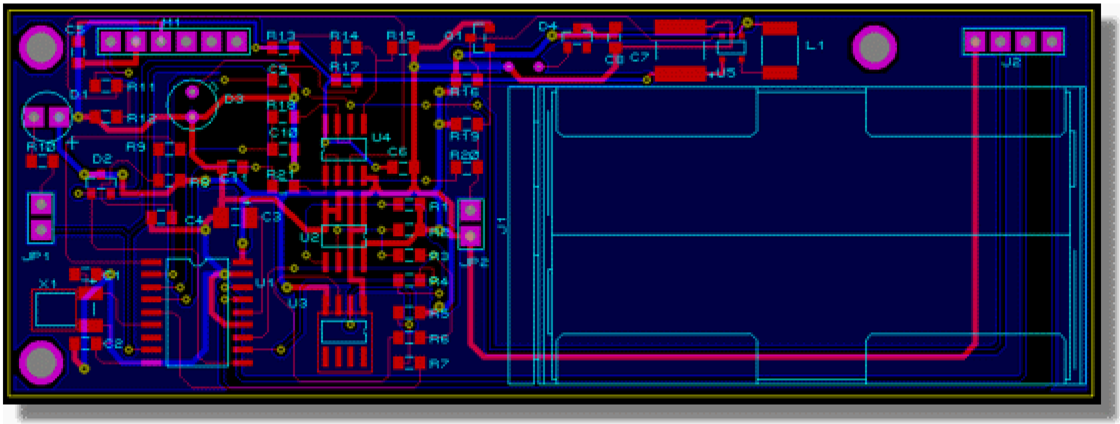


From the resulting dialogue form select the `GND=POWER` net, keeping the layer as bottom copper and setting the boundary style to be `T10`. This is the trace style in which the inner and outer boundaries of the zone are drawn and also determines the thinnest section of copper by which the power plane can make a connection. Setting this larger will prevent the copper flowing through small gaps (e.g. between pins) but making it smaller means that connectivity may be made only by thin sections of copper.

We can leave the clearance between the power plane and the board edge at the default value.



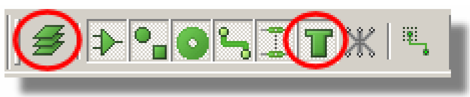
After you exit the dialogue form you should see that the power plane is generated across the entire board.



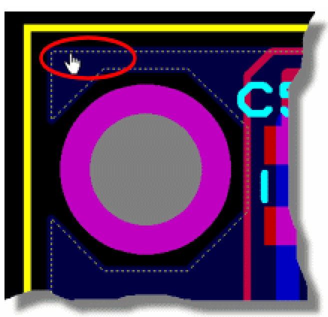
Nesting and Islands

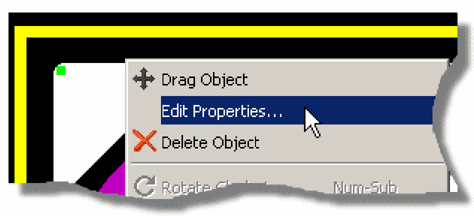
There are several additional configuration options available to us now that we have placed the zone. Editing a zone is slightly different from most other objects in ARES in that you must right click on the **border** of the zone (to avoid continuous unwanted selections).

Firstly, check the selection filter and make sure that the zone object type is selectable (or switch to selection mode).



Next, zoom in and move the mouse over the edge of the zone, right clicking the mouse when the zone is shown as active under the mouse. Select the *Edit Properties* from the resulting context menu.





The main options of interest are towards the bottom of the dialogue form and are explained below:

Relieve Pins

When checked pins on the same net as the zone will have thermal relief applied to them; the thickness of the thermal relief is determined by the *Relief* field of the dialogue form.

- The software will prevent you from using a relief track style that is larger than the boundary style thickness; this is to protect against the reliefs 'sticking out' of the boundary.

- The topology of the relief stems on a particular pad can be changed to a diagonal 'X' by editing the pad itself after placement. This is sometimes useful to maximize contact with a zone.

This option should almost always be left in its default checked state.

Exclude Tracking

If this option is checked, the zone will treat tracks on its own net as obstacles. Otherwise the zone flows over such tracking, effectively ignoring it. Tracks on other nets, or loose pieces of track on no net are always treated as obstacles.

This option is normally left unchecked.

Route to this Zone

When checked this option allows the auto-router to route appropriate SMT pads to this zone through an auto-via process.

Suppress Islands

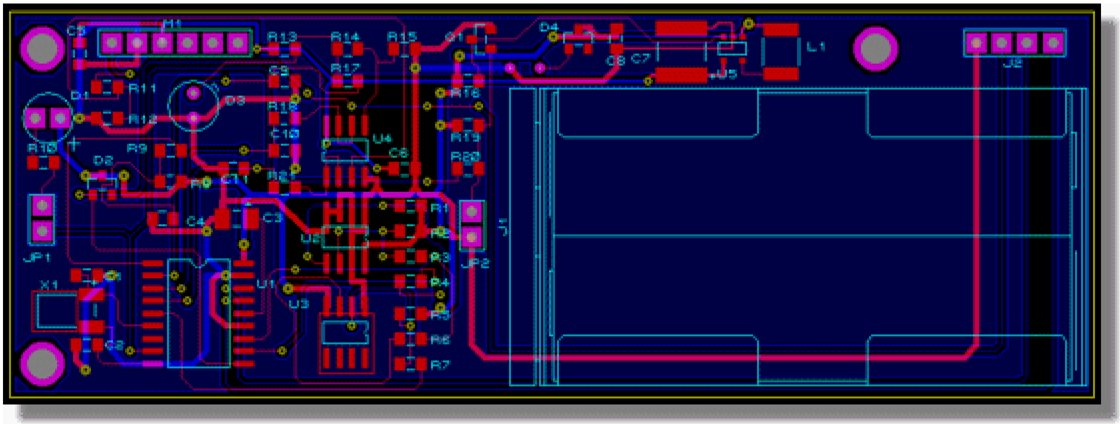
An island is defined in ARES as a block of copper or zone area in which no valid connections can be made. When checked, ARES will remove all such blocks from the board, leaving only areas of copper with connections.

Allow Nesting

It is quite common, particularly with larger boards, that the flow of a zone will encounter an obstacle through which it cannot pass. When checked, the zone will jump over the obstacle and continue flowing across the board.

This is a very useful option for example on busy boards or where you want to connect a pad where the topology of the board makes it impossible for a single zone flow to get through.

Given our relatively simple board, the default options are ideal for our purposes. Selecting the nesting option will have no effect for example as there are no inner boundaries on the board. However, if you want to experiment you should find that if you uncheck suppress islands option, the resulting zone populates far more of the board that in our initial configuration.

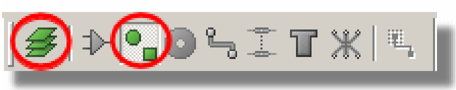


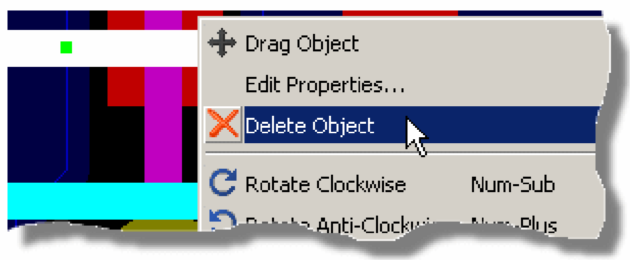
We have covered only the basics of power plane functionality here relevant to our current design. For more information, including split planes, zone keepouts, stitching and bridge tracking please see the reference manual.

Slots

To complete the layout of the board we need to return to our temperature/humidity sensor (U3). In order to make accurate measurements, we want a thermal cutout around this part, ensuring that the temperature we are measuring is actually the environmental temperature and not conducted heat from the PCB.

You may remember that we placed a keepout area around this part to prevent the autorouter from placing tracks through the area we want to cut out. Start by zooming in around U3 and removing the keepout. You'll need to either switch to Selection Mode or to change the selection filter such that you can select 2D Graphics.

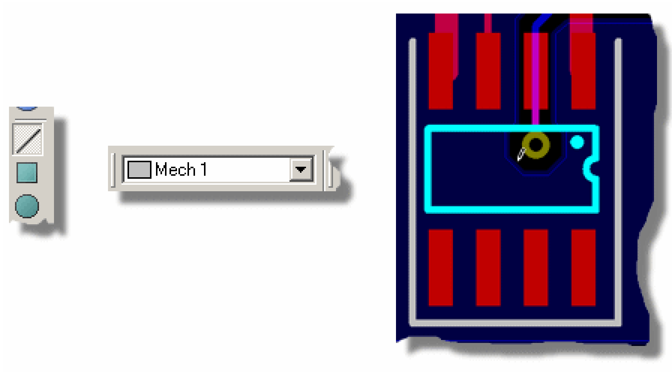




While we are it, we might as well move across and repeat the process by deleting with the other keepout around the crystal.

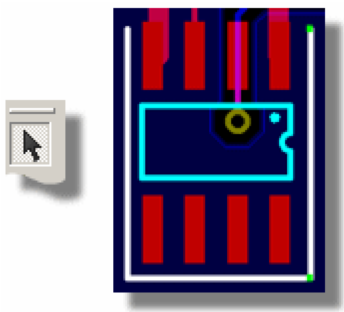
Returning to U3, we now need to outline the cutout region. In ARES this is a two stage process; we need to place graphics appropriately on a mechanical layer and then designate the mechanical layer as the slotting layer when we generate output for manufacture.

Select the 2D Graphics Line icon, change the layer selector to be `MECH1` and then place three lines to form a ‘U’ shaped cutout.

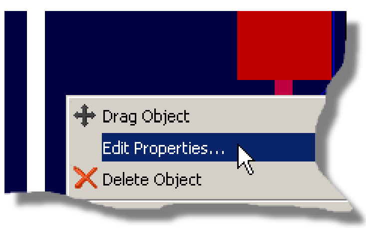


Finally, we can thicken these three lines to something more appropriate as follows:

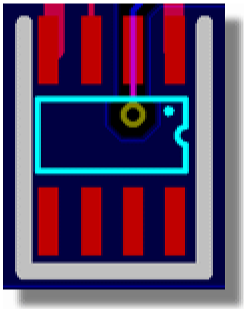
- Enter selection mode, hold the `CTRL` button down on the keypad and left click once on each line. This will select all three lines.



- Right click on any line and select *Edit Properties* from the resulting context menu,



- Uncheck the *Follow Global* checkbox, change the width to be `20th` and then apply to all tagged graphics.

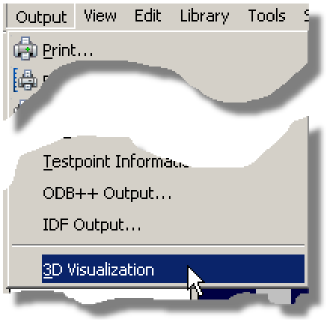


We will cover how to specify `MECH1` as the slotting later in the documentation in the section on manufacturing outputs.

Now that the physical layout of the board is complete we can look at visualization and output for manufacture. If you have not worked through the full tutorial to date you may want to load the pre-supplied layout file (`dsPIC33_REC_COMPLETE.lyt`) from the `../Samples/Tutorials` section of your Proteus installation before continuing.

3D VISUALISATION

Now that that the board is now routed and ready for production we first want to examine it in 3D in order that we can properly preview how it will look in real life and possibly make final design alterations prior to prototyping. Start by invoking the 3D Visualisation Engine from the *Output* menu in ARES.



Basic Navigation

The first thing we can do is view the board from different preset angles. Five preset views are supplied: top view, front view, back view, left view and right view and these are accessible via any of the following methods:

- Menu options on the *View* menu in the 3D Viewer
- From the navigation toolbar at the bottom of the 3D Viewer
- From keyboard shortcuts F8 through F12 whilst in the 3D Viewer.

The 3D Navigation Toolbar.

Now that we can look at the board from a number of angles the next thing is to be able to look at it at a specific zoom level. Again, there are numerous ways to zoom in and out of the board:

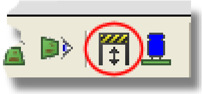
- Roll the middle mouse wheel in and out (recommended)
- Menu options on the *View* menu
- From the icons on the Navigation Menu
- From keyboard shortcuts F6 (zoom in) and F7 (zoom out)

Experiment now with custom zoom levels and different preset views – whilst it is pretty subjective we envisage most users changing views via the navigation toolbar or keyboard shortcuts and using the middle mouse wheel to zoom in and out.

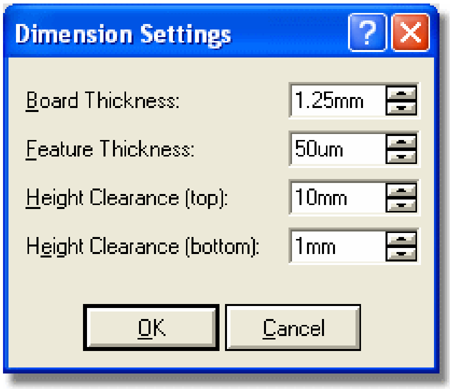
Bareboard View and Height Clearances

For inspection of resist and hole depth it is often useful to view the board without components. Selecting the bareboard view will remove all the physical components from the board.

By contrast, if you need to fit the PCB into a chassis and wish to check height clearances you can enable the height boundary box via the icon at the bottom of the display

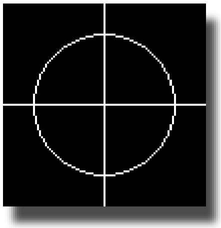


Specification of the height takes place from the *Dimensions* command on the *Settings* menu of the viewer.



Custom Views

The next logical step is to be able to customise the view. This works conceptually by ‘attaching’ the mouse to the camera such that as you move the mouse the camera moves to the area of the board that you are interested in. You can invoke the Navigation mode either from the *View* menu, the crosshairs icon on the *Navigation* toolbar or simply by clicking the left mouse button.



The mouse cursor when navigation mode is invoked.

You will know as soon as you are in Navigation mode as a crosshair cursor will appear over the mouse and your view of the board will change as you move the mouse. Using this together with the middle mouse wheel zoom will allow you to both ‘fly pass’ the board and to easily zoom in to closely examine a particular area of the board. Exiting navigation mode is as simple as right clicking the mouse.

For example, if we start in *Front View* (use the F9 keyboard shortcut) and we want to examine the resistors on the right we might proceed as follows:

1) Left click the mouse to enter navigation mode.
2) Move the mouse over the resistors.
3) Roll the middle mouse button to zoom in as required.
4) Right click the mouse to exit navigation mode.

The final necessary piece to completely customise the view is to allow users to spin or ‘orbit’ the board. This is done in navigation mode by holding down the left mouse button and moving the mouse. Essentially this will spin the board as you move the mouse – when you release the mouse button the camera will follow the mouse around the current view of the board as normal. Try this now, experimenting with different views of com[ponents on the board.

Remember that, if you are struggling to get the view you want you can use the keyboard shortcuts or navigation toolbar to return to one of the preset views. You should find however, that with only a little practise you become quite proficient at navigation.

To summarise:

- Left click enters navigation mode.
- Camera follows mouse around the board in navigation mode.
- Using the middle mouse wheel (or shortcut keys) allows you to zoom as you move the camera.
- Left depressing the mouse in navigation mode allows you to spin/orbit the entire board.
- Right click of the mouse exits navigation mode.

When you are finished close the 3D Viewer from the *File* Menu to return to ARES.

More information on 3D Visualization including creation your own 3D models, customizations and applying 3D data to legacy designs can be found in the 3D Viewer section in the online reference manual (Help Menu in ARES – Help Index command)..

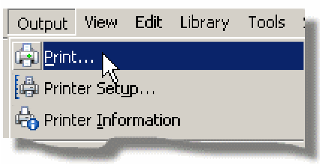
BOARD OUTPUT OPTIONS

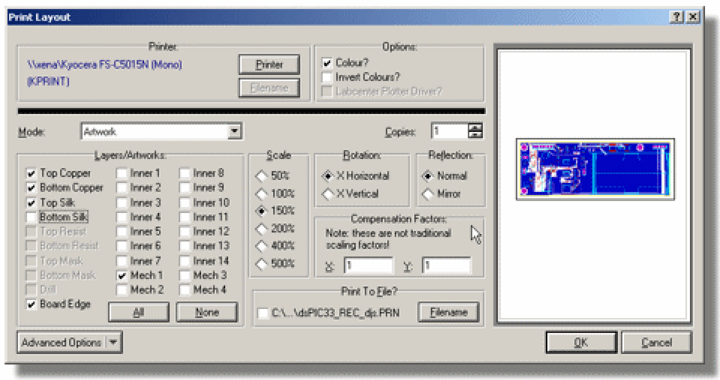
Last, but by no means least, we come to the crucial business of reproducing the pretty on-screen graphics on paper or film. Under Windows, most hard copy devices are supported through the normal Windows printer drivers. Additionally, we supply our own drivers for pen-plotters, Gerber photoplotters and Excellon NC drill machines.

Printing

We will deal here firstly with printing to an ordinary Windows printer device - it is unlikely that you will have a photoplotter to hand! The first step is to select the correct device to print to using the *Printer Setup* command on the *Output* menu. This activates the Windows common dialogue for printer device selection and configuration. The details are thus dependent on your particular version of Windows and your printer driver - consult Windows and printer driver documentation for details.

Then, with a layout loaded, invoke the *Print* command from the *Output* menu.





The dialogue forms offer a number of controls, all of which have context sensitive help associated with them (context sensitive help on dialogue forms is accessed with a ‘point and shoot’ mechanism via the ‘?’ key at the top right of the dialogue form). The default settings should do for getting something and you commence output generation by clicking on OK. Output can be aborted by pressing ESC, although there may be a short delay before everything stops whilst ARES and your printer/plotter empty their buffers.

With plotters in particular, you will probably need to experiment with pens, paper, and the various settings on the *Set Devices* dialogue form in order to get optimum results. Full details may be found under the chapter *Hard Copy Generation* in the reference manual.

ARES will remember your printer settings from the *Printer Setup* dialogue and maintain them independently of your printer settings for other applications. This means that you can configure a default set of printer options solely for use with the ARES application.

Output for Manufacture

ARES provides two main output options for board manufacture:

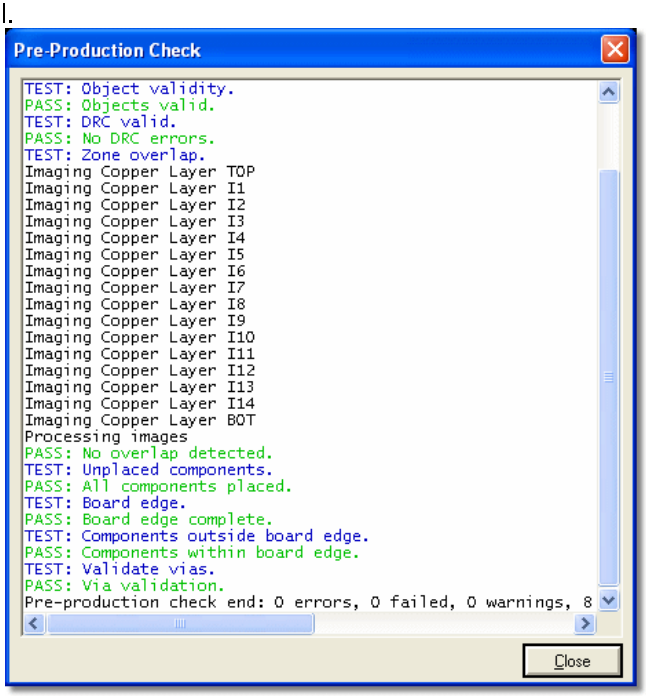
- Traditional Gerber/Excellon (available in all professional versions).
- ODB++ Manufacturing Output (available in Advanced Feature Set only).

From a user interface perspective both options are very similar but from a manufacturing perspective, the ODB++ option provides far more information than the older Gerber formats. Examples include:

- Inclusion of the connectivity information (the netlist) with the output fileset.
- Explicit support for plated/unplated specification on pads.
- Explicit support for fiducials.

All of this means that when viewing the output fileset from ODB++ the verification process is simpler and more complete. However, traditional Gerber/Excellon output is still prevalent in manufacture and would be sufficient for most purposes.

Regardless of your option the basic procedure is the same. When you invoke either of the output options you will most likely be prompted to run a pre-production check. This runs an automatic checklist to test for some common design errors and will report either a pass or a fail.

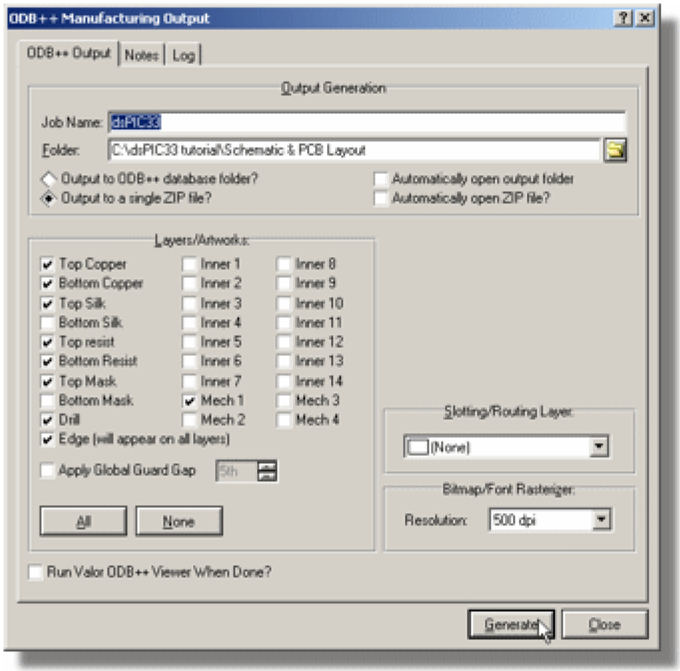


Pre-preduction checker

If your pre-production check reports errors we strongly recommend that you resolve them before proceeding to the manufacturin[g output dialogue. Please also note that pre-production check is an aid to the designer in quality assurance but not a guarantee; manual inspection of the layout is recommended and a prototype should always be made and tested before mass production.

More information on the Pre-production Check can be found in the CADCAM Output section of the online reference manual (Help Menu).

Assuming a pass you will then be provided with the manufacturing output dialogue form

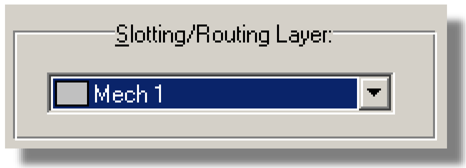


The options in the top section are self-explanatory but there are some points of note regarding the rest of the configuration options.

The software attempts to populate the layer set with those layers used on the layout but this should always be verified. One of the most common problems with manufacture occurs when the manufacturer is not supplied with the full board information.

The *Apply Global Guard Gap* option will set the expansion of the resist plot around pads and vias to the distance specified when this option is checked. This will take effect on **all** pads and vias **except** those which have been manually altered on the layout. Some board manufacturers prefer to create the resist plot in house and this option could then be used to remove any resist expansion pre-manufacture. Unless otherwise directed we recommend that you leave this option unchecked.

The *Slotting /Routing* Layer option specifies explicitly which layer of the board is to be used for defining the routing strokes for cutouts and slots. In our case, we have used `MECH1` and we must therefore set this via the selector.



The *Bitmap/Font Rasterizer* option controls the thickness of trace used to render bitmaps and, more importantly, power planes. The higher the resolution the better tonality of the resulting bitmaps but the larger the files. It is also possible that some manufacturers have a minimum width requirement and in such cases it may be necessary to reduce the DPI settings to conform with this. Generally speaking however, the default settings are fine.

The option at the bottom left allows you to automatically load your output into either the Labcenter Gerber Viewer (CADCAM Output) or the Valor ODB++ Viewer (ODB++ Output). This is useful if you want to verify the output fileset before passing to your manufacturing house.

The ODB++ Viewer option will be disabled unless you have downloaded and installed the free Valor viewer from the Valor website.

If you intend to panellise the board you should use the CADCAM Output option, run the viewer when done and select *Panelization Mode* from the resulting dialogue form. See the reference manual for more information.

Finally, the *Notes* tab allows us to insert any relevant information or special considerations for the manufacturer. This is very important when we have specified a slotting layer as there is no standard way to pass this information through. A simple note that the information on MECH1 is for slotting will suffice.

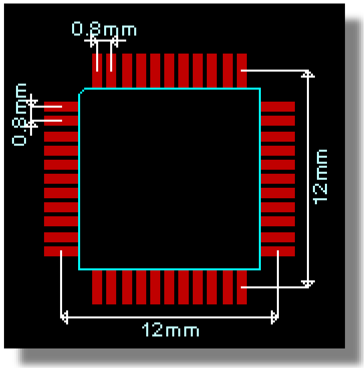
Having configured the necessary option we can generate the files and send for manufacture.

APPENDIX: CREATING NEW PACKAGES

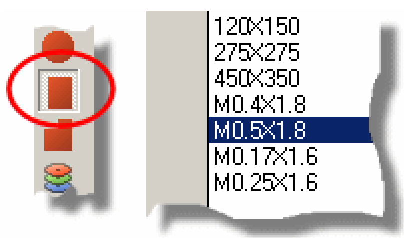
ARES comes pre-supplied with a large quantity of footprints, and we have seen previously how to select and place these parts on to a layout. However, it may be necessary at times to create your own custom footprints or symbols – also a simple task with ARES – and this process is detailed below.

Drawing the Footprint

As an example, we will create a `SQFP44` footprint with `0.8mm` pitch and `12mm` width.

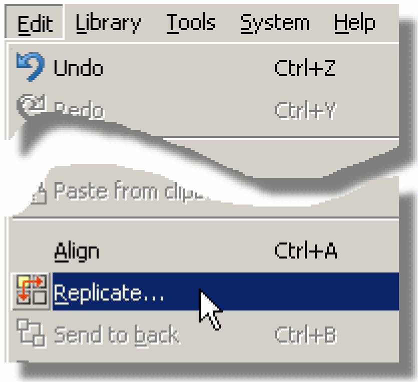


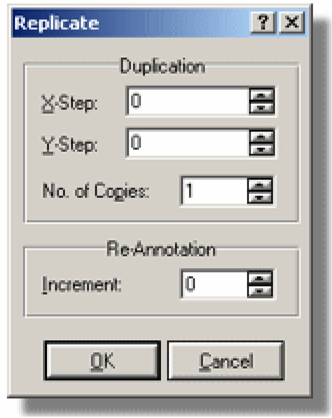
Start by selecting rectangular SMT mode. We want a pad `0.5mm` by `1.8mm` that should already exist as `M0.5x1.8.`



If the pad style does not exist you can easily create it as discussed earlier in the tutorial (Click on the `'C'` button above the Object Selector).

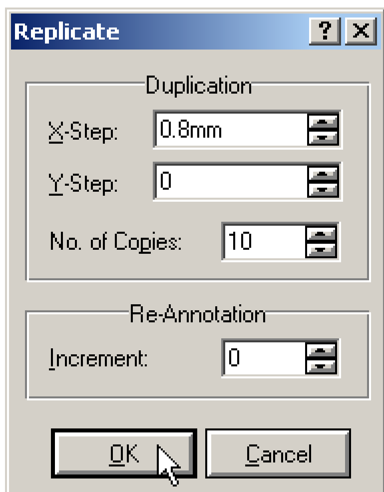
Make sure that the *Layer Selector* is on Top Copper and place one of the pads in the usual way. Right click to cancel placement immediately after placing a single pad and then left click on the pad to highlight it. With the pad highlighted, go to the *Edit Menu* and select the replicate command. Alternatively you can right click on the pad and select replicate from the resulting context menu.





The replicate command will action on tagged objects so you should ensure that you have only the placed pad highlighted before invoking this command.

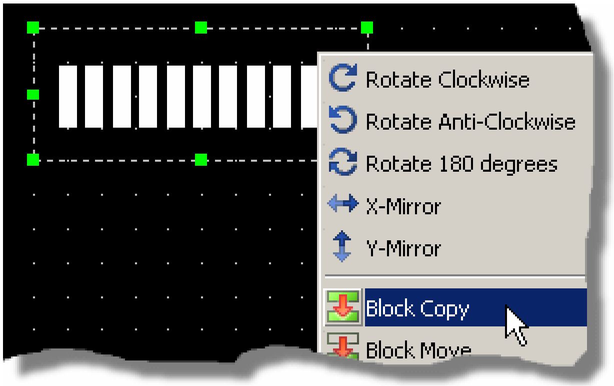
There are 11 pads on each side of the footprint so we need to replicate `10` times with an X-Step of `0.8mm` as per the following screenshot.



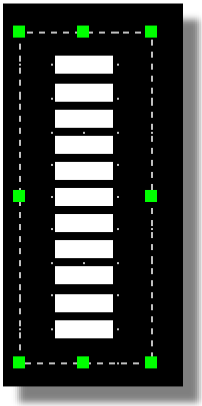
This will give us a single row of pads of the correct pitch (zoom in, change the snap settings and measure if you would like to confirm). The next step is to duplicate this row of pads to form the bottom of the footprint.

Start by right dragging a tagbox around the entire row of pads and then right click inside the selection box and launch the replicate dialogue form from the context menu. This time we want only one copy, 12mm down (or up) from the current row. Use negative co-ordinates if you want the duplicated row beneath the current one or positive co-ordinates if you want it above.

We now need to repeat the process with the other two rows. Drag a tagbox around a full row, right click inside the tagbox and select *Block Copy* from the resulting context menu,

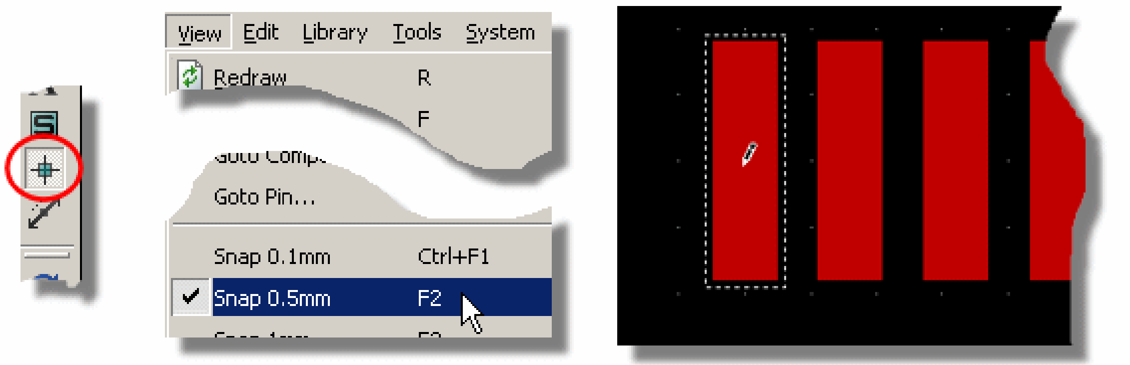


Move the mouse and drag the copy to a free area, left clicking to place and then right clicking to exit copy mode. Now, drag a tagbox around the row, right click and select the *Block Rotate* option, specifying a 90 degree rotation to align the pads correctly.



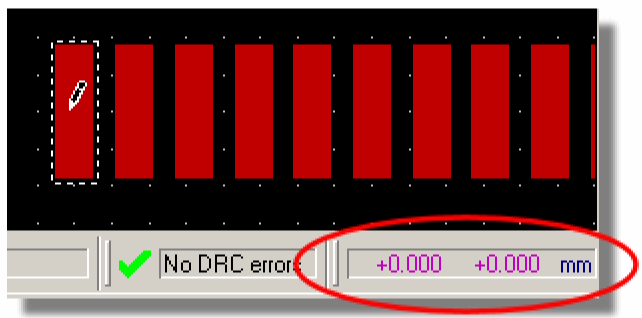
In order to position this row of pads correctly it's best to first set place a marker on the location we want to move the row of pads to. For this footprint the center of the top pad on the left hand side is 2mm below and 2mm to the left of the center of the left hand most pad on the top row. This gives us enough information to accurately position the row.

Select marker mode in preparation and then move the mouse over the left hand most pad on the top row until it is encircled. You'll want to be on a fairly precise snap setting for this (e.g 0.5mm or F2 shortcut key).

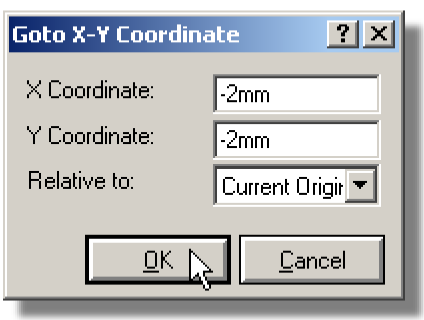


If you find your snap settings in thou you need to switch to Metric mode, either by hitting the 'M' key on the keyboard or via the Metric option on the View Menu,

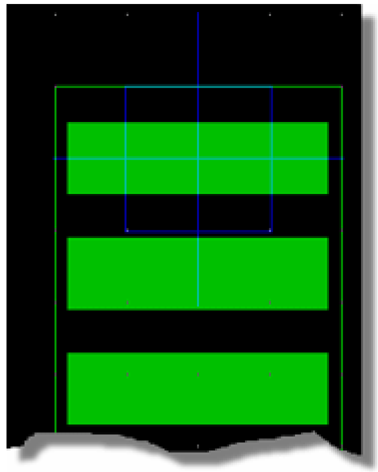
Now, hit the '`0`' key on the keyboard to set a false origin at this location and then invoke the Goto-XY command from the *Tools Menu.*



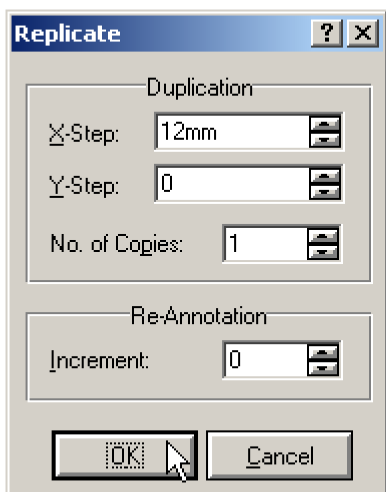
Type in `-2mm` in both the X and the Y co-ordinate fields (i.e. down and left) and make sure that the offset is relative to the current origin. Then exit the dialogue form and click the mouse twice to place the marker at this location.



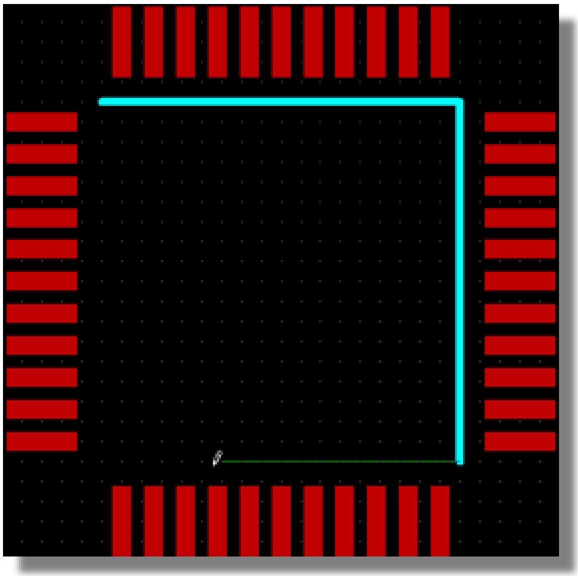
The marker we have placed will serve for now as the location we want to center the top most pad of our third row. Simply draw a tagbox around the row, left depress the mouse inside the selection box and drag into position. You will almost certainly need to use the zoom commands (`F6,F7` or the middle mouse wheel) and possibly also adjust the snap setting (`CTRL+F1`) to get accurate positioning.



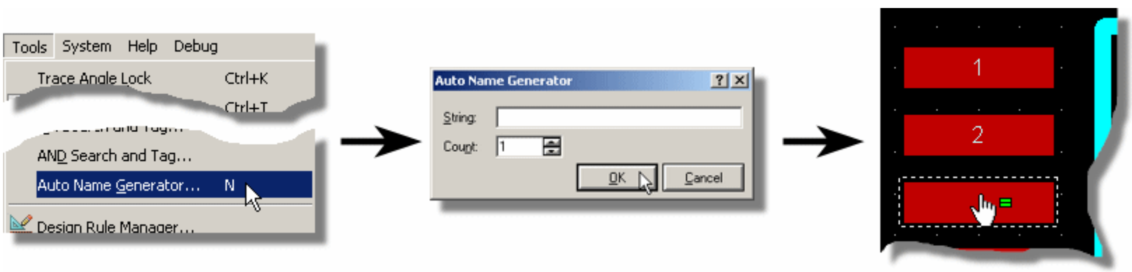
Having placed the row of pads right click on the marker and delete it – we'll come back to marker placement later on. The final stage of pad placement is to replicate the newly placed row of pads onto the right hand side. As before, draw a selection box around the row of pads and use the replicate command with an [X offset of `12mm` to duplicate the row.



Adding the silkscreen graphics is now straightforward. Select the 2D Graphics Line icon, make sure the *Layer Selector* is on top silk and place four lines along the inside edges of the pads to form a box. You'll find this much easier to do if you change the snap settings upwards (e.g F2).

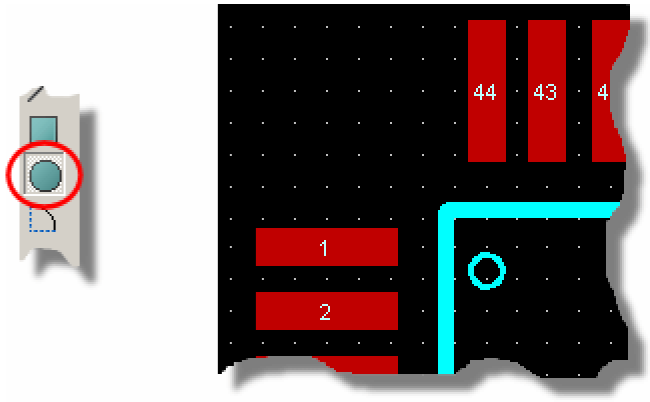


The next job we have is to number the pads. Start by invoking the *Auto Name Generator* from the *Tools Menu*. We don't need to enter anything in the string field here; simply leave the defaults and click on the pads consecutively to number them, Number 1 is the top pad on the left hand row.



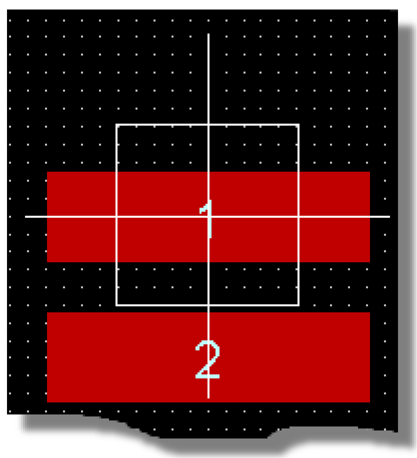
Remember to hit the escape key on your keyboard when you have finished numbering pin 44 to exit assignment mode.

It's common to place a small dot beside pin one and you can do this via the 2D Graphics circle icon. Make sure that the Layer Selector is on Top Silk and turn the snap setting down to minimum (CTRL+F1) to achieve finer control over the size.

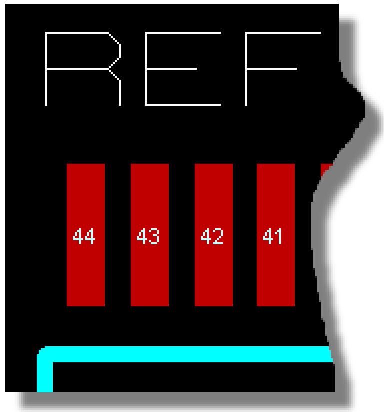


The final step is to specify where we want the origin of the device to be (for placement purposes) and where we want the silkscreen graphics for the package reference to appear. Both of these are done with markers.

Select *Marker mode* and make sure that `ORIGIN` marker is highlighted in the Object Selector. You'll recall that we used this as a reference point earlier but it's real job is to specify the origin of the component for placement and rotation. This is really a decision for the user but typically the origin is either pin 1 or the center of the component. For simplicity here, we will place the origin on pin 1; left click to begin placement, move the mouse over the center of pin 1 and left click again to commit the origin marker.



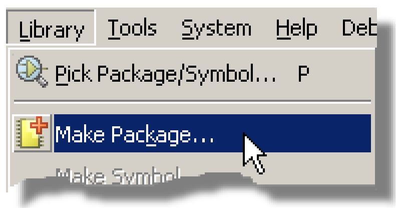
Now, change the marker type to be REFERENCE in the Object Selector. The Reference marker dictates where the component reference (e.g `U1, R12, C3`) is placed by the software relative to the component. Again, this is very subjective but we'll set it above and slightly to the left of the part.



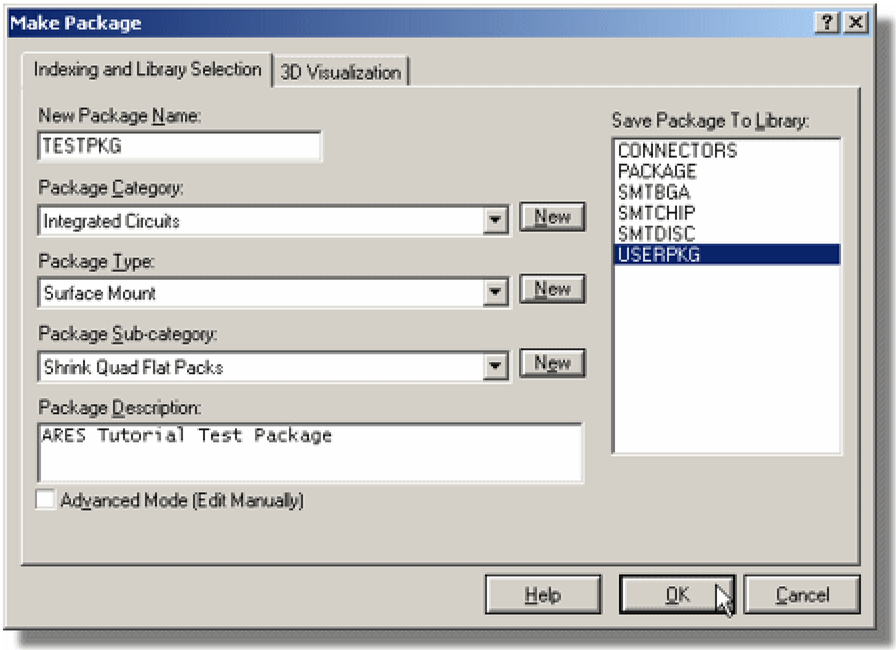
Having completed the layout of the footprint, we can now move on to the (far simpler) process of packaging the part into the libraries.

Packaging the Footprint

Drag a selection box around the entire footprint and then select the *Make Package* command from the *Library Menu*.



The first screen is fairly self-explanatory and similar to that we've seen in the ISIS application. Note that the package description is searchable when we are browsing for footprints so a little effort to make this as descriptive as possible is worthwhile. You will also want to create the part in the `USERPKG` library which is the default library supplied for user footprints. For our purposes, we'll call this package TESTPKG and give it some basic entries.

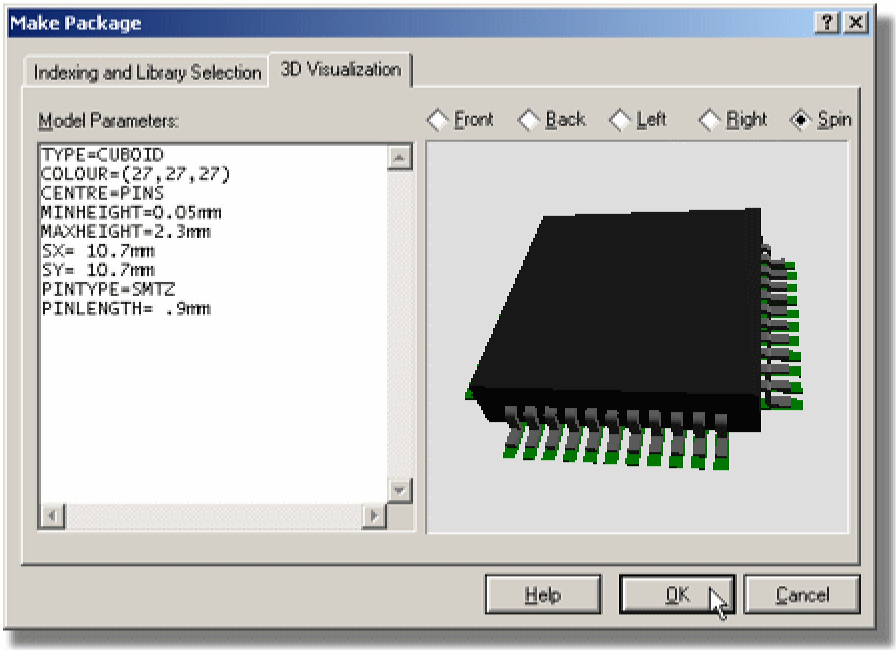


You can create your own libraries via the *Library Manager* – please see the reference manual for details. You should **not** make your own parts into the other pre-supplied libraries as Labcenter may overwrite these libraries during upgrade installs.

When this is filled out, switch tabs to the 3D Visualisation tab (do not hit the OK button at this stage – we still have some work to do). Essentially what we need to do here is provide as much information as possible in order to get a sensible 3D image of the part which can then be used when we use the 3D Viewer to examine a board. This job is greatly aided by a 3D Preview on the dialogue form that will update live as we adjust parameters. Discussion of parameters and values is beyond the scope of this tutorial and is discussed in some depth in the online reference manual (Help Menu in ARES – Help Index). For our purposes, simply fill out the property fields as shown in the following screenshot.

3D Visualisation

When this is filled out, switch tabs to the 3D Visualisation tab. Essentially what we need to do here is provide as much information as possible in order to get a sensible 3D image of the part which can then be used when we use the 3D Viewer to examine a board. This job is greatly aided by a 3D Preview on the dialogue form that will update live as we adjust parameters. Discussion of parameters and values is beyond the scope of this tutorial and is discussed in some depth in the online reference manual (Help Menu in ARES – Help Index). For our purposes, simply fill out the property fields as shown in the following screenshot.



When you are finished click the OK button to commit the changes to the library.

If you now select the *Package Icon*, you will see that TESTPKG has appeared in the Object Selector, and can be placed like any of the packages you have used so far. Also, if you place the part on the layout and invoke the 3D Visualisation engine from the *Output* Menu you will see the 3D rendered image of the part.