



***Intelligent Schematic
Input System***

User Manual

Issue 6.3 - November 2003

© Labcenter Electronics

ICON REFERENCE CHART

File And Printing Commands



Display commands



Main Mode Icons



Gadget Icons



2D Graphics



Design Tools


Real Time Snap


Wire Autorouter


Search & Tag


Property Assignment Tool


New Sheet


Delete Sheet


Goto Sheet


Zoom to Child


Return to Parent


Bill of Material


Electrical Rules Check


Netlist to Ares

Editing Commands

These affect all currently tagged objects.


Block Copy


Block Move


Block Delete


Block Rotate


Pick Device/Symbol


Make Device Decompose


Package Tool


Undo


Redo


Cut


Copy


Paste

Rotate And Mirror Icons


Rotate Clockwise


Rotate Anti-clockwise


Flip X axis


Flip Y axis

TABLE OF CONTENTS

INTRODUCTION	1
ABOUT ISIS.....	1
ISIS AND PCB DESIGN.....	2
ISIS AND SIMULATION.....	2
ISIS AND NETWORKS.....	4
HOW TO USE THIS DOCUMENTATION.....	4
TUTORIAL	5
INTRODUCTION	5
A GUIDED TOUR OF THE ISIS EDITOR.....	5
PICKING, PLACING AND WIRING UP COMPONENTS	7
LABELLING AND MOVING PART REFERENCES	9
BLOCK EDITING FUNCTIONS	11
PRACTICE MAKES PERFECT	11
ANNOTATING THE DIAGRAM	12
The Property Assignment Tool (PAT).....	12
The Automatic Annotator.....	13
CREATING NEW DEVICES	14
FINISHING TOUCHES	16
SAVING, PRINTING AND PLOTTING.....	17
MORE ABOUT CREATING DEVICES.....	17
Making a Multi-Element Device.....	17
The Visual Packaging Tool.....	19
Making Similar Devices.....	21
Replacing Components in a design.....	21
SYMBOLS AND THE SYMBOL LIBRARY	21
REPORT GENERATION.....	22
A LARGER DESIGN.....	22
GENERAL CONCEPTS	25
SCREEN LAYOUT	25
The Menu Bar.....	25
The Toolbars.....	25
The Editing Window	26
The Overview Window.....	27
The Object Selector.....	28

Co-ordinate Display	28
CO-ORDINATE SYSTEM	28
False Origin	30
The Dot Grid.....	30
Snapping to a Grid	30
Real Time Snap	30
FILING COMMANDS	31
Starting a New Design	31
Loading a Design	32
Saving the Design	32
Back-up and Last Loaded Files	32
Import / Export Section.....	33
Quitting ISIS.....	33
GENERAL EDITING FACILITIES	33
Object Placement.....	33
Tagging an Object	34
Deleting an Object	34
Dragging an Object	34
Dragging an Object Label.....	34
Resizing an Object.....	35
Reorienting an Object	35
Editing an Object	36
Editing an Object Label	36
Copying all Tagged Objects	37
Moving all Tagged Objects.....	37
Deleting all Tagged Objects.....	38
WIRING UP.....	38
Wire Placement.....	38
The Wire Auto-Router	38
Wire Repeat.....	39
Dragging Wires	39
THE AUTOMATIC ANNOTATOR.....	41
Value Annotation	42
MISCELLANEOUS	42
The Sheet Border.....	42
The Header Block.....	43
Send to Back / Bring to Front.....	46
Keyboard Configuration.....	46
GRAPHICS AND TEXT STYLES	49
INTRODUCTION	49
TUTORIAL.....	50

Editing Global Styles.....	50
Editing Local Styles.....	52
Working With The Template.....	53
TEMPLATES AND THE TEMPLATE MENU.....	54
PROPERTIES	55
INTRODUCTION	55
OBJECT PROPERTIES.....	55
System Properties	55
User Properties	56
Property Definitions (PROPDEFS)	56
SHEET PROPERTIES	57
Introduction.....	57
Defining Sheet Properties.....	57
Scope Rules for Sheet Properties	58
DESIGN PROPERTIES	60
PARAMETERIZED CIRCUITS.....	60
Introduction.....	60
An Example.....	61
Property Substitution	62
Property Expression Evaluation	64
The Rounding Functions E12 (), E24 ().....	65
THE PROPERTY ASSIGNMENT TOOL.....	65
The PAT Dialogue Form.....	66
PAT Actions.....	66
PAT Application Modes	68
The Search and Tag Commands	68
Examples	69
PROPERTY DEFINITIONS.....	70
Creating Property Definitions.....	70
Default Property Definitions.....	71
Old Designs.....	71
OBJECT SPECIFICS.....	73
COMPONENTS	73
Selecting Components from the Device Libraries	73
Placing Components.....	74
Replacing Components	75
Editing Components.....	75
Component Properties	76
Hidden Power Pins.....	76
DOTS.....	76
Placing Dots	77

Auto Dot Placement	77
Auto Dot Removal.....	77
WIRE LABELS.....	77
Placing And Editing Wire Labels	77
Deleting Wire Labels	78
Using a Wire Label to Assign a Net Name.....	79
Using a Wire Label to Assign a Net Property.....	79
Wire Label Properties.....	79
SCRIPTS	81
Placing and Editing Scripts.....	81
Script Block Types	82
Part Property Assignments (*FIELD).....	82
Sheet Global Net Property Assignments (*NETPROP).....	83
Sheet Property Definitions (*DEFINE)	83
Parameter Mapping Tables (*MAP ON varname).....	84
Model Definition Tables (*MODELS).....	84
Named Scripts (*SCRIPT scripttype scriptname).....	85
SPICE Model Script (*SPICE)	86
BUSES	86
Placing Buses.....	86
Bus Labels	87
Wire/Bus Junctions	88
Bus Properties	89
SUB-CIRCUITS.....	89
Placing Sub-Circuits	89
Editing Sub-Circuits	91
Sub-Circuit Properties	91
TERMINALS.....	91
Logical Terminals	91
Physical Terminals	92
Placing Terminals	92
Editing Terminals	93
Terminal Properties.....	94
PIN OBJECTS.....	94
Placing Pin Objects.....	94
Editing Pin Objects.....	95
Pin Object Properties	95
SIMULATOR GADGETS.....	96
2D GRAPHICS	96
Placing 2D Graphics	96
Resizing 2D Graphics	98
Editing 2D Graphics.....	99

MARKERS	99
Marker Types	99
Placing Markers.....	100
LIBRARY FACILITIES	101
GENERAL POINTS ABOUT LIBRARIES.....	101
Library Discipline	101
The Pick Command.....	101
SYMBOL LIBRARIES.....	102
Graphics Symbols	102
User Defined Terminals	103
User Defined Module Ports.....	104
User Defined Device Pins	106
Editing an Existing Symbol.....	107
Hierarchical Symbol Definitions	107
DEVICE LIBRARIES	107
Making a Device Element.....	108
The Make Device Command.....	111
The Visual Packaging Tool.....	116
Making a Single Element Device	119
Making a Multi-Element Homogenous Device.....	120
Making a Multi-Element Heterogeneous Device	121
Making a Device with Bus Pins.....	122
Property Definitions and Default Properties.....	123
Dealing with Power Pins.....	124
Editing an Existing Device.....	126
MULTI-SHEET DESIGNS	129
MULTI-SHEET FLAT DESIGNS.....	129
Introduction.....	129
Design Menu Commands	129
HIERARCHICAL DESIGNS.....	129
Introduction.....	129
Terminology	130
Sub-Circuits.....	131
Module-Components.....	132
External Modules	133
Moving About a Hierarchical Design	133
Design Global Annotation.....	134
Non-Physical Sheets	134
NETLIST GENERATION	135
INTRODUCTION	135

NET NAMES.....	135
DUPLICATE PIN NAMES	136
HIDDEN POWER PINS	136
SPECIAL NET NAME SYNTAXES.....	137
Global Nets	137
Inter-Element Connections for Multi-Element Parts	138
BUS CONNECTIVITY RULES	139
The Base Alignment Rule.....	139
Using Bus Labels to Change the Connectivity Rule	139
Using Bus Terminals to Interconnect Buses	140
Connections to Individual Bits.....	141
Tapping a Large Bus	143
General Comment & Warning	143
GENERATING A NETLIST FILE	144
Format	144
Logical/Physical/Transfer	144
Scope.....	144
Depth.....	144
Errors.....	145
NETLIST FORMATS.....	145
SDF	145
BOARDMAKER	145
EEDESIGNER	145
FUTURENET	145
MULTIWIRE	145
RACAL.....	146
SPICE.....	146
SPICE-AGE FOR DOS	146
TANGO	146
VALID	146
VUTRAX.....	146
REPORT GENERATION	147
BILL OF MATERIALS	147
Generating the Report	147
Report Formats	147
Bill Of Materials Configuration	148
Adding Bill of Materials Parts	154
ASCII DATA IMPORT	155
The IF...END Command.....	156
The DATA...END Command.....	157
ELECTRICAL RULES CHECK.....	159

Generating the Report	159
ERC Error Messages	159
HARD COPY GENERATION	163
PRINTER OUTPUT	163
Specifying a Printer Device	163
Print Preview	163
Configuring Margins	163
Scaling	164
PLOTTER OUTPUT	164
Plotter Pen Colours	165
CLIPBOARD AND GRAPHICS FILE GENERATION	165
Bitmap Generation	165
Metafile Generation	165
DXF File Generation	165
EPS File Generation	166
ISIS AND ARES	167
INTRODUCTION	167
PACKAGING	167
Default Packaging	167
Manual Packaging	168
Automatic Packaging	168
Using the Bill of Materials to Help with Packaging	169
The Package Verifier	170
Packaging with ARES	170
NET PROPERTIES AND ROUTING STRATEGIES	170
FORWARD ANNOTATION - ENGINEERING CHANGES	171
Adding New Components	171
Removing Existing Components	172
Changing the connectivity	172
Re-Annotating Components, and Re-Packaging Gates	173
PIN-SWAP/GATE-SWAP	173
Specifying Pin-Swaps and Gate-Swaps for ISIS Library Parts	174
Specifying Pin-Swaps in Single Element Devices	174
Specifying Pin-Swaps in Multi-Element Devices	175
Specifying Gate-Swaps in Multi-Element Devices	175
Performing Manual Pin-Swaps and Gate-Swaps in ARES	176
The Gate-Swap Optimizer	177
RE-ANNOTATION	179
BACK-ANNOTATION WITH ISIS	180
Semi-Automatic Back-Annotation	180
Fully-Automatic Back-Annotation	180

INDEX181

INTRODUCTION

ABOUT ISIS

Many CAD users dismiss schematic capture as a necessary evil in the process of creating PCB layout but we have always disputed this point of view. With PCB layout now offering automation of both component placement and track routing, getting the design into the computer can often be the most time consuming element of the exercise. And if you use circuit simulation to develop your ideas, you are going to spend even more time working on the schematic.

ISIS has been created with this in mind. It has evolved over twelve years research and development and has been proven by thousands of users worldwide. The strength of its architecture has allowed us to integrate first conventional graph based simulation and now – with PROTEUS VSM – interactive circuit simulation into the design environment. For the first time ever it is possible to draw a complete circuit for a micro-controller based system and then test it interactively, all from within the same piece of software. Meanwhile, ISIS retains a host of features aimed at the PCB designer, so that the same design can be exported for production with ARES or other PCB layout software.

For the educational user and engineering author, ISIS also excels at producing attractive schematics like you see in the magazines. It provides total control of drawing appearance in terms of line widths, fill styles, colours and fonts. In addition, a system of templates allows you to define a 'house style' and to copy the appearance of one drawing to another.

Other general features include:

- Runs on Windows 98/Me/2k/XP and later.
- Automatic wire routing and dot placement/removal.
- Powerful tools for selecting objects and assigning their properties.
- Full support for buses including component pins, inter-sheet terminals, module ports and wires.
- Bill of Materials and Electrical Rules Check reports.
- Netlist outputs to suit all popular PCB layout tools.

For the 'power user', ISIS incorporates a number of features which aid in the management of large designs. Indeed, a number of our customers have used it to produce designs containing many thousands of components.

- Hierarchical design with support for parameterized component values on sub-circuits.
- Design Global Annotation allowing multiple instances of a sub-circuit to have different component references.
- Automatic Annotation - the ability to number the components automatically.
- ASCII Data Import - .this facility provides the means to automatically bring component stock codes and costs into ISIS design or library files where they can then be incorporated or even totalled up in the Bill of Materials report.

ISIS AND PCB DESIGN

Users of ARES, or indeed other PCB software will find some of the following PCB design specific features of interest:

- Sheet Global Net Properties which allow you to efficiently define a routing strategy for all the nets on a given sheet (e.g. a power supply needing POWER width tracks).
- Physical terminals which provide the means to have the pins on a connector scattered all over a design.
- Support for heterogeneous multi-element devices. For example, a relay device can have three elements called RELAY:A, RELAY:B and RELAY:C. RELAY:A is the coil whilst elements B and C are separate contacts. Each element can be placed individually wherever on the design is most convenient.
- Support for pin-swap and gate-swap. This includes both the ability to specify legal swaps in the ISIS library parts and the ability to back-annotate changes into a schematic.
- A visual packaging tool which shows the PCB footprint and its pin numbers alongside the list of pin names for the schematic part. This facilitates easy and error free assignment of pin numbers to pin names. In addition, multiple packagings may be created for a single schematic part.

A full chapter is provided on how to use ISIS and ARES together.

ISIS AND SIMULATION

ISIS provides the development environment for PROTEUS VSM, our revolutionary interactive system level simulator. This product combines mixed mode circuit simulation, micro-processor

models and interactive component models to allow the simulation of complete micro-controller based designs.

ISIS provides the means to enter the design in the first place, the architecture for real time interactive simulation and a system for managing the source and object code associated with each project. In addition, a number of graph objects can be placed on the schematic to enable conventional time, frequency and swept variable simulation to be performed.

Major features of PROTEUS VSM include:

- True Mixed Mode simulation based on Berkeley SPICE3F5 with extensions for digital simulation and true mixed mode operation.
- Support for both interactive and graph based simulation.
- CPU Models available for popular microcontrollers such as the PIC and 8051 series.
- Interactive peripheral models include LED and LCD displays, a universal matrix keypad, an RS232 terminal and a whole library of switches, pots, lamps, LEDs etc.
- Virtual Instruments include voltmeters, ammeters, a dual beam oscilloscope and a 24 channel logic analyser.
- On-screen graphing - the graphs are placed directly on the schematic just like any other object. Graphs can be maximised to a full screen mode for cursor based measurement and so forth.
- Graph Based Analysis types include transient, frequency, noise, distortion, AC and DC sweeps and fourier transform. An Audio graph allows playback of simulated waveforms.
- Direct support for analogue component models in SPICE format.
- Open architecture for 'plug in' component models coded in C++ or other languages. These can be electrical., graphical or a combination of the two.
- Digital simulator includes a BASIC-like programming language for modelling and test vector generation.
- A design created for simulation can also be used to generate a netlist for creating a PCB - there is no need to enter the design a second time.

Full details of all these features and much more are provided in the PROTEUS VSM manual.

ISIS AND NETWORKS

ISIS is fully network compatible, and offers the following features to help Network Managers:

- Library files can be set to Read Only. This prevents users from messing with symbols or devices that may be used by others.
- ISIS individual user configuration in the windows registry. Since the registry determines the location of library files, it follows that users can have individual USERDVC.LIB files in their personal or group directories.

HOW TO USE THIS DOCUMENTATION

This manual is intended to complement the information provided in the on-line help. Whereas the manual contains background information and tutorials, the help provides context sensitive information related to specific icons, commands and dialog forms. Help on most objects in the user interface can be obtained by pointing with the mouse and pressing F1.

ISIS is a vast and tremendously powerful piece of software and it is unreasonable to expect to master all of it at once. However, the basics of how to enter a straightforward circuit diagram and create your own components *are* extremely simple and the techniques required for these tasks can be mastered most quickly by following the tutorial given in *Tutorial* on page 5. We strongly recommend that you work through this as it will save you time in the long run.

With some of the more advanced aspects of the package, you are probably going to find some of the *concepts* are new, let alone the details of how ISIS handles them. Each area of the software has been given a chapter of its own, and we generally start by explaining the background theory before going into the operation and use of the relevant features. You will thus find it well worthwhile reading the introductory sections rather than jumping straight to the how-to bits.

INTRODUCTION

The aim of this tutorial is to take you through the process of entering a circuit of modest complexity in order to familiarise you with the techniques required to drive ISIS. The tutorial starts with the easiest topics such as placing and wiring up components, and then moves on to make use of the more sophisticated editing facilities, such as creating new library parts.

For those who want to see something quickly, ISISUT.DSN contains the completed tutorial circuit. This and other sample designs are installed to the SAMPLES directory.

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 the shortcuts mentioned may not be valid. Information on configuring your own keyboard shortcuts can be found in the section on Keyboard Configuration on page 46.

A GUIDED TOUR OF THE ISIS 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.

To start the ISIS program, click on the *Start* button and select *Programs, Proteus 6 Professional*, and then the *ISIS 6 Professional* option. The ISIS schematic editor will then load and run. Along the top of the screen is the Menu Bar.

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 wire-up components. The smaller area at the top right 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.

You can adjust the area of the drawing displayed in the *Editing Window* in a number of ways:

- To simply 'pan' the *Editing Window* up, down, left or right, position the mouse pointer over the desired part of the *Editing Window* and press the F5 key.
- Hold the SHIFT key down and bump the mouse against the edges of the *Editing Window* to pan up, down, left or right. We call this *Shift Pan*.

- Should you want to move the *Editing Window* to a completely different part of the drawing, the quickest method is to simply point at the centre of the new area on the *Overview Window* and click left.
- You can also use the *Pan* icon on the toolbar.

To adjust the scale the drawing is displayed at in the *Editing Window* you can:

- Point with the mouse where you want to zoom in or out and press the F6 or F7 keys respectively or else scroll the mouse wheel.
- Press the F8 key to display the whole drawing.
- Hold the SHIFT key down and drag out a box around the area you want to zoom in to. We call this *Shift Zoom*.
- Use the *Zoom In*, *Zoom Out*, *Zoom Full* or *Zoom Area* icons on the toolbar.

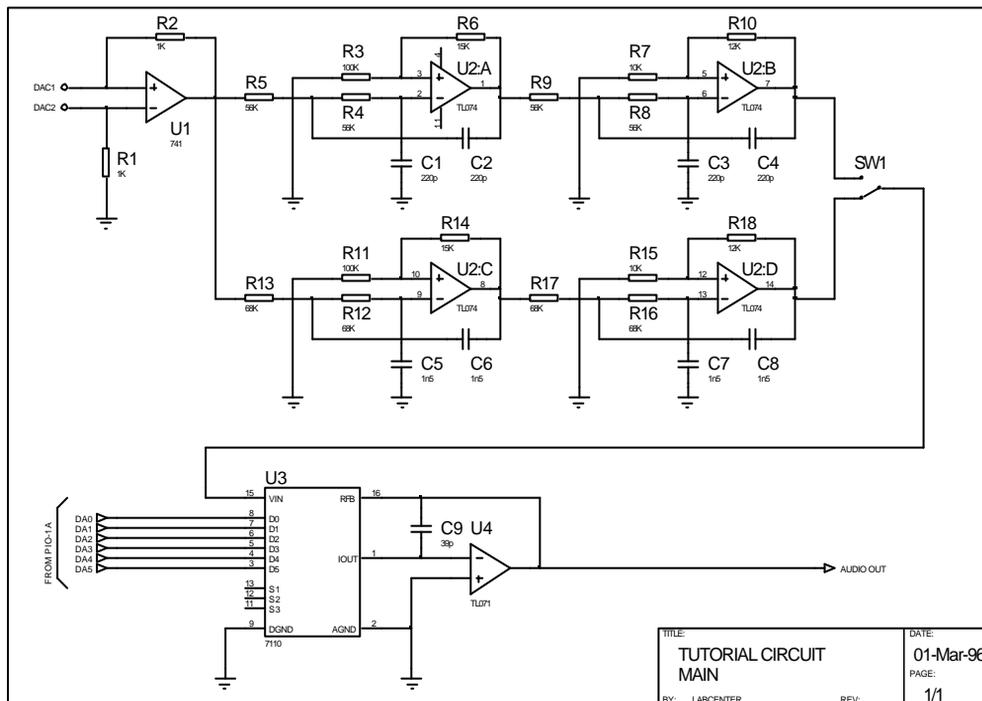
A grid of dots can be displayed in the *Editing Window* using the *Grid* command on the *View* menu, or by pressing 'G', or by clicking the *Grid* icon on the toolbar. The grid helps in lining up components and wires and is less intimidating than a blank screen. If you find it hard to see the grid dots, either adjust the contrast on your monitor slightly (by default the grid dots are displayed in light grey) or change their colour with the *Set Design Defaults* on the *Template* menu.

Below the *Overview Window* is the *Object Selector* which you use to select devices, symbols and other library objects.

At the bottom right of the screen is the co-ordinate display, which reads out the co-ordinates of the mouse pointer when appropriate. These co-ordinates are in 1 thou units and the origin is in the centre of the drawing.

PICKING, PLACING AND WIRING UP COMPONENTS

The circuit we are going to draw is shown below. It may look quite a lot to do, but some parts of it are similar (the four op-amp filters to be precise) which will provide opportunity to use the block copying facilities.



The Tutorial Circuit

We shall start with the 741 buffer amplifier comprising U1, R1, R2. Begin by pointing at the **P** button at the top left of the *Object Selector* and clicking left. This causes the *Device Library Selector* dialogue form to appear and you can now select devices from the various device libraries. There are a number of selectors labelled *Objects*, *Libraries*, *Extensions* and a *Browser*, not all of which may be shown:

- The *Library* selector chooses which of the various device libraries (e.g. DEVICE, TTL, CMOS) you have installed is current.

- The Objects selector displays all the parts in the currently selected library according to the settings in the *Extensions* selector, if it is shown (see below). Click left once on a part to browse it or double-click a part to 'pick' it in to the design.
- The *Browser* displays the last selected part in the *Parts* selector as a means of browsing the contents of the library.

We need two devices initially - OPAMP for the 741 op-amp and RES for the feedback resistors. Both these are in the DEVICE library, so, if it is not already selected, start by selecting the DEVICE library in the *Library* selector. Then, double-click on the OPAMP and RES parts from the *Parts* selector to select each part. The devices you have picked should appear in the *Object Selector*.

Whenever you select a device in the *Devices* selector or use the *Rotation* or *Mirror* icons to orient the device prior to placement, the selected device is shown previewed in the *Overview Window* with the orientation the selected device will have if placed. As you click left or right on the *Rotation* and/or *Mirror* icons, the device is redrawn to preview the new orientation. The preview remains until the device is placed or until the another command or action is performed.

Ensure the OPAMP device is selected and then move the mouse pointer into the middle of the *Editing Window*. Press and hold down the left mouse button. An outline of the op-amp will appear which you can move around by dragging the mouse. When you release the button, the component will be *placed* and is drawn in full. Place the op-amp in the middle of the *Editing Window*. Now select the RES device and place one resistor just above the op-amp as in the diagram and click left once on the anti-clockwise *Rotation* icon; the preview of the resistor shows it rotated through 90°. Finally, place the second (vertical) resistor, R1 as before.

Unless you are fairly skilful, you are unlikely to have got the components oriented and positioned entirely to your satisfaction at this first attempt, so we will now look at how to move things around. In ISIS, objects are selected for further editing by 'tagging' them. Try pointing at the op-amp and clicking right. This tags the object, causing it to be highlighted. Now, still keeping the pointer over it, hold the left button down and drag it around. This is one of the ways to move objects. Release the left button, and click right on the op-amp a second time. Clicking right on a tagged object deletes it. Select *Undo* on the *Edit Menu* (or press 'U') to recover it. Tag it again, and click first left and then right on the *Rotation* icon whilst watching the op-amp itself. The rotation of the last object you tagged can be adjusted in this way; the *Mirror* icon can similarly be used to reflect the last object tagged. Armed with the above knowledge, you should now be able to adjust the three components you have placed such that they match the diagram. When you have finished editing, point at a free space in the *Editing Window* (i.e. somewhere where there is no object) and click right to untag all tagged objects.

We can now move on to place some wires. Start by pointing at the tip of the upper end of R1 and clicking left. ISIS senses that you are pointing at a component pin and deduces that you wish to connect a wire from it. To signify this, it displays a green line which goes from the pin to the pointer. Now point at the tip of the inverting input of the op-amp and click left again. ISIS takes this as the other end for the wire and invokes the *Wire Auto Router* (WAR) to choose a route for the wire. Now do the same thing for each end of R2, following the diagram. Try tagging objects and moving them around whilst observing how the WAR re-routes the wires accordingly.

If you do not like a route that the *Wire Auto Router* has chosen, you can edit it manually. To do this, tag the wire (by pointing at it and clicking right) and then try dragging it first at the corners and then in the middle of straight runs. If you want to manually route a wire you can do so by simply clicking left on the first pin, clicking left at each point along the required route where you want a corner, and then finish by clicking left on the second pin. After a while, you will get a feel for when the WAR will cope sensibly and when you will need to take over.

To complete this first section of the drawing, you need to place the two generic and one ground terminals and wire them up. To do this, select the *Terminal* icon; the *Object Selector* changes to a list of the terminal types available. Select the *Ground* terminal, ensure its preview shows it correctly oriented, and place it just under R1. Now select the *Default* terminal from the selector and place two terminals as in the diagram. Finally wire the ground terminal to the bottom of R1 and the two default terminals to the corners of the wires going into the op-amp. ISIS will place the junction dots where required, sensing automatically that three wires are meeting at these points.

LABELLING AND MOVING PART REFERENCES

ISIS has a very powerful feature called *Real Time Annotation* which can be found on the *Tools Menu* and is enabled by default. Full information can be found on page 12 but basically, when enabled, this feature annotates components as you place them on the schematic. If you zoom in on any resistor you have placed you will see that ISIS has labelled it with both the default value (RES) and a unique reference. To edit/input part references and values click left on the *Instant Edit* icon and then click left on the object you wish to edit. Do the resistors first, entering R1, 1k and R2, 1k as appropriate. Now do the op-amp and the two terminals. To move the 'U1' and the '741' labels to correspond with the diagram, press F2 to reduce the snapping grid to 50th (it starts off at 100th) and then tag the op-amp. Now point at the label 'U1' and with the left button depressed, drag it to its correct position under the op-amp. Then do the same with the '741' label.

When you have finished positioning the labels, put the snap back to 100th by pressing F3. Although with the *Real Time Snap* feature ISIS is able to locate pins and wires not on the

current snap grid, working consistently with the same snap grid will keep drawings looking neat and tidy.

BLOCK EDITING FUNCTIONS

You may have noticed that the section of circuit you have drawn so far is currently located in the middle of the sheet, whereas it should be in the top left hand corner. To move it there, first tag all the objects you have placed by dragging a box round them using the right mouse button: point at a position above and to the left of all the objects; then press and hold down the right button and drag the mouse pointer to a position below and to the right of the objects. The selected area is shown by a cyan *tag-box* and (as the initial right click automatically untags any previously tagged objects) all and only those objects wholly within the *tag-box* will be tagged after the operation.

Now click left on the *Block Move* icon. A box will appear round all the tagged objects, and you can now begin to move this up towards the top left hand corner of the sheet. The sheet border appears in dark blue so you can now re-position the buffer circuit up at the top left of the drawing. Click left to effect the move, or else you can abort it by clicking right. You should also note how, when you moved the pointer off the *Editing Window* to the top or left, ISIS automatically panned the *Editing Window* for you. If you want to pan like this at other times (i.e. when *not* placing or dragging an object), you can use the *Shift Pan* feature.

The group of objects you have moved will remain tagged, so you might as well experiment with the *Copy* and *Delete* icons which similarly operate on the currently tagged objects. The effect of these icons can be cancelled by *immediately* following their use by pressing the 'U' key for Undo.

PRACTICE MAKES PERFECT

You should be getting the hang of things now, so get some more practice in by drawing the next section of circuitry centred around the op-amp U2:A. You will need to get a capacitor (CAP). A quick method of picking devices whose names you know is to use the *Pick Device/Symbol* command. Press the 'P' key (for *Pick Device/Symbol*) and then type in the name - CAP. Use the various editing techniques that have been covered so far to get everything in the right place. Move the part reference and value fields to the correct positions, but do not annotate the parts yet - we are going to use the *Automatic Annotator* to do this.

When you have done one op-amp filter to your satisfaction, use a *tag-box* and the *Block Copy* icon to make three copies - four filters in all - as there are in the diagram. You may find it useful to use the zoom commands on the *View* menu (or their associated short-cut keys) so as to be able to see the whole sheet whilst doing this. When you have the four filters in position, wire them together, and place a SW-SPDT device (SW1) on the drawing.

ANNOTATING THE DIAGRAM

ISIS provides you with four possible approaches to annotating (naming) components:

- **Manual Annotation** - This is the method you have already used to label the first op-amp and resistors. Any object can be edited either by selecting the *Instant Edit* icon and then clicking left on it, or by clicking right then left on it in the normal placement mode. Whichever way you do it, a dialogue box then appears which you can use to enter the relevant properties such as Reference, Value and so forth.
- **The *Property Assignment Tool* (PAT)** - This tool can generate fixed or incrementing sequences and assign the resulting text to either all objects, all tagged objects (either on all sheets or the current sheet) or to the objects you subsequently click left on. Using the PAT is faster than manual annotation, though slower than using the *Automatic Annotator*. However, it does leave you in control of which names are allocated to which parts.
- **The *Automatic Annotator*** - Using the *Automatic Annotator* leads to the whole design being annotated in a matter of seconds. The tool is aware of multi-element parts like the 7400 TTL NAND gate package and will allocate gates appropriately. However, the whole process is non-interactive so you get far less control over the names that are allocated than with the other two methods.
- **Real Time Annotation** – This feature, when enabled, will annotate components as you place them on the design, obviating any need for you to place references and values in your design. As with the *Automatic Annotator*, however, it makes the whole process non-interactive and offers no user control over the annotation process. *Real Time Annotation* can be toggled on and off through the *Real Time Annotation* command on the *Tools* Menu or via the shortcut key (default mapping is CTRL+N – more information on shortcut keys is given on page 46).

In practice you can use a mix of all four methods, and in any order you choose. The *Automatic Annotator* can be set to leave alone any existing annotation so that it is possible to fix the references of certain parts and then let ISIS annotate the rest by itself. As the *Real Time Annotation* is enabled by default, we shall leave it on and use the other three methods to edit the existing annotation of the design.

The Property Assignment Tool (PAT)

Let us suppose, for the sake of argument, that you wished to pre-annotate all the resistors using the PAT. Given that you have already manually annotated R1 and R2, you need to generate the sequence R3, R4, R5 etc. To do this, select the *Property Assignment Tool* option on the *Tools* menu. Enter REF=R# in the *String* field, then move the cursor to the next field (the *Count* field) and key in the value 3. Ensure the *On Click* button is selected and then click

left on the *OK* button or press ENTER. The hash-character (#) in the *String* field text will be replaced with the current *Count* field value each time the PAT assigns a string to an object and then the *Count* field value is incremented.

ISIS automatically selects the *Instant Edit* icon so that you can annotate the required objects by clicking left on them. Point at resistor R3 and click left. The PAT supplies the R3 text and the part is redrawn. Now do the same for the resistor below it, R4 and see how the PAT's *Count* field value increments each time you use it. You can now annotate the rest of the resistor references with some panache. When you are done with this, cancel the PAT, by calling up its dialogue form (use the 'A' keyboard shortcut for speed) and then either clicking on the CANCEL button or by pressing ESC.

The PAT can also be used to assign the same *String* to several tagged objects, for example the part values of resistors or capacitors that all have the same value. Consider the capacitors C1 to C4 which all have the value 220p. To assign this value, first ensure that only the capacitors are tagged by first clicking right on a free area of the *Edit Window* to untag all objects, then clicking right on each capacitor. Now invoke the PAT and enter VAL=220p in the *String* field, select the *Local Tagged* button and click OK. That's it - you do not need to cancel the PAT as it is not in its 'On Click' assignment mode.

Try this on your own for the rest of the diagram until you are clear about how the PAT works - although a little tricky at first, it is an extremely powerful tool and can eliminate a great deal of tedious editing. Do not forget that, when used in its *On Click* mode, you need to cancel the tool when finished.

The Automatic Annotator

ISIS features an automatic annotator which will choose component references for you. It can be made to annotate all the components, or just the ones that haven't yet been annotated - i.e. those with a '?' in their reference.

Since you have already annotated some of the parts, we will run the *Automatic Annotator* in 'Incremental' mode. To do this, invoke the *Global Annotator* command on the *Tools* menu, click on the *Incremental* button, and then click on OK. After a short time, the diagram will be re-drawn showing the new annotation. Since the OPAMP device is not a multi-element part like a true TL074, the annotator annotates them as U2 to U5 which is not what is wanted. To correct this, edit each one in turn and key in the required reference. We will see how to create and use a proper TL074 later on.

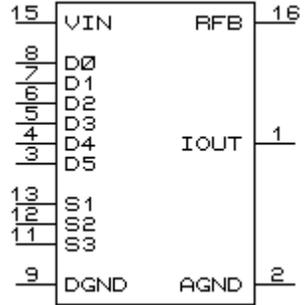
Even with the automatic annotator, you still have to set the component values manually, but try this for speed - instead of moving around the drawing to edit each component in turn, simply key 'E' for *Edit Component* (on the *Edit* menu) and key in a component's reference. This automatically locates the desired part and brings up its *Edit...* dialogue form. You should also try out the using the *Property Assignment Tool* as described in the above section.

CREATING NEW DEVICES

The next section of the circuit employs a 7110 digital attenuator, and this provides an opportunity to learn how to make new devices in ISIS.

In ISIS new devices are created directly on the drawing - there is no separate device editor mode, let alone a separate program. The new device is created by placing a collection of 2D graphics and pins, annotating the pins, and then finally tagging them all and invoking the *Make Device* command.

You will find it helpful when creating new devices to sketch out on paper how you want the device to look, and to establish roughly how big it needs to be by considering how many pins there will be down each side and so on. In this case you can use the diagram opposite as a guide. The first thing we need to do is to locate a free area of your design where the new device can be created - click the left mouse button on the lower-right region of the *Overview Window* to position the *Editing Window* on that area of the design.



Begin by drawing the device body of the new device. Select the *Box* icon. You will see that the *Object Selector* on the right displays a list of *Graphics Styles*. A graphics style determines how the graphic we are about to draw will appear in terms of line colour, line thickness, fill style, fill colour, etc. Each style listed is a different set of such attributes and define the way different parts of the schematic appear.

ISIS supports a powerful graphics style system of local and global styles and the ability of local styles to ‘follow’ or ‘track’ global styles that allows you to easily and flexibly customise the appearance of your schematic. See the section *Graphics And Text Styles* on page 49 for a complete explanation of how styles work and how they are used.

As we are drawing the body of a component, select the *COMPONENT* graphics style and then place the mouse pointer over the *Editing Window*, press and hold down the left mouse button and drag out a rectangle. Don't worry about getting the size exactly right - you can always resize the rectangle later. You will see that, as a result of choosing the *COMPONENT* graphics style, the rectangle appears in the same colour, fill, etc. as all the other components on the schematic.

The next thing to do is to place the pins for the new device. To do this, first select the the *Device Pin* icon. The *Object Selector* lists the types of available pins (note that you can also create your own pin objects in ISIS, though we will not cover that in this tutorial). Select the *Default* pin type from the selector; the *Overview Window* provides a preview of the pin with the pin's name and number represented by the strings NAME and 99 and its base and end

indicated by an *Origin* marker and cross respectively - the cross represents the end to which you will eventually connect a wire. Use the *Rotation* and *Mirror* icons to orient the pin preview ready to place the left-hand pins and then click the left mouse button in the *Editing Window* on the left edge of the rectangle where you want each pin's base to appear. Place pins for the VIN, D0..D5, S1..3 and DGND pins. Note that you can use the DOWN key to move the mouse pointer down one grid square and the ENTER key as a substitute for the left mouse button - it is sometimes quicker to use these keys instead of the mouse. Now click left on the *Mirror* icon and then place the three right-hand pins: RFB, IOU and AGND. To finish, place two pins, one on the top edge and one on the bottom edge of the rectangle, adjusting the *Rotation* and *Mirror* icons before placing them in order that they point outwards from the device body; these pins will be the VDD and VBB power pins and will eventually be hidden (this is why they are not shown in the figure).

At this stage, you can reposition the pins or resize the rectangle as required. To move a pin, tag it with the right mouse button and then drag it with the left button; to re-orient it, use the *Rotation* and *Mirror* icons. To adjust the size of the device body rectangle, tag it with the right mouse button, click and hold down the left mouse button on one of the eight 'drag handles' (the small white boxes at the corners and mid-points of the rectangle's edges) and drag the handle to the new position. If you adjust its width, you will also need to draw a *tag-box* (with the right mouse button) around the pins and then use the *Move* icon to re-position them.

So, having arranged the device body rectangle and pins as required, we now need to annotate the pins with names and numbers, and to assign them an electrical type. The electrical type (input, power, pull-up, etc.) is used by the *Electrical Rules Check* to ensure that only pins with the correct electrical type are inter-connected.

We will first assign names, electrical types and visibility. To do this, we have to tag each pin by clicking right on it and then edit it by clicking left on the tagged pin; the pin displays its *Edit Pin* dialogue form.

Edit each pin in turn, as follows:

- Enter the pin's name in the *Name* field. Leave the *Number* field empty as we will assign the pin numbers with the *Property Assignment Tool*.
- Select the appropriate electrical type for the pin - *Output* for the IOU pin, *Power* for the VDD, VBB, AGND and DGND pins, and *Input* for the remainder.
- Select whether the pin is to be hidden by unchecking its *Draw body* checkbox - the VDD and VBB pins are both standard power pins and can be hidden. The AGND and DGND pins are non-standard and so need to remain visible in order that they can be wired up as appropriate to the design the device is being used in.
- Select the OK button when finished.

To assign the pin numbers, we will use the *Property Assignment Tool*. To initialise the PAT, select the *Property Assignment Tool* command from the *Tools* Menu, and enter NUM=# in the *String* field and the value 1 in the *Count* field. Select the *On Click* button, and then close the dialogue form with the OK button. Now carefully click on each pin in order of its number (IOUT, AGND, etc.). As you click on each pin, it is assigned a pin number by the PAT. When done, don't forget to cancel the PAT by bringing up its dialogue form and selecting the CANCEL button.

All we do now is actually make the device. To do this, tag all the pins and the body rectangle - the easiest way is to drag out a tag-box with the right mouse button around the whole area being careful not to miss out the two hidden power pins. Finally, select the *Make Device* command from the *Library* menu to display the *Make Device* dialogue form. Key in the name 7110 in the *Name* field and the letter U in the *Prefix* field. Then press the *Next* button until the list of writable device libraries is displayed, select an appropriate library and then click the OK button to save the new device.

FINISHING TOUCHES

Now that you have defined a 7110 you can place, wire up and annotate the remainder of the diagram - use the *Automatic Annotator* in Incremental mode to annotate the new parts without disturbing the existing annotation.

The labelling and bracket around the six input terminals DA0-DA5 is done with 2D graphics. ISIS provides facilities for placing lines, boxes, circles, arcs and text on your drawings; all of which offered as icons on the *Mode Selector* toolbar.

The bracket is made from three lines - place these by selecting the *Line* icon and then clicking at the start and end of each line. Then place the text FROM PIO-1A as shown by selecting the *Text* icon, setting the *Rotation* icon to 90° and then clicking left at the point where to want the bottom of the 'F' character to appear. You can of course tag and drag 2D graphics objects around to get things just how you want.

Finally, you need to place a sheet border and a header block. To do the former, select the *Box* icon, zoom out till you can see the whole sheet outline (dark blue) and then place a graphics box over it. It is important to realise that the dark blue sheet outline does not appear on hard copy - if you want a bounding box you must place one as a graphics object.

The header block is worthy of more discussion. It is, in fact, no different from other symbols such as you might use for your company logo (see section [Symbols And The Symbol Library] for more on symbols). A default header block called HEADER is provided but you can re-define this to suit your own requirements - see *The Header Block* on page 43.

To actually place the header, select the *Symbol* icon and then click left on the **P** button of the *Object Selector* to display the *Symbol Library Selector* dialogue form. Picking symbols from

symbol libraries is similar to picking devices from device libraries except that there is no *Prefix* selector. Select the HEADER object from the SYSTEM symbol library and close down the dialogue form. With HEADER now the current symbol, point somewhere towards the bottom left of the drawing, press the left mouse button, and drag the header into position.

Some of the fields in the header block will fill in automatically; others such as the Design Title, Sheet Title, Author and Revision need to be entered using the *Edit Design Properties* and *Edit Sheet Properties* commands on the *Design* menu. Note that the *Sheet Name* field on the *Edit Sheet Properties* dialogue form is different from the *Sheet Title* - the *Sheet Name* is a short label for the sheet that is used in hierarchical design. The *Sheet Title* is a full description of the circuitry on that sheet and it is this that will appear in the header block.

You will need to zoom in on the header to see the full effects of your editing.

SAVING, PRINTING AND PLOTTING

You can save your work at any time by means of the *Save* command on the *File* menu, and now is as good a time as any! The *Save As* option allows you to save it with a different filename from the one you loaded it with.

To print the schematic, first select the correct device to print to using the *Printer Setup* command on the *File* 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. When you have selected the correct printer, close the dialogue form and select the *Print* option on the *File* menu to print your design. Printing can be aborted by pressing ESC, although it may be a short time before everything stops as both ISIS and possibly your printer/plotter have to empty their buffers.

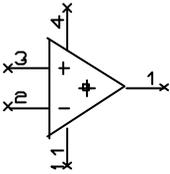
Further details regarding printer and graphics output are given under *Hard Copy Generation* on page 163.

If you have the demo version, please note that you can only print un-modified sample designs. To try this now, use the Load command on the File menu to load a sample design.

MORE ABOUT CREATING DEVICES

Making a Multi-Element Device

We shall now define a proper library part for the TL074 quad op-amp. As there are four separate op-amps to a single TL074 package our tutorial will be showing you how to create *multi-element devices* using the *Visual Packaging Tool*.



The illustration on the left shows the new op-amp device before it is made. The op-amp is made from some 2D graphics, five pins and an origin marker. We will look at two ways to construct the op-amp graphics. The easiest approach uses the pre-defined OPAMP symbol. Proceed as follows:

- Click on the *Symbol* icon and then click the **P** button at the top left of the *Symbols Object Selector*. This will launch the *Symbol Library Selector* dialogue form.
- Double-click on OPAMP in the *System Library* and close the dialogue form using the *Windows Minimise* button on the right of the title bar.
- Position the mouse pointer in an empty area of the *Editing Window* and use the left mouse button to place the op-amp. The op-amp automatically appears in the *COMPONENT* graphics style as this style was used to create the symbol.

Now place the pins around the component body. This is the same process as for creating the 7110 attenuator earlier:

- Select the *Device Pin* icon to obtain a list of available pin types and select the *Default* type.
- Use the *Rotate* and *Mirror* icons to orient the pins before placing them on the design.
- Once all the pins are in the correct positions, edit each pin in turn by tagging it with the right mouse button and then clicking left on it. Use the resulting *Edit Pin* dialogue form to annotate the pin with the correct electrical type and pin name. We have to give the pins names so that we can reference them in the *Packaging Tool* however, we don't want the name to be displayed (as the op-amp pins' uses are implicit from the graphics) so ensure that the *Draw Name* check-box is not checked. Note, there is no need to specify pin numbers as these will entered using the *Packaging Tool*.

The power pins have the names V+ and V- and have the electrical type of *Power*; if you place them just in from the left edge of the op-amp, you will find they just touch the sloping sides of the OPAMP graphic whilst keeping their pin ends (marked by an 'X') on a grid-square. If in a similar situation, they didn't touch, you could 'extend' the base of the pin by placing short lines in 2D Graphic Mode and with the mouse snap off. The input pins have the names +IP and -IP and the electrical type *Input*. The output pin has the name OP and the electrical type *Output*.

The final stage is to place an *Origin* marker. Select the *Marker* icon to display a list of system marker symbols in the *Object Selector*. Select the *Origin* marker in the *System Library* and then place the marker symbol at the centre of the op-amp graphics. The *Origin* marker is

displayed as a rectangle with cross-hairs and it indicates to ISIS how the new device should appear around the mouse pointer when the device is dragged or placed in a design.

We have now completed making the device. Tag the constituent parts - the op-amp symbol, pins and the *Origin* marker - by dragging out a *tag-box* around them using the right mouse button, and then invoke the *Make Device* command on the *Library* menu. Then proceed as follows:

- Enter the *Device Name* as TL074 and the Prefix as 'U'
- Click *Next* button display the *Packaging* page and click *Add/Edit* to launch the packaging tool itself.

The Visual Packaging Tool

The visual packaging tool provides a graphical environment in which to assign one or more PCB footprints to a schematic part. For each packaging, a table of pin numbers to pin names is created such that different packagings can have different pin numbers for the same schematic pin.

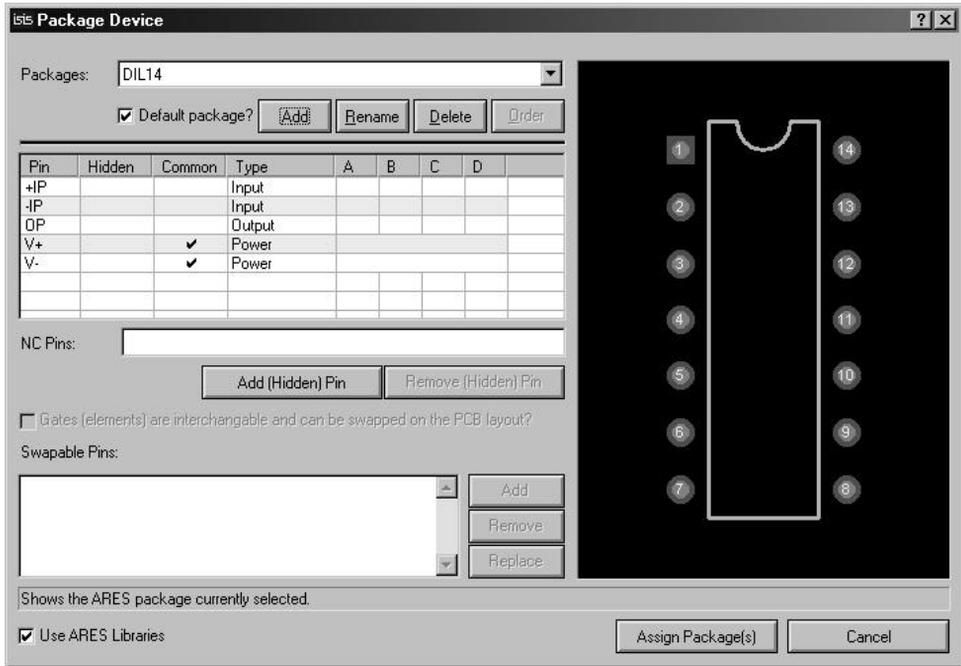
Having launched the packaging tool, the first thing to do is to create a packaging:

- Click the *Add* button. This will launch the ARES library browser.
- Select the PACKAGE library, and double click the DIL14 part.

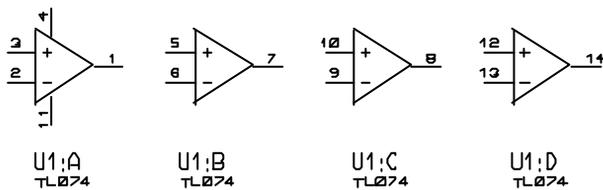
Then, you need to make the following changes to the default settings for the packaging:

- Change the number of elements from 1 to 4. This corresponds with the fact that there are four op-amps within the one physical DIL14 package.
- Mark the V+ and V- pins as common pins. This means that they will have the same pin number on each element, and that you can wire to any or all of the pins on the elements on the schematic. All such wiring will be deemed interconnected.
- Click the *Gates can be Swapped* checkbox. This specifies that the elements are identical and the ARES can perform gateswap operations on this part.

You should then see a display similar to that below:



Now to assign the pin numbers. The actual pinning of the op-amps that we are aiming at is shown below:



Proceed as follows:

- Click left in the 'A' column box for pin +IP.
- Either, click pin '3' on the package display or enter '3' from the keyboard and press TAB. Either way, pin '3' will highlight on the package to show that you have assigned it, and the cursor will move to the -IP row.

- Now, repeat the process for the other pin numbers, until all the pins on the package display are highlighted. This of course, provides a visual cue that you haven't missed any pins.

Finally, click the *Assign Packages* button to return the *Make Device* wizard, and then store the device into your USERDVC library, as you did with the 7110.

There is, as you have probably realized, a great deal of functionality built into the packaging tool. For a detailed discussion and further examples, see *DEVICE LIBRARIES* on page 107.

Making Similar Devices

Having defined a TL074, you can almost instantly define types TL064 and TL084 as well. Simply place a TL074, tag it, and invoke the *Make Device* command; change the name to TL064 (or whatever) and save it. What could be simpler? If you needed to add something to the basic TL074 - perhaps some more graphics, you could simply add them on top of the placed TL074 before invoking the *Make Device* command. Alternatively, if the TL074 was almost right but needed some slight editing before it was suitable for the new device, you could decompose it back into its constituent parts by tagging it, selecting the *Decompose* command on the *Library* menu, editing and/or adding to them, and then making the new device.

Replacing Components in a design

You can now replace the four filter op-amps and with proper TL074 parts. To replace a component with one of a similar type, pick the new device, ensure the mouse is over the device you want to replace, click and hold down the left mouse button and drag the new device such that one or more pin-ends overlap. ISIS will then transfer the wires from the old component to the new component whilst keeping all other information about the old component (e.g. its reference, etc.) intact.

SYMBOLS AND THE SYMBOL LIBRARY

Tag the three lines that form the bracket around the inputs to the 7110. Then invoke *Make Symbol* on the *Library* menu, key in TEST for the symbol name and press ENTER. Now select the *Symbol* icon. You will see that the item TEST has appeared in the *Object Selector*. Pick this and try placing it on the drawing. Common uses for this are things like OPAMP, which is needed for many device types, and logos, emblems etc.

A special use of symbols is for the HEADER block. The default symbol was created out of 2D lines, a box and several special text primitives that are automatically replaced by properties associated with the current design and sheet. For example, a text object with the string:

@DTITLE

will automatically appear as the design title entered in the *Edit Design Properties* command's dialogue form. The complete list of keywords is presented in *The Header Block* on page 43.

REPORT GENERATION

Now that the diagram is complete, you can generate Netlist, Bill of Materials (BOM) and Electrical Rules Check (ERC) reports from it. Each report is generated by invoking the appropriate command from the *Tools* menu. The report output is displayed in a pop-up text viewer window from where it can be saved to a file by selecting the 'Save as' button or placed on the clipboard for use in other packages using the 'Copy to clipboard' button; the 'Close' button clears the report display and returns you back to the editor. Note that the last report or simulation log generated is maintained by ISIS - to view a report again, select the *Text Viewer* command on the *System* menu.

The Bill of Materials report should be fairly self explanatory, although you can get a lot more out of the facility - see *Bill Of Materials* on page 147.

The Electrical Rules Check report will contain quite a few errors, since the tutorial circuit is not a complete design - of particular note is that the VBB pin of the 7110 is flagged as undriven, which could easily be forgotten in a real situation.

Further information regarding Report Generation is given in *Report Generation* on page 147 whilst *Netlist Generation* on page 135 details in intricacies of producing and using a netlist. For those of you who have bought ISIS to use with ARES, details of how to link the two packages together are provided in *Isis And Ares* on page 167.

A LARGER DESIGN

In this last section, we shall take a look at a pre-prepared design - EPE.DSN. This is a multi-sheet, hierarchical schematic for micro-processor controlled EPROM programmer/emulator (EPE). As such, it represents a substantial piece of electronics at the lower end of design complexity that you might expect to design with your ISIS system.

The EPE design is structured as three A3 sheets (Processor, Emulator and PSU). Sub-sheets are used to represent an Emulation RAM bank (of which there are 4, giving 32 bit emulation capability) and a Programmable Power Supply (PPSU) of which 6 are required to deal with the range of 27 series EPROM pinouts.

Load the design into ISIS by using the *Load* command on the *File* menu, and then selecting EPE.DSN from the file selector. You will find it in the "Samples\Schematic & PCB Design" folder of your Proteus installation. Alternatively, you can view any of the sample designs by launching the SAMPLES help file from the *Help* menu.

The first sheet is the CPU - take a look round this with the usual pan and zoom facilities. Then, to see more of the design, invoke the *Goto Sheet* command on the *Design* menu. Select the second item from the selector and after some disk activity the Emulator Control sheet will be loaded. Zoom out so that you can see all of it. The 4 big blue boxes are the sub-circuits. The labelling text at the top is the sub-circuit ID (like a part reference) and the text at the bottom is the circuit name.

You can also zoom into the sub-circuits: point at one of the sub-circuits and press the shortcut key for “*Zoom to Child*” (the default mapping for this is CTRL+C – information on changing keyboard shortcuts can be found on page 46). ISIS swaps out the Emulator Control sheet and loads an ERAM bank sheet. Take a look round the ERAM bank circuit and in particular, take note of a few of the component numbers. To get out, use the keyboard mapping for “*Exit to Parent*” – the default mapping for this is CTRL+X. Zoom into another ERAM bank and compare the component numbers in this one with the first - although both sub-circuit instances share the same circuit (if you modify one instance of the circuit, this will be instantly reflected in the others which simplifies design modification) each has its own set of component annotations; this is *Design Global Annotation* at work.

Now that you know about loading the various sheets and hierarchy roaming you may as well explore the rest of the EPE design. It is a good mix of analogue, digital and microprocessor circuits which shows how ISIS is well suited to all types of schematic.

GENERAL CONCEPTS

SCREEN LAYOUT

The Menu Bar

File View Edit Library Tools Design Graph Source Debug Template System Help

The *Menu Bar* runs across the top row of the screen and its main purpose (the selection of commands from the menus) is the same as with any other Windows application. In addition, the title bar area above the menu names is used to display certain prompt messages which indicate when the program has entered a particular editing or processing mode.

The Toolbars

As with other modern Windows applications, ISIS provides access to a number of its commands and modes through the use of toolbars. The toolbars can be dragged to any of the four edges of the ISIS application window.

Command Toolbars

The tools located along the top of the screen (by default) provide alternative access to the menu commands, as follows:

File/Print commands



Display Commands



Editing Commands



Design Tools



If you are working on a relatively small monitor, you can hide any or all of the command toolbars using the *Toolbars* command on the *View* menu.

Mode Selector Toolbar

The toolbar located down the left hand edge of the screen select the editor mode, i.e. what happens when you click the mouse on the *Editing Window*.

Main Modes



Gadgets



2D Graphics



Note that the mode toolbar cannot be hidden, as its functions are not duplicated on the menus.

Orientation Toolbar

The orientation toolbar displays and controls the rotation and reflection for objects placed onto the layout.

Rotation



Reflection



The edit box allows you type a rotation angle in directly; but note that in ISIS, only orthogonal angles may be entered.

When an existing object is tagged, the rotation and reflections icons highlight in red to show that they will modify the orientation of an object on the layout. When the icons are not highlighted, they serve to determine the orientation for new objects.

The Editing Window

The *Editing Window* displays the part of the schematic that you are currently editing. The contents of the *Editing Window* may be redrawn using the *Redraw* command on the *View* menu. This also redraws the *Overview Window*. You can use this feature after any other command that has left the display somewhat untidy.

Panning

You can reposition the *Editing Window* over different parts of the design in several ways:

- By clicking left at a point on the *Overview Window* - this re-centres the *Editing Window* about the marked point.
- By moving the mouse over the *Editing Window*, holding down the SHIFT key, and 'bumping' the pointer against one of its edges. This pans the display in the appropriate direction. We refer to this feature as *Shift-Pan*.
- By pointing in the *Editing Window* and pressing the Zoom key (see below). This re-centres the display about the cursor position.

- By using the *Pan* icon on the toolbar.

Zoom In / Zoom Out

You can magnify or reduce the display of the board using the *Zoom In* and *Zoom Out* commands which are also mapped onto shortcut keys (by default F6 will zoom in and F7 will zoom out). Similarly you can view the entire board via the shortcut key for the *Zoom to view entire sheet* command (default mapping is F8). You can also use the corresponding icons on the toolbar or, if you have a mouse wheel, you can zoom in and out around the mouse pointer with the mouse wheel.

If the keyboard is used to invoke the command, then the *Editing Window* will be redrawn to display a region centred around where the mouse cursor was pointing before. This also provides a way to effect a pan by pressing the zoom key for the current level and simultaneously pointing with the mouse at where you want centre of the new display to be.

Variable Zoom

An arbitrary degree of magnification can be achieved using the Shift-Zoom feature. A given area of the board can be selected to fill the *Editing Window* by holding down the SHIFT key, pressing the left mouse button and dragging out a box around the desired area. The area can be marked on either the *Editing Window* or the *Overview Window*.

You can also zoom to an area by clicking the *Zoom Area* icon on toolbar or by configuring a shortcut key for this command. Details on custom keyboard configuration can be found on page 46

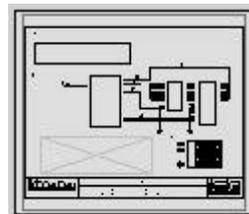
The Overview Window

This window normally shows a simplified representation of the whole drawing, and has a half-inch grid on it. The cyan box marks the outline of the sheet border, whilst the green box indicates the area of the design currently visible in the *Editing Window*.

Clicking left at a point on the grid re-centres the *Editing Window* around this point, and redraws the *Editing Window*.

At other times, *Overview Window* is used to display a preview of an object that is selected for placement. This *Place Preview* feature is activated in the following circumstances for any object which may be oriented:

- When an object is selected from the object selector.
- When the rotate or mirror icons are adjusted.



- When an object type icon is selected for an object whose orientation can be set (e.g. the *Component* icon, *Device Pin* icon, etc.).

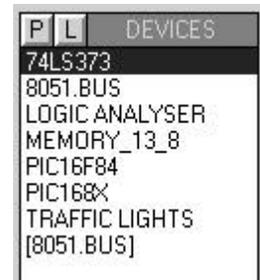
The place preview display is cleared automatically as soon as you proceed the place the object, or when you perform any operation other than those listed above.

The width and height of the *Overview Window* can be adjusted by dragging its borders. If you drag the vertical border right over to the other side of the application Window, ARES will re-organize the display so that the *Overview Window* and *Object Selector* are located at the right hand side.

The Object Selector

The Object Selector is used for picking components, terminals, generators, graph types and so on from those that are available. It always carries a label indicating what it is listing and this serves as a prompt additional to the state of the Icon Panel as to which mode is current.

The width and position of the *Object Selector* can be adjusted in conjunction with the width and height of the *Overview Window*, as described above.



The buttons at the top left of the object selector activate various functions such as library browsing and library management. The exact functions available depend on the current editor mode.

Co-ordinate Display

The current co-ordinates of the mouse pointer are displayed down at the bottom right of the screen by default. The read-out can be in imperial or metric units and a false origin may be set. Further details are given under CO-ORDINATE SYSTEM on page 28.

The *X-Cursor* command will display a small or large cross, in addition to the mouse arrow, at the exact location of the current co-ordinates. This is particularly helpful when used in conjunction with the Real Time Snap feature, since it gives you an immediate indication of what ARES thinks you are pointing at.

CO-ORDINATE SYSTEM

All co-ordinates in ISIS are actually held in 10nm units, purely to be consistent with ARES. However, the coordinate read-out is restricted to 1 thou units. The origin is held to be in the centre of the work area and so both positive and negative values are used. The co-ordinates of the pointer are displayed at the bottom right of the screen.

We refer to a 1 thou increment as a unit.

False Origin

Although the *Origin* command appears on the *View* menu, it should only be used via its keyboard short cut (key 'O'). Its function is to zero the co-ordinate display at the current mouse position, a feature that is most useful when laying out a complex pattern of pads given a set of dimensions on a drawing of the component.

When a false origin is set, the co-ordinate display changes colour from black to magenta as a reminder.

Cancelling a false origin is done by invoking the *Origin* command a second time.

The Dot Grid

A grid of dots is displayed in the *Editing Window* - this can be toggled on and off using the *Grid* command on the *View* menu. The dot spacing reflects the current snap setting., unless this would result in a ridiculous number of dots, in which the spacing is increased.

Snapping to a Grid

You will notice that when the pointer is over the *Editing Window*, the increments of the co-ordinate display are in fixed steps - initially 100th. This is called *snapping* and enables you to position components and other objects on a neat grid. The degree of snap may be selected with the *Snap* commands on the *View* menu, or via the shortcut keys assigned to the snap commands (by default these are F4, F3, F2 and CTRL+F1 – information on customising the keyboard can be found on page 46).

If you wish to see exactly where the snapped position is, you can use the *X-Cursor* command on the *View* menu that will display either a small or large cross at this location.

Real Time Snap

Furthermore, when the pointer is positioned near to pin ends or wires, the cursor location will also be snapped onto these objects. This function is called *Real Time Snap* and allows you to connect to or from pins and wires that are not on the currently selected snap grid. You can toggle this function using the *Real Time Snap* command on the *Tools* menu, or via a keyboard shortcut (the default keyboard mapping for this command is CTRL+S).

With a very large drawing on a slow computer, real time snap may cause some lag between the cursor and the pointer. You may find it best to disable the feature in these circumstances.

FILING COMMANDS

ISIS makes use of the following file types:

Design Files	(.DSN)
Backup Files	(.DBK)
Section Files	(.SEC)
Module Files	(.MOD)
Library Files	(.LIB)
Netlist Files	(.SDF)

Design files contain all the information about one circuit, and have the file extension 'DSN'. Previous versions of ISIS have used 'ISS', 'IDS' and 'IWS'; these can be converted automatically provided that you have the file converters IDSCVT40.DLL and/or IWSCVT40.DLL installed. Backup copies of design files made when saving over an existing file are given the extension "DBK".

A section of a drawing can be exported to a section file and subsequently read into another drawing. Section files have the extension 'SEC'. and are read and written by the *Import* and *Export* commands on the *File* menu.

Module files have the extension 'MOD' and are used in conjunction with the other features for hierarchical design. See *Hierarchical Designs* on page 129 for further information.

Symbol and device libraries have the extension 'LIB'.

Netlist files produce when exporting connectivity to ProSPICE and ARES have the extension 'SDF'; other extensions are used when producing netlist files in 3rd party formats.

 The Proteus VSM simulation system uses other file types as well. See the Proteus VSM manual for further details.

Starting a New Design

The *New Design* command will clear out all existing design data and present a single, empty A4 sheet. The design name is set to UNTITLED.DSN and this name will be used by the *Save Design* command and also as the default filestem in other file selectors.

Should you wish to start a new design and give it a name at the same time, you can use the *Load Design* command and enter the new filename in the file selector.

Loading a Design

A design may be loaded in three ways:

- From the DOS prompt as in:

```
ISIS <my_design>
```
- By using the *Load Design* command once ISIS is running.
- By double clicking the file in *Windows Explorer*.

Saving the Design

You can save your design when quitting ISIS via the *Exit* command, or at any other time, using the *Save Design* command. In both cases it is saved to the same file from which it was loaded. The old version will be prefixed with the text ‘Backup of’.

The *Save As* command allows you to save the design to a different file.

Back-up and Last Loaded Files

In addition to the auto-save ISIS provides extra security for your designs via a *Backup* and *Last Loaded* mechanism.

Whenever a design is saved any existing design file on the disk is backed up to either a last-loaded or back-up file. For a design with the file name MYFILE.DSN the last-loaded file has a file name of “Last Loaded MYFILE.DBK” and the back-up a file name of ‘Backup Of MYFILE.DBK’. Note that a different file extension (DBK) is used for back-up and last-loaded files so they don’t clutter the *Load Design* dialogue form. To see these files when loading a design you must change the *Files Of Type* field (at the bottom of the *Load Design* file selector dialogue form) to “*Backup Design Files*”.

For a new design, the back-up process works as follows. The first save operation saves the design file. For each subsequent save, any existing back-up file is deleted, the existing design file is renamed to the back-up file name and the design is then saved.

For an existing design, the back-up process is slightly different. The first save operation after loading the design deletes any last-loaded file and renames the current design file as the last-loaded file. For each subsequent save (during the editing session) the normal back-up scheme applies – that is, for each save, any existing back-up file is deleted, the existing design file is renamed as the back-up file and the design is then saved.

In summary, the design file (MYFILE.DSN) is always the last saved version of the design and the back-up file (Backup of MYFILE.DBK) is usually the version of the file prior to the last save. The last-loaded file (Last Loaded MYFILE.DBK) is a version of the design that is

known to load successfully and represents the version of a design at the start of an editing session regardless of how many times you save the design during that editing session.

Import / Export Section

The *Export* command on the *File* menu creates a section file out of all currently tagged objects. This file can then be read into another sheet using the *Import* command. After you have chosen the section file, operation is identical to the *Block Copy* function.

These commands have nothing to do with graphics export to DTP packages. The *Export Graphics* commands handle this functionality.

Quitting ISIS

When you wish to end an ISIS session, you should use the *Exit* command on the file menu, key 'Q'. If you have modified the design, you will be prompted as to whether you wish to save it.

GENERAL EDITING FACILITIES

Object Placement

ISIS supports many types of object, and full details of the purpose and behaviour of each type is given in the next chapter. However, the basic steps for placing an object are the same for all types.

To place an object:

1. Select the appropriate icon from the *Mode Selector* toolbar for the category of object that you want to place.
2. If the object type is Component, Terminal, Pin, Graph, Symbol or Marker, select the name of the object that you want from the selector. For Components, Terminals, Pins and Symbols, this may first involve picking it from the libraries.
3. If the object is orientable, it will have appeared in the *Overview Window*. You should now adjust its orientation to that which you require by clicking on the *Rotation* and *Mirror* icons.
4. Finally, point on the *Editing Window* and click left to place or drag the object. The exact procedures vary for each object type but you will find it all fairly intuitive and similar to other graphics software. If you really want to read about the details, they are given in *Object Specifics* on page 73.

Tagging an Object

Any object may be tagged by pointing at it and clicking right. This action highlights the object and selects it for further editing operations.

- Any wires connected to an object that is tagged are also tagged.
- A group of tagged objects may be assembled either by clicking right on each object in turn, or by dragging a box around the objects using the right button. Only objects wholly enclosed in the box will be tagged.
- All the objects may be untagged by pointing at no object and clicking right.

Deleting an Object

You can delete any tagged object by pointing at it and clicking right. All wires connected to the object will also be deleted, except in the case of a dot connected to exactly 2 wires, in which case the wires will be joined.

Dragging an Object

You can drag (i.e. re-position) any tagged object by pointing at it and then dragging with the left button depressed. This applies not only to whole objects, such as components, but also individually to their labels.

- If the *Wire Auto Router* is enabled and there are wires connected to it, then these will be re-routed or 'fixed up'. This can take some time (10 seconds or so) if the object has a lot of connected wires; the pointer becomes an hour glass while this is happening.
- If you drag an object by mistake, and all the wiring goes horribly wrong, you can use the Undo command, key U to restore things to their original state.

Dragging an Object Label

Many object types have one or more *labels* attached to them. For example, each component has a reference label and a value label. It is very easy to move these labels in order to improve the appearance of your schematics.

To move a label:

1. Tag the object by pointing at it (or the label) and clicking right.
2. Point at the label, press the left mouse button.

3. Drag the label to the required position. If you need to position it very finely, you can change the snap resolution (by default these commands are mapped onto the F4, F3, F2, CTRL+F1 keys) even whilst dragging.
4. Release the mouse button to finish.

Resizing an Object

Sub-circuits, graphs, lines, boxes and circles may be resized. When you tag these objects, little white squares called *handles* will appear and the object can be re-sized by dragging the handles.

To resize an object:

1. Tag the object by pointing at it and clicking right.
2. If the object can be resized, a set of little square handles will appear on it.
3. Resize the object by pointing at a handle, pressing the left mouse button, and dragging it to a new position. The handles disappear whilst you are dragging so that they do not obscure your view of the object itself.

 See the section *Resizing 2d Graphics* on page 98 for information about advanced use of handles when resizing 2D graphics path objects.

Reorienting an Object

Many of the object types may be oriented - that is rotated through 0°, 90°, 270° and 360° and reflected in x and/or y. If such an object is tagged, the *Rotation* and *Mirror* icons will change colour from blue to red, and will then affect the tagged object.

To reorient an object:

1. Tag the object by pointing at it and clicking right.
2. Click left on the *Rotation* icons to rotate it anti-clockwise, right to rotate it clockwise.
3. Click left on the *Mirror* icon to toggle its reflection in x, and right to toggle its reflection in y.

It is worth noting that if the *Rotation* and *Mirror* icons are red, operating them will affect an object somewhere on the diagram, even if you cannot currently see it. This becomes important if, in fact, you wish to manipulate a new object which you are about to place. If the icons are red, first untag the existing object by pointing at an empty area of the design in the *Editing Window* and clicking right. The icons will then revert to blue, indicating that it is 'safe' to adjust them.

Editing an Object

Many of the objects have graphical and/or textual properties that may be edited through a dialogue form, and because this is a very common operation we have provided a variety of ways to achieve it.

To edit a single object using the mouse:

1. Tag the object by pointing at it and clicking right.
2. Click left on it, as if to drag, but release the mouse button immediately, without moving the mouse.

To edit a succession of objects using the mouse:

1. Select the *Instant Edit* icon.
2. Point at each object in succession and click left.

To edit an object and access special edit modes:

1. Point at the object.
2. Use the shortcut key for “*Edit Object Under Cursor*” – by default this is mapped to the CTRL+E key combination.

For text scripts, this will invoke the external text editor. Also, if the mouse is not over any object, this command will edit the current graph, if any.

To edit a component by name:

1. Use the keyboard shortcut for “*Find and Edit Component*”. The default keyboard assignment for this command is E. Information on changing and configuring keyboard shortcuts can be found on page 46.
2. Type in the reference name (part ID) of a component.

This will locate and bring up the dialogue form for any component in the design, not just those on the current sheet. After the edit, the screen is re-drawn with the component in the centre. You can thus use this command to locate a component, even if you do not actually want to edit it.

 Details pertaining to the operation of the dialogue forms associated with each object type are given in *Object Specifics* on page 73.

Editing an Object Label

Component, terminal, wire and bus labels can all be edited in much the same way as objects:

To edit a single object label using the mouse:

1. Tag the object by pointing at it and clicking right.
2. Click left on the label, as if to drag, but release the mouse button immediately, without moving the mouse.

To edit a succession of object labels using the mouse:

1. Select the *Instant Edit* icon.
2. Point at each label in succession and click left.

Either way, a dialogue form with *Label* and *Style* tabs is displayed. The editing of local text styles is fully covered in the tutorial on graphics and text styles- see *Editing Local Styles* on page 52.

Copying all Tagged Objects**To copy a section of circuitry:**

1. Tag the required objects either individually, or by dragging out a tag-box as described in *Tagging An Object* on page 34.
2. Click left on the *Block Copy* icon.
3. Drag the copy outline to the require position and click left to place a copy.
4. Repeat step [3] as required to place multiple copies.
5. Click right to finish.

When components are copied, their references are automatically reset to the un-annotated state so as to ready them for automatic annotation, and prevent the occurrence of multiple instances of the same part IDs.

Moving all Tagged Objects**To move a set of objects:**

1. Tag the required objects either individually, or by dragging out a tag-box as described in *Tagging An Object* on page 34.
2. Click left on the *Block Move* icon.
3. Drag the outline to the required position and click left to place it.

The behaviour of wires during block move is somewhat subtle. Essentially, ISIS will move all wires or parts of wires enclosed in the tag-box without re-routing them, and then, where wires

cross the boundaries of the tag-box, it will reroute from the last point inside the tag-box to the first point outside it. It follows that you can control whether a section of wiring is preserved or re-routed according to whether you include it in the tag-box or not.

 You can also use block move to move just sections of wiring, without moving any objects at all. Further discussion of this is given in *Dragging Wires* on page 39.

Deleting all Tagged Objects

To delete a group of objects:

1. Tag the required objects either individually, or by dragging out a tag-box as described in *Tagging An Object* on page 34.
2. Click left on the *Delete* icon.

If you delete something by mistake, you can recover it using the *Undo* command.

WIRING UP

Wire Placement

You may have noticed that there is no *Wire* icon. This is because ISIS is intelligent enough to detect automatically when you want to place a wire. This avoids the tedium of having to select a wire-placement mode.

To connect a wire between two objects:

1. Click left on the *connection point* of the first object.
2. If you want ISIS to auto-route the wire, just click left on a second connection point. On the other hand, if you wish to determine the wire's route yourself, you can click left on one or intermediate points which will become corners in the wire's route.

A connection point can connect to precisely one wire. For components and terminals there is a connection point at the end of each pin. A dot has four connection points at its centre so that four wires can be joined at a junction dot.

Since it is common to wish to connect to existing wires, ISIS also treats wires as continuous connection points. Furthermore, as such a junction invariably means that 3 wires are meeting at a point it also places a dot for you. This completely avoids ambiguities that could otherwise arise from missing dots.

You can abort the routing of a wire by pressing ESC at any stage of the process.

The Wire Auto-Router

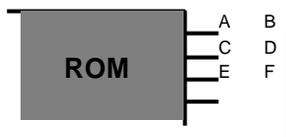
The *Wire Auto-Router* (WAR) saves you the chore of having to mark out the exact route of each wire. The feature is enabled by default, but can be overridden in two ways.

If you simply click left at two connection points, the WAR will attempt to choose a sensible path for the wire. If, however, you click on one connection point, and then click at one or more positions which are not connection points, ISIS will assume you are manually routing the wire and will let you click at each corner of the wire's route. The route is completed by clicking left on a second connection point.

The WAR can be completely disabled using the *Wire Auto-Router* command on the *Tools* menu. This is useful if you want to route a diagonal wire directly between two connection points.

Wire Repeat

Suppose you have to connect the data bus of an 8 bit ROM to the main data bus on the circuit diagram and that you have placed the ROM, bus and bus entries as shown overleaf.



You would first click left at A, then B to place a horizontal wire between them. By clicking twice at C, you will invoke the *Wire Repeat* function which will then place a wire between C and D. Clicking left twice on E will join E and F and so forth.

Wire Repeat copies exactly the way the previous wire was routed. If the previous wire was automatically routed then the repeated wire will also be automatically routed. On the other hand, if the previous wire was manually routed then its exact route will be offset and used for the new wire.

Dragging Wires

Although wires follow the general scheme of tag then drag, there are various special techniques that you can apply to them. In particular:

- If you point at a corner and drag then the corner simply follows the pointer.
- If you point in the middle of a horizontal or vertical wire segment, the segment will be dragged vertically or horizontally respectively and the adjacent segments will be stretched to maintain connectivity.
- If you point in the middle of a diagonal segment, or at either end of the wire, then a corner will be created and then dragged. Note that in order for the latter to work, the

LABCENTER ELECTRONICS

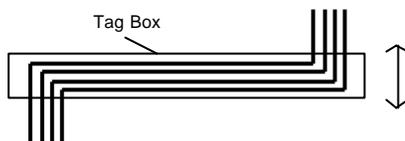
object to which the wire connects must not be tagged, as otherwise, ISIS will think you are trying to drag the object.

It is also possible to move a wire segment, or a group of wire segments using the block move command.

To move a wire segment or a group of segments:

1. Drag out a tag-box around the wire segment(s) you wish to move. It is quite acceptable for this 'box' to be a line, lying along a single segment, if this is convenient.
2. Click left on the *Move* icon.
3. Move the tag-box in the direction orthogonal to the wire segments, as shown in the diagram, opposite.
4. Click left to finish.

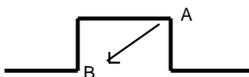
If it all goes wrong, you can use the *Undo* command to recover the situation.



A further technique provides a quick way of eliminating unwanted kinks in wires, perhaps where they have been routed around an object which has since been moved.

To remove a kink from a wire:

1. Tag the wire you wish to manipulate.
2. Point at one corner of the kink and press the left mouse button.
3. Drag the corner such that the kink is doubled over itself, as in the diagram, below.
4. Release the left mouse button. ISIS will remove the kink from the wire.



THE AUTOMATIC ANNOTATOR

ISIS can automatically chose component references for all or some of the components in a design - this process is called *Auto-Annotation*.

The process is initiated using the *Global Annotator* command on the *Tools* menu.

Note that the Global Annotator cannot annotate heterogeneous multi-element parts. This is because with several un-annotated relay coils and contacts, for example, there is no way for it to know what goes with what.

Value Annotation

This facility is intended for use where an analysis program computes some values for a standard circuit, and you want to import them. An example block would be:

```
VALUES
R1 , 10k
C1 , 100n
END
```

The value of R1 would be set to 10k and C1 to 100n.

MISCELLANEOUS

The Sheet Border

When a new sheet is created either as the first sheet of a new design, or by using the *New Sheet* command, it starts off at the size currently selected on the *Set Sheet Sizes* dialogue on the *System* menu. The extent of the sheet is shown by a dark blue sheet outline but this in itself will not actually appear on hard copy.

If a sheet border is required on the final output, you must place a graphics box (or whatever) over the sheet outline.

The *Set Sheet Sizes* command is worthy of further discussion since it serves two distinct purposes:

- To change the paper size for the current sheet, you simply invoke the command and click on the button for the required sheet size.
- To re-define the dimensions of a paper size, highlight the appropriate data entry field(s) and key in the new dimension(s). If you change the dimensions of the current sheet size, this will affect the current sheet immediately but not any other sheets in the design. To affect these, you must invoke *Set Sheet Sizes* and click OK for each sheet in turn.

The *Set Sheet Sizes* dialogue form allows you to define the sizes of five standard sheet sizes (A4 through to A0) as well as a non-standard size and to select which one is the size if the current sheet.

For each sheet size (*A4-A0* and *user*) there are two edit fields. The left field defines the width (x-dimension) and the right side defines the height (y-dimension). Whenever ISIS loads a sheet that does not match any of the standard (A4-A0) sheet sizes it places the dimensions of the sheet into the *User* field and sets that to be the default sheet size for the loaded sheet.

The default values for the standard sheet sizes should work for most printers but please note that all printers have 'margins' in which they cannot print – if you attempt to print something that impinges on the margins it will most likely not be drawn. The size of the margin where a printer cannot print varies between printers and the best way to determine it is to draw a sample design with a box that surrounds your chosen sheet size and then printing it. If one or more edges of the box do not appear on the printed output then your sheet size extends into the printers margins and you should reduce it.

There is one more point to note about sheet sizes: Every sheet in a design carries its actual dimensions with it in the design file. If a design is loaded into a copy of ISIS with a different sheet size configuration, this will not have any effect unless and until the *Set Sheet Sizes* command is used.

The Header Block

It is common practice to have on each sheet of a drawing a header block which shows details such as the design and sheet titles, the document, revision and page numbers, and the design's author.

In order that you have full control over how this information is presented, the header block is defined as a symbol library entry called HEADER. This was made in the usual way by placing graphics objects, tagging them and invoking the *Make Symbol* command. The significant thing is where, for example, the current design title is required, a text object with the string **@DTITLE** was placed and this is automatically replaced with the actual design title at display/print time.

The complete list of such keywords is shown overleaf.

@DTITLE	The design title taken from the <i>Edit Design Properties</i> command form.
@STITLE	The sheet title taken from the <i>Edit Sheet Properties</i> command form. Do not confuse this with the Sheet Name.
@DOCNO	The design document number taken from the <i>Edit Design Properties</i> command form.
@REV	The design revision number taken from the <i>Edit Design Properties</i> command form.
@AUTHOR	The design author taken from the <i>Edit Design Properties</i> command form.
@CDATE	The design creation date - generated automatically and in a fixed format.

@MDATE	The design modification date - generated automatically and in a fixed format.
@WS_CDATE	The design creation date - generated automatically and formatted according to the Windows 'short date format' (see below).
@WL_CDATE	The design creation date - generated automatically and formatted according to the Windows 'long date format' (see below).
@WS_MDATE	The design modification date - generated automatically and formatted according to the Windows 'short date format' (see below).
@WL_MDATE	The design modification date - generated automatically and formatted according to the Windows 'long date format' (see below).
@CTIME	The design creation time - generated automatically and formatted according to the Windows 'time format' (see below).
@MTIME	The design modification time - generated automatically and formatted according to the Windows 'time format' (see below).
@PAGENUM	The current sheet page number within the design.
@PAGECOUNT	The total number of sheets in the design.
@PAGE	The sheet page number within the design expressed as X/Y where X is the sheet page number and Y is the total number of sheets in the design.
@FILENAME	The current design's filename.
@PATHNAME	The full path and filename of the current design file.

The Windows long and short date formats and Windows time formats are set using the *Regional Settings* or *International* applets in Windows *Control Panel*.

Note that the above keywords must appear at the start of a 2D text string and that the string should not contain additional text. For example, do not place strings of the form:

```
AUTHORED BY @AUTHOR ON @WS_MDATE
```

as these will not work! To achieve the above you need to place four strings along side each other (AUTHORED BY, @AUTHOR, ON and @WS_MDATE).

By use of the 2D graphics objects and such special text objects it is possible to define any kind of header block you want. In particular, you can incorporate your company logo into the

header. Once defined, the header can be placed onto each sheet of the drawing like any other graphics symbol.

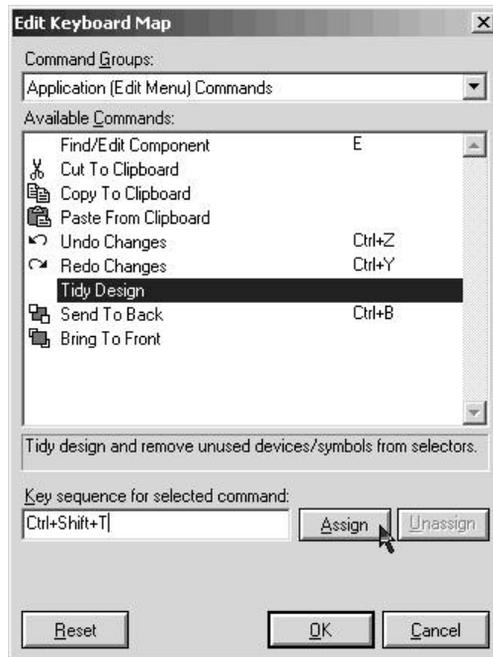
Send to Back / Bring to Front

Sometimes several objects (especially Graphics) overlap and it becomes difficult to point at the one you want. By default, ISIS takes the most recently placed object, but you can re-arrange the order using the *Send to Back* and *Bring to Front* commands. Each command operates on the currently tagged objects.

You can also use these commands to order the draw order when creating new devices. For example, if you are creating a new op-amp symbol, you may need to send the op-amp body (the triangle graphic) ‘to the back’ in order to ensure it’s fill pattern doesn’t obscure the ‘+’ and ‘-’ input symbols, which of course, need to be ‘in front’.

Keyboard Configuration

ISIS provides an interface for you to map the keyboard to a command set. This allows you to ‘bring out’ the functionality you commonly use according to personal preference. The configuration dialogue, accessible via the *System Menu – Set Keyboard Mapping* command is shown below (context sensitive help is available for all fields within the dialogue itself):



Notes :

- Accelerators can use any combination of CTRL, SHIFT and ALT keys. For example, CTRL+X, CTRL+ALT+T,SHIFT+CTRL+I, etc.
- A *Reset* option is provided to reload all Proteus standard shortcuts. This will overwrite any configuration you currently had.
- Some European keyboards have an ALT GR key (normally found directly to the right of the space bar). Where present this translates to be CTRL+SHIFT for the purposes of assigning keyboard shortcuts.
- If you assign an ALT accelerator that is used as a menu accelerator then you will no longer be able to accelerate the menu from the shortcut. For example, ALT+T by default expands the *Tools Menu* within ISIS. If you assign ALT+T to the *Tidy command* then you would have to have to access the *Tools Menu* using the mouse.
- Some commands are locked and cannot be assigned. This is particularly true of those on Maximised Graphs.
- If you assign a keyboard shortcut (e.g. CTRL+K) in the Application and then assign the same shortcut in a simulation popup window then the functionality executed by the shortcut is dependant on the focus at the time you use the accelerator. To avoid confusion we recommend you avoid multiple assignments of the same accelerator combination.
- When the NUM LOCK is on the keys 0-9 on the numeric keypad are distinct from the 0-9 keys on the main keyboard. For example, you could assign CTRL+0 up to CTRL+9 on the main keyboard as shortcuts to the Most Recently Used file list (MRU list) and on the numeric keypad you can assign CTRL+NUM-0 up to CTRL+NUM-9 to Goto Sheet 0-9.
- When the NUM LOCK is off keys 0-9 on the numeric keypad action their secondary functions and these functions are considered identical to those on the main keypad. The key for the number 5 on the numeric keypad has no secondary function when NUM LOCK is off.
- You can save your custom keyboard configuration via the '*Export Keyboard Map*' command on the *Options* tab. This allows you, for example, to port your keyboard configuration to another computer. Similarly, the '*Import Keyboard Map*' command allows you to reload a custom keyboard configuration that has previously been saved.

GRAPHICS AND TEXT STYLES

INTRODUCTION

ISIS implements a sophisticated scheme to allow you to customise the appearance of a schematic in terms of line styles, colour fills, text fonts, text effects, etc. The system is extremely powerful and allows you to control some or all aspects of the schematic appearance globally whilst allowing certain objects to carry their own local appearance attributes.

All graphical objects in ISIS (component bodies, wires, junction dots, etc.) are drawn according to a *graphics style*. A graphics style is a complete description of how to draw and fill a graphical shape (such as a line, box, circle or whatever) and includes attributes for line styles (solid, dotted, dashed etc.), thickness, colour, fill style, fill foreground and background colour, etc. Similarly, all labels and script blocks in ISIS (terminal labels, pin names, etc.) are drawn according to a *text style*. A text style is a complete description of how to draw some text and includes attributes for the font face (e.g. Arial, Times Roman, etc.), character height, width, colour, etc.

In ISIS, most objects, such as 2D graphics, wires, terminal labels, etc., each have their own *local* style in order that they can be customised on an object-by-object basis - that is, for example, that one wire can have a different appearance to another wire. The term *local* is used as the style settings are local to the object. Other objects such as pin names, sub-circuit bodies, etc. are always drawn in one of the predefined styles and therefore these objects can only be customised on an all-or-nothing basis - that is, for example, sub-circuits can have whatever appearance you want, but all sub-circuits must appear the same.

Most objects that have their own style have their *local* style initialised from the most appropriate of the *global* styles when they are placed. For example, when you place a terminal, its label's text style is automatically initialised with the *TERMINAL LABEL* style and when you place a wire its graphics style is automatically initialised with the *WIRE* style. 2D Graphics Objects are slightly different in that, for these objects, the right-hand *Object Selector* displays a list of available graphics styles and the newly-placed graphics object is initialised with whatever style is currently selected.

Now comes the clever part. Each *local* style keeps a track of the *global* style used to initialise it. In addition, each such local style also has a set of *Follow Global* options with one such option for each attribute in the style. The *Follow Global* option, when selected, indicates that the associated style attribute should always assume the value of the *global* style and, when not selected, that the value of the local style attribute be used. By default, for all newly-placed

objects, all the *Follow Global?* options are selected such that the appearance of the newly-placed object fully and completely follows the global style it is based on.

The benefits of having local and global styles and the ability of local styles to follow some or all the attributes of a global style are that:

- it allows you to edit the overall appearance of a design via a single edit of the global style - you do not need to edit all objects concerned individually.
- it allows you to define library symbols which will automatically 'blend in' with the appearance of the drawing onto which they are placed.
- it allows you to 'fix' some or all parts of the appearance of a component or other object.

For example, suppose you create a new component and put it in a library. Providing you draw the component's graphics in the *COMPONENT* style, then when the component is subsequently loaded from the library in to a new drawing, it will automatically take on and follow the *COMPONENT* style for that drawing.

TUTORIAL

As a starting point for the tutorial on graphics and text styles, load the sample design *STYLETUT.DSN* from the "*Samples\Tutorials*" directory of your Proteus installation. As you can see, the basic colour scheme in this design is blue outlines with yellow fills for components and red for terminals, etc. and if you zoom in about some text, you will see that it is all in the Labcenter Electronics 'vector' font.

Editing Global Styles

We will begin by looking at how to edit the global styles. These styles are, as their name implies, global to the design and editing these styles allows you to make global changes to the appearance of a schematic.

Let us start by changing the component colour scheme a little. All the components in the design were picked from the standard libraries and as such, the component graphics are all drawn in the *COMPONENT* style. We can change this style using the *Set Graphics Styles* command on the *Template* menu. Select this command and you will see the *Edit Global Graphics Styles* dialogue form displayed. This form gives access to all the graphics styles defined for the current design - drop the *Style* selector down to see the full list. As the buttons below the *Style* list indicate, it is possible to create, edit and delete new styles in addition to the editing the 'predefined' styles.

Make sure the *COMPONENT* style is selected in the *Styles* list. You will see in the *Line Attributes* and *Fill Attributes* sections the attributes for the style. Under *Line Attributes*

select a red colour and then under *Fill Attributes* select a *Fill Style* of 'none'. After both changes you will see that the sample at the right of the dialogue form updates to show a new preview of the style.

Now select the *TERMINAL* style in the *Styles* list. The changes you made to the *COMPONENT* style are automatically saved and the dialogue changes to display the settings of the *TERMINAL* style. This time, under *Line Attributes* select a blue colour for lines and then under *Fill Attributes* select a *Fill Style* of 'solid' which enables the *Fg. Colour* control where you can select a yellow fill colour.

Close the form using the *Close* button and the schematic is redrawn to show the changes. The new colour scheme would probably work better on a white background and as you will probably print your designs on white paper, let's now select white as our 'paper' colour. From the *Template* menu select the *Set Design Defaults* command. The *Edit Design Defaults* dialogue form allows you to edit most of the standard colours used by ISIS including the background or 'paper' colour. Change this to a light grey or white and close the form.

So we've seen how to change graphics styles. How about text styles? Changing these is pretty much the same as changing text styles except that style attributes you get to change are different.

However, before we start changing the text styles of individual elements, lets change the text font used in the drawing 'en mass'. As you've already noted, the drawing is currently displaying all its text in the Labcenter Electronics 'vector' font. To change the default design font, again select the *Set Design Defaults* command from the *Template* menu to display the *Edit Design Defaults* dialogue form. Under *Font Face For 'Default' Font* you will see *Vector Font* selected - change this to the standard Windows *Times New Roman* font face and close the form. The design is redrawn and all text can now be seen to be in *Times New Roman*.

Wherever ISIS offers a list of TrueType™ fonts (e.g. as part of editing a text style) there are always two additional font faces listed at the top of the list. These are *Default Font* and *Vector Font*. The *Default Font* option is a place holder for whatever font is selected on the *Edit Design Defaults* dialogue form whilst the *Vector Font* option selects the internal Labcenter Electronics *Vector Font*. In the case of STYLETUT.DSN all the text and labels in the design has been placed and/or edited to use the *Default Font* face which means that it all displays in the font set on the *Edit Design Defaults* form - select a new font face there and all the text in the design is changed immediately!

The main use of the *Vector Font* is where you need to both produce hard copy output on a pen plotter. Windows does not properly support the use of TrueType fonts on plotters, and the *Labcenter Plotter Driver* will output all text in the *Vector Font* whatever it is defined as on the drawing. The *Vector Font* has the additional advantage that it is guaranteed to come

out *exactly* the same size on all hard copy devices as it does on the screen. This is not always the case with TrueType fonts.

On the other hand, if you only need to use bitmap devices (i.e. printers) these issues do not arise and you can use all your TrueType™ fonts as you see fit. It has to be said that schematics which use more than two font faces are likely to look very odd!

Having changed our designs 'default font' to be *Times New Roman* lets make some object specific text style changes. From the *Template* menu, select the *Set Text Styles* command to display the *Edit Global Text Styles* dialogue form. Notice how similar this dialogue form is to the *Edit Global Graphics Styles* dialogue form you saw earlier - as we said, editing text styles is very similar to editing graphics styles.

By default, *COMPONENT ID* should be the *Style* selected for editing and you will see that the *Font Face* is set to *Default Font* which is, as we explained, the placeholder for the font selected on the *Edit Design Defaults* dialogue form, currently *Times New Roman*. Because of this, some fields on the dialogue form that aren't appropriate to a TrueType™ font, such as character width, are greyed out. Change the *Font Face* to be *Courier New* and select *Bold?* under the *Effects* options. Now select the *PIN NUMBER* style in the *Style* list (the changes to the *COMPONENT* style are automatically saved) and then uncheck the *Visible?* checkbox under the *Effects* options. Use the *Close* button to close the dialogue form.

You will see that on the redrawn schematic the component IDs are now in bold *Courier New* and that the pin numbers of the OP-AMP are no longer visible. The latter effect is often useful when producing drawings for illustration or as block diagrams.

Editing Local Styles

So far we have only changed 'global' styles. Changing these has had an effect right across the schematic. We will now look at making changes to 'local' styles by editing the op-amp's **U1** component reference label.

Lets start by taking a look at a local style: click right on the **U1** label to tag it (and the op-amp) and then click left on it to edit it causing the *Edit Terminal Label* dialogue form to be displayed. Finally, select the *Style* tab to see the 'local' style settings. You will see the following: a *Global Style* selector at the top, a set of style attributes (*Font face*, *Width*, etc.) to the left and for each attribute a *Follow Global?* check box to the right. Both text and graphics local styles have this format. Wherever a *Follow Global?* checkbox is checked, the associated style attribute is initialised from the global style indicated in the *Global Style* selector. Unchecking the *Follow Global?* checkbox enables the dialogue control for the associated style attribute allowing you to specify a local (object-specific) value for that attribute. Not all the *Follow Global?* checkboxes are enabled as, in this case, the *Width* attribute is meaningless for a TrueType™ font face.

The change we are going to make is to make the **U1** label a different colour. Start by unchecking the *Follow Global?* checkbox to the right of the *Colour* control and you will see the *Colour* control is enabled and displaying the current colour set in the whatever global style this local style is following (in this case, the *COMPONENT ID* style). Change the colour to be dark blue, then close the form and untag the op-amp so that it is redrawn in its normal colours. The **U1** label is now dark blue.

To verify that the label's colour is indeed now independent of the global style but that the remainder of the attributes are still following its global style, select the *Set Text Styles* command on the *Template* menu again, change the colour of the *COMPONENT ID* style (selected by default) to magenta, check (enable) the *Italic* option and then close the form. The op-amp's component label remains dark blue as its colour is no longer following the global *COMPONENT ID* style however its *Italic* attribute is still following the global style, so it is now italicised. All the other components' IDs, which have remained following the global *COMPONENT ID* style, have both changed colour and become italicised.

For completeness, we will now show you how to edit a local graphics style. Select the *Box* icon. Now make sure the *COMPONENT* style is selected and then, placing the mouse at the top left of the circuit, click left and drag out a box that surrounds (and covers!) the circuit diagram. We now need to change the local style of the box to be transparent. Tag the box and then click left on it to edit it - the *Edit Box's Graphic Style* dialogue form is displayed. Apart from the different style attributes, notice how this form is similar to that of the *Edit Terminal Label* dialogue form's *Style* tab. The operation of this form is the same as the operation of that form. Uncheck the *Follow Global?* checkbox for the *Fill Style* control and then, it now being enabled, change the fill style to 'none'. Close the form with the *OK* button and untag the box.

This is a simple example of the power of local and global styles though in general use of ISIS you will never need to modify a local style. The main use of local styles is with 2D graphics objects (lines, boxes, etc.) when creating a new library part and even then it is recommended that you avoid setting up local styles unless absolutely necessary. As you have seen, once an object has a local setting its appearance is fixed unless it is re-edited and, when applied to new library parts, this means that some or all of a library part will not blend in to the drawing in which it is used. In proper use, this is usually the intention - for example, that the base of a transistor always have a solid fill regardless of the fill selected in the global *COMPONENT* graphics style. However, when misused, it can lead to library parts that simply don't port between different colour schemes or users.

Working With The Template

We will finally touch on the subject of templates. All the information about text styles, graphics styles, junction dot shape and size, paper colour, etc. - in fact everything you can

change via the *Template* menu commands - is called a design template. When installed, ISIS comes with a default template that is stored in the file **DEFAULT.DTF** in the library directory of your Proteus installation. When ISIS is started or you select the *New* command, ISIS loads its initial template settings from this file and any modifications made to the template settings are only saved back to this file if you select the *Save Default Template* command from the *Template* menu. The existing template is not backed-up so don't *Save Default Template* unless you particularly want to reuse the template settings in the future (for example, you have a new 'house' style you want all your designs to conform to).

For portability and consistency, ISIS also stores a copy of the current template in schematic design files and recovers it when the file is loaded. You've already seen this behaviour in loading the tutorial design and if you (later) load some of the other sample designs supplied with ISIS you will see that they employ a wide variety of styles.

So what happens if you set up a 'house' style, use *Save Default Template* to make it the default template, design and save lots of schematic designs, and then decide to change the 'house' style? The simplest answer is to use either the *Apply Template From Design* or *Apply Default Template* command from the *Template* menu. Both these commands load a new template in to the current design, either from another design file or from the default template file though note that objects with local style settings will continue to use them. In this way you can set up a new default template and then apply it to all your old designs. Alternatively, if you lose your default template, you can recover it from an old design and then save it using the *Save Default Template* command.

TEMPLATES AND THE TEMPLATE MENU

All the information that controls the appearance of a schematic - graphics styles, text styles, design colours, wire junction dot size and shape, etc. - is termed the *template*. All the commands for modifying the template are on the *Template* menu are described subsequently.

Note that any changes to the template only affect the currently running copy of ISIS though they will be saved and preserved in any schematic design saved. To make the changes available the next time you start a design you need to use the *Save Default Template* command on the *Template* menu to update the default template.

INTRODUCTION

ISIS makes extensive use of the concept of properties. A property is considered to consist of a *keyword* which identifies a particular property and a *value* which is assigned to that property for a particular object. For example, in conjunction with ARES, we make use of a property called **PACKAGE** which contains the PCB footprint.

Properties can be associated with objects, sheets and the design itself and the relationships between the various types need to be clearly understood if you are to get the best from what is an extraordinary powerful scheme, unmatched by any other schematics software we have seen.

OBJECT PROPERTIES

There are two distinct types of object property - system properties and user properties. The former comprise a set of reserved keywords which have special functions within ISIS itself, whereas the latter relate either to external programs such as ARES and ProSPICE, or may relate to your own particular use of the software.

System Properties

System properties are properties whose keywords have special meaning within ISIS. For example, the **DEVICE** property of a component object directly determines the library part assigned to it. Some of these properties are textual - e.g. component **REF** and **VALUE** labels are directly accessible from the *Edit Component* dialogue form, but others such as the aforementioned **DEVICE** property are manipulated as a result of graphical operations.

In general, you need only concern yourself with system properties if you wish to read their values with the search and tag commands, or modify them with the *Property Assignment Tool*. For example, you might wish to tag all the 7400 components in your design. This requires you to know that the system property that will hold the library part name is called **DEVICE**.

 Details of the system properties used by each type of object are given in *Object Specifics* on page 73.

User Properties

Components, sub-circuits and VSM gadgets can carry an unlimited number of extra properties, in addition to their standard system properties. These user properties are held in a block of text known as a *property block* and consisting of lines such as:

```
SUPPLIER=XYZ Electronics
```

You can edit these user property blocks directly on the dialogue forms of the various objects, as well as manipulating them with the *Property Assignment Tool*.

To edit an objects user properties

1. Bring up the object's dialogue form by tagging it and clicking left.
2. If the object can hold user properties, the dialogue form will have a text editor box labelled *Properties*. Point below any existing text in this box and click left.
3. Edit the text as required. Each property should consists of a single line comprising a keyword and value separated by an equals sign (=).

User property keywords should, as a rule, consist of letters, numbers and underscores only. *Under no circumstances should they contain spaces, commas, double quotes or equals signs (, " =).*

In accordance with the general behaviour of ISIS text, if a property assignment is enclosed in curly brackets ('{' and '}') then it will not be displayed on the screen. For example, entering:

```
{PRIMITIVE=DIGITAL}
```

will define an object as requiring digital simulation, but this text will not actually appear.

Occasionally, one may want just the value to appear, in which case you can do:

```
{MODFILE=}OPAMP
```

In theory you can put curly brackets in other places. However, when the *Property Assignment Tool* modifies property blocks, it works on the assumption that if used at all, curly brackets have been placed as shown in the above examples. If they have been placed elsewhere, then you may get unexpected results.

Property Definitions (PROPDEFS)

It is possible to provide extra details for specific user properties of a device. For example, common user component properties are **PACKAGE** and **MODFILE**. Given that appropriate Property Definitions have been created for the component libraries, these properties are displayed in their own fields on the *Edit Component* dialogue form. The property definitions include a description of the property, a data type (such as integer, floating point or string)

and for numeric data types a valid input range. A default value may also be (and often is) specified.

This scheme makes it much easier to see what properties are valid for a particular model, and what those properties mean. It is also then possible to supply alternate package types and simulator models in a way that is obvious to the end user of a particular library part.

For further information on how to create and manage property definitions for your own library parts, see page 70.

Properties which are not known for a particular device (or all properties for a device which does not have property definitions) still appear in the textual properties block as described above, and the scheme is thus backward compatible with old designs and/or library parts which do not have property definitions.

SHEET PROPERTIES

Introduction

Each sheet of a schematic can hold a set of property assignments. These may be considered as defining constants (either numeric or textual) which may be used within object property assignments on the particular sheet. In themselves they are not terribly useful, but their real power comes to light when they are used in the context of object property expressions.

For example, if a sheet property is defined in a block such as:

```
*DEFINE  
PI=3.142
```

you could then define a resistor as having the value:

```
VAL=EVAL(500/PI)
```

At netlist time, this syntax causes the netlist compiler to evaluate the property, and the resistor's value will appear in the netlist or bill of materials appear as 159.134.

 For further information about property expressions see *Property Expression Evaluation* on page 64.

Defining Sheet Properties

Sheet properties can be defined in the following ways:

- Directly, using a **DEFINE** script block. You can use this to define constants for use in expressions, as in the above example.

- As a result of parameter mapping using a **MAP ON** script block. In this case, the value of the parent property specified by the **MAP ON** statement is used to select a set of sheet property definitions from a table. This is most commonly used for creating universal simulation models, in which several similar devices are modelling using the same circuit but with different sets of sheet properties for each device type.

Further discussion of this is given in the Proteus VSM manual.

- By inheriting the properties of a parent object. In other words, if the parent object has the object property assignment:

R3=10k

then the child sheet will automatically acquire this as a sheet property. This provides the basis for parameterized circuits (section [*Parameterized Circuits*]) in which several instances of a given hierarchy module can be assigned different component values.

If the same property is defined both on a child sheet in a **DEFINE** or **MAP ON** block and in the parent object property block, the value from the parent will be used. This provides the means to provide default values for sheet properties which can be overridden as required.

Scope Rules for Sheet Properties

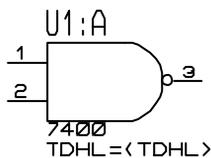
It is important to realise that sheet properties may only be referenced on the sheet for which they are defined. In particular, the sheet properties on a parent sheet are not accessible by any of its children unless they are passed though the parent object property block. If you do need to access a property in this manner, you can add a line such as:

```
TDHL=<TDHL>
```

to the appropriate parent object(s). If **TDHL** is defined as a sheet property on the parent sheet, it will then become an object property of the parent object, and thus be defined as a sheet property for the child sheet, where it can in turn appear in further object property expressions.

This arrangement is analogous to passing parameters in a C computer program.

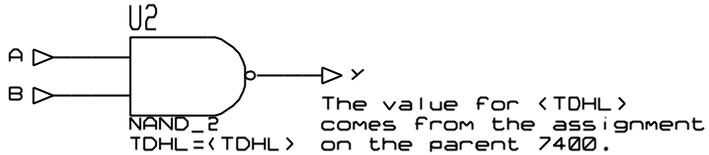
PARENT SHEET



```
*DEFINE  
TDHL=10n
```

The value for <TDHL> here comes from the sheet property defined in the *DEFINE block.

CHILD SHEET



DESIGN PROPERTIES

The design properties for a given schematic are determined by accumulating all the sheet properties that are declared on the root sheets of the design. Since root sheets cannot have parents, it follows that design properties can only be declared using **DEFINE** script blocks on the root sheets.

For SDF netlist output, the design properties appear in the **PROPERTIES** block of the netlist, and may be interpreted by whatever application is reading the netlist. In the case of Proteus VSM, design properties are used to define simulator options such as the number of steps, the operating temperature and so forth. The specifics of this are given in the Proteus VSM manual.

To create a list of design properties:

1. Go to the root sheet of your design by selecting the *Goto Sheet* command from the *Design* menu.
2. Select the *Script* icon from the *Mode Selector* toolbar.
3. Point where you want the property definition block to appear and click left.
4. Type in the first line of the script as


```
*DEFINE
```
5. Type in the property assignments as required.

Note that design properties are also sheet properties for the sheets on which they are defined. However, the standard scope rules for sheet properties still apply, and design properties are not accessible in property expressions on other sheets.

PARAMETERIZED CIRCUITS

Introduction

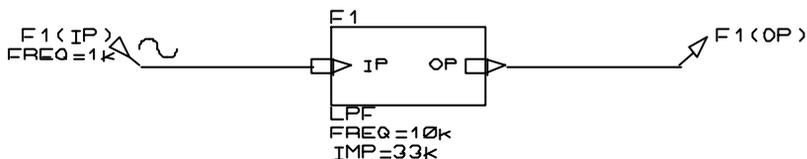
ISIS has a unique and tremendously powerful feature which combines sheet properties, object properties and hierarchical design to facilitate parameterized circuits. A parameterized circuit

is one in which some of the component values (or other properties) are given as formulae rather than as constant values. Naturally, the formulae contain variables or *parameters* and the values for these are taken from the sheet properties defined for that particular instance of the circuit. It then follows that, in the context of a hierarchical design, different instances of the circuit can have different parameters and therefore different component values.

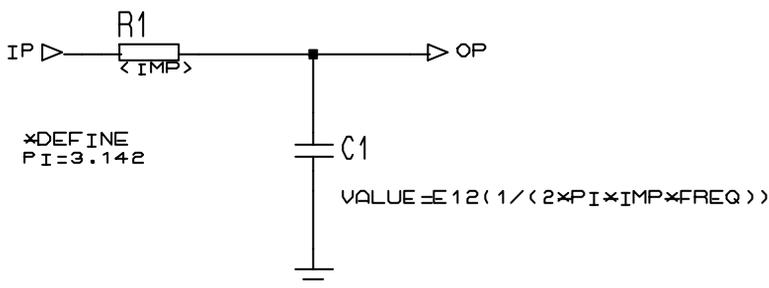
📖 For more information about hierarchical design see *Hierarchical Designs* on page 129. If you have no idea what a hierarchical design is, we suggest you skip this section entirely for the time being!

An Example

Parameterized circuits are best illustrated by an example design which you will find as LPF.DSN in the "Samples" directory of your Proteus installation. The root sheet of this design is shown below:



It consists of a single sub-circuit which has two user properties defining the desired frequency and impedance for the filter. The module attached to the sub-circuit is the actual parameterized circuit:



There are several points to note about it:

- The **DEFINE** block defines one sheet property: **PI**. This serves as a constant in the property expression for the capacitor value.
- The resistor's value field contains the string '<IMP>'. This syntax with the chevrons (<>) causes the netlist compiler to substitute the '<IMP>' with value of the **IMP** sheet

property, that is 33k. In this case, no expression evaluation is performed - the substitution is purely literal.

- The capacitor has a user property assignment (in its property block) which specifies its value as an expression. The **E12** function specifies both that the expression should be evaluated by the netlist compiler, and also that it should be rounded to the nearest E12 value. The other options are **EVAL** (no rounding) and **E24** (round to **E24** value).

PI, **IMP** and **FREQ** are all sheet properties. **PI** comes from the **DEFINE** block whilst **IMP** and **FREQ** come from the parent sub-circuit.

If you generate the Bill of Materials, you will see the following:

QTY	PART-REFS	VALUE
---	-----	-----
Resistors		

1	R1	33k
Capacitors		

1	C1	470p

ISIS has evaluated $1/(2*3.142*33000*10000)$ to get approximately 0.00000000482 and then rounded this to the nearest E12 value - 470p.

There are in fact two distinct processes going on in the above example:- *property substitution* and *property expression evaluation*. Both have their advantages and disadvantages and the following sections discuss each in more detail.

Property Substitution

This is the mechanism that is used for the resistor value and will operate whenever the netlist compiler encounters a property value containing a property keyword enclosed in chevrons (<>). If the keyword matches a sheet property, then the expression in chevrons is replaced with the value of the sheet property. If no sheet property exists, then a netlist warning is generated and the property is removed from object.

There are two main cases where property substitution is useful:

- You can use it in a parameterized circuit in which the parameters are not numeric. Packages for PCB design are perhaps the most common example of this - it is all very

well getting ISIS to compute that a 470pF capacitor is to be used, but you may still need to package it for PCB design. If you attach to the capacitor the user property:

```
PACKAGE=<C1_PACKAGE>
```

then you can add the property:

```
C1_PACKAGE=CAP10
```

to the sub-circuit. When netlisted, the C1 will appear with the property

```
PACKAGE=CAP10
```

Property expression evaluation cannot be used for this, because CAP10 will not evaluate as a number.

- The other major use for property substitution is for setting up sweep analyses with Proteus VSM. In this case, you want the simulator to evaluate the expressions, not ISIS and it may then be appropriate to build up component properties using property substitution rather than property expression evaluation. Further discussion of this is given in the VSM manual.

Property Expression Evaluation

In contrast to property substitution which is a purely textual process, property expression evaluation involves ISIS in numerically evaluating a property value that is in the form of a formula, and replacing it with its value. In addition, ISIS can also round the resulting number to an E12 or E24 series value.

There are three forms of syntax:

```
EVAL ( . . . )
```

```
E12 ( . . . )
```

```
E24 ( . . . )
```

In all cases, the parentheses should contain a mathematical expression which may include the operators plus (+), minus (-), times (*) and divide (/) and values. Values can be constant numbers, or the names of sheet properties. Multiplication and division have higher precedence but further levels of parentheses may be used to override this as required.

Some example expressions are shown below:

```
EVAL ( 1 / ( A+B ) )
```

A and B are sheet properties.

```
E12 ( 20k+2 * F * PI )
```

20k automatically treated as 20000.

```
E24 ( 3+4 * 5 )
```

Evaluates to 24.

Although in some ways more powerful than property substitution, there are some limitations:

- The evaluator can only handle numeric values - expressions involving strings are not allowed.
- You can only reference sheet properties in the formula - you cannot access other object properties or the values of other components.
- There is **no** support for mathematical functions (e.g. sin, cosine, square-root) etc.

We may eliminate some or all of these deficiencies in a future version.

The Rounding Functions E12 (), E24 ()

The property expression evaluation mechanism supports the ability to round the resulting value to the nearest E12 or E24 value. This prevents parameterized circuits ending up with non-available or multi-digit floating point values.

You should note:

- Rounding will be disabled for expressions that yield zero or negative values. This is unlikely to be a problem as negative valued resistors and capacitors are hard to come by anyway.
- The rounding is done on a geometric rather than an arithmetical basis so the mid-point between 3k3 and 4k7 is taken to be approximately 3.94. This, we feel, is in keeping with the thinking behind the E12 and E24 series themselves.
- Bear in mind that if a parameterized circuit contains several rounded values, there is no mechanism to round them all in the optimum directions, taking into account the way they may interact. This means that for critical filter designs and such like, you may be better computing the values manually (considering the various possible mixes of values) and using parameter substitution to load your chosen values directly into the circuit.

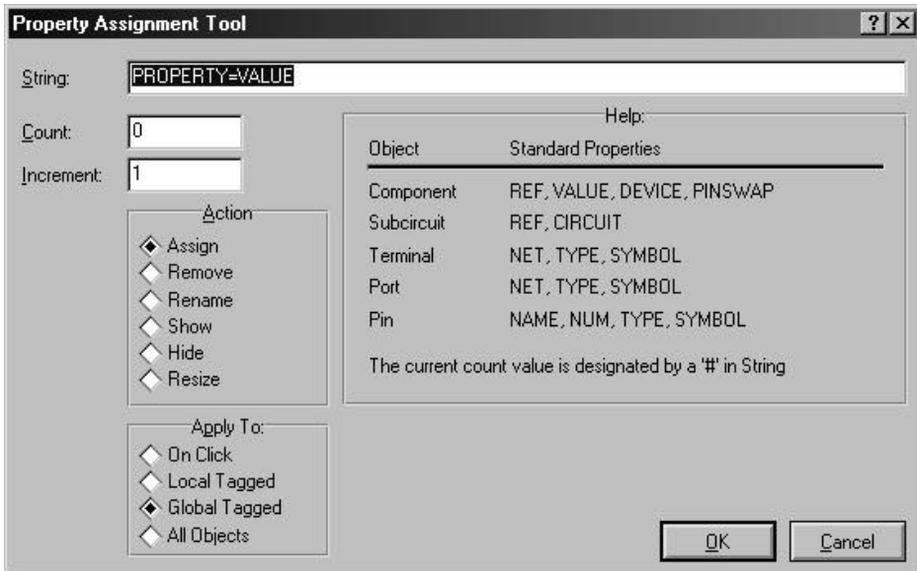
Of course, if you have Proteus VSM, you can run a simulation and see how the values ISIS has chosen affect the circuit performance.

THE PROPERTY ASSIGNMENT TOOL

The *Property Assignment Tool* (PAT) is a feature which enables you to assign, remove, show and hide object properties belonging selected objects in your design.

The PAT Dialogue Form

The PAT is operated from a modestly complex dialogue form with the following fields:



- String** The property assignment, or property keyword on which the selected action will be performed for each object.
- Count** The initial value for the counter. The counter is incremented on each application of the PAT. The current value of the counter may be included in the string by entering a hash ('#') character.
- Action** The action you wish to perform. See *Pat Actions* on page 66 for descriptions the various types of action.
- Apply** The application mode in which you want the PAT to operate. See *Pat Application Modes* on page 68 for descriptions the various types of action.

PAT Actions

The *Property Assignment Tool* can perform the following actions:

- ASSIGN** The string should contain a property assignment of the form:

keyword=value

and this property will be assigned to the selected objects.

If you want to assign values that run in a sequence, such as D0, D1, D2 etc. then use the hash (#) character in the value and set the initial count as required.

Both user and system properties may be assigned; assigning system properties may cause graphical changes to your drawing.

REMOVE

The string should contain a property keyword only and this property will be removed from the selected objects.

Only user properties can be removed.

RENAME

The string should contain both a property assignment of the form:

```
current_keyword=new_keyword
```

The name to the left of the assignment is the existing property name you wish to rename; the name to the right is the new name you wish to rename it to.

Only user properties can be renamed.

SHOW

The string should contain a property keyword only and this property will be made visible for the selected objects.

All user properties and system properties can be shown.

HIDE

The string should contain a property keyword only and this property will be made invisible for the selected objects.

All user properties and system properties can be hidden.

RESIZE

The string should contain an assignment such as

```
REF=20,16
```

and the given property will be assigned a new height and width for the selected objects. Only textual system properties may be resized.



Changing a labels text size implicitly sets the *Height* and *Width* attributes of the labels text style to be local and that, thereafter, changes to the global text style for the label will not be reflected in these labels.

PAT Application Modes

The selected PAT action can be applied to the design in the following ways:

ON CLICK

On selecting the OK button, the *Instant Edit* icon selected. Each object on which you click left has the selected action performed upon it.

When you select a different icon, the PAT is cancelled.

This mode is the only way to apply the PAT to wires, in order to assign wire labels. ISIS cannot perform PAT assignments on tagged wires because there would be no way for it to know where to put the wire labels.

LOCAL TAGGED

The selected action is performed on all tagged objects on the current sheet only.

You can select the tagged objects either individually, or with the *Search & Tag* commands.

GLOBAL TAGGED

The selected action is performed on all tagged objects right across the entire design.

You can select the tagged objects either individually, or with the *Search & Tag* commands.

ISIS is capable of remembering different tag states for different instances of an object in a hierarchical design, should this be an issue. However, the user property blocks are shared between instances so you cannot change a user property in just one instance.

ALL OBJECTS

The selected action is performed on all the objects in the design.

The Search and Tag Commands

The search and tag commands facilitate the selection of particular groups of objects for subsequent processing by the *Local Tagged* or *Global Tagged* options of the PAT.

There are three search commands:

SEARCH

This is the normal search operation which you should use to begin all new search tasks. It sets the tags of objects which meet the search condition and clears the tags of objects which do not.

AND SEARCH

This search operation can be used to eliminate objects from the currently tagged set. It clears the tags of all objects which do not meet the search condition, and leaves alone the tags of objects which do.

OR SEARCH

This search operation can be used to include further objects into the currently tagged set. It sets the tags of all objects which meet the search condition, and leaves alone the tags of those which do not.

Examples

The *Property Assignment Tool* and *Search & Tag* commands provide great power and flexibility when it comes to manipulating object properties. However, they can seem somewhat daunting to beginners. With this in mind, some step by step examples to get you started are given overleaf.

To label a set of bus taps

1. Invoke the PAT by keying 'A'.
2. Set the string to NET=D# and click OK. The action will have defaulted to *Assign* and the mode to *On Click*.
3. Click left on each wire where you want the wire label to appear. The cursor keys and the enter key can be used as an alternative to the mouse. The wires you click on will be assigned new wire labels in the sequence D0, D1, D2, etc.

To assign a package to all the BC108s in a design

1. Invoke the *Search and Tag* command by keying 'T'.
2. Set the property to VALUE and the string to BC108 , then click OK. The mode will have defaulted to *Equals*. All the components with the value BC108 will be tagged.
3. Invoke the PAT by keying 'A'.
4. Set the string to PACKAGE=TO18 and click OK. The action will have defaulted to *Assign* and the mode to *Global Tagged* (assuming there were tagged BC108s). All the tagged BC108s will be given the new user property.

To rename all ITEM properties to CODE properties:

1. Invoke the PAT by keying 'A'.
2. Set the string to ITEM=CODE, the action to *Replace* and the mode to *All Objects*, then click OK. All objects with the property ITEM=*value* will be replaced with a property CODE=*value*.

To hide all the package properties

1. Invoke the PAT by keying 'A'.
2. Set the string to PACKAGE, the action to *Hide* and the mode to *All Objects*, then click OK. All the **PACKAGE** properties will be hidden.

To resize the component references

1. Invoke the PAT by keying 'A'.
2. Set the string to REF=10 , 8, the action to *Resize*, and the mode to *All Objects*, then click OK. All the component references will be shrunk to the new size.

To assign larger packages to 1000uF capacitors

1. Invoke the *Search & Tag* command by keying 'T'.
2. Set the property to DEVICE and the string to CAP ELEC, then click OK. This will tag all the electrolytic capacitors.
3. Invoke the *AND Search* command from the Tools menu.
4. Set the property to VALUE, the string to 1000u, and the mode to *Begins*, then click OK. This will take care of capacitors with the values 1000u or 1000uF.
5. Invoke the PAT by keying 'A'.
6. Set the string to PACKAGE=ELEC-RAD30 and click OK. The action will have defaulted to *Assign* and the mode to *Global Tagged* (assuming there were tagged capacitors). All the tagged capacitors will be given the new package.

PROPERTY DEFINITIONS

Creating Property Definitions

Property definitions are entered using the *Component Properties* page in the *Make Device* dialogue form. For more detailed information use the context sensitive help on the dialogue form and/or refer to the instruction on making a device on page 108.

Default Property Definitions

A number of properties will be applied to most devices that you create. For example, anything that is going onto a PCB will need a **PACKAGE** property, and anything that has a simulator model will need a **MODFILE**, **MODEL** or **SPICEMODEL** property. You may also want to apply your own properties such as **STOCKCODE**, **SUPPLIER** or **COST** to most of the devices that you create.

To make this easier, there is a list of default property definitions which can be defined using the *Set Property Definitions* command on the *System* menu. The properties defined with this command are available on the *Name* field combo on the *Edit Device Properties* dialogue form.

The information manipulated by this command is held in the file PROPDEFS.INI located in library directory of your Proteus installation.

Old Designs

Note that components and library parts in designs created with PROTEUS versions prior to 4.5 will not contain property definitions. If you wish, you can have ISIS apply the default property definitions to such components 'on the fly' i.e. as they are edited.

To enable this feature, invoke the *Set Property Definitions* command from the *System* menu and enable the *Apply Default Properties to Components in Old Designs* checkbox.

OBJECT SPECIFICS

COMPONENTS

A component is an instance of a device, picked from one of the device libraries. Since some devices are *multi-element* it follows that in some cases, several components on the schematic may, in fact, all belong to one physical component on the PCB. In such cases, the logical components are annotated with names such as U1:A, U1:B, U1:C, U1:D to indicate that they all belong to the same physical part. This form of annotation also enables ISIS to select the correct set of pin numbers for each element.

Apart from physical terminals, components are the only objects you can place which will generate physical entities in the PCB design. Everything else you place serves either to specify connectivity, or else is present only for the purpose of human readability. This is an important point, since attempts to use other ISIS objects (in particular, graphics symbols) to represent objects on the PCB will fail.

Selecting Components from the Device Libraries

When you start ISIS on an empty drawing, the device selector is empty. Before you can place any components, you must first pick them from the libraries into the selector.

To select components from the device libraries:

1. Ensure that the *Object Selector* is showing devices by selecting the *Component* icon from the *Mode Selector* toolbar.
2. Click left on the **P** button on the *Object Selector*. This will cause the *Device Library Pick Form* to appear.
3. In the *Library* selector, select the library you wish to pick devices from.

The *Prefixes* selector will be displayed or hidden according to whether or not the library you have selected contains prefixes for its parts (see below).

The *Extensions* selector will be displayed or hidden depending on whether or not the parts in the library you select have two or more extensions in their names. An extension is any text at the end of the part name following a dot or full-stop and serves to differentiate different versions of the same device (e.g. normal, DeMorgan, IEC-617, etc. parts). Where the library only contains parts with out an extension or only contains parts with the same extension, the *Extensions* selector is not shown.

The *Objects* selector is updated to show the contents of the library according to the settings of the *Extensions* selector. Where the *Extensions* selector is hidden, all library parts will be shown.

4. Where the *Extensions* selector is shown, check or uncheck one or more of the extensions to show or hide parts with that extension.
5. Where the *Prefixes* selector is shown select the correct prefix for the part you require.
When you pick the part, the currently selected prefix will be inserted in front of the part name. For example, with the TTL.LIB library selected and the 74LS prefix selected in the *Prefixes* selector, picking a 249 part will in fact pick a 74LS249.
6. Single click a part in the *Objects* selector to browse the part in the *Browser* window or double-click a part in the *Objects* selector to pick it in to the design.
7. When you have finished picking devices, click the *Minimise* button of the pick form to

An alternative method for picking library parts is to use the *Pick* command, key 'P'. With this command, you can type in the name of the device you want directly.

 You can resize and position the pick form window and then save its position via the *Save Window Position* command on Windows 'system' menu (click on button on the far left of the window's title bar).

 Further information about libraries, and the procedures for making your own library parts is given in *Library Facilities* on page 101.

Placing Components

Component placement follows the general scheme outlined in *Object Placement* on page 33.

To place a component:

1. If the device type you want is not listed in the *Object Selector*, first pick it from the libraries as described in the previous section.
2. Highlight the device name in the *Object Selector*. ISIS will show a preview of the device in the *Overview Window*.
3. Use the *Rotation* and *Mirror* icons to orient the device according to how you want to place it.
4. Point at the position in the *Editing Window* where you want the device to appear, and click left. If you hold the mouse button down, you can drag the device around until you have it exactly where you want it. When you release the mouse button, it will be placed onto the drawing.

Replacing Components

Since deleting a component removes the wires attached to it, changing a component by deleting it and then placing a new one can be somewhat tedious. Instead ISIS provides a special mechanism to facilitate this type of operation.

To replace one component with another:

1. Pick the new device type and highlight its name in the *Object Selector* as described above.
2. Use the *Rotation* and *Mirror* icons to orient the new device according to how you want to place it.
3. Position the mouse pointer inside the old component, and then place the new one such that at least one of its pin ends touches one of the pin ends of the old component. *The mouse pointer must have been inside the old component when the new one was placed, if auto-replace is to be activated.*

ISIS will attempt to replace the old component with the new one whilst preserving as much of the old wiring as it can. It matches pins first by position, and then by pin name. Attempts to over-place wildly different parts are unlikely to give useful results, but the *Undo* command will recover the situation if such a replace is performed by accident.

 It is also possible to effect a replacement by assigning to the **DEVICE** property using the *Property Assignment Tool*. See *Properties* on page 55 for details of how to use this feature.

Editing Components

A component may be edited using any of the general editing techniques (see *Editing An Object* on page 36), and also specifically by using the *Edit Component* command on the *Edit* menu, key 'E'. As with most editing features in PROTEUS a dialogue form will appear with the appropriate editing fields. However, with the *Edit Component* dialogue form the available fields are dependant on the given properties of the component. You will find, therefore, that this dialogue form can be radically different for different components. Context Sensitive Help is available for the dialogue form but please note that, for the reasons given above, some fields may not have a help topic associated with them.

You can customise the appearance of components, pins, pin names, etc. by editing the associated global graphics and text style. See *Editing Global Styles* on page 50 for more information.

Component Properties

Components have the following system properties:

Property Name	Description
REF	Component reference designator label.
VAL	Component value label.
VALUE	Component value label or the VALUE user property if the assigned text is too long for the label.
DEVICE	Library part. If assigned, this will invoke the automatic replacement logic which will attempt to maintain connectivity.

If any other property names (e.g. STOCKCODE, TOLERANCE, etc.) are assigned, then these will create user properties in the component's text block.

Hidden Power Pins

If a component has been defined with hidden power pins, then the nets to which these will be connected can be viewed or edited by clicking the *Hidden Pins* button on the *Edit Component* dialogue form.

By default, hidden pins connect to a net of the same name - for example, a hidden VDD pin will be connected to VDD, a hidden VSS pin to VSS and so on.

DOTS

Junction dots are used to mark interconnection between wires. In general, ISIS will place and remove them automatically but it is sometimes useful to place a dot at a specific position and then route wires to and from it.

Wires which touch or cross are *never* considered connected unless there is a dot at the point of contact. Conversely, where there is a dot, there will always be a connection unless you have contrived to superimpose wires and dots by dragging or moving them into contact with each other.

 You can customise the appearance of junction dots by editing the *WIRE DOT* graphics style. See *Editing Global Styles* on page 50 for more information. You can also set their size and shape using the *Set Junction Dots* command.

Placing Dots

To place a dot:

1. Select the *Dot* icon from the *Mode Selector* toolbar.
2. Point where you want the dot to be placed in the *Editing Window*.
3. Click left to place the dot.

Auto Dot Placement

ISIS will automatically place a dot whenever you route a wire off an existing wire such that there are then 3 dots at that point.

Auto Dot Removal

When a wire or wires are deleted, ISIS will detect where this leaves dots with two or no connected wires. Such dots are automatically removed unless this would result in an unbroken wire loop.

WIRE LABELS

Wire labels are used to assign particular net names to groups of wires and pins, and also to assign *net properties* to specific nets. They are not in fact proper objects in the general scheme of the software. Instead, their behaviour is similar to other labels such as the reference and value labels of a component. It follows that the procedures for their placement and removal are somewhat different from those for other objects.

 See *Netlist Generation* on page 135 for information about the role of wire labels in netlisting and connectivity.

Placing And Editing Wire Labels

To place or edit a wire label:

1. Select the *Wire Label* icon from the *Mode Selector* toolbar.
2. If placing a new label, point at the wire at the position where you want the label to be placed or, for an existing label, point anywhere along the wire or at the label itself.
3. Click left to place the label. The *Create Wire Label* or *Edit Wire Label* dialogue form will be displayed.

 See *Editing Local Styles* on page 52 for more details of the controls on the *Style* tab.

4. Type in the required text for the wire label.
5. Click OK or press ENTER to close the dialogue form.

Note the following:

- You cannot place a wire label other than on a wire.
- You can place more than one label on a wire. If you wish them all to have the same name, and for all of them to update automatically whenever any of the names are changed, then click the *Auto-Sync* checkbox.
- ISIS will orient the wire label according to the orientation of the wire segment on which it is placed. This can be changed on the *Edit Wire Label* dialogue form.

To change the appearance of a wire:

1. Ensure the *Wire Label* icon is not selected.
2. Tag the wire by pointing at it and clicking right.
3. Click left on the tagged wire.

The *Edit Wire Style* dialogue box is displayed.

4. Uncheck the *Follow Global?* checkboxes of those style attributes of the graphics style you want to change. If a style attribute and its *Follow Global?* checkbox are both disabled it is because the style attribute is not meaningful given other attribute settings, either locally or through following the global style attribute.
5. Set the style attribute to the required setting.
6. Press ENTER or click the OK button to close the dialogue form and keep the changes. Press ESC or click the CANCEL button to close the dialogue form and abandon changes.

 To change the appearance of all wires on the schematic, use the *Set Graphics Style* command on the *Template* menu to edit the *WIRE* graphics style.

 See the section *Graphics And Text Styles* on page 49 for information on styles.

Deleting Wire Labels

To delete a wire label:

1. Tag the wire and wire label by pointing at either and clicking right.
2. Bring up the *Edit Wire Label* dialogue form by clicking left on the **label**.

Note that clicking left on the tagged wire or bus will, except when the *Wire Label* icon is selected, bring up the *Edit Wire Style* dialogue. To display the *Edit Wire Label* dialogue you must click on the label itself.

3. Ensure the label string text is fully selected (it is by default) and press DEL to delete it.
4. Click OK or press ENTER to close the dialogue form and save changes.

Using a Wire Label to Assign a Net Name

The normal use of wire labels is to indicate that a particular wire, and all the pins to which it is graphically connected, belong to a given net. Then, that group of pins are deemed connected to any other groups of pins which have also been assigned the same net name, even where there is no graphical connection. Occasionally, one may also label particular nets purely for human benefit.

 See *Net Names* on page 135 for more information about net names.

Using a Wire Label to Assign a Net Property

Net properties are used to assign special information to a net. In the PROTEUS system, the primary usage for this feature is to assign routing strategies for use in ARES. A net property assignment has the form:

<prop>=<value>

For example, a wire label with the string

STRAT=POWER

would result in the **STRAT** property for the connections being given the value POWER. In the ISIS/ARES context, this would result in ARES using the POWER strategy for the connections.

 See *Net Properties And Routing Strategies* on page 170 for more information about strategies.

Wire Label Properties

A wire or wire label (they are equivalent for this purpose) has the following property:

Property Name	Description
NET	The wire label text.

LABCENTER ELECTRONICS

Wire labels may only be assigned via the *Property Assignment Tool* when it is used in *On Click* mode. This is because there is no way for ISIS to determine where a wire label should be placed on a tagged wire, unless you mark the position with the mouse.

SCRIPTS

A major feature of ISIS is its ability to support free format text scripts, and the numerous uses it puts them to. These uses include:

- Defining variables for use in property expressions and parameter mapping.
- Defining primitive models and scripts for use with the ProSPICE simulator.
- Annotating designs with substantial quantities of text.
- Storing property and packaging information when a component is decomposed.

Placing and Editing Scripts

The procedures for placing and editing scripts are almost identical as both operations involve the *Edit Script Block* dialogue form.

To place a script :

1. Select the *Script* icon from the *Mode Selector* toolbar.
2. In the *Editing Window*, point to where you want the top-left of the script to be and click left.

An *Edit Script Block* dialogue form is displayed.

2. Enter the text for the new script in the *Text* field. You can also adjust the attributes of a script at this time - see *To edit a script* below for full details of the dialogue form.
3. Close the dialogue form with the OK button to place the new script or with the CANCEL button to quit the form and cancel the script placement.

To edit a script:

1. Either:
 - (a) Tag the script by pointing at it and clicking right and the click left on the tagged script (without moving the mouse).
 - (b) Point at the script with the mouse and use the configured shortcut command for “*Edit Object Under Cursor*” to edit it (by default this is CTRL+E).

The *Edit Script Block* dialogue form is displayed.

2. Adjust the script attributes as required.

The *Edit Script Block* dialogue form has two tabs: *Script* and *Style*. See the graphics and text style tutorial (*Editing Local Styles* on page 52) for more details of editing local text styles.

3. Close the form.

To close the form and save changes either click the OK button or use the CTRL+ENTER keys. To close the form and abandon changes, either click the CANCEL button or use the ESC key.

 You can resize the *Edit Script Block* dialogue form to make the *Text* field bigger and then save the size using the *Save Window Size* command on the Window's 'system' menu (click on the button at the far left of the title bar).

Script Block Types

ISIS currently supports the following script block types:

SCRIPT BLOCK TYPE	BLOCK HEADER
Part Property Assignment	*FIELD
Sheet Global Net Property Assignment	*NETPROP
Sheet Property Definition	*DEFINE
Parameter Mapping Table	*MAP ON <i>varname</i>
Model Definition Table	*MODELS
Named Script	*SCRIPT <i>type name</i>
SPICE model scripts	*SPICE <i>type name</i>

A brief description of the usage and format of each type is given in the following sections.

Part Property Assignments (***FIELD**)

Field blocks are primarily intended to facilitate the assignment of properties to connectors made from *physical terminals*. A special means to do this is required since when physical terminals are used, there is no single entity on the schematic that represents the connector and to which properties could be directly attached.

An example block is shown below:

```
*FIELD
J1 , PACKAGE=CONN-D9
J2 , PACKAGE=CONN-D25
```

This assigns connector J1 the PCB package for a 9 pin D connector, and J2 the package for a 25 way connector.

You can assign properties to ordinary components as well, but there is generally little point since you can add them directly to the components.

 See *Physical Terminals* on page 92 for more information about physical terminals.

 See *Isis And Ares* on page 167 regarding packaging considerations for PCB design.

Sheet Global Net Property Assignments (*NETPROP)

A net property is normally assigned by the placement of a wire label with the syntax:

```
prop=value
```

However, there are occasions when you wish a large number of nets to carry the same property. Most often, the requirement is for all the circuitry on a particular sheet to be of a particular type (perhaps extra wide tracking for a power supply) and it is for this purpose that a **NETPROP** block may be used. For example, if the block:

```
*NETPROP  
STRAT=POWER
```

is included on a sheet, then all nets whose wires do not carry explicitly net properties will be assigned the net property STRAT=POWER.

 See *Using A Wire Label To Assign A Net Property* on page 79 for more information about net properties.

Sheet Property Definitions (*DEFINE)

A sheet property is essentially a variable which is defined on a given sheet and may be used in *property expressions* within components' property lists on that sheet. Sheet properties defined on the root sheets of a design, also appear in the netlist and may be used as control parameters by software such as the VSM simulators that reads the netlist.

An example block is shown below:

```
*DEFINE  
TEMP=40  
MINSTEPS=100
```

This defines two properties **TEMP** and **MINSTEPS**, and would be placed on the root sheet of a design such that they would be passed to VSM as simulation control parameters.

 See *Properties* on page 55 for an in depth discussion of property management.

Parameter Mapping Tables (*MAP ON varname)

This is considered a highly advanced topic, and one which is only of great importance if you are involved in creating universal models for use with VSM. Further information on this is given in the VSM manual, but for completeness, we provide an example PMT here:

```
*MAP ON VALUE
7400   : TDLH=12n,TDHL=7n
74LS00 : TDLH=10n,TDHL=6n
74S00  : TDLH=5n,TDHL=3n
```

This table would be placed on the model schematic for a 2 input NAND gate, and would select different values for **TDLH** and **TDHL** according to the value (i.e. type) of the parent object. Also on the child sheet would be a NAND_2 primitive with the properties:

```
TDLH=<TDLH>
TDHL=<TDHL>
```

At netlist time, the PMT would examine the parent object's **VALUE** property and if, say, it were 74LS00, then the sheet properties TDLH=10n and TDHL=6n would be defined for that instance of the model. Then, on processing the NAND_2 primitive, these values would be substituted for <TDLH> and <TDHL> such that the primitive would acquire the correct timing to model a 74LS00.

A DEFAULT case is now supported.

Model Definition Tables (*MODELS)

These blocks are used only with the VSM simulators (currently, in fact, only with the Analogue simulator) and provide a shorthand method for managing the large numbers of properties that some of the simulator primitives use. For example, the bipolar transistor model has over 30 different properties and it would be fairly hopeless if you had to assign all of these individually for each transistor on the circuit.

Instead, you can use a **MODELS** block to define the properties for a particular type of transistor, and then refer to that set of properties using a model name.

An example block is shown below:

```
*MODELS
741_NPN : BETAF=80, ISAT=1E-14, RB=100, VAF=50, \
        TAUF=0.3E-9, TAUR=6E-9, CJE=3E-12, CJC=2E-12
741_PNP : BETAF=10, ISAT=1E-14, RB=20, VAF=50, \
        TAUF=1E-9, TAUR=20E-9, CJE=6E-12, CJC=4E-12
```

This defines the two transistor types used in a 741 IC. Individual transistors could then be given the values 741_NPN or 741_PNP and VSM would automatically give them the characteristics defined in the model table.

 See the VSM manual for further discussion of primitive models.

Named Scripts (*SCRIPT scripttype scriptname)

In certain circumstances it is useful to be able to export a named block of text to an external application. In particular the SPICE and SPICE-AGE netlist formatters make use of the named scripts **PSPICE** and **SPICE-AGE** to supply control information for the simulations.

A typical named script block is shown below. Note that the end of the named script is marked by an **ENDSCRIPT** keyword, though this is optional if there are no other blocks in the script object.

```
*SCRIPT PROGRAM 7493

// Declare linkage to pins and local variables:
PIN CKA, CKB, RA, RB
PIN QA,QB,QC,QD
INT counta = 0, countb = 0

// Handle single-bit 'A' counter and assign output:
IF RA=H THEN
    counta = 0
ELIF CKA=HL THEN
    counta = (counta+1) % 2
ENDIF
QA = counta & 1

// Handle three-bit 'B' counter and assign output:
IF RB=H THEN
    countb = 0
ELIF CKB=HL THEN
    countb = (countb+1) % 8
ENDIF
QB = countb & 1
QC = countb & 2
QD = countb & 4

*ENDSCRIPT
```

 See the SPICE and SPICE-AGE application notes for information about the syntax of the named scripts associated with these simulators.

SPICE Model Script (*SPICE)

These scripts allow SPICE input cards to be typed on the schematic and loaded directly into ProSPICE at simulate time. This can be handy if you want to develop or test SPICE models in native SPICE format in the native SPICE format.

 Further information on this is given in the Proteus VSM manual.

BUSES

A bus is a kind of shorthand for a large number of wires, and is commonly used on microprocessor schematics. ISIS has an unprecedented degree of support for buses, including not only the ability to run buses between hierarchy modules but also the facility to define library parts with *bus pins*. Thus it is possible to connect a CPU to an array of memories and peripherals by single bus wires, rather than a complex arrangement of wires, bus entries and buses.

Placing Buses

Buses are placed in very much the same way as ordinary wires except that they must run to and from bus connection points, rather than wire connection points. Also, unlike wires, buses may be placed in isolation from any other objects.

To place a bus:

1. Select the *Bus* icon from the *Mode Selector* toolbar.
2. Point where you want the bus to start. This may be a bus pin, an existing bus, or free space on the drawing.
3. Click left to start the bus, then click again at each corner of the desired bus route.
4. To finish the bus on a bus connection point (a bus pin or an existing bus) point at it and click left. To finish the bus in free space, click right.

Bus Labels

A bus may be labelled in exactly the same way as a wire. However, ISIS defines a special syntax for bus labels.

To place or edit a bus label:

1. Select the *Wire Label* icon from the *Mode Selector* toolbar.
2. Point on the bus at the position where you want the label to be placed.
3. Click left to place the label. The *Create Wire Label* or *Edit Wire Label* dialogue form will be displayed.

 See *Editing Local Styles* on page 52 for more details of the controls on the *Style* tab.

4. Type in the required text for the wire label. This should be something like $D[0..7]$ or $A[8..15]$. If you omit the range specification then the bus will take its base as zero and its width from the width of the widest bus pin connected to it. In general, however, you should always use range specification.
5. Click OK or press ENTER.

Note the following:

- You cannot place a wire label other than on a wire or a bus.
- You cannot place more than one wire label on one section of a bus. Attempting to do so will edit the existing wire label.
- ISIS will orient the wire label according the orientation of the wire segment on which it is placed. This can be changed on the *Edit Wire Label* dialogue form.

To delete a bus label:

1. Tag the wire and wire label by pointing at either and clicking right.
2. Bring up the *Edit Wire Label* dialogue form by clicking left on the label.
Note that clicking left on the tagged wire or bus will, except when the *Wire Label* icon is selected, bring up the *Edit Wire Style* dialogue. To display the *Edit Wire Label* dialogue you must click on the label itself.
3. Ensure the label string text is fully selected (it is by default) and press DEL to delete it.
4. Click OK or press ENTER to close the dialogue form and save changes.

To change the appearance of a bus:

1. Ensure the *Wire Label* icon is not selected.
2. Tag the bus by pointing at it and clicking right.
3. Click left on the tagged wire.
The *Edit Wire Style* dialogue box is displayed.
4. Uncheck the *Follow Global?* checkboxes of those style attributes of the graphics style you want to change. If a style attribute and its *Follow Global?* checkbox are both disabled it is because the style attribute is not meaningful given other attribute settings, either locally or through following the global style attribute.
5. Set the style attributes to the required settings.
6. Press ENTER or click the OK button to close the dialogue form and keep the changes. Press ESC or click the CANCEL button to close the dialogue form and abandon changes.

 To change the appearance of all buses on the schematic, use the *Set Graphics Style* command on the *Template* menu to edit the *BUS WIRE* graphics style.

Wire/Bus Junctions

Sometimes, even with the provision of bus pins, it is necessary to tap off a single signal from a bus, perhaps for decoding purposes. From a purely graphical point of view this is just a matter of placing a wire that ends on the bus:

To place a bus tap:

1. If you plan to route the wire *from* the bus to another object, ensure that the *Bus* icon is not selected.
2. Place the wire in the usual way. The bus will behave like an ordinary wire in these circumstances.

It may be helpful to you to understand that when you place a wire in this way, automatic dot placement will operate to give the wire and bus something to connect to. However, you will not see the dot as it adopts the same colour and width as the bus.

Having got a wire joined to a bus, you must specify which signal from the bus you are tapping.

To annotate a bus tap:

1. Ensure that the bus carries a bus label such as D[0 . . 7] . This label would define eight nets called D0, D1, ... , D7.
2. Place a wire label on the wire to indicate which signal it is tapping.

If the bus doesn't connect to any bus pins or bus terminals, you can omit step [1]. In these circumstances, the bus itself plays no part at all in the determining the design connectivity.

Placing a bus tap without annotating it will be reported as a netlist error as it is a wholly ambiguous situation. ISIS cannot know which signal you are trying to tap. Nor can a human reader for that matter!

Bus Properties

A bus or bus label (they are equivalent for this purpose) has the following property:

Property Name	Description
NET	The bus label text.

Bus labels may only be assigned via the *Property Assignment Tool* when it is used in *On Click* mode. This is because there is no way for ISIS to determine where a bus label should be placed on a tagged bus, unless you mark the position with the mouse.

SUB-CIRCUITS

Sub-circuits are used to attach lower level sheets to higher level sheets in a hierarchical design. Each sub-circuit has a name that identifies the child sheet, and a circuit name that identifies the child circuit. On any given sheet, all the sub-sheets should have different sheet names, but may - and often will - have the same circuit names. More information about hierarchical design is given in *Hierarchical Designs* on page 129.

Sub-circuits can also have property lists, and this leads to the possibility of *parameterized circuits* in which different instances of a given circuit can have different part values (or other properties) as well as independent annotations. Further information about this powerful feature are given in *Parameterized Circuits* on page 60

Placing Sub-Circuits

Placing a sub-circuit involves laying out the actual sub-circuit box, and then placing sub-circuit ports upon it. The same icon is used for both operations; ISIS determines what happens according to whether you point it free space, or at an existing sub-circuit box.

To place a sub-circuit box:

1. Select the *Sub-Circuit* icon from the *Mode Selector* toolbar.
2. Point where you want the top left corner of the box. This must be a point not occupied by an existing sub-circuit.
3. Click left and drag out the box - then release the mouse button.

 You can customise the appearance of subcircuits by editing the *SUBCIRCUIT* graphics style. See *Editing Global Styles* on page 50 for more information.

To place sub-circuit ports:

1. Select the *Sub-Circuit* icon from the *Mode Selector* toolbar.
2. Select the type of port you want from the *Object Selector*.
3. Point where you want the port. This must be a point on the left or right edge of the sub-circuit to which you want to attach the port.
4. Click left to place the port. ISIS will orient it automatically according to which edge of the sub-circuit you have placed it on.

Having placed a port or ports, you must annotate them. Hierarchical design works by connecting the ports on the parent object with like named logical terminals on the child sheet. It follows that both the ports and the terminals must be given unique names. This can be achieved in a variety of ways:

- Edit the terminal label using any of the methods described in *Editing An Object* on page 36.
- Use the *Property Assignment Tool* to assign the **NET** property of one or more ports. This is especially effective if a group of ports have names which run in an alphanumeric sequence.

It is quite legal to connect buses to ports. In this case, the name for the port should generally define the range for the bus, as in $D[0..7]$, although this is not mandatory. If no range is given, ISIS will use the range given for the bus section that connects to the port, or if that has no name or range then the width will be taken from whatever bus pins connect to the bus.

Placing a port without annotating it will be reported as a netlist error since such an object has no meaning and cannot be tied up with anything on the child sheet.

Editing Sub-Circuits

A sub-circuit may be resized using the general procedure described in *Resizing An Object* on page 35 and edited using any of the general editing techniques described in *Editing An Object* on page 36.

 See *Parameterized Circuits* on page 60 for further information about parameterized circuits.

Sub-Circuit Properties

Sub-circuits have the following system properties:

Property Name	Description
NAME	The sub-circuit instance name. This is also used as the name for the child sheet.
CIRCUIT	The name of the child circuit. If you do not assign this, ISIS will choose an automatic name when you first enter the child sheet.

If any other property names are assigned, then these will create user properties in the sub-circuit's text block. Such properties then become sheet properties for the child sheet and may be used in property expressions.

TERMINALS

Terminals are used for specifying the interface to a circuit - ISIS does not allow a wire to 'float' - both ends must connect to something so all the inputs and outputs in your design will be indicated by terminals.

There are two types of terminal - *Logical Terminals* and *Physical Terminals*. The two are distinguished purely by the syntax of their labels.

 You can customise the appearance of terminals by editing the *TERMINAL* graphics style. See *Editing Global Styles* on page 50 for more information.

Logical Terminals

A logical terminal serves merely to donate a net name to the wire to which it connects. Groups of wires with one or more net names in common are taken to be connected by the netlist generator. Logical Terminals thus provide a means to connect things together without using

wires. In particular they provide the means make connections between the sheets in a multi-sheet design.

As with Wire Labels and Bus Entries, the net name can contain any alphanumeric characters plus the hyphen ('-') and underscore ('_'). Spaces can be used within PROTEUS but may cause problems for other software.

Logical terminals may also connect to buses. This provides an extremely efficient way to run buses up and down a hierarchical design.

 See *Net Names* on page 135 for discussion of the role of logical terminals in netlisting

 See *Hierarchical Designs* on page 129 for discussion of the role of logical terminals in hierarchical designs.

Physical Terminals

A physical terminal represents a pin on a physical connector. For example, a terminal with the name:

J3 : 2

is taken to be pin 2 of connector J3. Whilst it is perfectly possible to deal with connectors in exactly the same way as all other components (i.e. define a device to represent them), using Physical Terminals has the advantage that the pins can be placed wherever it is most convenient.

For PCB design, where it is necessary to specify the package type for the connector, a **FIELD** property assignment block (see *Part Property Assignments (*Field)* on page 82) must be used as there is no actual component to edit.

Note that a bus terminal may not be physical, as there is no means to specify the pin numbering for the individual pins.

Placing Terminals

ISIS supports an unlimited variety of terminal symbols. However, when you first select the *Terminal* icon, a basic set of 7 types are automatically pre-loaded into the *Object Selector*.

To place a terminal:

1. If the terminal type you want is not listed in the *Object Selector*, first pick it from the symbol library.
2. Highlight the terminal name in the *Object Selector*. ISIS will show a preview of the terminal in the *Overview Window*.

3. Use the *Rotation* and *Mirror* icons to orient the terminal according to how you want to place it.
4. Point at the position in the *Editing Window* where you want the terminal to appear, and click left. If you hold the mouse button down, you can drag the terminal around until you have it exactly where you want it. When you release the mouse button, it will be placed onto the drawing.

Having placed a terminal, you must then annotate it since an unannotated terminal will be ignored by the netlist compiler. There are a variety of approaches:

- Edit the terminal label using any of the general methods described in *Editing An Object* on page 36.
- Use the *Property Assignment Tool* to assign the **NET** property of one or more terminals. This is especially effective if a group of terminals have names which run in an alphanumeric sequence.

It is quite legal to connect buses to terminals. In this case, the name for the terminal should generally define the range for the bus, as in $D[0..7]$, although this is not mandatory. If no range is given, ISIS will use the range given for the bus section that connects to the port, or if that has no name or range then the width will be taken from whatever bus pins connect to the bus.

Placing a terminal without annotating it will be reported as a netlist error since such an object has no meaning, logically or physically.

Editing Terminals

A terminal may be edited using any of the general editing techniques (see *Editing An Object* on page 36). In addition, since terminals often appear in groups, the *Property Assignment Tool* can be put to good use for annotating and setting the electrical type of terminals. The *Edit Terminal* dialogue form has the following fields:

Name	The net name for a logical terminal or the pin name for a physical terminal.
Type	The electrical type of the terminal.

Terminal Properties

Terminals have the following system properties:

Property Name	Description
NET	The terminal net label.
SYMBOL	The symbol used for the terminal. This can be one of the standard terminal symbols, or else the name of a user defined terminal symbol.
TYPE	The electrical type of the terminal. This can be any of PASSIVE, INPUT, OUTPUT, BIDIR or POWER.

PIN OBJECTS

A full description of how to create and edit your own devices is given in *Library Facilities* on page 101. Here, we just discuss the placement and editing of pin objects. These are used to represent each drawn pin of a device element and consist of some graphics (often just a single line) plus the capacity to carry and display a pin name and a pin number.

Please note that you cannot wire off a pin object - you can only connect to pins that belong to fully constituted components that have been placed in the usual way.

Placing Pin Objects

An unlimited number of pin object types may be defined, and a good selection are provided in SYSTEM.LIB. However, when first select the *Device Pin* icon, a basic set of 6 types are automatically pre-loaded into the *Object Selector* and these will suffice for most purposes.

To place a pin:

1. If the pin type you want is not listed in the *Object Selector*, first pick it from the symbol library.
2. Highlight the pin type name in the *Object Selector*. ISIS will show a preview of the pin in the *Overview Window*.
3. Use the *Rotation* and *Mirror* icons to orient the pin according to how you want to place it.
4. Point at the position in the *Editing Window* where you want the pin to appear, and click left. If you hold the mouse button down, you can drag the pin around until you have it exactly where you want it. When you release the mouse button, it will be placed onto the drawing.

Having placed a pin, you will generally want to modify it to give it a pin name, number and electrical type. There are a variety of possibilities:

- Edit the pin manually using any of the general methods described in *Editing An Object* on page 36.
- Use the *Property Assignment Tool* to assign the **NAME**, **NUMBER** and **TYPE** properties of one or more pins. This is especially effective if a group of pins have names which run in an alphanumeric sequence such as buses.

If a pin represents a data or address bus, you may want to use a bus pin. In this case, pin numbers can only be specified using the *Visual Packaging Tool*. Equally, if the device has multiple elements, (e.g. a 7400) you must again specify pin numbers for each element using the packaging tool. In either of these cases, you should leave the pin numbers of the pins blank.

Editing Pin Objects

A pin may be edited using any of the general editing techniques (see *Editing An Object* on page 36). In addition, since pins often appear in groups, the *Property Assignment Tool* can be put to good use for defining pin names, numbers and electrical types.

Pin Object Properties

Pin objects have the following system properties:

Property Name	Description
NAME	The pin name.
NUM	The pin number.
SYMBOL	The symbol used for the pin. This can be one of the standard pin symbols, or else the name of a user defined pin symbol.
TYPE	The electrical type of the terminal. This can be any of PASSIVE, INPUT, OUTPUT, BIDIR, TRISTATE, PULLUP, PULLDOWN or POWER.

SIMULATOR GADGETS

The interface to the ProSPICE simulator makes use of certain special objects within ISIS. The complete set consists of:

GENERATORS TAPES
VOLTAGE PROBES CURRENT PROBES
GRAPHS

Instructions pertaining to the use of these objects are given in the Proteus VSM manual.

2D GRAPHICS

ISIS supports the following types of 2D graphics objects: lines, boxes, circles, arcs, scalable text and composite symbols. These are intended for use both directly on the drawing, for example to draw division lines and sectional boxes around parts of a design, and also for creating new library parts (devices, symbols, pins and terminals).

Placing 2D Graphics

The following are the procedures for placing the various types of graphic objects.

To place a line:

1. Select the *Line* icon from the *Mode Selector* toolbar.
2. Select the *Graphics Style* you want the line drawn in from the *Object Selector*.
3. Click left to mark the start of the start of the line.
4. Click left again to mark the end of the line.

To place a box:

1. Select the *Box* icon from the *Mode Selector* toolbar. .
2. Select the *Graphics Style* you want the box drawn in from the *Object Selector*.
3. Point where you want the top left corner of the box and press the left mouse button.
4. Drag the mouse to where you want the bottom right corner of the box and release the button.

To place a circle:

1. Select the *Circle* icon from the *Mode Selector* toolbar.
2. Select the *Graphics Style* you want the circle drawn in from the *Object Selector*.

3. Point where you want the centre of the circle and press the left mouse button.
4. Drag the mouse to a point on the circumference of the desired circle and release the button.

To place an arc:

1. Select the *Arc* icon from the *Mode Selector* toolbar.
2. Select the *Graphics Style* you want the line drawn in from the *Object Selector*.
3. Consider the arc as lying in one quadrant of an ellipse - you will first define this quadrant. Point at the position where one end of the quadrant will lie and press the left mouse button.
4. Drag the mouse roughly along the path of the quadrant and release the button when you reach the other end.
5. A pair of 'clipping lines' will now appear, allowing you to define which section of the quadrant you wish the arc to be drawn. Move the mouse around until you have just the desired section visible, and then click left.

To place a path:

1. Select the *Path* icon from the *Mode Selector* toolbar.
2. Select the *Graphics Style* you want the path drawn in from the *Object Selector*.
3. Position the mouse over the *Editing Window* where you wish the first vertex of the path to be and click left to place it.
4. To enter a straight-line segment, simply move the mouse; to enter a curved segment, press and hold down the CTRL key and then move the mouse.

As the mouse is moved a rubber-banded line is displayed showing the type of segment (straight or curved) that will be created and its position.

5. Click left to place the second vertex. During placement, a placed vertex can not be deleted or 'undone' though the path may be edited after placement and unwanted vertices removed or segments altered.
6. Repeat steps four and five to complete your path or press ESC to cancel the path entry.

The path is not completed until you place a final vertex at the same point as the first vertex, so closing the path.

To place graphics text:

1. Select the *Text* icon from the *Mode Selector* toolbar.
2. Select the *Graphics Style* you want the text drawn in from the *Object Selector*.
3. Use the *Rotation* and *Mirror* icons to orient the text according to how you want it to appear on the drawing.
4. Point at the position in the *Editing Window* where you want the bottom-left of the text to appear and click left.

The *Edit 2D Graphics Text* dialogue form is displayed.

 See *Editing 2d Graphics* on page 99 for a full reference this dialogue form.

5. Type the text into the dialogue form and set the justification, text size, etc. if required.
6. Press ENTER or click on the OK button to place the text, press ESC or click on the CANCEL button to abort placing the text.

To place a symbol:

1. Select the *Symbol* icon from the *Mode Selector* toolbar.
2. Select the symbol you wish to place from the *Object Selector*. If the symbol you want is not in the selector, you must first pick from the symbol library. The *Symbol Library Pick* form can be displayed by clicking on the 'P' toggle on the selector.
3. Use the *Rotation* and *Mirror* icons to orient the symbol according to how you want it to appear on the drawing.
4. Point at the position in the *Editing Window* where you want the symbol to appear, and click left. If you hold the mouse button down, you can drag the symbol around until you have it exactly where you want it. When you release the mouse button, it will be placed onto the drawing.

Resizing 2D Graphics

Lines, boxes, circles, arcs and paths may be resized by tagging them with the right mouse button and dragging one or more of the displayed 'handles' as follows:

- Lines have two handles which adjust the start and end points.
- Boxes have eight handles which adjust the corners and edges.
- Circles have four handles, all of which adjust the radius.
- Arcs have two handles which adjust the endpoints and the two Bezier control points.

- Paths have one handle per vertex (the point between two segments) plus two Bezier control points for each curved segment.

Paths support additional editing operations that allow the path to be modified without having to be deleted and re-entered. All these operations require you to hold the ALT (think of ALT=ALTer) key down:

- Click right on a vertex handle to delete that handle. The segments either side of the handle are also removed and a straight line replaces them.
- Click left on a line or curve segment to break it in to two straight lines with a common vertex at the point clicked.
- Hold the CTRL key down and click left on a line or curve segment to break the segment in to two Bezier segments with a shared vertex at the point clicked.

Don't forget that for all three of the above path editing operations, the ALT key must be held down!

The other types of graphic objects cannot be resized - instead you should delete and replace them to effect modifications.

Editing 2D Graphics

All 2D graphics objects can be edited in the usual way by first tagging it with the right mouse button and then clicking left on it (without moving the mouse).

All 2D graphic objects except 2D graphics text (discussed below) displays an *Edit Graphics Style* dialogue box that allows you to specify local, fixed, values.

 See *The Header Block* on page 43 for details of how to use 2D graphics text to display design information.

MARKERS

Marker Types

Markers are used in the creation and editing of devices, symbols, terminals and pins. The following fixed set of types is provided:

TYPE	PURPOSE
ORIGIN	Defines the anchor point of any library part. The anchor point is the point around which the object may be rotated and corresponds to the mouse position at the time of placement.

NODE	Defines the position of the wire connection point for a pin or terminal.
BUSNODE	As above, but defines the pin or terminal as being of bus type. A bus line (thick, blue) will be drawn from the busnode to the origin.
LABEL	Defines the position and orientation of the label for a terminal.
PINNAME	Defines the position and orientation of the pin name for a pin.
PINNUM	Defines the position and orientation of the pin number for a pin.
INCREMENT	Used in creating Active Component simulator models, this marker defines a hot-spot for incrementing the state variable.
DECREMENT	Used in creating Active Component simulator models, this marker defines a hot-spot for decrementing the state variable.

Placing Markers

Markers are placed in the same way as graphics symbols (which is really what they are!).

To place a marker:

1. Select the *Marker* icon from the *Mode Selector* toolbar.
2. Select the marker you wish to place from the *Object Selector*.
3. Use the *Rotation* and *Mirror* icons to orient the marker according to how you want it to appear on the drawing. This is only appropriate for *Label*, *Pinname* and *Pinnum* markers - orientation is irrelevant for the other types.
4. Point at the position in the *Editing Window* where you want the marker to appear, and click left. If you hold the mouse button down, you can drag the symbol around until you have it exactly where you want it. When you release the mouse button, it will be placed onto the drawing.

LIBRARY FACILITIES

GENERAL POINTS ABOUT LIBRARIES

As supplied there are two symbol libraries and over 25 device libraries. For an up to date listing of all the supplied libraries and library parts, read the file LIBRARY.PDF in the library directory of your Proteus installation. You will need to install the Adobe Acrobat reader if you have not already done so.

Library Discipline

USERSYM.LIB and USERDVC.LIB are set to read/write; the rest are set to read-only. The idea is that you should only add things to USERSYM.LIB (new symbols) and USERDVC.LIB (new devices). This means that we can supply you with updates to the parts we have defined without risk of overwriting similarly named objects in your own libraries.

You can, of course, create further libraries of your own using the *Library Manager*.

Should you really need to change things in the read-only libraries, you can set them to read/write in *File Manager* or *Explorer* under Windows or using the *Library Manager* from within Proteus.

Under no circumstances should you remove things from SYSTEM.LIB.

The Pick Command

The *Pick* command serves as an alternative to the library browser, primarily for when you know the name of the device or symbol you are looking for. You can search for an exact match, or else use the various pattern matching options to search for a part when you have a rough idea of its name.

If you pick a device or symbol that is already loaded into the layout, ISIS will update it from the libraries on disk.

Note also:

- Where devices are concerned, the replacement algorithm will match pin positions or pin names, so connectivity will be maintained even if you move or renumber a devices pins.
- Where a there are two or more devices or symbols with the same name spread across several libraries, the *Pick* command will load the newest one. This is particularly helpful

if you have changed one of our parts and put in USERDVC.LIB, since your version will be deemed newer than ours.

 See the section *Picking, Placing And Wiring Up Components* in the tutorial on page 7 for details of how to pick parts using the library pick forms.

SYMBOL LIBRARIES

The symbol libraries are used to hold both general graphical symbols for direct placement onto drawings, and also symbols for terminals, module ports and device pins.

The various types of symbol are created with different name prefixes, and thus appear to all intents and purposes to be stored in separate 'compartments' of the symbol library. A set of these objects is defined in SYSTEM.LIB and is pre-loaded into the various object selectors when you start ISIS. Thus, when you select the *Terminal* icon, and you see the names DEFAULT, INPUT, OUTPUT etc. you are actually accessing symbols called \$TERDEFAULT, \$TERINPUT, \$TEROUTPUT and so on. When you place a terminal on the drawing, a terminal object is created and the appropriate symbol is assigned to it.

The significance of all this comes down to two points:

- Graphics symbols, terminals, module ports and device pins are all stored in symbol libraries (e.g. SYSTEM.LIB and the user library USERSYM.LIB). There are no special libraries for terminals, ports or pins.
- The procedures for making the various symbol types are all very similar.

Graphics Symbols

A symbol is a group of 2D graphics objects which are treated as a single object. For instance, using three lines and two arcs you can form an AND gate symbol.



To make a graphics symbol

1. Select one of the 2D graphics icons - e.g. *Line*, *Box* etc.
2. In the *Object Selector* select a graphics style appropriate to the type of symbol you are creating. For a symbol that will form the basis of future components, this is generally the *COMPONENT* style though for some symbols, such as OP AMP, where a small line is needed to link the body of the symbol to the base of a pin this would be drawn in the *PIN* style.

-  If the outline of the symbol consists of lines and arcs and you want the symbol to be filled, use the *path* object to create the outline.
3. Select and place graphic objects as required to form the symbol.

Any graphic that needs to have a fixed appearance should be edited, the appropriate *Follow Global?* checkboxes unchecked and the graphics style attribute changed.
4. If you want to define the origin for the symbol, select the *Markers* icon, click on the *Origin* marker in the object selector and place it where you want the origin to be. If you don't place an *Origin*, ISIS will default the origin to the centre of the symbol.
5. Tag the objects that will comprise the symbol by dragging a tag-box around them with the right mouse button.
6. Invoke the *Make Symbol* command from the *Library* menu, select a name and library for the new symbol.
7. Click O.K. to complete the operation.

A symbol can only consist of 2D graphics objects - you cannot tag a whole section of circuitry including components, wires etc. and make that into a symbol. To manipulate circuit sections in this way, you should use the *Import* and *Export* section commands on the *File* menu.

User Defined Terminals

ISIS permits the definition of user defined symbols for use as logical or physical terminals. These are made in the same way as ordinary symbols except that you must place a *Node* marker to specify the position of the terminal's connection point, and a *Label* marker to specify the position and orientation of its net label.

To make a user defined terminal

1. Select an appropriate 2D graphics icon - typically the *Line* icon - from the *Mode Selector* toolbar.
2. In the *Object Selector* select an appropriate graphics style. This will nearly always be the *TERMINAL* style though the *BUS WIRE* style may also be appropriate for small parts of the symbol.
3. Select and place graphics objects as required to form the body of the terminal.

Any graphic that needs to have a fixed appearance should be edited, the appropriate *Follow Global?* checkboxes unchecked and the graphics style attribute changed.

4. Select the *Markers* icon. Place a *Node* or *Busnode* marker where you want the wire or bus to connect the terminal, and a *Label* marker where you want its net label to be. You can also place an *Origin* marker to define where its origin will be.
5. Tag the objects that will comprise the terminal by dragging a tag-box around them with the right mouse button.
6. Invoke the *Make Symbol* command from the *Library* menu, set the type to *Terminal* and then select a name and library for the new terminal.
7. Click OK to complete the operation.

Note that the electrical type of user defined terminals will always default to passive. If you need user defined terminals to carry different electrical types you must assign them after placement with the *Property Assignment Tool*.

User Defined Module Ports

Module ports are the connectors used to attach wires to sub-circuits and it is possible to create user defined symbols for them. These are made in the same way as ordinary symbols except that you must place a *Node* marker to specify the position of the port's connection point, and a *Label* marker to specify the position and orientation of its label.

To make a user defined module port

1. Select an appropriate 2D graphics icon - typically the *Line* icon - from the *Mode Selector* toolbar.
2. In the *Object Selector* select an appropriate graphics style. This will nearly always be the *PORT* style though the *BUS WIRE* style may also be appropriate for small parts of the symbol.
3. Select and place graphics objects as required to form the pin symbol.

Any graphic that needs to have a fixed appearance should be edited, the appropriate *Follow Global?* checkboxes unchecked and the graphics style attribute changed.

In drawing the port, you should orient it in a manner suitable to be placed on the left hand edge of a sub-circuit - ISIS will mirror it in X when it is placed on the right hand edge.

4. Select the *Markers* icon. Place a *Node* or *Busnode* marker where you want the wire or bus to connect the port, and a *Label* marker where you want its net label to be. You should also place an *Origin* marker to correspond with where the edge of the sub-circuit will be.
5. Tag the objects that will comprise the port by dragging a tag-box around them with the right mouse button.

6. Invoke the *Make Symbol* command from the *Library* menu, set the type to *Module Port* and then select a name and library for the new terminal.
7. Click OK to complete the operation.

User Defined Device Pins

Device pins are, in fact, drawn as symbols and consequently you can define your own symbols for them. A variety of device pin symbols are supplied in SYSTEM.LIB. Nevertheless, there may be some situations in which it is appropriate to define your own.

To make a user defined device pin

1. Select an appropriate 2D graphics icon - typically the *Line* icon - from the *Mode Selector* toolbar.
2. In the *Object Selector* select a graphics style. This will nearly always be the *PIN* style for a standard pin or the *BUS WIRE* style for a bus pin.
3. Select and place graphics objects as required to form the body of the terminal. Any graphic that needs to have a fixed appearance should be edited, the appropriate *Follow Global?* checkboxes unchecked and the graphics style attribute changed.
4. Select the *Markers* icon. Place a *Node* or *Busnode* marker where you want the wire or bus to connect to the pin, a *Pinname* marker where you want the pin name to be, and a *Pinum* marker where you want the pin number to be. You can also place an *Origin* marker to define where its origin will be.
5. Tag the objects that will comprise the pin by dragging a tag-box around them with the right mouse button.
6. Invoke the *Make Symbol* command from the *Library* menu, set the type to *Device Pin* and then select a name and library for the new pin.
7. Click OK to complete the operation.

Editing an Existing Symbol

Any type of symbol may be edited by placing an instance of it and then using the *Decompose* command from the *Library* menu.

To edit a symbol:

1. Place an instance of the symbol. This will be a Graphics Symbol, Terminal, Module Port or Device Pin, as appropriate.
2. Tag the object by pointing at it and clicking right.
3. Select the *Decompose* command from the *Library* menu. This will break the symbol into graphics and markers.
4. Add, delete or edit the graphics and markers as required.
5. Reconstitute the symbol according to the procedure from the previous sections appropriate to its type.

Hierarchical Symbol Definitions

ISIS quite happily allows a symbol to contain other symbols and/or other graphic objects. This allows you to make, for instance, a NAND gate out of the previously defined AND gate plus a circle. Note that symbols defined from other symbols are not "linked" to them in any way - if the lower level symbols are changed or even deleted, this will not affect the new one. This also means that a symbol can be defined as a modification of itself without difficulty, although this will destroy the old version.

DEVICE LIBRARIES

A device (in ISIS terminology) is a type of real world component such as an NPN transistor or a PIC microprocessor. It follows that a component placed on the drawing is an instance of a device. There are essentially three types of device:

- Single element devices. These are parts for which there is a one to one correspondence between the schematic symbol and the PCB package. Each pin has a single name and a single pin number.
- Homogenous multi-element devices. These are parts for which there are several *identical* elements in the one PCB package. Typical examples would be a 7400 quad NAND gate or a TL072 dual op-amp. The same pin will have a different pin number for each element, except for the power pins which tend to be *common*.
- Heterogeneous multi-element devices. These are parts where there are several *different* elements in the one PCB package, but where you wish to draw each element as separate component on the schematic. The most common example by far is a relay, in which you

wish to have the coil and one or more sets of contacts on different parts of the schematic.

ISIS also provides support for *bus pins*. Devices such as microprocessors and their associated peripherals can thus be drawn in very compact forms since their data and address buses may be represented by single pins. Wiring them up becomes a lot less tedious, too.

Whereas a single element device will have just one physical pin number associated with each schematic pin, multi-element parts and parts with bus pins all have multiple pin numbers for each device pin. This aspect of the device creation process is handled by the *Visual Packaging Tool*. In addition, this tool will allow you to create alternate packagings (each with its own set of pin numbers) for the same schematic part. A typical application of this would a microprocessor chip such as the PIC16F877 which is available in both DIL40 and PLCC44 packages.

Making a Device Element

Most of the devices you will encounter will involve only a single element. That is to say that by placing one component object, you account for all the pins in the physical part. This is in contrast to a multi-element part like a 7400 for which you need to place four gates to account for all the pins. Either way, the first stage in making a new library part is to place the graphics and pins for the device element or elements.

To make a device element:

1. Place graphics objects to define the device body in the appropriate graphics style(s).
2. Place device pin objects to represent the pins.
3. Annotate the pins to assign name and types using any of the standard editing techniques (see *Editing An Object* on page 36) or the *Property Assignment Tool*.
4. Tag all the objects that comprise the element. Then invoke the *Make Device* command and assign any default properties.

These stages are worthy of further discussion:

Defining the Device Body

The device body is essentially the complete graphic for the device, excluding its reference designator and pins. Very often, it will just be a box, in which case you should simply select and place a graphics box of the appropriate size in the *COMPONENT* graphics style. For components with more complex graphics such as transistors, op-amps and so forth you can use any of the graphic objects that ISIS provides in whatever graphics styles are appropriate and perhaps editing the objects and assigning local, fixed values to some or all of the graphics style attributes.

It is important to think carefully about what graphics styles you choose for the graphics in the new component. In general most graphics will be placed with the *COMPONENT* style selected and the graphic objects will not need editing as they default to fully following this parent *COMPONENT* style. Occasionally, however, you will want part of the graphics of the new device to be ‘fixed’. For example, you may want the solid body of a transistor to always be filled, perhaps to black. In these cases it is appropriate to edit the graphic object after placement and uncheck some or all of the graphics style attributes and to set local values though where possible you should endeavour to leave as many attributes as possible following the parent, *COMPONENT*, style. Your other consideration should be how the graphic object placed will fit in to a schematic. For example, if you are designing an OPAMP you may wish to use short lines between the slope of the body and the base of power, compensation or some such pins in which case it is best if these lines are placed in the *PIN* style so that they look like part of the pin.

If you wish to define the origin for the device, you should also place an *Origin* marker at the appropriate point. If you don't specify an origin, ISIS will default it to the top leftmost pin end.

 See *2d Graphics* on page 96 for more information about placing and editing graphics objects.

Placing the Pins

When making a device, you use the special device pin objects, available with the *Device Pin* icon selected to place each pin in turn. Several types of device pin are pre-loaded into the *Object Selector* when you first select the *Device Pin* icon, and further types can be picked from the symbol libraries as required. You may also define pin types of your own.

As you place each pin object, you need to be sure that it is oriented correctly. The blue cross that appears at one end of each pin object designates the connection point for the pin; the other should generally be in contact with some part of the device body.

 For further information about pin objects see page 94.

Annotating the Pins

This third phase of the device creation process is probably the trickiest, and can lead to some obscure problems much later on in the design cycle if not carried out correctly. *You have been warned!*

Each pin can carry a pin name, pin number and electrical type. The latter is used for electrical rules checking, and also by the ProSPICE simulator. The electrical type of pins must be correctly specified for digital simulator models in particular.

You have two basic approaches available to you in annotating the pins:

- Edit each pin in turn (the easiest way to do this is to point at it and use the keyboard shortcut for “*Edit Object Under Cursor*” - CTRL+E by default), and proceed to edit its properties using the dialogue form.
- Use the *Property Assignment Tool* to assign to the **PINNAME**, **PINNUM** and **TYPE** properties of the pins.

In most cases, you will find it appropriate to use a mix of these techniques.

In assigning pin names and numbers, bear in mind the following:

- A pin must always have a name. If you type in a number when there is no name, the pin name will automatically be made the same as the pin number.
- If you give two or more pins the same name, they will be deemed to be electrically interconnected in the netlist and thus on the PCB layout.
- To place pins with overbars in their names, , use dollar (\$) characters to mark the start and end of the overbar. For example RD/\$WR\$ would display as RD/ \overline{WR} .
- In general, it is easier to assign pin numbers using the *Visual Packaging Tool*, and this is the only way to assign pin numbers for a multi-element part or a part with bus pins. However, for a simple single element device, you can enter pin numbers at this stage, if you wish.

In assigning pin types, the following table may be helpful:

Pin Type	TYPE ID	Example Uses
Passive	PS	Passive device terminals
Input	IP	Analogue or digital device inputs
Output	OP	Analogue or digital device outputs
Bidir	IO	Microprocessor or RAM data bus pins
Tri-state	TS	ROM output pins
Pull Down	PD	Open collector/drain outputs
Pull Up	PU	Open emitter/source outputs
Power	PP	Power/Ground supply pins

If you are unclear how to actually perform the required editing operations, the following sections of the manual are also relevant:

 See *Pin Objects* on page 94 for a full discussion of the *Edit Pin* dialogue form.

 See *The Property Assignment Tool* on page 65 for examples of how to use the *Property Assignment Tool*.

Invoking the Make Device Command

The final stage of creating a single element device is to tag all the objects (graphics and pins) which comprise the device, and then invoke the *Make Device* command from the *Library* menu. The use of this command is discussed in detail in the following sections.

The Make Device Command

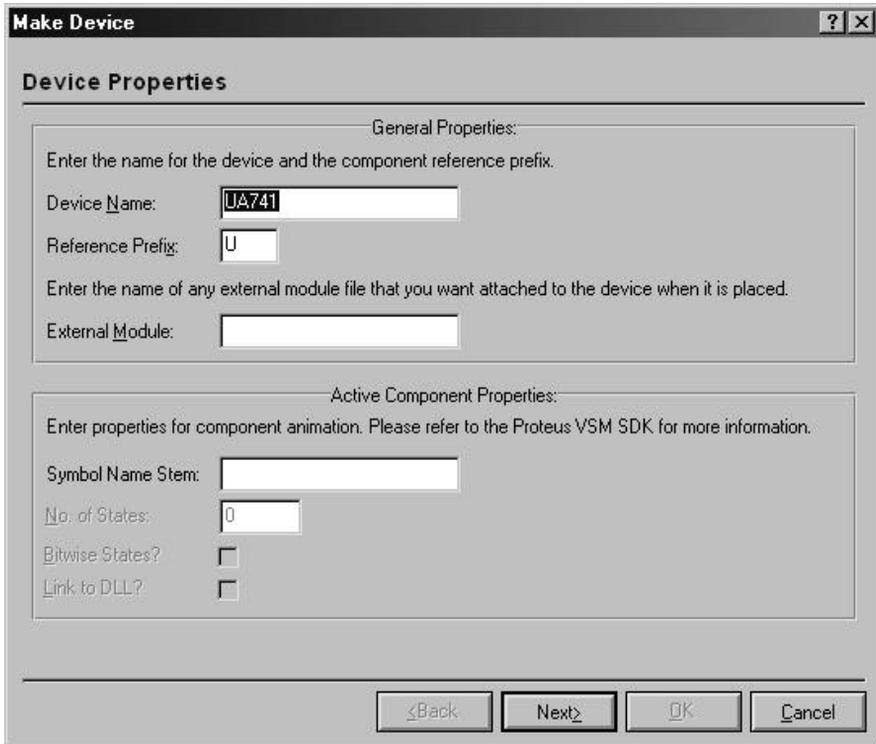
The *Make Device* command is a multi-stage dialogue form sometimes referred to as a *Wizard* in other applications. There are four pages:

- Device properties - these are things like the name of the device, the prefix to use for new components and also properties associated with component animation as used by *Proteus VSM*
- Packaging - this page displays the packagings that have been defined (for an existing component) and provides access to the *Visual Packaging Tool*.
- Component Properties - this page provides the means to create and edit property definitions and default property values.
- Library selection - the final screen allows you to choose the library in which the device will be stored.

Detailed context sensitive help is provided for all the fields on these pages. If you are unsure of the purpose of a particular field, click the '?' at the top right of the dialogue form and then click the field itself to see an explanation of its purpose. Given this, we shall restrict the discussion here to general information about each page.

Device Properties Page

This page has two major sections:- *General Properties* and *Active Component Properties*.

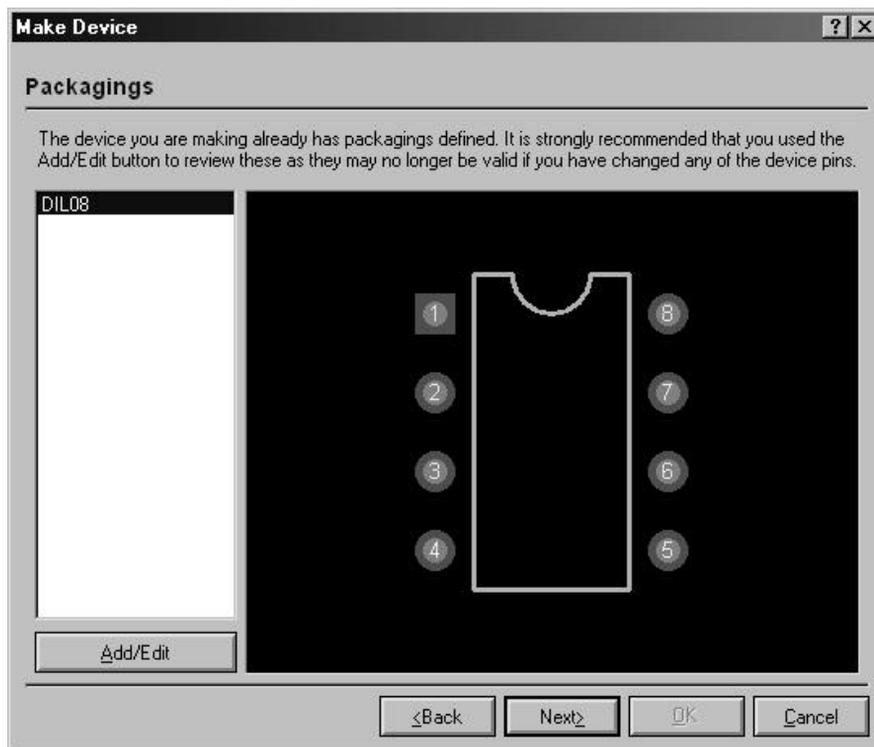


The *General Properties* section determines the name of the device, and the reference prefix. This is the letter or letters that appear in front of the part ID for newly placed components. Note that if you make this blank, newly placed components will be un-annotated and that their part value and properties text will also be hidden. This is useful for block diagram type drawings, or for components such as the *Virtual Oscilloscope*, which are not really part of the design.

The *Active Component Properties* section is used for creating animated components for use with Proteus VSM. See the Proteus VSM SDK documentation for more information.

Packagings Page

This page displays the set of packagings that have been defined for the device; for a new device the list will be empty.

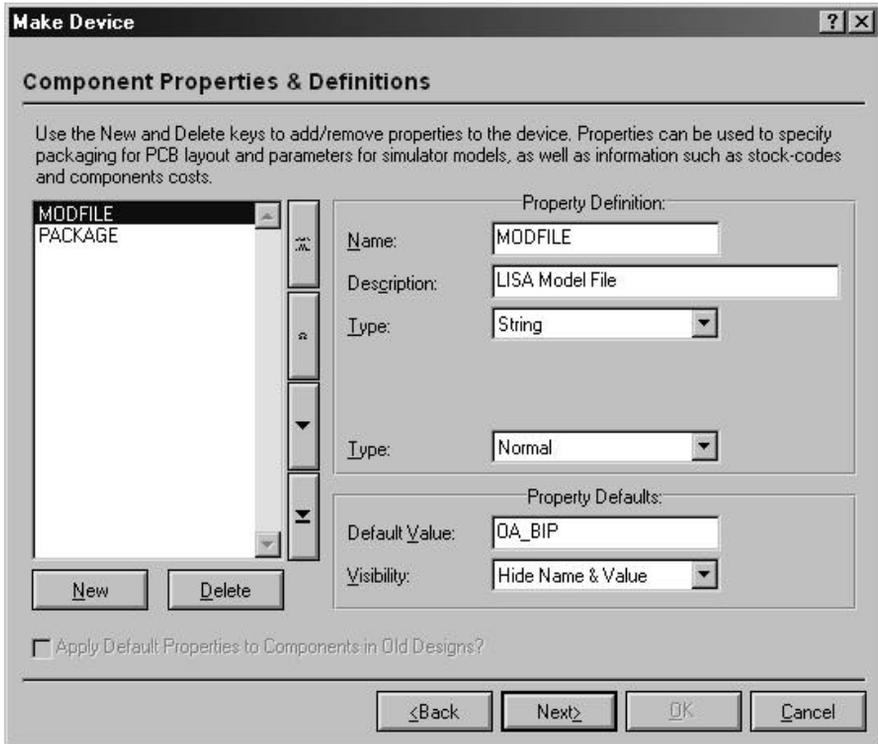


Pressing the *Add/Edit* button will launch the *Visual Packaging Tool* which is described in more detail on page 116.

Note that a different procedure is required to package a heterogeneous multi-element part. In this case, the packaging tool must be invoked for the complete set of elements as placed on the schematic. See page 121 for more information.

Component Properties & Definitions Page

This page is used to define property definitions and default values for the component's properties. The selector on the left shows the properties that have been defined, whilst the *Property Definition* and *Property Defaults* sections determine the type, visibility and default value of the property.



See *Property Definitions* on page 70 for more information about component properties and property definitions

Data Sheet and Help Page

This page allows you to associate a data sheet (PDF file) and/or a help topic with the device. If a data sheet is defined, a *Data* button will appear on the *Edit Component* dialogue form and if a help topic is defined, a *Help* button will appear. You will see these buttons for many of the components in the supplied libraries.

Make Device [?] [X]

Device Data Sheet & Help File

You can link your device to a data sheet (Acrobat .PDF file) and/or a help file. These can then be accessed via special buttons on the 'Edit Component' dialogue form.

Data Sheet:

Data Sheet Filename:

FTP Server:

ETP Path:

FTP User Id:

FTP Password:

CD Title:

CD Path:

Help Topic:

Help File:

Context Number:

<Back Next> OK Cancel

Data sheets can be located in one of three places:

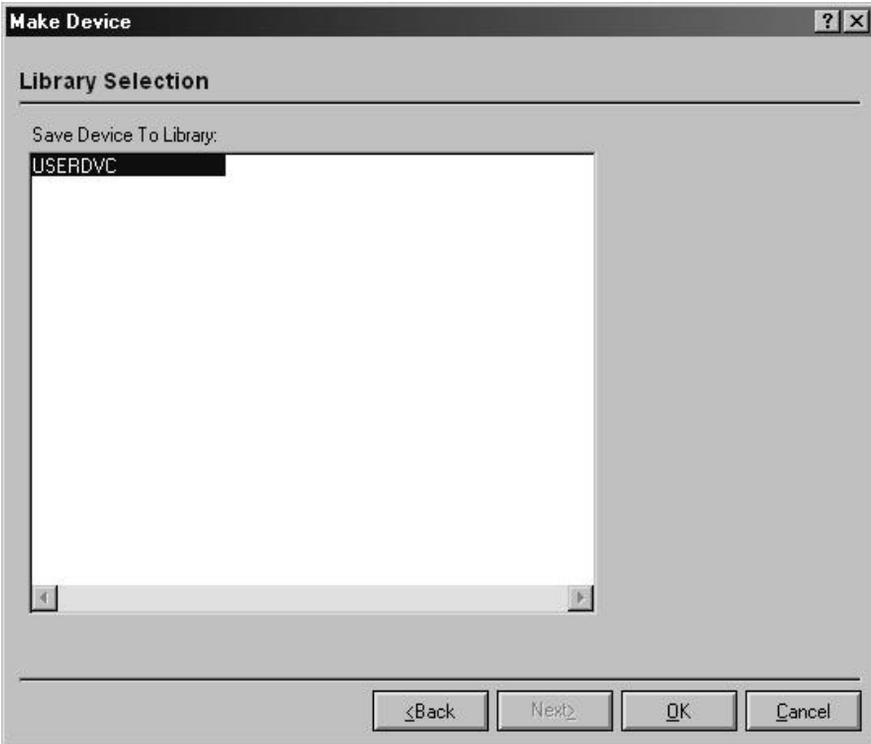
- In the *Data* directory of your Proteus installation. You can change the location that Proteus search for data sheets by clicking the *Path Settings* button on the *Data Sheet Not Found* dialogue form that appears when Proteus cannot find a data sheet.
- On an FTP server - either on the Internet or on your company Intranet. You may need to enter a user ID and password for some servers.
- On a CD or CD-R.

The general idea is that there should be a central repository for data sheets, and that Proteus copies them to individual users installations on as-needed basis.

Referencing a help topic is most useful if you are creating complex models for the ProSPICE simulator, and need to create documentation to be associated with the model. This is likely to be of relevance only to advanced users and model developers.

Library Selection Page

The final page allows you to select into which library the new device will be stored. Only read/write (as opposed to read only) device libraries are displayed.

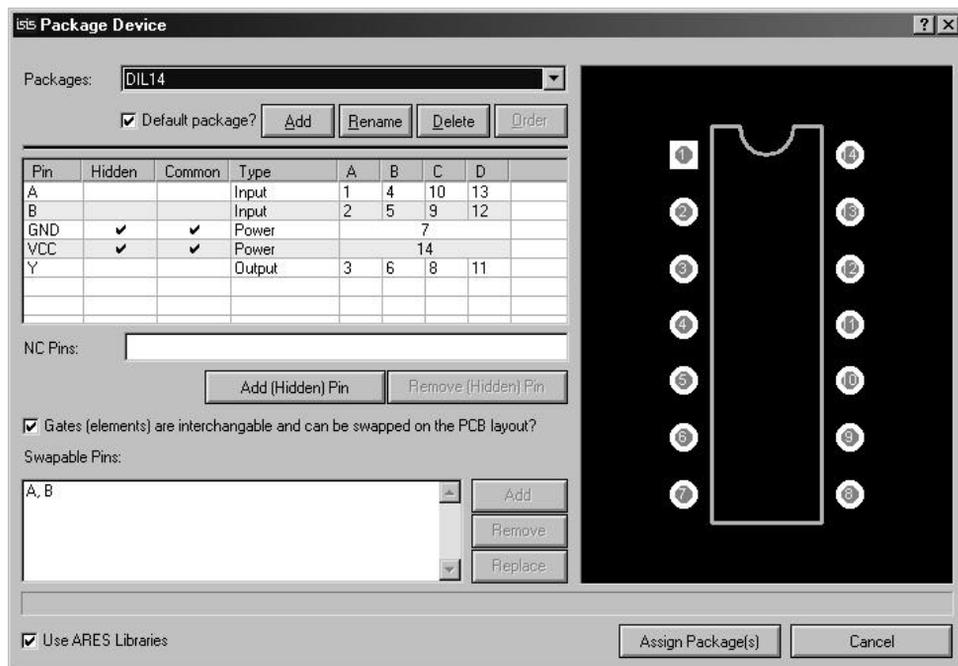


Once you click the OK button, the device will be stored to the selected library. Also, if it exists within the design you will be prompted as to whether you wish to update all the components that use it with the new definition.

The Visual Packaging Tool

The visual packaging tool provides a graphical environment in which to assign one or more PCB footprints to a schematic part. For each PCB package, a table mapping pin numbers to pin names is created such that different packagings can have different pin numbers for the same schematic pin.

The packaging tool also facilitates the entry of different pin numbers for each element in a multi-element part, and each bit of a bus pin.



The packaging tool can be invoked in one of two ways:

- By clicking the *Add/Edit* button of the *Packaging Page* in the *Make Device* dialogue form.
- By placing the element or elements of the device to be packaged on the schematic, tagged them and selecting the *Packaging Tool* command from the *Library* menu.

The second method is the only way to package heterogeneous multi-element parts.

As with the *Make Device* command, detailed context sensitive help is available on all the fields. To see the help on a particular field, click the '?' at the top right of the form, and then click the field itself.

Packaging Selector

The *Packages* section of the packaging tool shows a list-selector for the packagings that have been defined so far and buttons for adding, renaming, deleting and ordering them. One package may be selected as the default, and this is the one that will be used for newly placed components.

Assuming that you have ARES installed, the *Add* button will launch a copy of the package library browser enabling you to choose a footprint for the new packaging.

Pin Grid

The main business of the packaging tool is carried out using the pin grid. Here you can enter pin numbers for each pin in each element, and also select which pins are common between the elements of a multi-element device.

The electrical type is displayed for information only, and cannot be changed here - you must define it when placing the individual pins around the device element.

Number of Elements

This field determines the number of pin-number columns that appear in the pin grid. For a multi-element heterogeneous part, it should represent the total number of elements.

Package Viewer

The package viewer displays the PCB footprint chosen for the current packaging. As each pin is assigned to the pin grid, it is highlighted such that you can see which pins remain unassigned. Also, if you point at a pin with the mouse, ISIS will display the associated name (if assigned) and the pin number.

If the text cursor is in the pin grid, then clicking on a pin in the package view will enter its number into the pin grid and the pin will highlight to show that it is now assigned.

Hidden Pins

Hidden pins are generally used to specify power supply connections to components in such a way that they do not clutter up the schematic. A hidden pin can be defined in one of two ways:

- By clearing the *Draw Body* checkbox in the *Edit Pin* dialogue form. This must be done when making the device element. Such a pin will appear in the *Pin Grid* when the packaging tool is first invoked, and will be present in all packagings.
- By clicking the *Add Hidden Pin* button. This creates a hidden pin that is specific to a given packaging, and is implicitly common to all its elements.

Common Pins

A common pin is one which has the same pin number on all the elements of a multi-element device. Typically this will apply to power pins and enable/strobe pins that are common to all buffers within a multi-element driver.

If a common pin is not hidden, then any wiring to it on one element will be connected wiring on the other elements.

A common pin is also deemed to be associated with all the elements in a heterogeneous multi-element part, even if it is not present on some of them. This has implications for the gate swap rules within ARES.

Gate Swap

Selecting this checkbox will flag to ARES that the gate elements can be swapped.

 See page 173 for more information about Pin and Gate Swaps.

Swappable Pins

The swappable pins section allows you to define groups of pins that are electrically interchangeable. For example, the two inputs A and B of a 7400 quad NAND gate are electrically identical and could be swapped on the PCB for convenience of wiring.

You can add a swap-group to the list by highlighting the pins in the pin grid (hold the CTRL key down and click left on the pin names), and then clicking the *Add* button.

 See page 173 for more information about Pin and Gate Swaps.

NC (Not Connected) Pins

When packaging a large device, it may be useful to 'prove' that all the pins have been accounted for by verifying that all the pins are highlighted in the *Package View*. Since some pins may be specified as non-connected, it is useful to be able to record that they have been accounted for. This can be achieved by entering their numbers in the *NC Pins* field. Use commas to separate the pin numbers.

Making a Single Element Device

Most of the devices you make will have a one-one correspondence between the schematic symbol and the PCB package. In these cases, the procedure is pretty much the same as that for creating a device element, as described on page 108.

To make a single element device

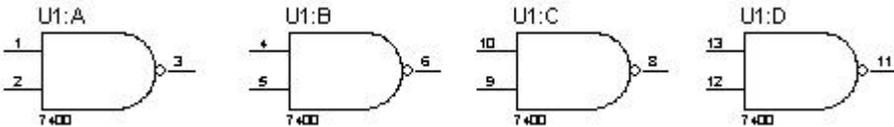
1. Place graphics objects to define the device body in the appropriate graphics style(s).
2. Place device pin objects to represent the pins.
3. Annotate the pins to assign name and types using any of the standard editing techniques (see *Editing An Object* on page 36) or the *Property Assignment Tool*.
4. Tag all the objects that comprise the device. Then invoke the *Make Device* command.
5. Enter a name and reference prefix for the device in the *Device Properties* page, then click *Next* to move to the *Packaging* page.

6. Assuming that the device has a PCB footprint, click the *Add/Edit* button and use the *Visual Packaging Tool* to assign a package and pin numbering. When you are done, click *Assign Packages* to return to the *Packaging* page.
7. Click *Next* to move to the *Component Properties* page and enter any property definitions and default values for other component properties.
8. Click *Next* to move to the *Data Sheet* page and specify the location of any data sheet or help topic that you wish to associate with the device.
9. Click *Next* to move to the *Library Selection* page and choose the library in which you wish to store the device.
10. Finally, click OK to complete the process.

If you prefer, you can also assign pin numbers for single element devices at step [3]. In this case, the pin numbers will appear automatically within the *Visual Packaging Tool*.

Making a Multi-Element Homogenous Device

A multi-element homogenous device is one like a 7400 in which the physical part consists of several *identical* elements which you wish to place as separate components on the schematic. To handle such a device, ISIS must allow for different sets of pin numbers to be applied to the same element. For a 7400, there are 4 sets of pin numbers - one for each gate - and which set is used for a given gate is determined by the component reference suffix. For example, if you label a 7400 gate as U1:C then ISIS will use the third set of pin numbers: 8, 9 & 10. This is all handled by the *Visual Packaging Tool*.



To make a multi-element homogenous device

1. Make the device body and place its pins as discussed on page 108.
2. Annotate the pins to assign name and types using any of the standard editing techniques (see *Editing An Object* on page 36) or the *Property Assignment Tool*.
3. Tag all the objects that comprise the device. Then invoke the *Make Device* command.
4. Enter a name and reference prefix for the device in the *Device Properties* page, then click *Next* to move to the *Packaging* page.
5. Click the *Add/Edit* button on the *Packaging* page to launch the *Visual Packaging Tool*.

6. Specify the number of elements in the device and then fill out the pin number fields for each element in turn. Check the *common* column to indicate pins (such as power pins) that have the same number on all elements.
7. Proceed as for a single element device from step 7.

Making a Multi-Element Heterogeneous Device

A multi-element heterogeneous device is defined as a part which consists of several *different* elements, each of which will be placed as a separate component on the drawing. The most common example is probably a relay where you have a coil and one or more sets of contacts. The requirement is to place the coil at one point in the drawing, and the set(s) of contacts elsewhere.

As with a multi-element homogenous device, the *Visual Packaging Tool* is used to deal with the assignment of pin numbers to each element.

To make a multi-element heterogeneous device

1. Place graphics objects and device pins to define appearance of the elements. Make sure the elements are sufficiently separated in the *Editing Window* so that you can tag the objects that comprise each one by themselves.

 The process for creating a device element is described in detail on page 108.

2. Annotate the pins to assign pin names and types only.
3. Tag all the objects that comprise the first element. Then invoke the *Make Device* command from the *Tools* menu. Enter the *Device Name* with in the form

NAME : A

where *NAME* is whatever you want to call the part as a whole. The suffix :A tells ISIS that this is the first element of a heterogeneous device.

4. Click *Next* twice to move to the *Component Properties* page and enter any property definitions and default values for other component properties. Note that you do not invoke the packaging tool at this stage for a heterogeneous part.
5. Click *Next* to move to the *Data Sheet* page and specify the location of any data sheet or help topic that you wish to associate with the device.
6. Click *Next* to move to the *Library Selection* page and choose the library in which you wish to store the device element and click OK to actually store it.
7. Repeat the procedure from step [3] for the other elements, giving them names such as

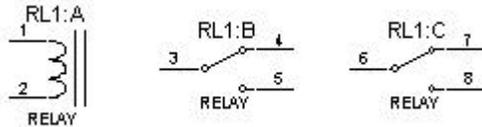
NAME : B , *NAME* : C

and so on.

8. Select the *Component* icon and place one instance of each element of the device in a free area of the schematic.
9. Drag a tag box around the placed elements, and then invoke the *Packaging Tool* from the *Library* menu. You should see that there are pin number columns for each element, but that pins which are unavailable for a particular element have '---' in the number column.
10. Create a packaging and enter the pin numbering in the usual way.
11. Click the *Assign Packages* button to store the packaging into the library parts. Note that the library part that you created for each element of the device will be updated.

An Example

Since this is a rather complex process, we will clarify it further with a simple example. Consider a relay coil and contacts as shown below.



Having made the device elements RELAY:A, RELAY:B and RELAY:C, you would invoke the *Visual Packaging Tool* and set up the *Pin Grid*, as follows:

Pin	Hidden	Common	Type	A	B	C	
C1			Passive	1	---	---	
C2			Passive	2	---	---	
COM			Passive	---	3	6	
NC			Passive	---	4	7	
NO			Passive	---	5	8	

Making a Device with Bus Pins

The ISIS device library system has the capability to support devices in which a single pin on the drawn component represents several pins on the physical part. Such a pin is called a *bus pin* and is intended to facilitate the efficient representation of microprocessors and their support chips. As with multi-element devices, the process of entering pin numbers for each bit of the bus is achieved with the *Visual Packaging Tool*.

To make a device with bus pins

1. Make the device body and place its pins in the same way as for a single element device. You must use the *Bus* pin, or a user defined pin with a *Busnode* marker, for the bus pins.
2. Annotate the pins to assign pin names and types only. The bus pins must be given their full bus specification as in $D[0..7]$. However, you can choose to hide the bus range - i.e. the $[0..7]$ part - from the schematic by unselecting the *Draw Bus Range* checkbox.
3. Tag all the objects that comprise the device, invoke the *Make Device* command.
4. Enter a name and reference prefix for the device in the *Device Properties* page, then click *Next* to move to the *Packaging* page.
5. Assuming that the device has a PCB footprint, click the *Add/Edit* button and use the *Visual Packaging Tool* to assign a package and pin numbering. You will see that each bit of the bus is given its own row in the *Pin Grid*. Beyond this, the process for assigning pin numbers is exactly the same as for an ordinary part.

When you are done, click *Assign Packages* to return to the *Packaging* page

6. Proceed as from step 7 for making a single element device.

Property Definitions and Default Properties

There are a variety of applications in which it is useful to arrange for library parts to carry default user properties which are automatically assigned to each component as it is placed. Some common examples include:

- **PACKAGE** properties for PCB design. Most of the library parts in the supplied libraries carry these properties pre-defined for through hole packaging.
- **PRIMITIVE**, **MODEL** and **MODFILE** properties for VSM simulation. Again, all our library parts for which there is simulator support carry these default properties.
- Stock and/or supplier order codes. If you add default properties for these to your libraries, then they can be included in the Bill of Materials report.

To add a default properties to a device whilst making it:

1. Follow the steps in *Making A Device Element* on page 108.
2. When you get to the *Make Device* dialogue form, use the *Component Properties* page to assign property definitions and default values for the properties..
3. Store the device to a library in the ordinary way.

To add to or edit the default properties of an existing device:

1. Pick, place and tag an instance of the device. There is no need to decompose it.

2. Invoke the *Make Device* command and click *Next* to move to the *Component Properties* page. Any existing property definitions will be displayed.
3. Edit the properties as required.
4. Click *Next* twice more and then store the device back to a library by clicking OK.

Default properties may also be assigned to library parts *en masse* by applying an ADI script from with *Library Manager*.

Dealing with Power Pins

The handling of power pins tends to be a somewhat confusing subject with different schematic capture packages doing different things. Further difficulty arises from the need to accurately define what happens with the power rails whilst not cluttering up the drawing with what is essentially trivial information.

ISIS provides you with a variety of approaches to handling power pins:

- Make the power pins visible and physically wire them to the appropriate points in the circuit. This has the advantage that you can see exactly what is connected where, but tends to be very inconvenient where there are lots of ICs on a schematic.
- Hide the power pins and let ISIS default their connections to like named nets. A hidden VCC pin will thus connect to the VCC net, and a hidden GND pin to the GND net.
- Hide the power pins and specify explicitly which nets they connect to using like named user properties. For example, the user property

VCC = +5V

applied to an object with a hidden VCC pin will cause that pin to be connected to the +5V net.

You can, of course, use a mix of these techniques as appropriate to different parts of a drawing. In general, we have found the use of hidden power pins to be almost essential for digital designs whereas in analogue work, it is often simplest just to wire up the power pins to the appropriate supply rails. This state of affairs is reflected in the construction of the supplied library parts.

To create a hidden pin on a simple, single element device:

1. Place a pin object for the pin in the usual way.
2. Bring up the *Edit Pin* dialogue from by clicking right then left on the pin.
3. De-select the *Draw body* checkbox. This hides the pin body, name and number irrespective of the settings of the *Draw name* and *Draw number* checkboxes.

4. If the pin is to be a power pins, set the pin type to *Power*.

The existence of the pin object will then be marked only by a dark blue cross at its node end, and the pin will not be drawn at all on any device in which it is included. At netlist time, it will be automatically connected to a net with the same name as its pin name, unless explicitly overridden with an appropriately named user property.

To create a hidden power pin within a packaging:

1. Create the device element(s) in the usual way, but do not place pin objects for the hidden pins.
2. When creating the packaging(s), use the *Add Pin* button to create an extra row within the *Pin Grid* for the hidden pin.
3. Type in a name for the hidden pin e.g. VCC, and then assign the pin a number

As with the other type of hidden pin, at netlist time ISIS will connect all hidden pins specified in the packaging to nets with the same name as the given pin names, unless you override this action with an appropriately named user property.

To override a hidden pin's net:

1. Tag *all* the elements of the component for which the hidden pin exists. This is most easily achieved using the *Search and Tag* command with its *Begins With* option.
2. Use the *Property Assignment Tool* to assign a user property of the form.

PINNAME=NET

where *PINNAME* is the name of the hidden pin, and *NET* is the name of the net you want it connected to.

Editing an Existing Device

Any device element, whether a self contained, single element device, or a constituent of a multi-element may be broken into its constituent pins and graphics using the *Decompose* command from the *Library* menu.

To edit device graphics or pins:

1. Place an instance of the device as a component.
2. Tag the object by pointing at it and clicking right.
3. Select the *Decompose* command from the *Library* menu. This will break the device into 2D graphics, pins and possibly an *Origin* marker. You will also get a text script which contains the device name, prefix, packagings and any default component properties.
4. Add, delete or edit the 2D graphics, pins and markers as required.
5. Reconstitute the device using the *Make Device* command as previously described. If you tag the text script generated by the *Decompose* command as well as the graphics before making the device then you can avoid having to re-enter the device properties.

If the library part is in use in the current drawing, you will get a prompt requesting whether to perform a *Design Global Update*. If you select this, then all components on the design that use the device will be updated with the new one using the same mechanism as described for the *Pick* command. Note that the properties of already placed components will not be modified as ISIS doesn't know which properties have been manually edited.

To edit device properties:

1. Pick, place and tag an instance of the device. There is no need to decompose it.
2. Invoke the *Make Device* command and click *Next* to move to the *Component Properties* page. Any existing property definitions will be displayed.
3. Edit the properties as required.
4. Click *Next* twice more and then store the device back to a library by clicking OK.

As with device elements, you will be prompted as to whether you want to update components in the current design.

To edit device packagings:

1. Pick, place and tag an instance of the device. There is no need to decompose it.
2. Invoke the *Packaging Tool* command from the *Library* menu.
3. Modify the packagings as required.
4. Click *Assign Package(s)* to store the changes back into the device library.

As with device elements, you will be prompted as to whether you want to update components in the current design.

Two other points are worth making:

- Remember that all design files carry with them their own copies of the library parts that they use. Consequently, changing a part in a library will not directly change any designs that make use of it. To effect such a change, you must load the drawing into ISIS and use the *Pick* command from the *Library* menu to effect a replace operation.
- When ISIS is installed, our own libraries are set to be read-only. This is to deter you from changing them - an action which would backfire on you when we issue updates. It follows that if you wish to change any of the supplied library parts, you should store them to USERDVC.LIB, or some other library of your own.

MULTI-SHEET DESIGNS

MULTI-SHEET FLAT DESIGNS

Introduction

With very large designs, or just to create some structure in smaller ones, it is common practice to draw the various sections of the design on separate sheets. Connections between the sheets are then indicated by means of common net names. For example, if two nets on different sheets are both labelled as MREQ, then they are assumed to be connected - see *Net Names* on page 135 for more information on net names.

Design Menu Commands

ISIS supports multi-sheet designs and keeps all the sheets of a design in one file. Three commands on the *Design* menu give you all the facilities you need:

- *New Sheet* - creates a new root sheet and loads it.
- *Goto Sheet* - presents a menu of the sheets enabling you to move about the design. For a hierarchical design, the selection consists of the entire hierarchy tree and you can thus move instantly to any sheet in the design.
- *Remove Sheet* - removes and deletes the current sheet. You can only delete root sheets, and you cannot delete the last root sheet.

The titles presented by the *Goto Sheet* command are taken from the *Sheet Title* field of the *Edit Sheet Properties* dialogue form or else the *Sheet Name* if no sheet title has been given.

The sheet names assigned to sheets also determine their ordinal position in the design - i.e. the order in which they will print if you print out the lot. New root sheets start with the names ROOT10 , ROOT20 etc. and this gives you room to insert ROOT15 or whatever if you want a sheet in between.

HIERARCHICAL DESIGNS

Introduction

A hierarchical design is one which consists of two or more levels of sheets. The highest level is likely to be a block diagram showing the structure of the overall system and each block will have a sub-sheet with a section of the design on it. Depending on the complexity of the design, these sub-sheets may themselves contain further black boxes or *modules*; ISIS sets

no limit on the hierarchy depth although you are doing something very odd if you need more than half a dozen levels.

A second use for hierarchy concerns the replication of a part of a design - a simple example would be a stereo amplifier which has two mono channels and a common power supply. There is nothing to stop you simply drawing one channel, exporting it as a SEC file and then importing it to a second sheet. However, should you then wish to alter the mono circuit - even if it is only a cosmetic change - you are going to have to alter both channels. Where more than two copies of a circuit are involved this can mean serious hassle. With a hierarchical approach you have two modules labelled LEFT and RIGHT but each one is associated with the same circuit data. Naturally you still need different references for the same component in each instance of the mono amplifier and this is catered for by means of *Design Global Annotation*.

In ISIS, hierarchy also facilitates the creation and use of *Parameterized Circuits*, and is of considerable use when developing simulator models for VSM. The former subject is covered at length in *Parameterized Circuits* on page 60 whilst the latter is dealt with in the VSM manual.

Terminology

Before we delve any further into what is quite an abstract concept, we need to define some terminology...

Circuit

A circuit is a collection of components, other objects and the associated wiring in the general case. For example we can talk about the mono amplifier circuit.

Sheet

A sheet is an instance of a circuit and has a unique set of annotation data that gets mapped onto the components in the circuit. Where a sheet is attached to a module in the next level up - the *parent sheet* - we can call it a *sub-sheet* or *child sheet*. Therefore, we can say that the left and right channels of our amplifier are drawn on the left and right sub-sheets. The sheets at the top level of the design are called the *root sheets*.

Module

A module is an object which has an associated sub-sheet. There are two types of module: *sub-circuits* and *module-components*.

Sheet Property

A sheet property is a property assignment that is attached to a particular sheet, and is available for use in property expressions for any of the objects on the sheet. In hierarchical design, any user properties of the parent module become sheet properties for the child sheet.

Parameterized Circuit

A parameterized circuit is one in which one or more component values or other object properties is given as property expression involving one or more sheet properties. Since these sheet properties can be specified on the parent module (be it a sub-circuit or a module-component), it follows that the circuit itself can have different component or property values from one instance to another. Typical applications for this are filter circuits in which some resistor and capacitor values are different for each instance.

Sub-Circuits

Editing a sub-circuit in the usual way allows you to enter a reference, circuit name and possibly some user properties which become sheet properties on the child sheet. The reference would be `LEFT` or `RIGHT` in our amplifier example and the circuit name could be `AMP`.

Connections between the parent sheet and the sub-sheet are made by means of like named module ports on the left and right edges of the sub-circuit, and terminals on the child sheet.

Sub-circuits are most useful in cases where the exact interface between parent and child sheets is not clear at the outset - you can easily add and remove ports and terminals.

To set up a hierarchy with a sub-circuit:

1. Select the *Sub-Circuit* icon and drag out a box for the sub-circuit body using the left mouse button.
2. From the *Object Selector*, select and place the appropriate types of module port on the left and right edges of the sub-circuit body. You will need one port for each interconnection between the parent and child sheets. By convention, one generally puts inputs on the left and outputs on the right.
3. Either directly, or using the *Property Assignment Tool*, assign names to the module ports. These names must correspond with the Logical Terminals that you will place on the child sheet.
4. Edit the sub-circuit itself and give it an instance name (e.g. `LEFT`) and a circuit name (e.g. `AMP`). Several sub-circuits may share the same circuit name, but on a given sheet, each should have a unique instance name.

5. Point at the sub-circuit and use the keyboard shortcut for “*Zoom to Child*” (CTRL+C by default). This will cause ISIS to load the child sheet. Unless you have specified the name of a circuit that already exists, you should now see a blank drawing.
6. Select the *Terminal* icon and place terminals to correspond with the module ports on the sub-circuit.
7. Again, either directly or with the *Property Assignment Tool*, annotate them to have corresponding name which correspond with the module ports. A netlist compiler warning is generate for any module port which is not matched by a terminal.
8. Draw the circuitry for the child sheet, connecting it where appropriate to the terminals.

Module-Components

Any ordinary component can be made into a module by setting the *Attach Hierarchy Module* checkbox on the *Edit Component* dialogue form. The component's value is taken to be the name for the associated circuit, and the component's reference serves as the instance name.

Connection between parent and child is achieved by having terminals on the child that correspond with the pin names of the parent module-component. This works for hidden power pins too, although this is irrelevant if the *Global Power Nets* option on the *Edit Design Properties* command form is selected.

Module-components are most suited to handling components which need to be expanded in some way for simulation, but kept as a component for PCB design. The *Depth* control on the *Netlist Generator* command form (*Depth* on page 144) provides the means to select what will happen.

To set up a hierarchy with a module-component:

1. Select and place the component itself in the ordinary way.
2. Bring up the component's *Edit Component Dialogue* form and set the *Attach Hierarchy Module* checkbox. Also ensure that the component's reference and value are suitable as an instance name and a circuit name respectively. In general, the module component will be an IC of some sort, and so its value will make sense as the name of an attached circuit.
3. Point at the component and use the keyboard shortcut for “*Zoom to Child*” (the default shortcut is CTRL+C- more information on Keyboard configuration can be found on page 46). This will cause ISIS to load the child sheet. Unless you have specified the name of a circuit that already exists, you should now see a blank drawing.
4. Select the *Terminal* icon and place terminals to correspond with the pins of the parent component.

5. Either directly or with the *Property Assignment Tool*, annotate the terminals to have corresponding name which correspond with the component's pins. If unknown and not drawn on the screen, the name of pin may be established by pointing at its end and keying the shortcut for “*Edit Object Under Cursor*” (by default this is CTRL+E). A netlist compiler warning is generated for any pin which is not matched by a terminal.
6. Draw the circuitry for the child sheet, connecting it where appropriate to the terminals.

External Modules

Having created a module-component and its child sheet, it is possible to store the child circuit externally to the design such that it can be re-used in other designs. Furthermore, you can modify the library part for the parent component such that newly placed instances automatically bind to the child module.

To set up an external module and an associated library part:

1. Set up a hierarchy with a module component as described in the previous section.
2. Zoom to the child sheet, invoke the *Edit Sheet Properties* command from the *Design* menu, and click the *External .MOD File* checkbox. This will create a MOD file with the same name as the library part associated with the parent component.

In the first instance, this file will be created in the same directory as the design file.

3. Return to the parent sheet, tag the parent component and invoke the *Make Device* command.
4. Enter the name of the module file in the *External Module* field.
5. Click *Next* until you come to the *Library Selector* page, then click *Ok* to store the device back into its library.

Newly placed instances of the device will then attach automatically to the MOD file.

Moving About a Hierarchical Design

There are two ways to move about the hierarchy:

- The *Goto Sheet* command will display the complete design hierarchy in graphical form and you can then move directly to any sheet in the design.
- The *Zoom to Child* and *Exit to Parent* commands on the *Design* menu allow you to move one step down or up the hierarchy. The form must be used via its keyboard shortcut (default assignation CTRL+C) whilst pointing at the module whose sheet you wish to enter.

If you zoom into a module which has not yet been given a circuit name, a new internal one will be chosen automatically. You can edit the circuit name later, but this will detach the old circuit from the module rather than re-naming the circuit. Any circuits that have been orphaned in this way remain in the design file, but can be removed using the *Tidy* command on the *Edit* menu.

Design Global Annotation

Where several sub-circuit objects share one circuit name, you will find that editing done through one of them is reflected in them all. It follows from this that you need only draw out each circuit type once. However, each instance has its own set of component references for the objects in the circuit. This is, of course, essential for PCB design where each instance will require separate physical components to make it. The automatic annotator handles this *Design Global Annotation* seamlessly, and you need only be aware that changing a component reference on one sub-sheet does not affect the references on other instances of the circuit. It is possible to disable this feature for specialist applications - see *Non Physical Sheets* below

Non-Physical Sheets

In some applications where there are multiple instances of a circuit it is preferable for all instances to carry the same annotation. If a flat netlist is produced the equivalent parts in each instance must be distinguished and this is achieved by means of Non Physical Sheets. In our amplifier example, the first IC in each channel would `LEFT_U1` and `RIGHT_U1` - these names being produced by concatenating the Sheet Name (not the sheet title) of the parent module, an underscore and the part reference.

A Non Physical Sheet is selected via the *Edit Sheet Properties* command, having first loaded the relevant sheet. Set the *Non-Physical Sheet* checkbox if you wish the naming of components to be local on a particular sheet.

This functionality is unlikely to be useful where a PCB design is the end result but in can be useful in simulation work, or where a module represents a daughter card.

Net names on sub-sheets are always prefixed in this way, unless they are power nets and the *Global Power Nets* option of the *Edit Design Properties* command form is selected. The result of this is that net names on sub-sheets are 'local'; this is more or less essential where several instances of the same circuit are used.

NETLIST GENERATION

INTRODUCTION

A schematic diagram contains two sorts of information: graphical and electrical. The process of generating a netlist is that of extracting the electrical data and writing it in a form that other CAD programs can use. Sadly there is no single standard for netlist files with most vendors 'doing their own thing'. In such circumstances only an international standards committee or a very large and successful vendor can hope to rectify the position. The former has produced EDIF which is so complex as to be virtually useless, and no single vendor has become large enough to set a *de facto* standard. Like many others we have decided to use our own format and provide also for conversion to some of the other file formats in common use.

Our format is called *SDF* standing for Schematic Description Format. SDF is designed to be compact, human readable, and exceptionally easy to process - Visual BASIC will do nicely. SDF is also intended to be an open format - the technical specification will be provided to anyone who asks.

NET NAMES

So what exactly is a netlist? A netlist is a list of nets and, before you ask, a net is a group of pins that are connected together. In ISIS, a pin is defined by the reference of the component to which it belongs, its type (determined when the relevant device was created) and the pin name or number.

A net can also be named and one of the jobs of the *netlist compiler* is to merge all nets that have been given the same name. Connections between groups of pins can thus be indicated without having to draw wires between them. This facility is useful for avoiding clutter within a sheet and essential for specifying connections between sheets in a multi-sheet design. The following cause a name to be associated with a net:

- Attaching a wire label to any wire in the net - the net takes a name from the wire label.
- Connecting to a Logical Terminal - the net takes a name from the Terminal.

If several of the above occur with different names, the net takes on all the names and will be merged with any other nets which have any of the names. The final SDF netlist will choose one name for the net and a precedence scheme based on the types of the various name-donors is used to choose it - specifically in order of decreasing priority the ranking is:

Power Rails & Hidden Power Pins (see below)
Bi-Directional Terminals

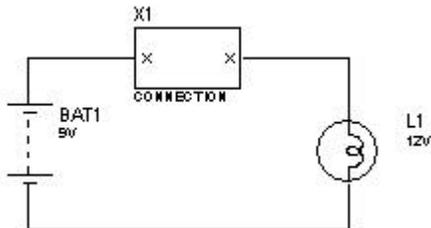
- Output Terminals
- Input Terminals
- Generic Terminals
- Bus Entries & Wire Labels

As a special case, unnamed *Power* Terminals assume the name **VCC** and unnamed *Ground* Terminals assume the name **GND**.

Net names can contain any alphanumeric characters plus the minus sign ('-') and the underscore ('_'). Spaces can be used in the PROTEUS environment but may cause problems for other software. The exclamation mark ('!') and asterisk ('*') characters have special meanings which are documented later in this section. Net name comparison is case insensitive.

DUPLICATE PIN NAMES

If a device element has more than one pin with the same name, these pins will be considered to be internally interconnected. For example, in a simulation of the circuit below, the light bulb will light, because the two X pins are deemed to be connected.



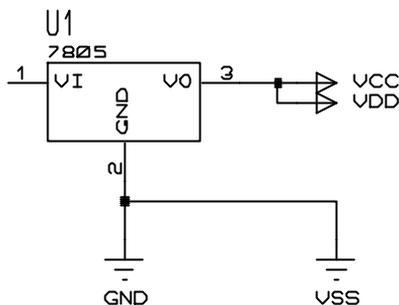
Typically, this is relevant for devices which have multiple power or ground pins, but it can also be handy when drawing schematics for wiring matrices such as keypad switch or led arrays.

HIDDEN POWER PINS

Many of the integrated circuits in the Device Libraries have hidden power pins. When the netlist generator encounters these, it creates a new net and assigns the name of the hidden pin to it. A 7400 gate will thus generate two nets - VCC carrying pin 14 and GND carrying pin 7. Since all like-named nets are merged, all like-named pins will get connected together.

You can make further connections to these nets by placing name-donor objects (e.g. Logical Terminals) with the same name and then connecting off them. For example, to connect a resistor to VCC, you would place an unnamed *Power* Terminal (which automatically takes the name VCC) and then wire from that to the appropriate end of the resistor.

In some designs, especially if there is a mix of CMOS and TTL logic, you may need to connect two groups of hidden power pins together -VCC and VDD / GND and VSS for example. This can be achieved by placing two *Generic* Terminals, connecting them together with a wire, and labelling them with the net names to be merged. A convenient place to effect this is often at the output end of the PSU circuit - the output of the regulator can connect to several terminals.



In some cases, you may want hidden power pins to connect to different nets from the pin name. This can be achieved by adding appropriately named user properties to the parts which carry the hidden power pin. For example, if attached to a 7400, the property

$$VCC = VCC1$$

would force pin 14 to be connected to VCC1. Note that in the case of a multi-element part such as a 7400, you must add the property to all the gates.

- You can view and edit the net names assigned to the hidden pins of a component by clicking the *Hidden Pins* button on the *Edit Component* dialogue form.
- Note that if you place several pins with the same name, and only some of them are hidden, then they will all be connected together, but not net name will be generated. Instead, they will be connected to the wiring attached to the visible pin(s).

SPECIAL NET NAME SYNTAXES

Global Nets

Occasionally, in a hierarchical design, it is useful to be able to make a connection on a child sheet directly to another sheet (root or child) without running up and down the hierarchy. Typically, this requirement arises either when debugging a design with VSM, or else relates to signals such as clocks which are global to the design. Either way, ISIS recognises a leading exclamation mark (!) in a net name as defining a global net. Hence a terminal labelled as !CLK will be deemed connected to any other terminal labelled !CLK, as well as to any terminals labelled just CLK on the root sheets.

Note also:

- It is not necessary to do this for power nets, unless you have de-selected the *Global Power Nets* option on the *Edit Design Properties* dialogue form.
- Unnamed power and ground terminals in fact donate the names **!VCC** and **!GND** and so are global unless labelled otherwise.

Inter-Element Connections for Multi-Element Parts

This feature was introduced specifically to deal with an obscure problem in the creation of VSM models and is unlikely to find general use. However, we document it here for completeness.

Consider a model for a 1458 dual op-amp. Clearly two instances of this model are required and they share the same power connections. However, the 1458 device only has power pins drawn on op-amp A. How then does one specify the power connections for op-amp B?

We solve this by declaring that the terminal net name *V+ on a child sheet specifies an interconnection between the nets on all child sheets attached to the same parent part, and between those nets and any V+ pins on the parent component elements. The trigger for this mechanism is the leading asterisk (*) character.

BUS CONNECTIVITY RULES

ISIS supports bus pins and wiring between bus pins. In the main, this operates intuitively, but it is necessary to be aware of the behaviour of ISIS in some of the more subtle cases that can arise with this feature.

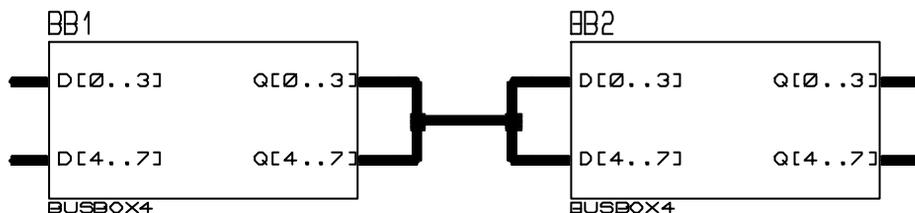
Users of ISIS 3.0X should take note that we changed the bus connectivity behaviour in release 3.1X, in order to make it more intuitive and fail safe.

The Base Alignment Rule

Within the netlist compiler, all bus entities (pins, terminals and module ports) are assigned a bus range. This is held internally in terms of a base and width so the bus D[0..7] has a base of 0 and a width of 8.

The fundamental basis for ISIS bus connectivity is that all entities on a bus (except for bus labels around a junction dot) are connected by *Base Alignment*. This means that, for example, if two bus pins D[0..3] and Q[4..7] are connected by an unlabelled bus wire, then D0 will be connected to Q4, D1 to Q5 and so on.

The base alignment rule applies, even if the bus pins being connected are different segments of the same bus. For example, the diagram below will be interpreted as making a 4 bit bus which connects Q0 to Q4 to D0 to D4 and Q1 to Q5 to D1 to D5 and so on.

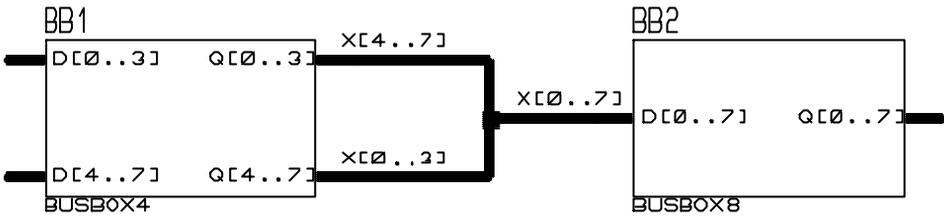


If this is not what is required, then you must use bus labels to designate the required connectivity as explained in the next section.

Using Bus Labels to Change the Connectivity Rule

The only exception to the Base Alignment Rule is in the situation where several labelled bus sections are combined at a bus junction dot. In this case, the bus sections are combined on a *Like Bit* basis.

The example overleaf shows how some bus pins can be cross-connected using bus labels:

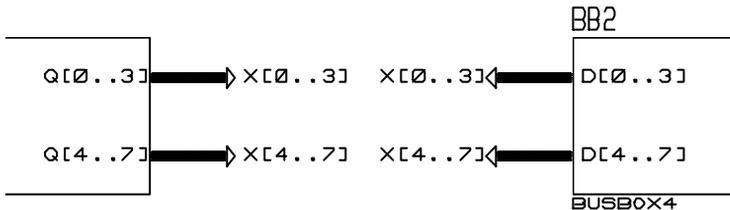


In this case, Q0 connects to D4, Q1 to D5, Q4 to D0, Q5 to D1 and so on. It is worth emphasising here that the choice of name stem for the bus labels is completely unconnected with the names of the bus pins - you could have used D[0..3] etc. but it would make no difference whatsoever to the connectivity.

It is also worth re-emphasizing that the Base Alignment Rule applies in all cases except bus labels at a junction dot, so that the connection between Q[0..3] and X[4..7] connects Q0 to X4 and so on.

Using Bus Terminals to Interconnect Buses

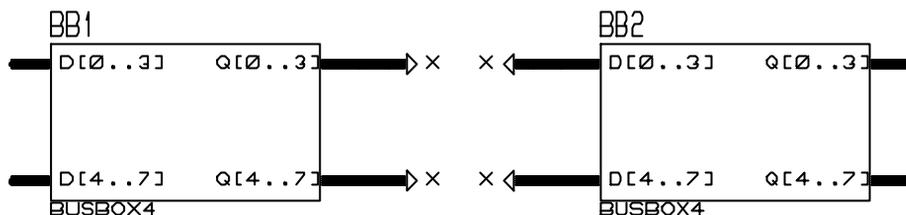
As with ordinary wires, it is possible to connect sections of bus without actually drawing in bus wiring. This can be achieved using bus labels and/or bus terminals as shown below:



If you omit the range specifier of a bus terminal or label, then it will take its range from the bus section to which it connects. The bus range is determined as follows:

- If there are bus labels present in the section, then these are combined on a like bit basis so that the meeting of X[0..3] and X[4..7] at a dot creates a range at the dot of X[0..7]. X[4..7] meeting X[8..11] would create a combined range of X[4..11].
- If there are no bus labels in a section, then the base of its range is 0 (since pins are always base aligned) and the width is that of the widest pin. To put this another way, an unlabelled bus section is always deemed to have a base of 0, irrespective of the ranges of the pins that connect to it.

The latter point does carry the potential to trap the unwary. Consider the diagram overleaf:



Since the X terminals all get ranged as X[0..3], the diagram in fact connects all four bus pins together on a 4 bit bus, rather than creating an 8 bit bus between Q and D. The moral of this is to use explicitly ranged bus terminal labels in all but the simplest cases and always if you are in any doubt.

Note that an isolated section of bus wiring in which there are no bus pins, and in which none of the labels or terminals carries a bus range, is not allowed since ISIS cannot then determine the names and number of individual bits to be interconnected. Instead, you should use a scheme such as the one shown below:



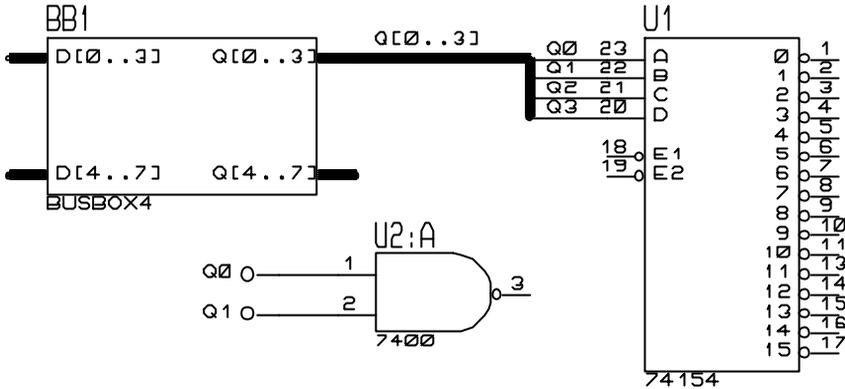
Failure to do this will result in a netlist compiler error.

Connections to Individual Bits

In most circuits, even where all the major ICs use bus pins, it becomes necessary to connect to individual bits of a bus. To do this, you need to know about the net names that ISIS generates when it encounters a bus label or terminal. You should by now appreciate that when the netlist compiler encounters an ordinary Logical Terminal, or a Wire Label, the object donates a net name to the partial net. All partial nets which have one or more net names in common are then taken to be connected.

When a bus label or terminal is encountered, it generates a set of net names, which are assigned to the partial nets that constitute each bit of the bus. The bus label D[0..7] thus generates the net names D0, D1 ... D7.

In the circuit overleaf, both the NAND gate and the 74154 are connected to Q[0..3] by this mechanism.



There are several things to note from this example:

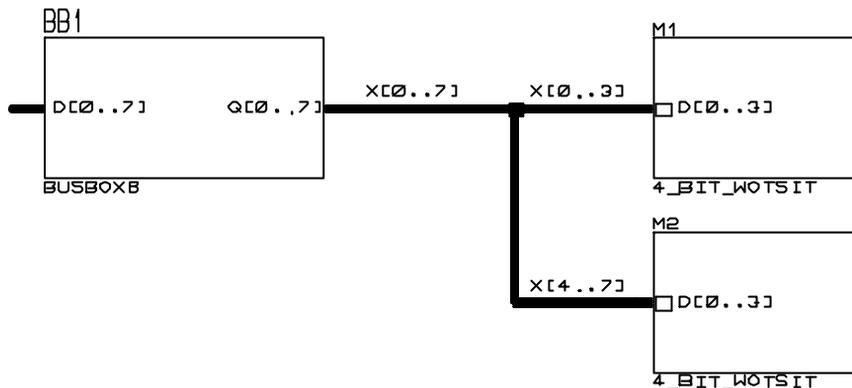
- Connections to the bus bits can be made without actually drawing a wire coming out of the bus. The bus label $Q[0..3]$ gives the bus bits the net names Q_0 - Q_3 and these can then be referred to and connected up in the usual way.
- The bus label $Q[0..3]$ is mandatory. The bus pin $Q[0..3]$ does not contribute any net names as this could lead to unwanted connectivity if there were several like named pins on the design. This behaviour is also consistent with the behaviour of ordinary, non-bus pins which do not donate net names either.

In fact, the junction between a wire and a bus has no significance at all, as far as ISIS is concerned.

- In the above circuit, you could just label the bus as Q , since the bus pin $Q[0..3]$ will contribute the range information.

Tapping a Large Bus

A not uncommon situation that arises in the use of buses is that of the need to break a large bus into several smaller buses which connect to chips which are perhaps 4 or 8 bits wide.



Here, the 8 bit Q output of the BUSBOX8 is split into two four bit buses which are then fed into the 4_BIT_WOTSIT sub-circuit modules. The Base Alignment Rule applies at the connection of X[4..7] to D[0..3] to give the desired result. The label X[0..7] is actually redundant in this case but it does no harm to be explicit.

This diagram also shows how bus connectivity can be combined with hierarchical design to give a very powerful scheme for representing array like circuitry. The module ports behave in the same way as pins for bus connectivity purposes.

General Comment & Warning

The preceding sections describe the behaviour of bus pins and labels in the contexts that we envisage them being used. Clearly, other (bizarre) possibilities may be drawn and the hope is that the explanations of how bus pins work will enable you to perceive what will happen.

However, if you are in any doubt whatsoever as to what connectivity will result from what you have drawn, we strongly recommend that you check the resulting netlist with a text editor before simply assuming that what you have drawn will result in the connectivity that you expect.

This said, you will not easily go wrong if you remember the following two points:

- The Base Alignment Rule always applies, except for the case of labelled bus sections being merged at a bus junction dot.
- Use un-ranged bus terminal labels only in simple cases; an unlabelled bus terminal or module port will take a base of 0 if there are no other bus labels in the bus section.

GENERATING A NETLIST FILE

The *Netlist Compiler* command on the *Tools* menu first presents the *Netlist Compiler* dialogue form and then a file selector from which you choose a filename for the netlist. In most cases, the default settings will suffice; they cause a flat, physical netlist to be generated in SDF format for all sheets in the design. The functions of the various controls are as follows:

Format

A variety of netlist formats can be generated - SDF is the Labcenter Native format, the others are used for interfacing to 3rd party software. Some brief notes are provided in *Netlist Formats* on page 145 and detailed application notes regarding some packages are available from our technical support department. Since 3rd party software is subject to changes beyond our control, it is preferable to supply up to the minute application notes rather than commit things to the printed manual.

Logical/Physical/Transfer

A logical netlist contains pin names whereas a physical netlist contains pin numbers. A more subtle effect is that in a physical netlist, the elements of multi-element parts such as the 7400 will be grouped (appearing, for example, as U1) whereas in a logical netlist they are kept separate (appearing, for example, as U1:A, U1:B, U1:C, U1:D). A logical netlist will generally be used for simulation whilst a physical netlist is best suited to PCB design.

Transfer mode is used in specialist applications of ISIS only, for which separate documentation is provided.

Scope

The default scope is the whole design. Current scope restricts the netlist generator to just the sheet that is loaded. This is generally used when you wish to extract the netlist from a child sheet - perhaps a daughter card which is to be represented as a layout in ARES, but which is part of a larger design that needs to be simulated whole. It is also possible to create a 'virtual test jig' by having a parent sheet which contains circuitry to simulate the components on the child sheet.

Depth

The default mode is to flatten the design. In this case objects with sub-sheets will be replaced by their implementation. If the netlist is not flattened then this replacement will not occur and objects with sub-sheets will appear as-is in the partslist and netlist.

The most common reason for not flattening a design is if some components have attached child sheets representing their simulator models, but you wish to produce a purely physical netlist for PCB design. Note that if you wish to use this approach, you cannot have a hierarchical design as well - even ISIS does not cater for selective flattening! However, with VSM, this is not an issue since you can easily link external model files to the design proper, using **MODFILE** properties. See the VSM manual for further details.

Errors

Various kinds of error can occur whilst a netlist is being generated - the most common is two parts with the same name. If any errors do occur, they will be displayed in the pop-up text viewer window.

NETLIST FORMATS

SDF

Schematic Description Format - the native Labcenter format - used by VSM, ARES and any future Labcenter EDA products. Also very easy to read in and process to other forms. Contains all textual/connectivity information contained in the DSN file.

Use *Physical* mode for ARES.

BOARDMAKER

Netlist format for Tsien Boardmaker II.

The user property PACKAGE is used for the package name if the file is generated from the *Netlist Compiler* dialogue form. If you want to use a different field, you must invoke the netlist generator from a script file.

Use *Physical* mode.

EEDESIGNER

EE Designer III netlist format.

Comment on packaging as for Boardmaker.

Use *Physical* mode.

FUTURENET

Netlist format used by Dash design tools. Also popular for general purpose netlist transfer.

Use *Physical* mode for Pin List, *Logical* mode for Net List.

MULTIWIRE

Multewire netlist format. Also used by EAGLE PCB design.

File format does not contain packaging data.

Use *Physical* mode.

RACAL

RACAL netlist format. Used by RedBoard, CADSTAR etc.

Comment on packaging as for Boardmaker.

Two files are created with CPT and NET extensions.

Use *Physical* mode.

SPICE

SPICE netlist format, also ideal for P-Spice.

Ground net will be node 0, unnamed nets start at 1000; numeric net will be fed straight through. The file SPICE.LXB can be re-named to SPICE.LIB to obtain a set of SPICE compatible models.

Use *Logical* format.

Do not use this format for creating PROSPICE models – use the normal MDF output from the *Model Compiler* – it is far more flexible.

SPICE-AGE FOR DOS

SPICE-AGE netlist format for Those Engineers analogue simulator.

The file SPICEAGE.LXB can be re-named to SPICEAGE.LIB to obtain a set of SPICE-AGE compatible models.

TANGO

Tango netlist format, also used by Protel and others. A good general purpose format too.

Comments on packaging as for Boardmaker.

Use *Physical* mode.

VALID

Valid netlist format used for transfer of ISIS designs to VALID Transcribe package.

Use *Transfer* mode.

VUTRAX

Netlist format for use with VUTRAX PCB design software as used by several design bureaux

Comment on packaging as for Boardmaker.

REPORT GENERATION

BILL OF MATERIALS

Generating the Report

ISIS can generate a Bill Of Materials report that lists all the components used in the current design. Facilities are provided to give you extensive control over the contents and arrangement of this report.

To actually generate the report, all you have to do is select the name of the Bill Of Materials configuration you want from the *Bill Of Materials* sub-menu on the *Tools* menu. The report will appear in an external *Viewer* and can then be saved or printed as required

Report Formats

ISIS is able to generate Bill Of Materials in four formats:

HTML (Hyper Text Mark-up Language) Format

The Bill Of Materials report will be an HTML or MHTML document based on a HTML template file. The template is an HTML document that contains markers for where information (such as quantities, component references, etc.) are to be inserted. The use of a template allows you to customise the look of the final report and even allows you to incorporate images such as company logos.

For a report based on a template without images, the output file is an HTML document that can be viewed in any web browser such *Internet Explorer*, *Netscape* or *Opera*.

However, HTML files cannot contain image data (a HTML file with images only contains 'links' to external image files) the use of an HTML file for the report output based on an HTML template with images is problematic. Specifically, moving the HTML report file - for example, emailing it a colleague - requires not only the document to be sent but also the associated image files. The need to always copy several (rather than one) file is both cumbersome and prone to error.

As a solution to this, where the HTML template file contains links to images, ISIS will output the report as a MIME-HTML (MHTML) document. An MHTML document is a single file (usually with a .MHT or .MHTML extension) that contains not only the HTML document but also all the associated images needed by the document.

The MHTML format is a formal internet standard (RFC2557) and is often used by e-mail packages for sending ‘pretty’ emails with images. However, at present, only Microsoft™ *Internet Explorer* directly supports the display of MHTML documents.

ASCII Format

The Bill Of Materials report will be a plain ASCII text file that uses spaces to pad the information in to columns.

Such files are easily ported (say, via email) but are also the least ‘pretty’ as you need to display them with a monospaced (fixed-pitch) font in order that the text lines up in to columns.

Compact Comma-Separated Variable (CSV) Format

The Bill Of Materials report will be a plain ASCII text file in *Comma Separated Variable* (CSV) format. This is a popular format of data exchange commonly used by spread-sheet programs such as Microsoft™ *Excel*.

As is common with CSV files, the first line is the name of the fields separated by commas. The remaining lines of the file is raw Bill Of Materials information (quantities, references, values, etc.) separated by commas (Information that contains a comma will be automatically enclosed inside double quotation marks).

The report is termed ‘compact’ because the normal Bill Of Materials operation of grouping like components together is still performed and the CSV output contains one such grouping per line. For example:

```
Resistors, 2, "R3, R15", 18k
```

Full Comma-Separated Variable (CSV) Format

This is a CSV output similar to the ‘compact’ version above except that the normal Bill Of Materials ‘grouping’ of like components is suspended and instead each component in the design is output separately, one component per line.

```
Resistors, "R3", 18k  
Resistors, "R15", 18k
```

Bill Of Materials Configuration

The contents and formatting of a Bill Of Materials report is determined by a configuration script. A number of such scripts (one each of the available formats) are created during installation. You can create, edit or delete scripts using the *Set BOM Scripts* command on the *System* menu. The command displays a *Edit BOM Scripts* dialogue form.

The top part of the *Edit BOM Scripts* form controls which script is currently selected for editing and includes three buttons (*New*, *Rename* and *Delete*) that let you create new scripts, rename the currently selected script or delete it.

The lower part of the form allows you to edit the various configuration parameters for the currently selected script:

Bill Of Materials Output Format

This drop down list lets you select the output format of any report generated for this script. There are four options: ASCII, HTML, CSV (Compact) and CSV (Full) and these correspond directly with the formats detailed in the section above.

Number Of Columns For References (ASCII Output)

This field is only displayed where an ASCII report format is selected and allows you to specify the number of character columns used to for listing component references. Depending on what other component fields you have chosen to display, you may wish to adjust this value to make best use of your page width.

The text to the right of number, *xxx columns used*, indicates the total number of character columns used so far and includes the (fixed) five character quantity column plus the total of all field columns. You may want to use this as a guide where trying to keep the total width of a line constant (e.g. at 80 character columns for printing purposes).

HTML Template File

This field is only displayed where an HTML report format is selected. The file specified in this field will be used as the 'template' for the report output. Presently, ISIS ships with two HTML templates (found in the *templates* sub-folder of the Proteus installation): DEFAULT.HTM and FANCY.HTM.

The `default.htm` template is the template used by the pre-configured HTML configuration. The template is reasonably neutral and doesn't contain links to images so the resulting reports are always plain HTML files (.HTM extension).

The `fancy.htm` template is intended as a sample of how to create your own HTML templates. The top of the file is an explanation of the template is written and how it is used by the Bill Of Materials generator.

The file contains (links to) two images (our Labcenter Electronics logo and a small blue bullet) and so any report generated using this template will be an MIME-HTML (MHTML) file (.MHT extension).

Categories

The *Categories* list and the four buttons beneath it (*Add*, *Delete*, *Edit*, and *Order*) control the addition, removal, editing and ordering of the categories of components in the Bill Of Materials report.

A category allows you to group components together within the report according to their component reference prefix. For example, you may want a *Resistors* category that contains all components whose reference begins with an the letter R (i.e. R1,R13,R27, etc.).

For each category you can specify four component reference prefixes so, for example, the *Resistors* category might contain not only components whose reference begins with the letter *R* but also those whose reference begins *RV* (usually, variable resistors).

Each configuration script must contain a *Miscellaneous* category. This category will list (contain) all components that are not grouped in to any other category. You can specify which category is the *Miscellaneous* category on the *Edit BOM Category* dialogue form.

The *Add*, *Delete*, *Edit* buttons are self-explanatory. Note that you can also edit a category by double-clicking the left mouse button on its name in the *Categories* list.

The *Order* button allows you to adjust the order of the categories in the list. The Bill Of Materials report will contain categories listed in the order they appear in the list so you may want to adjust the ordering to suit your preferences (say, IC's first, discrete components second, miscellaneous last).

Fields (Columns)

The *Fields (Columns)* list and the four buttons beneath it (*Add*, *Delete*, *Edit*, and *Order*) control the addition, removal, editing and ordering of the data fields (columns) that are output for each component.

A field is the value of a component property, for example, its value or perhaps a property you have assigned yourself such as a stock-code, tolerance, or cost.

For each field, you specify the name of the component property that is to be output. Additionally, you can specify the field width (for ASCII report formats), a prefix and suffix to go around the value (for example, a currency symbol), a default value (if the property isn't defined), as well as whether or not the field values should be sub-totalled/totalled.

The *Add*, *Delete*, *Edit* buttons are self-explanatory. Note that you can also edit a field by double-clicking the left mouse button on its name in the *Categories* list.

The *Order* button allows you to adjust the order of the fields in the list. The Bill Of Materials report will contain field values in columns ordered left-to-right in the order they appear in the list.

Editing Bill of Materials Categories

To edit a Bill Of Materials category select the name of the category from the *Categories* list on the *Edit BOM Scripts* dialogue form and then click the *Edit* button below the list. This displays the *Edit BOM Category* dialogue form.

Category Heading

This is the name of the category and is used in the report as a heading for the category. For example, a heading *Resistors* might be used to head up a category containing all the resistors in the design.

Reference(s) to match

These four fields allow you to specify the component reference prefixes that must be matched for a component to appear in this category. For example, prefixes of *R* and *RV* will cause resistors (R1, R2, R17, etc.) and potentiometers (RV1, RV2, etc.) to be listed in this category.

Mark this category as the miscellaneous category?

When checked, this category will be used as the ‘miscellaneous’ category that contains all components in the design that do not fit in to any other category.

Note that the *Reference(s) to match* fields are disabled for the miscellaneous category.

Output separate references for each component?

By default, the Bill Of Materials generator groups runs of component references together and outputs them as the first and last reference separated by a hyphen. For example, R3,R4,R5,R6,R7 would be output as R3-R7.

When this option is checked components will be listed individually and not grouped. For large numbers of components this may produce a considerably longer report.

Editing Bill of Materials Fields (Columns)

To edit a Bill Of Materials field select the name of the field from the *Fields (Columns)* list on the *Edit BOM Scripts* dialogue form and then click the *Edit* button below the list. This displays the *Edit BOM Field* dialogue form.

Component Property Name

This is the name of the component property whose value is to be displayed. You can use the *List* button to list all component properties in the current design. The list includes all manually added properties as well as many system properties. To select a property, just double click the left mouse button on its name in the list.

Title For The Column Heading

This is the ‘English’ title that will be used as a column heading. It may well be the same as the property name or it can be something more explanatory. Remember that for ASCII formatted output, the column must be at least as wide as the heading text so for this format, avoid verbose heading names!

Property Validation

This drop-down list lets you choose whether the report generation should be aborted if the component property doesn't exist in a component or if the value of the property is blank. For most properties it is probably best to leave this at *No Validation*. However, for something like a 'cost' field that is being totalled to give the design cost, you may want to set this to *Must exist/cannot be blank* in order to avoid components with no cost assigned to them not being accounted for.

Column Width (In Characters)

For ASCII formatted output, this is the width of the column (in characters) for the field (the column will be widened automatically to account for the width of title text).

For other formats, the values output will be clipped to this number of characters.

Prefix (to go in front of value)

Suffix (to go after value)

These two fields allow you to specify arbitrary text that will prefix and suffix the property values. For example, if the property values represent monetary costs then you may want to set the *Prefix* to be a currency symbol such as '£' or '\$'.

Apply prefix/suffix to blank (empty) values?

This option allows you to choose whether or not an empty field value (i.e. where the property doesn't exist or where the property exists but has no value) is still prefixed/suffixed.

Default Field Value

This field allows you to specify a default value for where a component property is not defined for a given component. The value is only used where the component property is not defined within a component – it is not used where a property is defined but has no value (i.e. is blank) as this is deemed intentional.

Note that a default value, when used, will not be prefixed or suffixed like a normal value. If you want this to be the case, simply duplicate the prefix/suffix text in to the default value.

Output Sub-totals/Totals For This Field?

When this field is checked the Bill Of Materials generator will attempt to parse floating-point values from the field values and will display a sub-total at the end of each category as well as a total for all categories.

Values are parsed as they are obtained from the component (i.e. the prefix/suffix is ignored) and any *Default Field Value* will also be parsed. A value that does not parse will be taken as zero (no error is reported).

For a value to parse successfully the value must begin with a floating-point number. Alpha or punctuation characters (except a decimal separator) cannot occur before the number and similar characters at the end of the value are ignored. For example, a value of “\$10.00” will not parse because of the leading dollar – instead, use a value of “10.00” and have the ‘\$’ inserted via the *Prefix* field.

Adding Bill of Materials Parts

It is sometimes useful for one schematic component to contribute two (or more) parts to the Bill Of Materials report. A fairly common case is a for a fuse component to appear in the Bill Of Materials report along with a fuse-holder part.

You can achieve this behaviour by adding one or **BOMPARTx** properties to a component. The format of the property is:

```
BOMPART=QTY , REF , VALUE [ , PROPNAME=PROPVAL , . . . ]
```

The property name must begin with BOMPART – thus ‘BOMPART’, ‘BOMPART1’, etc. are all valid property names.

The QTY parameter represents the quantity of parts to add. This will usually be ‘1’ but can be any number as appropriate.

The REF parameter represents the component reference prefix these new parts will be given. Note that if other parts exist in the report with the same prefix, the new parts will have prefixes that follow (numerically) from the last count used. Further, parts added in this way will still be grouped by category so if you want them to appear in a particular section of the report you will need to add a *Category* to handle this prefix.

The VALUE parameter is the ‘value’ that the new part should have. This is synonymous with the ‘name’ you want the part to have (e.g. FUSEHOLDER).

The PROPNAME=PROPVAL pairs are optional and allow you specify properties for the new components. For example, if your report contains a COST field to display the COST property of components, you might add ‘COST=0.11’.

The following is an example of a **BOMPART** property you might add to a transformer such that the for each transformer in the design, the Bill Of Materials report will contain four M4 bolts costing 0.04 each and with a ‘LENGTH’ property of 40 (mm):

```
BOMPART=4 , BL , M4BOLT , COST=0 . 04 , LENGTH=40
```

The additional parts would appear in the Bill Of Materials report as BL1,BL2,BL3, and BL4, for example.

You can add a **BOMPART** property to a device when you create it (on the *Component Properties & Definitions*) tab of the *Make Device* dialogue form.

Alternatively you can add it to components once they're already in the design:

- You can add it manually by tagging and edit a component and then adding the property to the *Other Properties* field.
- You can use the *Property Assignment Tool* on the *Tools* menu to assign the property to, for example, tagged components. Use the *Search & Tag* command to first select the components to be assigned and then use the *Property Assignment Tool* to assign them. You can then use the *Property Assignment Tool* a second time to hide the property so that it doesn't clutter your design.
- You can use *ASCII Data Import* to import one or more of the properties in to the design. This method may be preferable if your **BOMPART** property contains cost/price information that fluctuates.

ASCII DATA IMPORT

Although not strictly to do with report generation we shall discuss this here as it is of most significance when viewed in conjunction with the Bill of Materials report.

The idea of ADI is that much of the data you would store within components (e.g. costs, stock codes, tolerances) will remain the same for each component type across all your designs. ADI allows you to specify data for different component types in a simple ASCII file and to import it all into a design with just one command.

The ADI source file can be created with any ASCII text editor - for example the Windows editor - NOTEPAD. The file can contain any number of separate commands, each of which starts on a new line with its command keyword and ends on a new line with the keyword **END**. There are two ADI commands, the **IF...END** command and the **DATA...END** command.

The ADI 'interpreter' is run by invoking the ADI command and then selecting an ADI file with the file selector. The commands within the ADI file are then loaded and pre-compiled - any errors will be written to the error log, which is automatically displayed if errors occur. ISIS then applies each command in the ADI file, in order, to each component in the design.

The **IF...END** Command

The **IF...END** command allows you to test the existing properties of each component via an expression, and if the expression evaluates **TRUE**, then the sub-commands within the **IF...END** block are applied to the component in succession.

The syntax is best explained by example:

```
IF DEVICE="CAP ELEC" AND NOT VALUE=10p
    VALUE=1n, HIDEKWD
    TOLERANCE, HIDE
    STOCKCODE, REMOVE
END
```

The expression to be applied to the component follows the keyword **IF**. Expressions consist of one or more terms separated by operators. Each term consists of a property value, optionally followed by an equals sign and a value.

- A term only evaluates **TRUE** if the property named exists, and in the case where a value has been specified, the property value matches the value specified character-for-character, case-sensitive.
- Operators consist of brackets - which may be used to enclose sub-expressions, and the keywords **AND**, **OR** and **NOT**. The operators **AND** and **OR** are executed left to right - there is no precedence. The **AND** operator evaluates **TRUE** only if the evaluation so far is **TRUE** and the term or sub-expression following it evaluates **TRUE**. The **OR** operator evaluates **TRUE** if the evaluation so far is **TRUE** or the term or sub-expression following it evaluates **TRUE**. The **NOT** operator is unary and has the effect of negating the result of the evaluation of the term or sub-expression that follows it.
- A special expression, consisting of the single keyword **TRUE** allows you to affect all components in the design.

In the example, the expression tests that the component is an instance of the CAP ELEC library device and that it does not have a value of 10p. If this is the case, then the three sub-commands after the **IF** expression and before the closing **END** keyword are applied to the component. Note that, because the device being tested for contains a space (CAP ELEC), it is enclosed in double quote (") characters.

Each sub-command consists of a property name optionally followed by an equals sign and a new value and/or a comma and a command. If a value is specified, the property named is either created with the new value or has its existing value modified. If a command is specified it is executed following any assignment, as follows:

```
SHOW          Both the property name and its value are made visible.
```

HIDE	Both the property name and it's value are hidden.
HIDEKWD	The property name is hidden; the visibility of the property value is unchanged.
HIDEVAL	The property value is hidden; the visibility of the property name is unchanged.
REMOVE	The property name and any value is removed from the component.

If no command is specified then the existing visibility of the property name and value is unchanged.

In the example, the property **VALUE** is assigned the value 1n and the property name is hidden, the value of the property **TOLERANCE** is left unchanged but the property/value pair is hidden and finally, the property **STOCKCODE** is removed from the component.

The DATA...END Command

The **DATA...END** command allows multiple test values to be tested against a fixed list of property names, and given a match, allows a set of separate values to be assigned to a separate set of fixed properties.

Consider the following example of a **DATA...END** command:

```
DATA DEVICE   +  VALUE : COST+, TOLERANCE, STOCKCODE-
RES           1k   : 0.01, 1%, 100-1001
RES           1k2  : 0.01, 1%, [REMOVE]
"CAP ELEC"    1n   : 0.03, 5%, 200-1001
SWITCH        *    : 0.25, [SKIP], 300-1001
END
```

The **DATA** keyword is followed by the name or names of the properties to be tested, separated by plus characters. This list is followed by a colon and then a list of the name or names of the properties to be assigned, separated by commas. Each property name may optionally be directly followed by a plus or minus character that reflects whether you want the property named and the value assigned to it to be shown or hidden respectively; no character indicates that existing visibility should be maintained.

In the example, the **DEVICE** and **VALUE** properties are tested. Given a match, new values are assigned to the **COST**, **TOLERANCE** and **STOCKCODE** properties; the **COST** property and its value being made visible and the **STOCKCODE** property and its value being hidden.

Each line between the **DATA** keyword line and the **END** keyword consists of a list of the property values that you want to test for (separated by one or more spaces), a colon, and then the list of property values you want to assign, separated by commas. Each line is

executed in turn; the value of each property named in the list to the left of the colon on the **DATA** line is tested against the corresponding test value on current line. If all property values match, then each property named on the right of the colon on the **DATA** line is assigned the corresponding assignment value on the current line.

In the example, the first line tests the **DEVICE** property for the value RES and the **VALUE** property for the value 1k; if both match then the **COST** property is assigned a value of 0.01, the **TOLERANCE** property a value of 1% and the **STOCKCODE** property a value of 100-1001. Similarly the third line tests the **DEVICE** property for the value CAP ELEC (because the device name contains a space it must be enclosed in double quotes) and the **VALUE** property for the value 1n; if both match then the **COST** property takes a value of 0.03, the **TOLERANCE** property a value of 5% and the **STOCKCODE** property a value of 300-1001.

Note that the CAP ELEC device name is enclosed in double quotes. You must do this for any of the match values to the left of the colon as these are space delimited. If this were not done, ISIS would have complained it was expecting a colon since it would have assumed you wanted to test **DEVICE** property against the value CAP and **VALUE** property against the value ELEC; the 1n would then be unexpected. Similarly, the assignment values to the right of the colon are comma delimited, so if you wanted to assign a new value that contains a comma, then you would have to enclose the value in double quotes.

There are two unusual features of the property values that ADI tests for.

- The values can contain the wild-card characters: '?' and '*'. The question mark indicates that any single character may appear in the property value being tested in same position as the question mark. The asterisk indicates that any number of characters may appear in the property value being tested between the sets of characters to the left and right of the asterisk. In our example above, we test for a **DEVICE** property with the value SWITCH but choose not to test such a component's **VALUE** property by specifying a test value of a single asterisk.
- Numerical values (including values that contain exponent suffixes such as 'k' or 'u'). Such values are converted by ISIS in to a textual representation with six decimal places and an exponent and the textual representations are then compared character by character. In our example, component placed from the library device RES and assigned the values 1k, 1000 and 1.0k would all match the test condition in the first line.

Whilst the list to the right of the colon normally consists of a new value to be assigned to the respective property named on the **DATA** line, you can instead specify a command, as follows:

[NULL]	Assign the respective property an empty string. Only the property name and the following equals sign will appear in the component's properties text block.
[REMOVE]	Remove the property; the property name and its value are removed from the component's text property block.
[SKIP]	Skip assignment to the respective property. This keyword is required as an empty assignment (nothing between the commas) will cause an error. Any existing property in the component's text property block is left unchanged.

The second line of the example specified that the **STOCKCODE** property of any component placed from a RES device with a **VALUE** of 1k2 should be removed. Similarly, line four, specifies that there should be no assignment to the **TOLERANCE** property of a component placed from a SWITCH device.

ELECTRICAL RULES CHECK

ISIS can check for simple errors in a design by examining the various pin types connected to each net. Obvious examples of errors are outputs connected together or several input pins connected together without a driving source. Terminals are also regarded as having an electrical type - an input terminal connected to an input pin provides a driving source for it.

Generating the Report

You can generate the ERC report by invoking the *Electrical Rules Check* command on the *Tools* menu. The results are displayed in the *Text Viewer* from where they can be saved or printed as required.

Do bear in mind that not all entries that appear in the report may actually be mistakes (e.g. some input pins might deliberately be left NC) and that more subtle mistakes such as wrong part values will not be detected. Nevertheless, many silly mistakes can be detected at an early stage.

ERC Error Messages

The first part of the ERC process involves compiling the netlist, and this in itself can result in warning or error messages.

The actual ERC process itself detects two basic categories of error:

- Nets so wired as to be likely to result in contentions - i.e. two outputs trying to drive in different directions, resulting in a large flow of current.

- Nets so wired such that there is no driving source. A net containing only INPUT pins will result in an **UNDRIVEN** error.

The detection of the first type of error is determined by the following table:

	PS	IP	OP	IO	TS	PU	PD	PP	GT	IT	OT	BT	PR
PS	ok												
IP	ok												
OP	ok	ok	er	wn	er	er	er	ok	ok	er	ok	wn	er
IO	ok	ok	wn	ok	ok	ok	ok	wn	ok	wn	ok	ok	wn
TS	ok	ok	er	ok	ok	ok	ok	er	ok	er	ok	ok	er
PU	ok	ok	er	ok	ok	ok	ok	wn	ok	er	ok	ok	er
PD	ok	ok	er	ok	ok	ok	ok	wn	ok	er	ok	ok	er
PP	ok	ok	ok	wn	er	wn	wn	ok	ok	ok	ok	wn	ok
GT	ok												
IT	ok	ok	er	wn	er	er	er	ok	ok	er	ok	wn	er
OT	ok												
BT	ok	ok	wn	ok	ok	ok	ok	wn	ok	wn	ok	ok	wn
PR	ok	ok	er	wn	er	er	er	ok	ok	er	ok	wn	wn

KEY:

- | | | |
|--------------------|------------------------------|---------------------------|
| PS : Passive Pin | GT : Generic Terminal | ok : No warning or error. |
| IP : Input Pin | IT : Input Terminal | wn : Warning issued. |
| OP : Output Pin | OT : Output Terminal | er : Error issued. |
| IO : I/O Pin | BT : Bi-directional Terminal | |
| TS : Tri-State Pin | PT : Passive Terminal | |
| PU : Pull Up Pin | PR : Power Rail Terminal | |
| PD : Pull Down Pin | | |
| PP : Power Pin | | |

HARD COPY GENERATION

PRINTER OUTPUT

Output through standard Windows device drivers is performed using the *Print* command on the *File* menu; the device to print to may be selected using the *Printer Setup* command.

Note that context sensitive help is available on all fields within these dialogue forms.

Specifying a Printer Device

You can specify the printer device in one of two ways :

- Use the *Printer Setup* command from the *File* Menu. All printer settings including the printer device specified will be remembered across ISIS sessions and the printer device selected will be the default for future ISIS sessions.
- Use the *Printer* button from the *Print* command on the *File* Menu. Printer settings will be retained but the default printer across ISIS sessions will remain unchanged. This is useful if you want to print to a different printer for a specific design.

Print Preview

The Print Preview appears on the far right of the *Print* dialogue form and displays the position and size of the schematic relative to the specified paper size. It will update automatically as you configure the dialogue (scaling, colour, etc..) to your requirements.

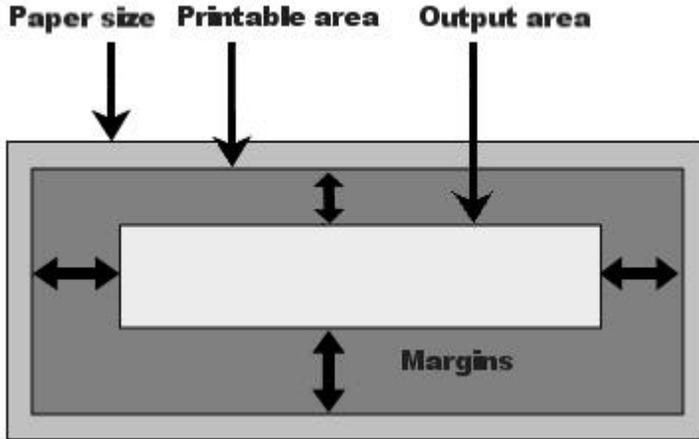
You can adjust the position of the printed schematic area by left depressing the mouse on the print preview and dragging to the desired position. Alternatively, you can choose a pre-configured position by right clicking the mouse on the preview pane and selecting the desired position from the context menu.

Note that Print Preview is disabled when '*All Sheets*' is selected from the '*What to Print*' section of the dialogue as it is not possible to show a preview of all sheets simultaneously. In this case printer output will be centred on the printable area.

Configuring Margins

When not printing all sheets you can manually configure the position of the output within the printable area. Right click on the Print Preview and select *Position Output Numerically* and enter the margins that you want between the edge of the printable area and the edge of the output area. This is, in effect, an absolute method of positioning the output area within the printable area. In general, however, we would anticipate most users to position the output

area via the mouse drag method detailed above. The graphical illustration below depicts the use of margins.



If you turn off the option to *Use Printer Margins* on the context menu the margins you specify will be from the edge of the paper itself. In most cases this is not recommended as, depending on your printer driver, you are likely to lose part of your schematic if you position it absolutely at the edge of the paper (outside of the printable area).

Scaling

In addition to a number of standard scaling options you can select *Fit to Page*.

In the case where the output area is larger than the printable area both dimensions are examined and both X and Y dimensions of the output area are shrunk according to the dimension with the largest offset (thus ensuring that the entire output area is within the printable area).

In the case where the output area is smaller than the printable area both the X and Y dimensions of the output area are expanded according to the dimension with the smallest offset (again ensuring that the resulting output area is entirely contained within the printable area). Note that this option is only available if you uncheck the checkbox for *Only Shrink to Fit*.

PLOTTER OUTPUT

Windows support for pen plotters is, unfortunately, very poor. Although drivers are supplied for HPGL and other plotters, the implementation of these drivers is very sketchy. Better drivers may be available for particular plotters, but we have not relied on this in designing the

plotter support within ISIS for Windows. Instead, we rely on the Windows plotter driver only to draw straight lines and then ISIS itself does the rest.

Plotter output is generated as for bitmap printer output using the *Print* command on the *File* menu. Where a plotter is selected as the output device, the *Labcenter Plotter Driver* option is enabled on the *Print Design* dialogue form. If this option is checked then ISIS only relies on the plotter driver's ability to draw a line on the plotter with ISIS linearising arcs, circles, etc. and rendering text in a vector font. If this option is not checked then ISIS treats the plotter driver just like any other driver and expects it to sensibly handle all Windows GDI (Graphics Device Interface) calls, including translation of functions the plotter doesn't support (for example, Bezier curves in to

Plotter Pen Colours

The plotter drivers are able to generate a multi-coloured plot according to the colours selected on the *Set Colours* command.

Unfortunately, the Windows plotter drivers do not (as far as we have been able to find out) support the direct selection of plotter pens by number. Therefore it is up to you to find out how your plotter driver maps colours onto pens and to select appropriate colours on the *Set Colours* dialogue.

CLIPBOARD AND GRAPHICS FILE GENERATION

As well as printing directly to Windows print devices, ISIS for Windows can generate output for use by other graphics applications. You have the choice of generating this output as either a Bitmap or a Windows Metafile, and you can transfer the output to the other applications either through the clipboard, or by saving it to a disk file.

Bitmap Generation

The *Export Graphics Bitmap* command on the *File* menu will create a bitmap of the board and place it either on the clipboard or in a disk file. The dialogue form offers several additional options, each of which has an associated help topic.

Metafile Generation

The Windows Metafile format has the advantage of being truly scaleable where a bitmap is not. However, not all Windows applications (e.g. *Paintbrush*) can read a metafile.

The *Export Graphics Metafile* command on the *File* menu will create a bitmap of the board and place it either on the clipboard or in a disk file.

DXF File Generation

The DXF format can be used to transfer output to DOS based mechanical CAD applications (it is better to use a clipboard metafile to transfer to Windows based CAD programs). The file is generated by a Labcenter output formatter, rather than by Windows, and as such many of the Windows based drawing appearance attributes will be lost.

Our current experience is of considerable incompatibility and disagreement between applications on what constitutes a valid DXF file. To put this another way, given six applications supporting DXF, only about 30% of file exchange pairings seem to work! This notwithstanding the fact that our DXF has been tested with the official Autodesk applications (AutoCAD, AutoSketch etc.). For Windows work, the Clipboard provides a much more reliable transfer medium.

EPS File Generation

An EPS file is a form of Postscript file that can be embedded in another document. Although popular in the world of DTP, for Windows based DTP work you are much better off transferring graphics using a clipboard metafile.

EPS file generation shares the same output options as *Printer Output* on page 163.

ISIS AND ARES

INTRODUCTION

ARES is Labcenter Electronics' high performance PCB design system offering the same user interface as ISIS and full netlist based integration with it. Using ISIS and ARES together you are guaranteed to produce a PCB layout which matches the schematic exactly and the use of a netlist in PCB design also saves you from worrying about the details of IC pin numbering when laying out the board. In addition, a netlist is a more or less essential where autorouting is to be used.

In ISIS, the *Netlist to ARES* command is located on the *Tools* menu and when invoked causes one of two things to happen:

- If no copy of ARES is running, then a copy will be started with command line arguments causing it to load the appropriate PCB file and read the netlist from ISIS.
- If a copy of ARES is already running, then ISIS locates it and sends it a message telling it to read the new netlist.

Whilst much of the discussion relates specifically to ARES, users of alternative PCB design systems will find much of interest in the following sections.

PACKAGING

In order that ARES knows which library packages must be used for given components, this information must be entered somewhere in the design process. With ISIS and ARES, the best time to enter it is either whilst creating new library parts or when editing the schematic. The property to use for this purpose is, not improbably, called **PACKAGE**. Most of the library parts in the supplied libraries have this property embedded.

Default Packaging

Most of the library parts supplied with ISIS have suitable packages for PCB layout associated with them, and these packages can be viewed when you browse the libraries. Note that some parts - e.g. simulator primitive models do not represent real world components and so do not have packages.

ISIS supports multiple packagings for a single schematic symbol, and where appropriate we have utilized this to offer both through hole (PTH) and surface mount (SMT) packagings for the parts in the libraries.

Note, however, that the responsibility lies wholly with yourself to check that the defaults we have used are suitable for your particular application. We cannot and will not accept any liability for losses due to boards being wasted as a result of unsuitable PCB packaging, however this may have arisen. We recommend most strongly that you make a one off prototype before commissioning the manufacture of large numbers of PCBs.

Manual Packaging

User properties including **PACKAGE** can be edited in a variety of ways. The simplest is to select the *Instant Edit* icon, click on the component whose field you want to edit, and then type in the new property information into the *PCB Package* text field. Where the components have already been given references, you can use the *Edit* command to access them by name. For example, the sequence 'E', Q1, ENTER will bring up the dialogue form for transistor Q1.

Where many components have the same package - e.g. resistors are almost always packaged with RES40, the *Property Assignment Tool* can be used to load this fixed value into the desired field of each component you click on. For example, to load the string RES40 into the **PACKAGE** field of a group of components you would set the PAT string to PACKAGE=RES40.

Automatic Packaging

The *ASCII Data Import* feature can be used for the automatic packaging of component types which always take the same package. As an example of this, the file DEVICE.ADI is supplied with ISIS and shows how packages and other properties were assigned to the parts in DEVICE.LIB.

The file is set up such that packages are loaded into the **PACKAGE** property which is then hidden. You can change this by editing the ADI block headers - see *Report Generation* on page 147 for more information on ADI.

Some components such as ordinary and electrolytic capacitors come in many different shapes and sizes. Furthermore, we cannot know what particular capacitor families you will want to use. Nevertheless, it is possible to build ADI files that operate on part values as well as device library names. For example, the extract below packages capacitors from our favourite suppliers, and at the same time, loads the order codes for them into property **ORDER**.

Note that packaging is only done for capacitors which are not of the default size specified in DEVICE.LIB. Running CAPS.ADI would override the default assignments to achieve the desired result.

```

;CAPACITOR PACKAGING FOR TYPES REQUIRING
;OTHER THAN CAP10 / ELEC-RAD10 PACKAGES
DATA DEVICE + VALUE : PACKAGE-
CAP 1u : CAP20
"CAP ELEC" 470u : ELEC-RAD20
"CAP ELEC" 1000u : ELEC-RAD30
"CAP ELEC" 2200u : ELEC-RAD30
END

```

```

;CAPACITOR ORDER CODES
;These are Ceramic, Polyester types and
;Electrolytics at 25V or better.
DATA DEVICE + VALUE : ORDERCODE-
CAP 100p : F146-447
CAP 680p : F146-457
CAP 1n : F149-100
CAP 1n5 : F149-101
CAP 3n3 : F149-103
.
.
.
"CAP ELEC" 47u : R11-0235
"CAP ELEC" 100u : R11-0245
"CAP ELEC" 220u : R11-0260
"CAP ELEC" 470u : R11-0280
END

```

In general, there will always be a final phase of manual packaging to do for unusual parts, or for special components used in that design only. We suggest that you make the **PACKAGE** properties of manually packaged parts visible as a reminder that they are non-standard.

Using the Bill of Materials to Help with Packaging

In the case of largish designs, it can become difficult to ascertain whether all the components have been packaged, and which packages have been assigned. This is especially true if you have made the **PACKAGE** property invisible during *ASCII Data Import*.

One way round this is to create a supplementary Bill of Materials configuration file which has the following **FIELD** records:

```

FIELD=VALUE, 15
FIELD=PACKAGE, 15

```

so that both the part value and the package appear for each component. The Bill of Materials then gives you a sorted list of all the components and their packages making for much easier checking.

Note that you can create multiple Bill of Materials configuration scripts using the *Set BOM Scripts* command on the *System* menu.

The Package Verifier

ISIS can verify that all the packages specified for components on a schematic exist in the ARES libraries, and that all the pin numbers specified in the ISIS library parts exist in the specified packages.

Whilst this facility is most useful when creating library parts *en masse*, it is also useful if you wish to verify that all the packaging is correct prior to netlisting to ARES.

The *Verify Packaging* command may be found on the *Library* menu.

Packaging with ARES

If you miss out packaging information from any of the components on the schematic, ARES will prompt for it when it loads the netlist. Indeed, you do have the option of doing all the package selections at this stage. However, packaging selections made this way will have to be re-entered - at least for unplaced components - each time the netlist is loaded into ARES.

Note that we do not recommend this method of working.

NET PROPERTIES AND ROUTING STRATEGIES

ARES associates with every net in the netlist a routing "strategy" which defines track and via styles, net type, routing layers and so forth. By default, all nets take the **SIGNAL** strategy except:

- The nets VCC and GND which take the POWER strategy.
- Nets with names such as D[0] which take the BUS strategy. It is the presence of the square brackets which is important. This syntax is really provided for use with schematics packages which could not otherwise convey a net property for a bus.

To override the default strategy assignment for a given net, you need to attach a *Net Property* to one of the wires on the net. This is achieved by placing a wire label of the form:

```
STRAT=strategy_name
```

If the named strategy does not exist, ARES will create it and add it to the strategy selector when the netlist is loaded. The strategy definition can then be edited by selecting the strategy and clicking on the strategy selector toggle.

It is perfectly legal to place an assignment such as:

```
STRAT=BUS
```

on a bus segment, though this will not work in the case of a 'cosmetic' bus. Such a bus is one which connects to no bus pins or terminals, and does not carry a bus net label.

Sometimes it is appropriate to specify a strategy for all the nets on a given sheet. For example, you might want all the nets on the power supply sheet to be assigned the POWER strategy.

This could be done by placing a script block as follows:

```
*NETPROP  
STRAT=POWER
```

on the appropriate sheet of the design. See page 83 for more information.

Wire and bus label net properties take precedence over sheet global net properties.

FORWARD ANNOTATION - ENGINEERING CHANGES

The term 'Engineering Change' refers to a situation in which an existing schematic is modified, and the resulting netlist re-loaded into ARES. This can occur both during the development of an original design, and also at a later date where a 'Mark II' version of the product is being produced. The PROTEUS system fully supports engineering changes, but it is important that you appreciate what the software does in the various circumstances that can arise.

Adding New Components

Adding new components and associated wiring to a design poses little problem *provided that you use the Auto-Annotator in incremental mode*. If you add components to the schematic and totally re-annotate the schematic (such that the part IDs of *existing* components get changed or exchanged) then ARES will not in general be able to make sense of the incoming netlist.

To add new components to a design

1. Place and wire up the components in ISIS in the usual way.
2. Use the Auto-Annotator in *incremental* mode to give the parts new unique component IDs. Alternatively, you can do this manually. Under no circumstances should you change the component IDs of existing components.
3. Transfer the changes to ARES using the *Netlist->ARES* command on the *Tools* menu. ARES will display the new components in the *Component Selector*.
4. Place the components in ARES in the usual way. ARES will then display ratsnest lines to indicate the required connections to the new components.

5. Route tracking to connect up the new components using automatic or manual routing as appropriate.

Removing Existing Components

Again, this is fairly straightforward to manage. ARES will tag the components that have been removed from the netlist so that you can see what has been removed before it is actually deleted from the PCB.

To remove components from a design

1. Delete the components from the schematic in the usual way. ISIS will automatically remove any wiring to them.
2. Transfer the changes to ARES using the *Netlist->ARES* command on the *Tools* menu. ARES will tag & highlight the components on the PCB that no longer appear in the netlist.
3. Examine the tagged components to ensure that you really do want to delete them. For each component that you want to delete, point at it and click right.
4. Remove residual tracking to the old components by using the usual route-editing tools in ARES.

Changing the connectivity

Where existing wiring is changed, ARES analyses the current connectivity of the board and marks sections of tracking that make connections no longer in the netlist as VOID. This is indicated by them appearing flashing yellow. Connections appearing in the netlist but not present on the PCB are shown as ratsnest lines in the usual way.

To change the connectivity of a design

1. Make changes to the wiring and connectivity of the schematic in ISIS. The change procedure is not affected by whether the changes involve drawn wires or syntactical changes such as wire and terminal labels.
2. Transfer the changes to ARES using the *Netlist->ARES* command on the *Tools* menu. ARES will mark tracks that make connections that are not in the netlist as VOID and they will appear in flashing yellow. Connections present in the netlist but not made on the PCB will appear as new ratsnest lines.
3. Inspect the void tracking to ensure that you really do want to remove it. If so, use the *Tidy* command to do so.
4. Form tracking to make the new missing connections using automatic or manual routing as you see fit.

Re-Annotating Components, and Re-Packaging Gates

This area is potentially the most confusing. There is a tendency to think that if you have a particular component in ISIS, say 'U35', that if you edit it in ISIS and call it 'U34' that this will simply be reflected in ARES when you next compile and load the netlist. *This is not the case.* What such a re-annotation means, in the context of PROTEUS, is that 'U35' has been removed, and that 'U34' has been added. Consequently, ARES will tag 'U35', and add a new part 'U34' to the parts selector.

Similarly, if two 7400 gates, say 'U1:A' and 'U2:B' are re-annotated on the schematic such that their names are exchanged, this will be seen by ARES as a change to the connectivity of the design, in that wires going to pins 1,2,3 will now go to pins 4,5,6 and vice-versa.

The key concept is that PROTEUS uses the part IDs in ISIS as the cross reference between the schematic and the PCB. If you change these IDs, then you will change the connectivity of the design rather than its annotation.

Changes to the annotation of the design must be made in ARES, from where they can be back-annotated into ISIS - see *Re-Annotation* on page 179.

PIN-SWAP/GATE-SWAP

Many types of component have either interchangeable pins - for example the two enable pins on a 74138, or interchangeable elements - for example the six inverters in a 7404. Some components such as the 7400 have both - there are four interchangeable gates each of which has two identical inputs. This phenomenon can be exploited when routing complex PCBs in that it may turn out to be easier to route to an alternate pin or gate to the one specified in the netlist. Clearly, making such a 'spur of the moment' decision will affect the connectivity of the design, and this change needs to be recorded and eventually reflected back on the schematic.

PROTEUS provides an extremely comprehensive - probably unrivalled - degree of support for this aspect of the EDA problem. Specifically, the following features are provided:

- A scheme for specifying swappable pins and gates within ISIS library parts.
- The ability to mark a pin or gate in ARES and see a graphical display of which other pins and gates it can be swapped with based on the data from the ISIS library parts.
- A gate-swap optimiser which will attempt to find the optimum gate allocations based on achieving the shortest possible ratsnest length.
- Automatic back-annotation of both manual and automatic changes into ISIS.
- A locking mechanism which prevents changes being made to both schematic and PCB simultaneously, thus preventing conflicts and ambiguities.

Specifying Pin-Swaps and Gate-Swaps for ISIS Library Parts

The following discussion assumes a detailed understanding of the creation of ISIS library parts. Please refer to the chapter entitled Library Facilities on page 101 if you need further information.

The first thing to appreciate is that pin-swaps can occur for both single element and multi-element devices whereas gate-swaps require the device to have multiple elements more or less by definition. Thus there are three cases.

Specifying Pin-Swaps in Single Element Devices

This is achieved through the use of the **PINSWAP** property; the *pin-names* (not numbers) of the interchangeable pins are listed. Thus, a resistor for which the two pins named '1' and '2' are interchangeable can be given the property

```
PINSWAP=1 , 2
```

whilst the 74138, which has identical input pins 'E2' and 'E3' carries the property:

```
PINSWAP=E2 , E3
```

Where more than one set of pins is interchangeable, a semicolon may be used to separate the pin sets. For example:

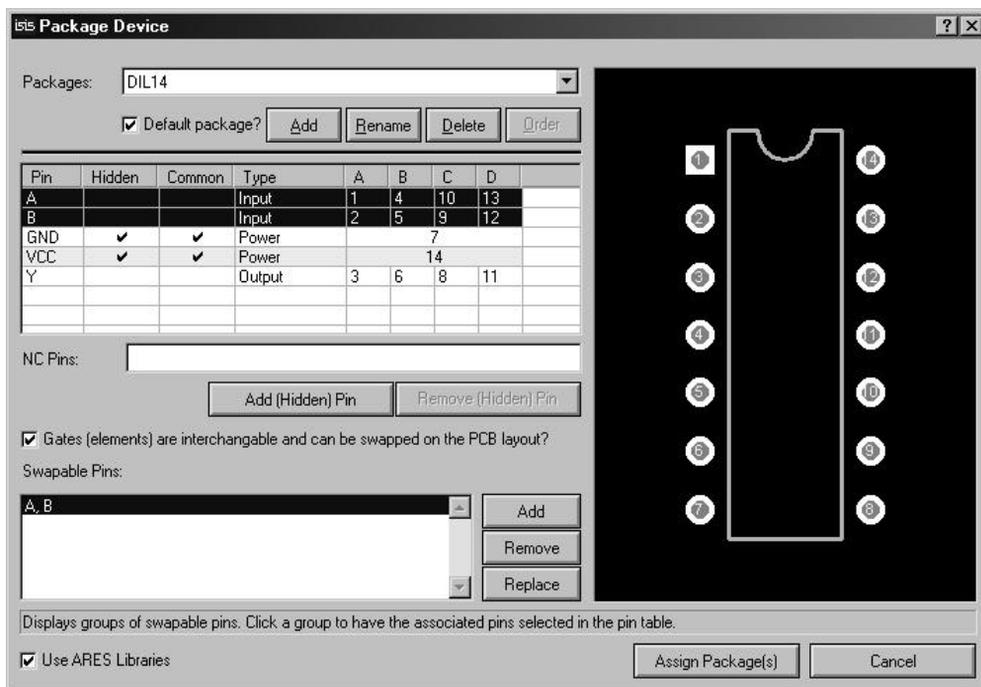
```
PINSWAP=A , B ; C , D
```

means that A can be swapped with B, and C can be swapped with D, but that the swaps A-C, A-D, B-C and B-D are still illegal.

Only one **PINSWAP** property may be used.

Specifying Pin-Swaps in Multi-Element Devices

For multi-element parts such as the 7400 shown below, you must use the *Visual Packaging Tool* to specify pinswap groups. This is achieved using the *Pinswap* section of the dialogue form.



To add a pinswap group to a packaging:

1. Highlight the pins to be swapped in the *Pin Grid* by holding down the CTRL key and clicking on each row in turn. In the above case, pins A and B are swappable, so you would highlight their rows as shown.
2. Click the *Add* button to create the pinswap group.

You can define multiple swap groups by repeating the above procedure.

Specifying Gate-Swaps in Multi-Element Devices

It is a happy fact that ISIS can work out the much of the gate-swap information for a multi-element device entirely by itself. For example, in the case of the 7400 (show overleaf), the

packaging script contains sufficient information to tell that the swappable element consists of pins A,B and Y and that there are 4 of them.

The only requirement is that we tell ISIS to allow gate swaps using the *Gateswap* checkbox in the *Visual Packaging Tool*. Note that the power pins VCC and GND are automatically excluded from the swappable element because they are hidden. They are also required to be on the same nets if a swap is to be made between different packages, since they are marked as being common.

Our final example is of a heterogeneous multi-element part - in this case a DPDT relay consisting of a coil and two pairs of contacts:

The image shows a schematic of a DPDT relay on the left and a dialog box on the right. The schematic has a coil labeled 'RL1:A' with pins 1 and 2, and two switch pairs labeled 'RL1:B' (pins 3, 4, 5) and 'RL1:C' (pins 6, 7, 8). The dialog box contains a table of pin configurations:

Pin	Hidden	Common	Type	A	B	C
C1		<input checked="" type="checkbox"/>	Passive	1	---	---
C2		<input checked="" type="checkbox"/>	Passive	2	---	---
COM			Passive	---	3	6
NC			Passive	---	4	7
NO			Passive	---	5	8

Below the table are fields for 'NC Pins:', 'Add (Hidden) Pin', and 'Remove (Hidden) Pin'. A checked checkbox 'Gates (elements) are interchangeable and can be swapped on the PCB layout?' is present. Under 'Swappable Pins:', a list contains 'C1, C2' with 'Add', 'Remove', and 'Replace' buttons.

In this case, both pins of the coil are both *common* and swappable. Omitting the *common* would, of course, be disastrous as it would allow swaps of contacts between different relays!

Performing Manual Pin-Swaps and Gate-Swaps in ARES

Pin-swaps and gate-swaps are both carried out in exact the same way using the *Ratsnest* mode in ARES.

To swap two pins or gates:

1. Load the netlist for the design from ISIS. ARES must have a copy of the netlist that is synchronized with the schematic before this feature may be used.
2. Select the *Ratsnest* icons in ARES.

3. Click right on the source pin. The ratsnest lines connected to it will highlight.
4. Click and hold down the left button. All pins with which the source pin can be swapped will highlight. This set includes all the valid pin-swaps within the source gate and all the same pin-sets in any valid gate-swaps.
5. Drag the source ratsnest lines to the destination pin you want to swap with. If this pin is in the same gate then just a pin swap will be performed. If it is in another gate, then a gate-swap and possible also a pin-swap will occur.

There is little more to it than that. Note, however, that ARES will not allow swaps where any of the source or destination pins have tracking attached. In the case of a gate swap, all pins of both gates must be unrouted.

WARNING

Pin-swaps and Gate-swaps constitute changes to the connectivity of your design. ARES uses the pin-swap and gate-swap data specified in the ISIS libraries to decide what is, and is not, a valid swap. If there are errors in this data, then ARES may well suggest illegal swaps. We will not, under any circumstances, be held liable for any costs incurred or losses arising as result of such mishaps, whether the error be in your library parts or ours or in the software itself. We strongly recommend that you check that the swaps you make are legal, and that you prototype your PCB prior to manufacturing large quantities.

The Gate-Swap Optimizer

In cases where a design has large numbers of swappable gates, it can be very hard to see what the best gate-allocation should be. The number of possible combinations can become literally astronomical even for small numbers of gates. The sample design SHIFT16 contains many more combinations than there are particles in the universe!

To help you find a near optimum solution, ARES incorporates an automatic gate-swap optimizer which will perform thousands of trial swaps in such a way as to find what is technically called a 'local minima' for a given board placement. In many cases this is very near optimum, and almost invariably gives some reduction in the total ratsnest length.

To use the Gate-Swap Optimizer:

1. Load the netlist for the design from ISIS. ARES must have a copy of the netlist that is synchronized with the schematic before this command may be used.
2. Place all the components in the usual way, aiming to achieve as low a ratsnest length as possible. The swap optimizer is no excuse for bad placement!
3. Invoke the *Gate-Swap Optimizer* command from the *Tools* menu.

The algorithm makes repeated passes until such time as it can make no further improvement. Run times can be quite long (30 minutes) if there lots of possible swaps.

WARNING

The Gate-Swap Optimizer relies entirely on the gate-swap data specified in the ISIS component libraries to decide what is, and is not, a valid swap. If there are errors in this data, then the swap-optimizer is likely to make erroneous changes to the connectivity of your design. We will not, under any circumstances, be held liable for any costs incurred or losses arising as result of such mishaps, whether the error be in your library parts or ours or in the software itself. We strongly recommend that this command be used only if you are going to prototype your PCB prior to manufacture.

RE-ANNOTATION

In some circumstances you may wish to renumber components on the PCB to suit manufacturing or assembly activities. This can be done either manually or automatically.

To re-annotate manually:

1. Ensure that an up-to-date copy of the netlist is loaded into ARES.
1. Select the *Edit Mode* icon in ARES, and the correct layer for the component(s) you wish to annotate.
2. Click on the component labels to edit them.

You will get an error message if you attempt to make changes which cannot be back-annotated to the schematic. Connectors made with physical terminals are the most significant example - because there is no named carrying entity in ISIS, there is nowhere for the change to be applied.

To re-annotate automatically:

1. Specify any components you do not wish to be renumbered by giving them a **NOANNOTATE=TRUE** property on the schematic.
2. Ensure that an up-to-date copy of the netlist is loaded into ARES.
3. Invoke the *Component Re-Annotator* command from the *Tools* menu in ARES. This will renumber the components based on their existing prefixes and their physical positions on the PCB.

Either way, any changes made will be reflected on the schematic the next time you save the PCB.

BACK-ANNOTATION WITH ISIS

Once pin-swaps, gate-swaps or annotation changes have been made, you will want to transfer the effects of these change back to ISIS so that the schematic accurately reflects both the annotation and the connectivity of the PCB. There are two ways of working this:

Semi-Automatic Back-Annotation

By default, you will be prevented from making changes to the schematic whilst open with unsaved changes. The menu bar in ISIS will contain the text '(Locked)' to indicate this state of affairs.

To re-synchronise and unlock the schematic, use the *Back-Annotate from ARES* command on the *Tools* menu in ISIS. This command will cause ARES to save its changes.

Fully-Automatic Back-Annotation

In this case, the schematic will update whenever you bring ISIS to the foreground. However, a consequence of this is that ARES will automatically save its changes to disk.

To select fully automatic operation, check the *Auto Sync/Save checkbox* on the *Set Environment* dialogue from the *System* menu in ISIS. The default mode is semi-automatic.

In both cases, back-annotation is automatic if the PCB is saved after the changes have been made.

A

Accelerators	41
Active Components	103
Annotating components	11
Apply Default Template Command	47
Apply Template From Design Command	47
Arc graphic	88
ARES	2, 155
ASCII Data Import	
in PCB design	156
reference	144
ASCII format	138
ATTRIB program	93
Auto Dot Removal	69
Automatic Annotator	
reference	37
tutorial	12
Automatic dot placement & removal	69
Automatic Wire Routing	35

B

Backup files	29
Bill of Materials	
adding parts	143
and parameterized circuits	55
ASCII Format	138
categories	140
columns	141
Compact CSV Format	138
configuring	138
Fields	141
Full CSV Format	138
generating	137
HTML format	137
report formats	137
tutorial	21
use in PCB design	157
Bitmap	153
Block	
commands tutorial	10
copy	34
delete	35

move	34
BMP file	153
Boardmaker (netlist format)	135
BOM Scripts	137
BOMPART property	143
Border	15, 38
Box graphic	87
Bring to Front command	41
Bus labels	
deleting	78
in netlist generation	130
placing and editing	78
Bus pins	107, 113
Buses	
connecting to individual bits	79, 131
connectivity rules for	129
definition	77
placing	77
BUSNODE marker	91, 113

C

CadStar	136
Capacitors	
packaging for PCB design	62, 156
Changing Schematic Appearance	44
Child sheet	120
Circle graphic	87
Circuits	
automatic creation of	123
definition	120
sharing in hierarchical design	124
Clipboard	153
Common pins	98, 109
Component Properties	104
Components	
annotating	11, 37
assigning packages for PCB design	155
definition	65
editing	11, 67
editing by name	33
finding	33
in hierarchical design	122
placing	7, 66

properties	68	Devices	98
replacing	20, 67	advanced tutorial	16
replacing from library	93	default properties	114
selecting from libraries	65	editing	116
Configuring		multi-element heterogeneous	112
Bill of Materials	138	multi-element homogenous	111
Connection points	35	single element	110
Connectors	83	tutorial	13
Coordinate Display	26	with bus pins	113
Co-ordinates	6, 26	Displaying design information on the schematic	39
Copy icon	34	Dots	68
Copying		Dragging	
between designs	30	general procedure	31
tagged objects	34	labels	31
Creating		objects	8
graphics symbols	94	wires	36
multi-element heterogeneous devices	112	Duplicate pin names	126
multi-element homogenous devices	111	DXF	
new design	28	file generation	153
single element device	99	E	
single-element devices	110	E12,E24 keywords	57
user defined device pins	97	Eagle PCB software	135
user defined module ports	96	EDIF (netlist format)	125
user defined terminals	95	Editing	
CSV format	138	components	67
Cursor types	26	devices	116
D		general procedure	32
Data Sheets	105	Pin objects	86
DATA...END (ADI command)	146	pinout	117
Default Font	45	sub-circuits	82
Default Properties	50	symbols	98
Default properties for device	114	terminals	84
DEFINE keyword	51, 74	Wire Labels	69, 70
Delete icon	35	Editing Global Styles	44
Deleting		Editing Local Styles	46
Bus Labels	78	Editing Window	5, 24
general procedure	31	EEDesigner (netlist format)	135
tagged objects	35	Electrical Rules Check	
Wire Labels	70	error messages	148
Design		generating	148
properties	53	tutorial	21
Design Global Annotation	120, 124	Electrical types	
Device pins		of device pins	14, 101
creating	97	of terminals	96
Device Properties	103		

Encapsulated Postscript	154
EPS file	154
Example files	5
Exit to Parent command	22, 123
Export	
Bitmap	153
DXF	153
EPS file	154
Windows Metafile	153
Export Section command	30
External module files	123
External text editor	33

F

False origin	27
FIELD keyword	
for packaging connectors	83
reference	73
File types	28
Files	
loading	29
saving	16, 29
Find and Edit command	33
Fonts	45
Footprint	107
Futurenet (netlist format)	135

G

Gate swap	110
Global net names	127
Global Styles	
See Styles	43
Goto Sheet command	119
Graphics	
placing	87
reference	87
resizing	89
text	89
tutorial	15
Graphics files	153
Graphics Styles	
See Styles	43
Graphs	3
Grid dots	6, 27

H

Hard copy	
tutorial	16
Header block	15, 20, 39
Header Block	
List of PROPERTIES	39
HEADER symbol	39
Help Topics	105
Heterogeneous multi-element devices	98, 112
Hidden pins	68, 109
Hiding	
power pins	115
text	50
Hierarchical design	53, 119
external modules	123
navigating	123
sub-circuits	82
tutorial	22
with module components	122
with sub-circuits	121
Homogenous multi-element devices	111
Homogenous multi-element devices	98
HTML format	137

I

Icons	6
IF...END (ADI command)	145
Import Section command	30
Incremental annotation	12
ISIS II	
converting files	28

J

Junction Dots	See Dots
Justifying	
labels	33

K

Keyboard Configuration	41
------------------------	----

L

LABEL marker	91, 95
Labels	
dragging	9, 31
editing	33

resizing	59	NC pins	110
Last Loaded files	29	Net names	
Library		assigning with logical terminals	83
read only	93	assigning with wire labels	71
selector	7	definition	125
supplied files	93	design global	127
Library parts	See Devices	in hierarchical design	124
Like bit connection rule	129	legal characters in	126
Line graphic	87	Net properties	
LISA		assigning with NETPROP keyword	74
introduction	2	assigning with wire labels	71
Load design command	29	use for routing strategies	158
Loading files	29	Netlist	
Locked Design	167	compiler command	134
Logical netlist	134	definition	125
Logical terminals	82	errors	135
		generating	134
M		NETPROP keyword	74, 159
Make Device command	102	Network compatibility features	4
Make Symbol command	95, 96	New Design command	28
MAP ON keyword	51, 75	New Sheet command	119
Mapping Keyboard Shortcuts	41	NODE marker	90, 95
Markers	90	Non –Physical Sheets	124
Menu bar	5, 23		
Metafile	153	O	
MHTML format	137	Object selector	26
Mirror icon	8, 24, 32	Object Selector	65
MOD files	123	Objects	
Mode selector toolbar	23	deleting	31
MODELS keyword	75	dragging	31
Module components	122	editing	32
Module ports		Editing Local Styles	46
creating	96	editing properties	50
placing	81	placing	30
Modules	120	reorienting	32
Move icon	34	resizing	32
Moving		shuffling database order	41
tagged objects	34	tagging	31
Multi-element devices	16	Ordinal number of sheets	119
Multi-element parts	98, 107	Orientation toolbar	24
Multi-sheet designs	21, 119	Origin command	27
Multiwire (netlist format)	135	ORIGIN marker	90, 95, 100
		Overbars	101
N		Overview Window	5, 25
Named scripts	76		
Navigating a hierarchical design	123		

P	
PACKAGE property	155
Package Verifier	158
PACKAGE.ADI file	156
Packages for PCB design	155
Packaging	
with hidden power pins	116
Packaging Tool	107
Panning	5, 24
Parameter Mapping Tables	75
Parameterized circuits	53, 121
Parent sheet	120
Parts bin	26
PCB design	2, 155
PCB Package	107
PDF files	105
Pen colours	153
Physical netlist	134
Physical terminals	82, 83
Pick command	10, 66, 93
Picking library parts	7
Pin	
NC	110
Pin grid	
in packaging tool	109
Pin objects	85
Pin swap	110
PINNAME marker	91, 97
PINNAME property	101
PINNUM marker	91, 97
PINNUM property	101
Pinout	
editing	117
Pins	
common	109
hidden	68
hiding	109
placing for new device	100
with duplicate names	126
with overbars	101
Place preview	25
Placing	
bus entries	79
buses	77
components	7, 66
dots	69
general procedure	30
graphics	15, 87
header block	15
markers	91
scripts	72
sheet border	15
sub-circuits	80
terminals	9, 83
text	15
wires	8
Plotter output	152
Postscript	154
Power nets	126
Power pins	68, 115
in netlist generation	126
Prefix	103
Printing	151
Configuring Margins	151
Fit to Page	152
Positioning the output area	151
Print Preview	151
Scaling	152
Specifying a Printer Device	151
Properties	
in PCB design	155
Property Assignment Tool	57
examples	61
tutorial	11
Property Definitions	50, 104
Property Expression Evaluation	56
Property Substitution	55
ProSPICE	2
Protel (netlist format)	136
PSPICE (netlist format)	136
Q	
Quit command	30
R	
Racal (netlist format)	136
Real Time Annotation	9, 11
Real Time Snap	9, 27
Re-annotation	166
RedBoard	136
Redraw command	24
Reference prefix	103

LABCENTER ELECTRONICS

Remove Sheet command	119	SPICE keyword	77
Reorienting objects	32	SPICE-AGE (netlist format)	136
Replacing components	20, 67	STRAT net property	158
Resizing		Strategies	158
general procedure	32	Styes	
graphics	89	Component creation	99
labels	33, 59	Styles	
Roaming a hierarchical design	123	Benefits Of	44
Root sheets	119, 120	Component creation	13
Rotation icon	8, 24, 32	Editing Global Styles	44
		Editing Local Styles	46
		Introduction	43
		Symbol creation	94, 95, 96, 97
		Tutorial	44
		Sub-Circuits	
		editing	82
		placing	80
		use in hierarchical design	121
		Symbol libraries	94
		Symbols	
		definition	94
		editing	98
		placing	89
		tutorial	20
		System properties	49
		T	
		Tagging	
		tutorial	8
		with mouse	31
		with Search & Tag commands	60
		Tango (netlist format)	136
		Tapping a bus	79
		Templates	47
		Terminals	
		creating	95
		definition	82
		editing	84
		in netlist generation	125
		placing	9, 83
		properties	85
		Text	
		graphic	89
		hiding	50
		placing	15
		scripts	72

Text Styles	
See Styles	43
Toolbars	23
TrueType Fonts	45
TYPE property	101

U

Units	26
Untagging	31
User properties	50

V

Valid (netlist format)	136
Value annotation	38
Variable zoom	25
Vector Font	45
Verify packaging	158
Visual Packaging Tool	107
VSM	2
file types	28
gadgets	87
Inter-element net names	128
models	75

netlist generation for	134
parameter mapping	75
Vutrax (netlist format)	136

W

Wire Auto Router	31, 35
Wire labels	
deleting	70
in netlist generation	125
properties	71
Wire Labels	
Editing	70
placing and editing	69
Wires	
and block move	34
auto repeat placement	36
dragging	9, 36
routing	8, 35
WMF file	153

Z

Zoom	25
Zoom to Child command	22, 123

PROTEUS

VSM

Virtual System Modelling

Incorporating PROSPICE - Berkeley SPICE3F5
with extensions for mixed mode simulation.

User Manual

Issue 6.3 - November 2003

© Labcenter Electronics

TABLE OF CONTENTS

INTRODUCTION.....	1
ABOUT PROTEUS VSM.....	1
ABOUT THE DOCUMENTATION.....	2
TUTORIALS.....	3
INTERACTIVE SIMULATION TUTORIAL.....	3
Introduction.....	3
Drawing the Circuit.....	4
Writing the Program.....	6
Debugging the Program.....	8
GRAPH BASED SIMULATION TUTORIAL.....	10
Introduction.....	10
Getting Started.....	11
Generators.....	11
Probes.....	12
Graphs.....	13
Simulation.....	16
Taking Measurements.....	16
Using Current Probes.....	18
Frequency Analysis.....	18
Swept Variable Analysis.....	20
Noise Analysis.....	20
INTERACTIVE SIMULATION.....	23
BASIC SKILLS.....	23
The Animation Control Panel.....	23
Indicators and Actuators.....	24
Setting Up an Interactive Simulation.....	24
TAKING MEASUREMENTS.....	25
Animation Effects.....	25
Point and Shoot Parameter Information.....	26
Voltage and Current Probes.....	27
Virtual Instruments.....	27
ANIMATION TIMESTEP CONTROL.....	27
Overview.....	27
Frames Per Second.....	28
Timestep Per Frame.....	28
Single Step Time.....	28

HINTS AND TIPS	28
Circuit Timescale.....	28
Voltage Scaling	29
Earthing.....	29
High Impedance Points	29
VIRTUAL INSTRUMENTS.....	31
VOLTMETERS & AMMETERS.....	31
OSCILLOSCOPE.....	31
Overview	31
Using the Oscilloscope.....	32
LOGIC ANALYSER.....	34
Overview	34
Using the Logic Analyser	35
COUNTER TIMER	36
Overview	36
Using the Timer Modes	37
Using the Frequency Mode.....	37
Using the Counter Mode.....	38
VIRTUAL TERMINAL.....	39
Overview	39
Using the Virtual Terminal.....	40
SIGNAL GENERATOR.....	41
Overview	41
Using the Signal Generator.....	41
DIGITAL PATTERN GENERATOR	43
Overview	43
Using the Pattern Generator.....	44
ThePattern Generator Component Pins.....	45
Clocking Modes	46
Trigger Modes	46
External Hold.....	48
Additional Functionality	49
VSM USER INTERFACE ELEMENTS	51
Rotary Dials	51
WORKING WITH MICROPROCESSORS.....	53
INTRODUCTION	53
SOURCE CODE CONTROL SYSTEM	53
Overview	53
Attaching Source Code to the Design.....	54

Working on your Source Code	54
Installing 3 rd Party Code Generation Tools	55
Using a MAKE Program.....	55
Using a 3 rd Party Source Code Editor	56
USING A 3RD PARTY IDE.....	56
Using Proteus VSM as an External Debugger	57
Using Proteus VSM as a Virtual In-Circuit Emulator (ICE).....	57
POPUP WINDOWS	58
SOURCE LEVEL DEBUGGING WITHIN PROTEUS VSM	59
Overview	59
The Source Code Window.....	59
Single Stepping.....	59
Using Breakpoints.....	60
Variables Window	60
Multi CPU Debugging.....	60
THE WATCH WINDOW.....	61
Adding items to the Watch Window.....	61
Modifying Items in the Watch Window	61
Watchpoint Conditions.....	62
BREAKPOINT TRIGGER OBJECTS.....	62
Overview	62
Voltage Breakpoint Trigger – RTVBREAK.....	62
Current Breakpoint Trigger – RTIBREAK.....	63
Digital Breakpoint Trigger – RTDBREAK.....	63
GRAPH BASED SIMULATION	65
INTRODUCTION	65
SETTING UP A GRAPH BASED SIMULATION.....	65
Overview	65
Entering The Circuit	65
Placing Probes and Generators	65
Placing Graphs	66
Adding Traces To Graphs.....	66
The Simulation Process.....	67
GRAPH OBJECTS	67
Overview	67
The Current Graph.....	68
Placing Graphs	68
Editing Graphs	68
Adding Traces To A Graph.....	68
The Add Trace Command Dialogue Form	70

Editing Graph Traces	71
Changing the Order and/or Colour of Graph Traces	71
THE SIMULATION PROCESS.....	71
Demand Driven Simulation	71
Executing Simulations.....	72
What Happens When You Press the Space-Bar	73

ANALYSIS TYPES 75

INTRODUCTION	75
ANALOGUE TRANSIENT ANALYSIS.....	76
Overview	76
Method of computation.....	76
Using Analogue Graphs	77
Defining Analogue Trace Expressions.....	78
DIGITAL TRANSIENT ANALYSIS.....	79
Overview	79
Method of Computation.....	79
Using Digital Graphs.....	80
How Digital Data is Displayed.....	81
MIXED MODE TRANSIENT ANALYSIS.....	82
Overview	82
Method of Computation.....	82
Using Mixed Graphs.....	83
FREQUENCY ANALYSIS.....	83
Overview	83
Method of computation.....	83
Using Frequency Graphs	84
DC SWEEP ANALYSIS.....	85
Overview	85
Method of computation.....	85
Using DC Sweep Graphs	86
AC SWEEP ANALYSIS	87
Overview	87
Method of Computation.....	87
Using AC Sweep Graphs.....	87
DC TRANSFER CURVE ANALYSIS.....	88
Overview	88
Method of Computation.....	88
Using Transfer Graphs.....	88
NOISE ANALYSIS.....	89
Overview	89

Method of calculation	89
Using Noise Graphs	90
DISTORTION ANALYSIS	90
Overview	90
Method of Computation	91
Using Distortion Graphs	91
FOURIER ANALYSIS	92
Overview	92
Method of Calculation	92
Using Fourier Graphs	92
AUDIO ANALYSIS	93
Overview	93
Method of Calculation	93
Using Audio Graphs	93
INTERACTIVE ANALYSIS	94
Overview	94
Method of Computation	94
Using Interactive Graphs	94
DIGITAL CONFORMANCE ANALYSIS	95
Overview	95
Method of Computation	95
Using Conformance Graphs	96
GENERATORS AND PROBES	99
GENERATORS	99
Overview	99
Placing Generators	100
Editing Generators	100
DC Generators	101
Sine Generators	101
Pulse Generators	102
Exponential Generators	103
Single Frequency FM Generators	103
Piece-wise Linear Generators	104
File Generators	105
Audio Generators	105
Digital Generators	106
PROBES	107
Overview	107
Placing Probes	107
Probe Settings	108

USING SPICE MODELS	109
OVERVIEW	109
USING A SPICE SUBCIRCUIT (SUBCKT DEFINITION)	110
USING A SPICE MODEL (MODEL CARD)	112
SPICE MODEL LIBRARIES	113
*SPICE SCRIPTS	114
HANDLING MODEL SIMULATION FAILURES	114
ADVANCED TOPICS.....	115
GROUND AND POWER RAILS	115
Why You Need a Ground.....	115
Power Rails	117
INITIAL CONDITIONS.....	118
Overview	118
Specifying Initial Conditions for a Net	118
Specifying Initial Conditions for Components	119
NS (NODESET) Properties	119
PRECHARGE Properties	120
TEMPERATURE MODELLING.....	120
THE DIGITAL SIMULATION PARADIGM	121
Nine State Model	121
The Undefined State.....	121
Floating Input Behaviour	122
Glitch Handling.....	122
MIXED MODE INTERFACE MODELS (ITFMOD).....	125
Overview	125
Using ITFMOD Properties	127
PERSISTENT MODEL DATA.....	127
TAPES AND PARTITIONING.....	128
Overview	128
Single Partition Operation.....	128
Tape Objects.....	129
Tape Modes	129
SIMULATOR CONTROL PROPERTIES.....	130
Overview	131
Tolerance Properties	131
Mosfet Properties	131
Iteration Properties	131
Temperature Properties	132
Digital Simulator Properties	132
TYPES OF SIMULATOR MODEL.....	134

How to tell if a Component Has a Model.....134
Primitive Models134
Schematic Models135
VSM Models135
SPICE Models.....136

TROUBLESHOOTING.....137

THE SIMULATION LOG.....137
NETLISTING ERRORS137
LINKING ERRORS137
PARTITIONING ERRORS.....138
SIMULATION ERRORS.....138
CONVERGENCE PROBLEMS.....139

INDEX141

INTRODUCTION

ABOUT PROTEUS VSM

Traditionally, circuit simulation has been a non-interactive affair. In the early days, netlists were prepared by hand, and output consisted of reams of numbers. If you were lucky, you got a pseudo-graphical output plotted with asterisks to show the voltage and current waveforms.

More recently, schematic capture and on screen graphing have become the norm, but the simulation process is still non-interactive – you draw the circuit, press go, and then study the results in some kind of post processor. This is fine if the circuit you are testing is essentially static in its behaviour e.g. an oscillator which sits there and oscillates nicely at 1MHz. However, if you are designing a burglar alarm, and want to find out what happens when a would-be burglar keys the wrong PIN into the keypad, the setting up required becomes quite impractical and one must resort to a physical prototype. This is a shame, as working 'in cyberspace' has so much to offer in terms of design productivity.

Only in educational circles has an attempt been made to present circuit simulation like real life electronics where it is possible to interact with the circuit whilst it is being simulated. The problem here has been that the animated component models have been hard coded into the program. Only limited numbers of simple devices such as switches, light bulbs, electric motors etc. have been offered, and these are of little use to the professional user. In addition, the quality of circuit simulation has often left much to be desired. For example, one major product of this type has no timing information within its digital models.

PROTEUS VSM brings you the best of both worlds. It combines a superb mixed mode circuit simulator based on the industry standard SPICE3F5 with animated component models. And it provides an architecture in which additional animated models may be created by anyone, including end users. Indeed, many types of animated model can be produced without resort to coding. Consequently PROTEUS VSM allows professional engineers to run interactive simulations of real designs, and to reap the rewards of this approach to circuit simulation.

And then, if that were not enough, we have created a range of simulator models for popular micro-controllers and a set of animated models for related peripheral devices such as LED and LCD displays, keypads, an RS232 terminal and more. Suddenly it is possible to simulate complete micro-controller systems and thus to develop the software for them without access to a physical prototype. In a world where time to market is becoming more and more important this is a real advantage.

It is also worth pointing out that the processing power of the modern PC is truly awesome. A 300MHz Pentium II PC can simulate simple micro-controller designs in real time, or even faster in some cases. And even where things slow down somewhat, the response time is more often

that not useable for software development. If you are serious about this game, you can go out and buy a 2GHz dual processor PC, which is far, far faster. This, then, debunks the other obvious objection to interactive simulation – that it would not be fast enough.

ABOUT THE DOCUMENTATION

This manual is intended to complement the information provided in the on-line help. Whereas the manual contains background information and tutorials, the help provides context sensitive information related to specific icons, commands and dialog forms. Help on most objects in the user interface can be obtained by pointing with the mouse and pressing F1.

PROTEUS VSM can be used in two rather distinct ways – either for *Interactive Simulation* or for *Graph Based Simulation* and this is reflected in the structure of the manual. Typically, you will use interactive simulation to see if a design works at all, and graph based simulation to investigate why it doesn't or to take very detailed measurements. However, there are no hard and fast rules. Some elements of the system, such as *Generators* are relevant to both modes of use and are given their own chapters for this reason.

Detailed, step by step tutorials are provided which take you through both types of simulation exercise. We strongly recommend that you work through these as they will demonstrate all the basic techniques required to get going with the software.

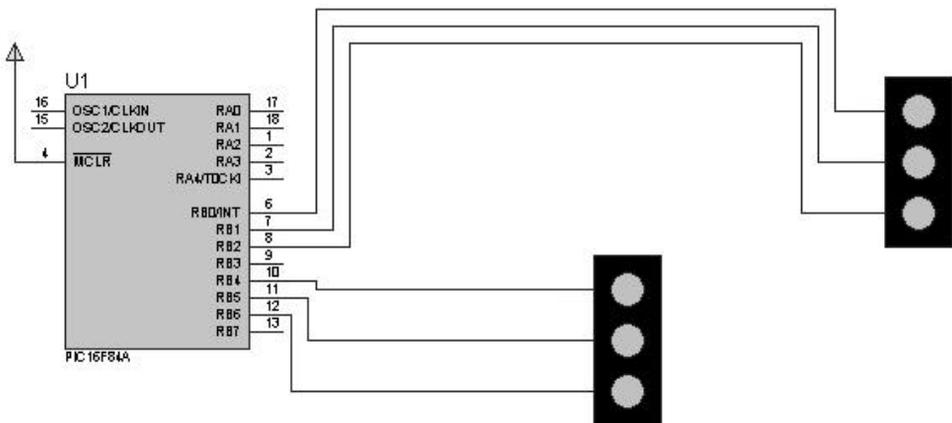
Information on creating VSM models is provided separately in the *VSM Software Development Kit (SDK)*, an on-line resource which is installed as standard with the professional version of Proteus.

INTERACTIVE SIMULATION TUTORIAL

Introduction

The purpose of this tutorial is to show you, through the creation of a simple schematic, how to conduct an interactive simulation using Proteus VSM. While we will concentrate on the use of Active Components and the debugging facilities of the ISIS Editor we will also look at the basics of laying out a schematic and general circuit management. Full coverage of these topics can be found in the ISIS Manual.

The circuit we will be using for the simulation is a pair of traffic lights connected to a PIC16F84 microcontroller as shown below.



Whilst we will be drawing the schematic from scratch, the completed version can be found as "Samples\Tutorials\Traffic.DSN" within your Proteus installation. Users who are familiar with the general operating procedures in ISIS may choose to use this ready made design and move on to the section on the microcontroller program. Please note, however, that this design file contains a deliberate error - read on for more information.

If you are unfamiliar with ISIS, the interface and basic usage are discussed at length in *A Guided Tour of the ISIS Editor* in the ISIS manual. and although we will touch on these issues in the following section you should take the time to familiarise yourself with the program before proceeding .

Drawing the Circuit

Placing the Components

We will start by placing two sets of traffic lights and a PIC16F84 on a new schematic layout. Start a fresh design, select the Main Mode and the Component icons (all the icons have both tooltip text and context-sensitive help to assist in their use) and then left click on the letter 'P' above the *Object Selector*. The *Device Library* selector will now appear over the *Editing Window* (for more information see *Picking, Placing and Wiring Up Components* in the ISIS manual).

The Traffic Lights can be found in the ACTIVE Library and the PIC microcontroller in the MICRO Library. To select a component into the design, highlight the component name in the *Objects* listbox and double click on it. A successful selection will result in the component name appearing in the *Object Selector*.

Once you have selected both TRAFFIC LIGHTS and PIC16F84 into the design close the *Device Library Selector* and click left once on the PIC16F84 in the *Object Selector* (this should highlight your selection and a preview of the component will appear in the *Overview Window* at the top right of the screen). Now left click on the *Editing Window* to place the component on the schematic - repeat the process to place two sets of traffic lights on the schematic.

Movement and Orientation

We now have the building blocks on the schematic but the chances are they are not ideally positioned. To move a component, point the mouse over it and right click (this should highlight the component), then depress the left mouse button and drag (you should see the component outline 'follow' the mouse pointer) to the desired position. When you have the outline where you want release the left mouse button and the component will move to the specified position. Note that at this point the component is still highlighted - right click on any empty area of the *Editing Window* to restore the component to it's normal status.

To orient a component, right click over it in the same way as before then click on one of the *Rotation* icons. This will rotate the component through 90 degrees - repeat as necessary. Again, it is good practice to right click on an empty area of the schematic when you are finished to restore the component to it's original state.

Set out the schematic in a sensible fashion (perhaps based on the sample given), moving and orientating the components as required. If you are having problems we advise you to work through the tutorial in the ISIS manual

Zooming and Snapping

In order to wire up the schematic it is useful to be able to zoom in to a specific area. Hitting the F6 key will zoom around the current position of the mouse, or, alternatively, holding down

the SHIFT key and dragging a box with the left mouse button will zoom in on the contents of the dragged area. To zoom back out again hit the F7 key, or, should you wish to zoom out until you can see the entire design, hit the F8 key. Corresponding commands can be accessed through the *View Menu*.

ISIS has a very powerful feature called *Real Time Snap*. When the mouse pointer is positioned near to pin ends or wires, the cursor location will be snapped onto these objects. This allows for easy editing and manipulation of the schematic. This feature can be found in the *Tools Menu* and is enabled by default.

More information on zooming and snapping can be found in *The Editing Window* in the ISIS manual.

Wiring Up

The easiest way to wire a circuit is to use the *Wire Auto Router* option on the *Tools Menu*. Make sure that this is enabled (a tick should be visible to the left of the menu option). More information on the functionality on the WAR can be found in the ISIS manual.

Zoom in to the PIC until all the pins are comfortably visible and then place the mouse pointer over the end of pin 6 (RB0/INT). You should see a small 'x' cursor on the end of the mouse. This indicates that the mouse is at the correct position to connect a wire to this pin. Left click the mouse to start a wire and then move the mouse to the pin connected to the red light on one of the sets of Traffic Lights. When you get an 'x' cursor again over this pin left click to complete the connection. Repeat the process to wire up both sets of Traffic Lights as given on the sample circuit.

There are a few points worth mentioning about the wiring up process:

- You can wire up in any mode - ISIS is clever enough to determine what you are doing.
- When enabled, the Wire Autorouter will route around obstacles and generally find a sensible path between connections. This means that, as a general rule, all you need to do is left click at both end points and let ISIS take care of the path between them.
- ISIS will automatically pan the screen if you nudge the edge of the *Editing Window* while placing a wire. This means that you can zoom in to the most suitable level and, so long as you know the approximate position of the target component, simply nudge the screen over until it is in view. Alternatively, you can zoom in and out while placing wires (using the F6 and F7 keys).

Finally, we have to wire pin 4 to a power terminal. Select *Gadgets Mode* and the *Terminal* icon and highlight 'POWER' in the *Object Selector*. Now left click on a suitable spot and place the terminal. Select the appropriate orientation and wire the terminal to pin 4 using the same techniques as before.

More useful information on the wiring up process is available in the tutorial in the ISIS manual.

At this point we recommend that you load the completed version of the circuit – this will avoid any confusion arising if the version you have drawn is different from ours!

Writing the Program

Source Listing

For the purposes of our tutorial, we have prepared the following program which will enable the PIC to control the traffic lights. This program is provided in a file called TL.ASM and can be found in the "Samples\Tutorials" directory.

```
LIST      p=16F84 ; PIC16F844 is the target processor

          #include "P16F84.INC" ; Include header file

          CBLOCK 0x10           ; Temporary storage
            state
            11,12
          ENDC

          org      0           ; Start up vector.
          goto    setports    ; Go to start up code.

          org      4           ; Interrupt vector.
halt      goto    halt       ; Sit in endless loop and do
nothing.

setports  clrw           ; Zero in to W.
          movwf  PORTA      ; Ensure PORTA is zero before
enabled.
          movwf  PORTB      ; Ensure PORTB is zero before
enabled.

          bsf   STATUS,RP0   ; Select Bank 1
          clrw           ; Mask for all bits as
outputs.
          movwf TRISB       ; Set TRISB register.
          bcf   STATUS,RP0   ; Reselect Bank 0.

initialise clrw           ; Initial state.
```

```

        movwf    state        ; Set it.

loop    call     getmask      ; Convert state to bitmask.
        movwf   PORTB        ; Write it to port.
        incf    state,W      ; Increment state in to W.
        andlw   0x04         ; Wrap it around.
        movwf   state        ; Put it back in to memory.
        call    wait         ; Wait :-)
        goto   loop         ; And loop :-)

; Function to return bitmask for output port for current state.
; The top nibble contains the bits for one set of lights and the
; lower nibble the bits for the other set. Bit 1 is red, 2 is
amber
; and bit three is green. Bit four is not used.

getmask    movf    state,W      ; Get state in to W.
           addwf   PCL,F        ; Add offset in W to PCL to calc.
goto.

           retlw   0x41         ; state==0 is Green and Red.
           retlw   0x23         ; state==1 is Amber and Red/Amber
           retlw   0x14         ; state==3 is Red and Green
           retlw   0x32         ; state==4 is Red/Amber and
Amber.

; Function using two loops to achieve a delay.
wait       movlw   5
           movwf   l1

w1         call    wait2
           decfsz  l1
           goto   w1
           return

wait2      clrf    l2
w2         decfsz  l2
           goto   w2
           return
           END

```

There is, in fact, a deliberate mistake in the above code, but more of that later...

Attaching the Source File

The next stage is to attach the program to our design in order that we can successfully simulate its behaviour. We do this through the commands on the *Source Menu*. Go to the *Source Menu* now and select the *Add/Remove Source Files* Command. Click on the *New* button, browse until you reach the “Samples\Tutorials” directory and select the TL.ASM file. Click open and the file should appear in the *Source Code Filename* drop down listbox.

We now need to select the code generation tool for the file. For our purposes the MPASM tool will suffice. This option should be available from the drop down listbox and so left clicking will select it in the usual fashion. (Note that If you are planning to use a new assembler or compiler for the first time, you will need to register it using the *Define Code Generation Tools* command).

Finally, it is necessary to specify which file the processor is to run. In our example this will be tl.hex (the hex file produced from MPASM subsequent to assembling tl.asm). To attach this file to the processor, right click on the schematic part for the PIC and then left click on the part. This will bring up the *Edit Component* dialogue form which contains a field for *Program File*. If it is not already specified as tl.hex either enter the path to the file manually or browse to the location of the file via the “?” button to the right of the field. Once you have specified the hex file to be run hit ok to exit the dialogue form.

We have now attached the source file to the design and specified which *Code Generation Tool* will be used. A more detailed explanation on the *Source Code Control System* is given on page 53.

Debugging the Program

Simulating the Circuit

In order to simulate the circuit point the mouse over the *Play Button* on the animation panel at the bottom right of the screen and click left. The status bar should appear with the time that the animation has been active for. You should also notice that one of the traffic lights is green while the other is red and the logic state of the pins can be seen on the schematic. You will notice, however, that the traffic lights are not changing state. This is due to a deliberate bug we have introduced into the code. At this stage it would be appropriate to debug our program and try to isolate the problem.

Debugging Mode

In order to ensure that we are thorough in the debugging of the circuit we will stop the current simulation. Once you have done that you can start debugging by pressing CTRL+F12. Two popup windows should appear - one holding the current register values and the other holding displaying the source code for the program. Either of these can be activated from the *Debug Menu* along with a host of other informative windows. We also want to

activate the *Watch Window* from which we can monitor the appropriate changes in the state variable. A full explanation of this feature is available in the section entitled *The Watch Window* on page 61.

Concentrating for now on the *Source* popup notice the red arrow at the far left. This, together with the highlighted line indicate the current position of the program counter. To set a breakpoint here hit the ENTER key (the breakpoint will always be set at the highlighted line). If we wanted to clear the breakpoint we could do so by hitting the ENTER key again but we will leave it set just now.

Setting a Breakpoint

Looking at the program it can be seen that it loops back on itself in a repeating cycle. It would therefore be a good idea to set a breakpoint at the beginning of this loop before we start. You can do this by highlighting the line (at address 000E) with the mouse, and then pressing F9. Then press F12 to set the program running. You should now see a message on the *Status Bar* indicating that a digital breakpoint has been reached and the Program Counter (PC) address. This should correspond with the address of the first breakpoint we have set.

A list of the debugging keys are given in the *Debug Menu* but for the most part we will be using F11 to single step through the program. Hit the F11 key now and notice that the red arrow at the left has moved down to the next instruction. What we have actually done is executed the 'clrw' instruction and then stopped. You can verify this by looking at the W register in the *Registers Window* and noticing that it has been cleared to zero.

What we now need to do is determine what we expect to happen on execution of the next instruction and then test to see if it actually does happen. For example, the next instruction should move the contents of the 'w' register in to PORT A i.e. Port A should be cleared. Executing this instruction and checking the *Register Window* verifies that this is indeed the case. Continuing in this vein until you reach our second breakpoint you should notice that both ports have been cleared ready for output (as dictated by the TRISB register) and that the state variable has correctly been set to 0.

As this is a function call we have the option of *Stepping Over* the function (by hitting the F10 key) but for completeness we will step through each instruction. Hitting F11 here will jump us to the first executable line of the getmask function. Stepping forward we see that the move operation was successful and that we 'land' in the correct place for adding an offset of zero onto our lookup table. When we return to the main program therefore, we have the mask that we would expect. Single stepping further and writing the mask to the port we can see the correct result on the schematic. Single stepping again to increment the state is also successful as evidenced by the *Register Window* where the value for the W register is incremented by 1.

The single step takes us onto the instruction designed to wrap the state around to zero when it is incremented above 3. This, as can be seen from the *Watch Window* , is not performing

as it should. The state should clearly be incremented to state 1 here in order for the mask to be set correctly on the next execution of the loop.

Finding the Bug

Closer investigation reveals that the problem is caused by ANDing with 4 instead of with 3. The states we want are 0,1,2, 3 and any of these when ANDed with 4 gives 0. This is why when running the simulation the state of the traffic lights does not change. The solution is simply to change the problem instruction to AND the state with 3 instead of 4. This means that the state will increment to 3 and when the W register is incremented to 4 the state will wrap around to 0. An alternative solution would be to simply test for the case when the 'W' register hits 4 and to reset it to zero.

While this short example of the debugging techniques available in Proteus VSM illustrates the basics there is a lot of additional functionality available. It is recommended that you look at the section on *Source Level Debugging* later in this documentation for a more detailed explanation.

GRAPH BASED SIMULATION TUTORIAL

Introduction

The purpose of this tutorial is to show you, by use of a simple amplifier circuit, how to perform a graph based simulation using PROTEUS VSM. It takes you, in a step-by-step fashion, through:

- Placing graphs, probes and generators.
- Performing the actual simulation.
- Using graphs to display results and take measurements.
- A survey of the some of the analysis types that are available.

The tutorial does **not** cover the use of ISIS in its general sense - that is, procedures for placing components, wiring them up, tagging objects, etc. This is covered to some extent in the *Interactive Simulation Tutorial* and in much greater detail within the ISIS manual itself. If you have not already made yourself familiar with the use of ISIS then you must do so before attempting this tutorial.

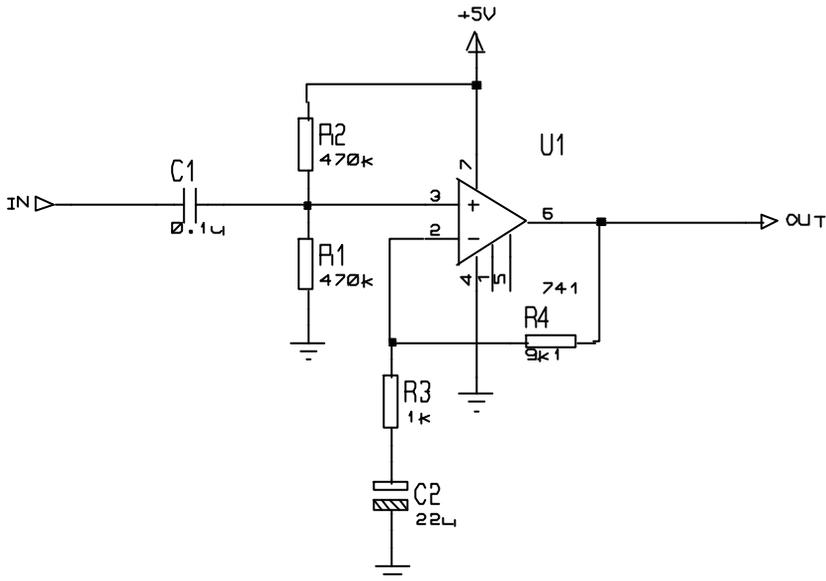
We do strongly urge you to work right the way through this tutorial before you attempt to do your own graph based simulations: gaining a grasp of the concepts will make it much easier to absorb the material in the reference chapters and will save much time and frustration in the long term.

Please note that graph based simulation is only available if you have purchased the Advanced Simulation Features option for your Proteus system.

Getting Started

The circuit we are going to simulate is an audio amplifier based on a 741 op-amp, as shown overleaf. It shows the 741 in an unusual configuration, running from a single 5 volt supply. The feedback resistors, R3 and R4, set the gain of the stage to be about 10. The input bias components, R1, R2 and C1, set a false ground reference at the non-inverting input which is decoupled from the signal input.

As is normally the case, we shall perform a *transient analysis* on the circuit. This form of analysis is the most useful, giving a large amount of information about the circuit. Having completed the description of simulation with transient analysis, the other forms of analysis will be contrasted.



If you want, you can draw the circuit yourself, or you can load a ready made design file from "Samples\Tutorials\ASIMTUT1.DSN" within your Proteus installation. Whatever you choose, at this point ensure you have ISIS running and the circuit drawn.

Generators

To test the circuit, we need to provide it with a suitable input. We shall use a voltage source with a square wave output for our test signal. A *generator* object will be used to generate the required signal.

To place a generator, first click left on the *Generator* icon: the *Object Selector* displays a list of the available generator types. For our simulation, we want a *Pulse* generator. Select the *Pulse* type, move the mouse over to the edit window, to the right of the IN terminal, and click left on the wire to place the generator.

Generator objects are like most other objects in ISIS; the same procedures for previewing and orienting the generator before placement and editing, moving, re-orienting or deleting the object after placement apply (see *GENERAL EDITING FACILITIES* in the ISIS manual or *GENERATORS AND PROBES* on page 99 in this manual).

As well as being dropped onto an existing wire, as we just did, generators may be placed on the sheet, and wired up in the normal manner. If you drag a generator off a wire, then ISIS assumes you want to detach it, and does not drag the wire along with it, as it would do for components.

Notice how the generator is automatically assigned a reference - the terminal name IN. Whenever a generator is wired up to an object (or placed directly on an existing wire) it is assigned the name of the *net* to which it is connected. If the net does not have a name, then the name of the nearest component pin is used by default.

Finally, we must edit the generator to define the pulse shape that we want. To edit the generator, tag it with the right mouse button and then click left on it to access its *Edit Generator* dialogue form. Select the *High Voltage* field and set the value to 10mV. Also set the pulse width to 0.5s.

Select the OK button to accept the changes. *GENERATORS AND PROBES* on page 99 gives a complete reference of the properties recognised by all the types of generator. For this circuit only one generator is needed, but there is no limit on the number which may be placed.

Probes

Having defined the input to our circuit using a generator, we must now place probes at the points we wish to monitor. We are obviously interested in the output, and the input after it has been biased is also a useful point to probe. If needs be, more probes can always be added at key points and the simulation repeated.

To place a probe, first click left on the *Voltage Probe* icon (ensure you have not selected a current probe by accident - we shall come to these later). Probes can be placed onto wires, or placed and then wired, in the same manner as generators. Move the mouse over to the edit window, to the left of U1 pin 3, and click left to place the probe on the wire joining pin 3 to R1 and R2. Be sure to place the probe on the wire, as it cannot be placed on the pin itself. Notice the name it acquires is the name of the nearest device to which it is connected, with the pin name in brackets. Now place the second probe by clicking left just to the left of the OUT terminal, on the wire between the junction dot and the terminal pin.

Probe objects are like generators and most other objects in ISIS; the same procedures for previewing and orienting the probe before placement, and editing, moving, re-orienting or deleting the probe after placement apply (see *GENERAL EDITING FACILITIES* in the ISIS manual or PROBES on page 107 in this manual). Probes may be edited in order to change their reference labels. The names assigned by default are fine in our case, but a useful tip when tagging probes is to aim for the *tip* of the probe, not the body or reference label.

Now that we have set up the circuit ready for simulation, we need to place a graph to display the results on.

Graphs

Graphs play an important part in simulation: they not only act as a display medium for results but actually define what simulations are carried out. By placing one or more graphs and indicating what sort of data you expect to see on the graph (digital, voltage, impedance, etc.) ISIS knows what type or types of simulations to perform and which parts of a circuit need to be included in the simulation. For a transient analysis we need an *Analogue* type graph. It is termed analogue rather than transient in order to distinguish it from the *Digital* graph type, which is used to display results from a digital analysis, which is really a specialised form of transient analysis. Both can be displayed against the same time axis using a *Mixed* graph.

To place a graph, first select the *Graph* icon: the *Object Selector* displays a list of the available graph types. Then select the *Analogue* type, move the mouse over to the edit window, click (and hold down) the left mouse button, drag out a rectangle of the appropriate size, and then release the mouse button to place the graph.

Graphs behave like most objects in ISIS, though they do have a few subtleties. We will cover the features pertinent to the tutorial as they occur, but the reference chapter on graphs is well worth a read. You can tag a graph in the usual way with the right mouse button, and then (using the left mouse button) drag one of the handles, or the graph as a whole, about to resize and/or reposition the graph.

We now need to add our generator and probes on to the graph. Each generator has a probe associated with it, so there is no need to place probes directly on generators to see the input wave forms. There are three ways of adding probes and generators to graphs:

- The first method is to tag each probe/generator in turn and drag it over the graph and release it - exactly as if we were repositioning the object. ISIS detects that you are trying to place the probe/generator over the graph, restores the probe/generator to its original position, and adds a trace to the graph with the same reference as that of the probe/generator. Traces may be associated with the left or right axes in an analogue graph, and probes/generators will add to the axis nearest the side they were dropped. Regardless of where you drop the probe/generator, the new trace is always added at the bottom of any existing traces.

The second and third method of adding probes/generators to a graph both use the *Add Trace* command on the graph menu; this command always adds probes to the current graph (when there is more than one graph, the current graph is the one currently selected on the *Graph* menu).

- If the *Add Trace* command is invoked without any tagged probes or generators, then the *Add Transient Trace* dialogue form is displayed, and a probe can be selected from a list of all probes in the design (including probes on other sheets).
- If there are tagged probes/generators, invoking the *Add Trace* command causes you to be prompted to *Quick Add* the tagged probes to the current graph; selecting the *No* option invokes the *Add Transient Trace* dialogue form as previously described. Selecting the *Yes* option adds all tagged probes/generators to the current graph in alphabetical order.

We will *Quick Add* our probes and the generator to the graph. Either tag the probes and generators individually, or, more quickly, drag a tag box around the entire circuit - the *Quick Add* feature will ignore all tagged objects other than probes and generators. Select the *Add Trace* option from the *Graph* menu and answer *Yes* to the prompt. The traces will appear on the graph (because there is only one graph, and it was the last used, it is deemed to be the *current* graph). At the moment, the traces consist of a name (on the left of the axis), and an empty data area (the main body of the graph). If the traces do not appear on the graph, then it is probably too small for ISIS to draw them in. Resize the graph, by tagging it and dragging a corner, to make it sufficiently big.

As it happens, our traces (having been placed in alphabetical order) have appeared in a reasonable order. We can however, shuffle the traces about. To do this, ensure the graph is *not* tagged, and click right over the name of a trace you want to move or edit. The trace is highlighted to show that it is tagged. You can now use the left mouse button to drag the trace up or down or to edit the trace (by clicking left without moving the mouse) and the right button to delete the trace (i.e. remove it from the graph). To untag all traces, click the right mouse button anywhere over the graph, but *not* over a trace label (this would tag or delete the trace).

There is one final piece of setting-up to be done before we start the simulation, and this is to set the simulation run time. ISIS will simulate the circuit according to the end time on the x-scale of the graph, and for a new graph, this defaults to one second. For our purposes, we want the input square wave to be of fairly high audio frequency, say about 10kHz. This needs a total period of 100 μ s. Tag the graph and click left on it to bring up its *Edit Transient Graph* dialogue form. This form has fields that allow you to title the graph, specify its simulation start and stop times (these correspond to the left and right most values of the x axis), label the left and right axes (these are not displayed on *Digital* graphs) and also specifies general

properties for the simulation run. All we need to change is the stop time from 1.00 down to 100u (you can literally type in 100u - ISIS will covert this to 100E-6) and select OK.

The design is now ready for simulation. At this point, it is probably worthwhile loading our version of the design ("Samples\Tutorials\ASIMTUT2.DSN") so as to avoid any problems during the actual simulation and subsequent sections. Alternatively, you may wish to continue with the circuit you have entered yourself, and only load the ASIMTUT2.DSN file if problems arise.

Simulation

To simulate the circuit, all you need do is invoke the *Simulate* command on the *Graph* menu (or use its keyboard short-cut: the space bar). The *Simulate* command causes the circuit to be simulated and the *current* graph (the one marked on the *Graph* menu) to be updated with the simulation results.

Do this now. The status bar indicates how far the simulation process has reached. When the simulation is complete, the graph is redrawn with the new data. For the current release of ISIS and simulator kernels, the start time of a graph is ignored - simulation always starts at time zero and runs until the stop time is reached or until the simulator reaches a quiescent state. You can abort a simulation mid-way through by pressing the ESC key.

A simulation log is maintained for the last simulation performed. You can view this log using the *View Log* command on the *Graph* menu (or the CTRL+'V' keyboard short-cut). The simulation log of an analogue simulation rarely makes for exciting reading, unless warnings or errors were reported, in which case it is where you will find details of exactly what went wrong. In some cases, however, the simulation log provides useful information that is not easily available from the graph traces.

So the first simulation is complete. Looking at the traces on the graph, its hard to see any detail. To check that the circuit is working as expected, we need to take some measurements...

Taking Measurements

A graph sitting on the schematic, alongside a circuit, is referred to as being *minimised*. To take timing measurements we must first *maximise* the graph. To do this, first ensure the graph is *not* tagged, and then click the left mouse button on the graph's title bar; the graph is redrawn in its own window. Along the top of the display, the menu bar is maintained. Below this, on the left side of the screen is an area in which the trace labels are displayed and to right of this are the traces themselves. At the bottom of the display, on the left is a toolbar, and to the right of this is a status area that displays cursor time/state information. As this is a new graph, and we have not yet taken any measurements, there are no cursors visible on the graph, and the status bar simply displays a title message.

The traces are colour coded, to match their respective labels. The OUT and U1(POS IP) traces are clustered at the top of the display, whilst the IN trace lies along the bottom. To see the traces in more detail, we need to separate the IN trace from the other two. This can be achieved by using the left mouse button to drag the trace *label* to the right-hand side of the screen. This causes the right y-axis to appear, which is scaled separately from the left. The IN trace now seems much larger, but this is because ISIS has chosen a finer scaling for the right axis than the left. To clarify the graph, it is perhaps best to remove the IN trace altogether, as the U1(POS IP) is just as useful. Click right on the IN label twice to delete it. The graph now reverts to a single, left hand side, y-axis.

We shall measure two quantities:

- The voltage gain of the circuit.
- The approximate fall time of the output.

These measurements are taken using the *Cursors*.

Each graph has two cursors, referred to as the *Reference* and *Primary* cursors. The reference cursor is displayed in red, and the primary in green. A cursor is always 'locked' to a trace, the trace a cursor is locked to being indicated by a small 'X' which 'rides' the waveform. A small mark on both the x and y -axes will follow the position of the 'X' as it moves in order to facilitate accurate reading of the axes. If moved using the keyboard, a cursor will move to the next small division in the x -axis.

Let us start by placing the *Reference* cursor. The same keys/actions are used to access both the *Reference* and *Primary* cursors. Which is actually affected is selected by use of the CTRL key on the keyboard; the *Reference* cursor, as it is the least used of the two, is always accessed with the CTRL key (on the keyboard) pressed down. To place a cursor, all you need to do is point at the trace data (*not* the trace label - this is used for another purpose) you want to lock the cursor to, and click left. If the CTRL key is down, you will place (or move) the *Reference* cursor; if the CTRL key is not down, then you will place (or move) the *Primary* cursor. Whilst the mouse button (and the CTRL key for the *Reference* cursor) is held down, you can drag the cursor about. So, hold down (and keep down) the CTRL key, move the mouse pointer to the right hand side of the graph, above both traces, and press the left mouse button. The red *Reference* cursor appears. Drag the cursor (still with the CTRL key down) to about 70u or 80u on the x -axis. The title on the status bar is removed, and will now display the cursor time (in red, at the left) and the cursor voltage along with the name of the trace in question (at the right). It is the OUT trace that we want.

You can move a cursor in the X direction using the left and right cursor keys, and you can lock a cursor to the previous or next trace using the up and down cursor keys. The LEFT and RIGHT keys move the cursor to the left or right limits of the x -axis respectively. With the control key still down, try pressing the left and right arrow keys on the keyboard to move the *Reference* cursor along small divisions on the time axis.

Now place the *Primary* cursor on the OUT trace between 20u and 30u. The procedure is exactly the same as for the *Reference* cursor, above, except that you do not need to hold the CTRL key down. The time and the voltage (in green) for the primary cursor are now added to the status bar.

Also displayed are the differences in both time and voltage between the positions of the two cursors. The voltage difference should be a fraction above 100mV. The input pulse was 10mV high, so the amplifier has a voltage gain of 10. Note that the value is positive because the

Primary cursor is above the *Reference* cursor - in delta read-outs the value is *Primary* minus *Reference*.

We can also measure the fall time using the relative time value by positioning the cursors either side of the falling edge of the output pulse. This may be done either by dragging with the mouse, or by using the cursor keys (don't forget the CTRL key for the *Reference* cursor). The *Primary* cursor should be just to the right of the curve, as it straightens out, and the *Reference* cursor should be at the corner at the start of the falling edge. You should find that the falling edge is a little under 10 μ s.

Using Current Probes

Now that we have finished with our measurements, we can return to the circuit - just close the graph window in the usual way, or for speed you can press ESC on the keyboard. We shall now use a current probe to examine the current flow around the feedback path, by measuring the current into R4.

Current probes are used in a similar manner to voltage probes, but with one important difference. A current probe needs to have a *direction* associated with it. Current probes work by effectively breaking a wire, and inserting themselves in the gap, so they need to know which way round to go. This is done simply by the way they are placed. In the default orientation (leaning to the right) a current probe measures current flow in a *horizontal* wire from left to right. To measure current flow in a vertical wire, the probe needs to be rotated through 90° or 270°. Placing a probe at the wrong angle is an error, that will be reported when the simulation is executed. If in doubt, look at the arrow in the symbol. This points in the direction of current flow.

Select a current probe by clicking on the *Current Probe* icon. Click left on the clockwise *Rotation* icon such that the arrow points downwards. Then place the probe on the vertical wire between the right hand side of R4 and U1 pin 6. Add the probe to the right hand side of the graph by tagging and dragging onto the right hand side of the minimised graph. The right side is a good choice for current probes because they are normally on a scale several orders of magnitude different than the voltage probes, so a separate axis is needed to display them in detail. At the moment no trace is drawn for the current probe. Press the space bar to re-simulate the graph, and the trace appears.

Even from the minimised graph, we can see that the current in the feedback loop follows closely the wave form at the output, as you would expect for an op-amp. The current changes between 10 μ A and 0 at the high and low parts of the trace respectively. If you wish, the graph may be maximised to examine the trace more closely.

Frequency Analysis

As well as transient analysis, there are several other analysis types available in analogue circuit simulation. They are all used in much the same way, with graphs, probes and generators, but they are all different variations on this theme. The next type of analysis that we shall consider is *Frequency* analysis. In frequency analysis, the x-axis becomes frequency (on a logarithmic scale), and both magnitude and phase of probed points may be displayed on the y-axes.

To perform a frequency analysis a *Frequency* graph is required. Click left on the *Graph* icon, to re-display the list of graph types in the object selector, and click on the *Frequency* graph type. Then place a graph on the schematic as before, dragging a box with the left mouse button. There is no need to remove the existing transient graph, but you may wish to do so in order to create some more space (click right twice to delete a graph).

Now to add the probes. We shall add both the voltage probes, OUT and U1(POS IP). In a frequency graph, the two y-axes (left and right) have special meanings. The left y-axis is used to display the *magnitude* of the probed signal, and the right y-axis the *phase*. In order to see both, we must add the probes to both sides of the graph. Tag and drag the OUT probe onto the left of the graph, then drag it onto the right. Each trace has a separate colour as normal, but they both have the same name. Now tag and drag the U1(POS IP) probe onto the left side of the graph only.

Magnitude and phase values must both be specified with respect to some reference quantity. In ISIS this is done by specifying a *Reference Generator*. A reference generator always has an output of 0dB (1 volt) at 0°. Any existing generator in the circuit may be specified as the reference generator. All the other generators in the circuit are ignored in a frequency analysis. To specify the IN generator as the reference in our circuit, simply tag and drag it onto the graph, as if you were adding it as a probe. ISIS assumes that, because it is a generator, you are adding it as the reference generator, and prints a message on status line confirming this. Make sure you have done this, or the simulation will not work correctly.

There is no need to edit the graph properties, as the frequency range chosen by default is fine for our purposes. However, if you do so (by pointing at the graph and pressing CTRL-E), you will see that the *Edit Frequency Graph* dialogue form is slightly different from the transient case. There is no need to label the axes, as their purpose is fixed, and there is a check box which enables the display of magnitude plots in dB or normal units. This option is best left set to dB, as the absolute values displayed otherwise will not be the actual values present in the circuit.

Now press the space bar (with the mouse over the frequency graph) to start the simulation. When it has finished, click left on the graph title bar to maximise it. Considering first the OUT magnitude trace, we can see the pass-band gain is just over 20dB (as expected), and the useable frequency range is about 50Hz to 20kHz. The cursors work in exactly the same manner as before - you may like to use the cursors to verify the above statement. The OUT phase

trace shows the expected phase distortion at the extremes of the response, dropping to -90° just off the right of the graph, at the unity gain frequency. The high-pass filter effect of the input bias circuitry can be clearly seen if the U1(POS IP) magnitude trace is examined. Notice that the x-axis scale is logarithmic, and to read values from the axis it is best to use the cursors.

Swept Variable Analysis

It is possible with ISIS to see how the circuit is affected by changing some of the circuit parameters. There are two analysis types that enable you to do this - the *DC Sweep* and the *AC Sweep*. A *DC Sweep* graph displays a series of operating point values against the swept variable, while an *AC Sweep* graph displays a series of single point frequency analysis values, in magnitude and phase form like the *Frequency* graph.

As these forms of analysis are similar, we shall consider just one - a *DC Sweep*. The input bias resistors, R1 and R2, are affected by the small current that flows into U1. To see how altering the value of both of these resistors affects the bias point, a *DC Sweep* is used.

To begin with place a *DC Sweep* graph on an unused space on the schematic. Then tag the U1(POS IP) probe and drag it onto the left of the graph. We need to set the sweep value, and this is done by editing the graph (point at it and press CTRL-E). The *Edit DC Sweep Graph* dialogue form includes fields to set the swept variable name, its start and ending values, and the number of steps taken in the sweep. We want to sweep the resistor values across a range of say $100\text{k}\Omega$ to $5\text{M}\Omega$, so set the *Start* field to 100k and the *Stop* field to 5M . Click on *OK* to accept the changes.

Of course, the resistors R1 and R2 need to be altered to make them swept, rather than the fixed values they already are. To do this, click right and then left on R1 to edit it, and alter the *Value* field from 470k to X. Note that the swept variable in the graph dialogue form was left at X as well. Click on *OK*, and repeat the editing on R2 to set its value to X.

Now you can simulate the graph by pointing at it and pressing the space-bar. Then, by maximising the graph, you can see that the bias level reduces as the resistance of the bias chain increases. By $5\text{M}\Omega$ it is significantly altered. Of course, altering these resistors will also have an effect on the frequency response. We could have done an *AC Sweep* analysis at say 50Hz in order to see the effect on low frequencies.

Noise Analysis

The final form of analysis available is *Noise* analysis. In this form of analysis the simulator will consider the amount of thermal noise that each component will generate. All these noise contributions are then summed (having been squared) at each probed point in the circuit. The results are plotted against the noise bandwidth.

There are some important peculiarities to noise analysis:

- The simulation time is directly proportional to the number of voltage probes (and generators) in the circuit, since each one will be considered.
- Current probes have no meaning in noise analysis, and are ignored.
- A great deal of information is presented in the simulation log file.
- PROSPICE computes both input and output noise. To do the former, an input reference must be defined - this is done by dragging a generator onto the graph, as with a frequency reference. The input noise plot then shows the equivalent noise at the input for each output point probed.

To perform a noise analysis on our circuit, we must first restore R1 and R2 back to 470kΩ. Do this now. Then select a *Noise* graph type, and place a new graph on an unused area of the schematic. It is really only output noise we are interested in, so tag the OUT voltage probe and drag it onto the graph. As before, the default values for the simulation are fine for our needs, but you need to set the input reference to the input generator IN. The *Edit Noise Graph* dialogue form has the check box for displaying the results in dBs. If you use this option, then be aware that 0dB is considered to be 1 volt r.m.s. Click on *Cancel* to close the dialogue form.

Simulate the graph as before. When the graph is maximised, you can see that the values that result from this form of analysis are typically extremely small (pV in our case) as you might expect from a noise analysis of this type. But how do you track down sources of noise in your circuit? The answer lies in the simulation log. View the simulation log now, by pressing CTRL+V. Use the down arrow icon to move down past the operating point printout, and you should see a line of text that starts

```
Total Noise Contributions at ...
```

This lists the individual noise contributions (across the entire frequency range) of each circuit element that produces noise. Most of the elements are in fact inside the op-amp, and are prefixed with U1_. If you select the *Log Spectral Contributions* option on the *Edit Noise Graph* dialogue form, then you will get even more log data, showing the contribution of each component at each spot frequency.

INTERACTIVE SIMULATION

BASIC SKILLS

The Animation Control Panel



Interactive simulations are controlled from a simple VCR like panel that behaves just like a normal remote control. This control is situated at the bottom right of the screen. There are four buttons that you use to control the flow of the circuits.

- The PLAY button is used to start the simulator.
- The STEP button allows you to step through the animation at a defined rate. If the button is pressed and released then the simulation advances by one time step; if the button is held down then the animation advances continuously until the button is released. The single step time increment may be adjusted from the *Animated Circuit Configuration* dialog box. The step time capability is useful for monitoring the circuits more closely and seeing in slow motion what is affecting what.

- The PAUSE button suspends the animation which can then be resumed either by clicking the PAUSE button again, or single stepped by pressing the STEP button. The simulator will also enter the paused state if a breakpoint is encountered.

The simulation can also be paused by pressing the PAUSE key on the computer keyboard.

- The STOP button tells PROSPICE to stop doing a real time simulation. All animation is stopped and the simulator is unloaded from memory. All the indicators are reset to their inactive states but the actuators (switches etc.) retain their existing settings.

The simulation can also be stopped by pressing SHIFT-BREAK on the computer keyboard.

During an animation, the current simulation time and average CPU loading are displayed on the status bar. If there is insufficient CPU power to run the simulation in real time, the display will read 100% and the simulation time will cease to advance in real time. Aside from this, no harm results from simulating very fast circuits as the system automatically regulates the amount of simulation performed per animation frame.

Indicators and Actuators

Aside from ordinary electronic components, interactive simulations generally make use of special *Active Components*. These components have a number of graphical states and come in two flavours: *Indicators* and *Actuators*. Indicators display a graphical state which changes according to some measured parameter in the circuit, whilst Actuators allow their state to be determined by the user and then modify some characteristic of the circuit.

Actuators are designated by the presence of small red marker symbols which can be clicked with the mouse to operate the control. If you have a mouse with a wheel, you can also operate actuators by pointing at them and rolling the wheel in the appropriate direction.

Setting Up an Interactive Simulation

Most of the skills required to set up and run an interactive simulation centre around drawing a circuit in ISIS. This subject is best grasped by working through the tutorial in the ISIS manual. However, the process may be summarized as follows:

- Pick the components you want to use from the device libraries using the ‘P’ button on the *Device Selector*. All the active components (actuators and indicators) are contained in the library ACTIVE.LIB, but you can use any components which have a simulator model of some kind.
- Place the components on the schematic.
- Edit them – click right then click left, or press CTRL-E – in order to assign appropriate values and properties. Many models provided context sensitive help so that information about individual properties can be viewed by placing the caret in the field and pressing F1.
- Micro-processor source code can be brought under the control of PROTEUS VSM using the commands on the *Source* menu. Don’t forget also to assign the object code (HEX file) to the micro-processor component on the schematic.
- Wire the circuit up by clicking on the pins.
- Delete components by clicking right twice.
- Move components by clicking right then dragging with the left button.
- Click the PLAY button on the *Animation Control Panel* to run the simulation.

Where you have used virtual instruments, or microprocessor models, popup windows related to this components may be displayed using the commands on the *Debug* menu.

TAKING MEASUREMENTS

There are several different ways of taking measurements during an interactive simulation. These include

- Animation effects.
- Point and shoot operating parameter information.
- Voltage and current Probes
- Virtual instruments.

Animation Effects

As well as any active components in the circuit, a number of other animation effects may be enabled to help you study the circuit operation. These options may be enabled using the *Set Animation Options* command on the *System* menu. The settings are saved with the design.

Pin Logic States

This option displays a coloured square by each pin that is connected to a digital or mixed net. By default, the square is blue for logic 0, red for logic 1 and grey for floating. Logic contentions are shown in yellow. The colours may be changed via the *Set Design Defaults* command on the *Template* menu.

Enabling this option incurs a modest extra burden on the simulator, but can be very, very useful when used in condition with breakpoints and single stepping as it enables you to study the state of a micro-controllers output port pins as the code is single stepped.

Show Wire Voltage as Colour

This option causes any wire that is part of an analogue or mixed net to be displayed in a colour that represents its voltage. By default, the spectrum ranges from blue at $-6V$ through green at $0V$ to red at $+6V$. The voltages may be changed on the *Set Animation Options* dialogue, and the colours via the *Set Design Defaults* command on the *Template* menu.

Enabling this option incurs a modest extra burden on the simulator but is very effective for helping novices understand a circuits operation, especially when used in conjunction with the *Show Wire Current as Arrows* option.

Show Wire Current as Arrows

This option causes an arrow to be drawn on any wire that carrying current. The direction of the arrow reflects conventional current flow, and it is displayed if the magnitude of the current exceeds a threshold. The default threshold is $1\mu A$, although this can be changed on the *Set Animation Options* dialogue.

Computing the wire currents involves the insertion of a zero value voltage source in every wire segment in the schematic (apart from inside models) and this can create a large number of extra nodes. Consequently the burden on the simulator can be significant. However, the arrows are most useful in teaching basic electricity and electronics where the circuits tend to be fairly simple.

Show Voltage & Current on Probes

This option cause Voltage and Current probe objects to provide a real time readout during interactive simulation. Voltage probes placed on digital or bus nets will show logic state or bus values as appropriate.

Enabling this option incurs a small performance overhead, especially if there a relatively large number of probes on the schematic.

Point and Shoot Parameter Information

You can interrogate any of the components in the schematic to obtain information about their current status whenever the simulation is paused. At the least this will be a summary of their node voltages and/or pin logic states but some components will display additional data such as relative (pin-pin) voltages and power dissipation.

To display parameter values:

1. Ensure that the simulation is paused using the *Animation Control Panel*.
2. Select the *Multimeter* icon in ISIS.
3. Click left on any component to view its parameters.

DC Operating Point

If you view parameter information at time zero, then you are viewing the DC operating point of the circuit. Otherwise, the parameters displayed represent instantaneous values as at the time you paused the simulation. This is particularly significant for power dissipation values; these numbers are not averaged in any way whatsoever and represent a simple multiplication of instantaneous voltage multiplied by instantaneous current.

Analogue or Digital?

Where a component pin is on a net which carries only digital logic states (i.e. there are no analogue components connected to it), Proteus VSM will display the terminal voltage as a logic state. Otherwise an analogue node voltage is displayed.

Voltage and Current Probes

The voltage and current probe objects within ISIS are most commonly used in graph based simulations where the voltages and currents being measured can be plotted out on a graph. However, you can also use them during interactive simulation to display node voltages or currents which update in real time.

The probe objects have two advantages over the virtual ammeters and voltmeters:

- They can be added and removed to any point in the circuit without drawing wires.
- They take up much less room on the schematic.

Voltage probes dropped onto nets which carry only digital signals (i.e. to which no analogue components are connected) will display logic states rather than node voltages. Voltage probes dropped onto buses will display a bus value.

To place voltage or current probes

1. Select the *Voltage Probe* or *Current Probe* icon within ISIS as appropriate.
2. Click left on the wire that you wish to measure to add the probe.

Note that current probes are directional, and measure conventional current as indicated by their arrow symbol. The arrow must be parallel to the wire segment on which the probe is placed.

The real time display of voltages and currents can be turned off via the *Set Animation Options* command on the *System* menu.

Virtual Instruments

A selection of virtual instruments can be displayed by clicking the *Multimeter* icon in ISIS. The *Object Selector* then displays a list of instrument types which you can select and place just like ordinary components within ISIS. Further information about how to use each instrument is given in the *VIRTUAL INSTRUMENTS* chapter on page 31.

ANIMATION TIMESTEP CONTROL

Overview

Two parameters control how an interactive simulation evolves in real time. The *Animation Frame Rate* determines the number of times that the screen is refreshed per second, whilst the *Animation Timestep* determines by how much simulation is progressed during each frame. For real time operation, the timestep should be set to the reciprocal of the frame rate.

Frames Per Second

Normally, it is unnecessary to adjust the frame rate as the default value of 20 frames per second gives smooth animation without overburdening the (considerable) graphics performance of a modern PC. It is, however, occasionally useful to slow it right down when debugging the operation of animated compute models.

Timestep Per Frame

The *Animation Timestep* can be used to make fast circuits run slowly, or slow circuits run quickly. For real time operation, the timestep should be set to the reciprocal of the frame rate.

Of course, the entered timestep will always be a desired value. Achieving it is dependent on there being sufficient CPU power to compute the necessary simulation events within the time allocated to each frame. The CPU load figure displayed during animation represents the ratio of these two times. If insufficient CPU power is available, the CPU load will be 100%, and the *achieved* timestep per frame will drop off.

Single Step Time

The remaining timestep control parameter is the *Single Step Time*. This is the time by which the simulation will advance when the single step button on the *Animation Control Panel* is pressed.

HINTS AND TIPS

Circuit Timescale

Interactive simulations will be generally be viewed in real time, so it is no use drawing circuits with 1MHz clocks, or 10kHz sine wave inputs unless you also adjust the *Timestep per Frame* value in the *Animated Circuits Configuration* dialog box.

If you do attempt to simulate something that operates very quickly, there are several things to bear in mind:

- Given finite CPU power, only a certain amount of simulation time can be computed in a fixed amount of real time. PROTEUS VSM is designed to maintain its animation frame rate (frames per second) whatever, and to cut short any frames which are not completed in the available time. The consequence of this is that very fast circuits will simulate slowly (relative to real time) but smoothly.
- Analogue component models simulate very much more slowly than digital ones. On a fast PC you can simulate digital circuitry operating up to several MHz in real time but analogue circuit electronics will manage only about 10-20kHz.

Consequently it is very silly to attempt to simulate clock oscillators for digital circuitry in the analogue domain. Instead, use a *Digital Clock* generate and set its *Isolate Before* checkbox so that the analogue clock is not simulated.

Voltage Scaling

If you intend to use coloured wires to indicate node voltages you need to give some thought as to the range of voltages that will occur in your circuit. The default range for displayed voltages is +/-6V so if the circuit operates at significantly different voltages from this you may need to change the *Maximum Voltage* value in the *Animated Circuits Configuration* dialog box.

Earthing

PROSPICE will attempt to define a sensible earth point for any active circuit which does not specifically include a ground terminal. In practice this is usually chosen as the mid-point of the battery, or the centre tap if the circuit has a split supply. It follows that the positive terminal of the battery will sit above ground and the negative terminal below it, corresponding to red and blue wire colours. However, if this behaviour is not what is required, you always have the option of explicitly defining the ground reference point using a ground terminal in ISIS.

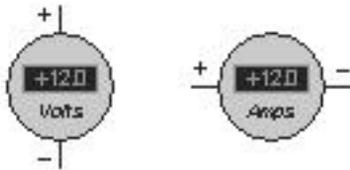
High Impedance Points

The automatic earthing logic also checks for any device terminals which are not connected to ground, and will insert high value resistors automatically in order to ensure convergence of the SPICE simulation. This means that ill formed circuits (e.g. circuits with unconnected, or partially connected) components will generally simulate – although there can sometimes be strange results.

VIRTUAL INSTRUMENTS

VOLTMETERS & AMMETERS

Proteus VSM provides both AC and DC voltmeters and ammeters. These operate in real time and can be wired into the circuit just like any other component. Once the simulation is started they display the voltage across their terminals or the current flowing through them in an easy to read digital format.



The FSD of the meters can be set to any of 100, 100m and 100u with a resolution of 3 significant figures and a maximum number of two decimal places by editing the meter component and selecting the required *Display Range* property.

The voltmeter models support an internal load resistance property which defaults to 100M but which can also be changed by editing the component, in the usual way. Leaving the value blank disables the load resistance element of the model.

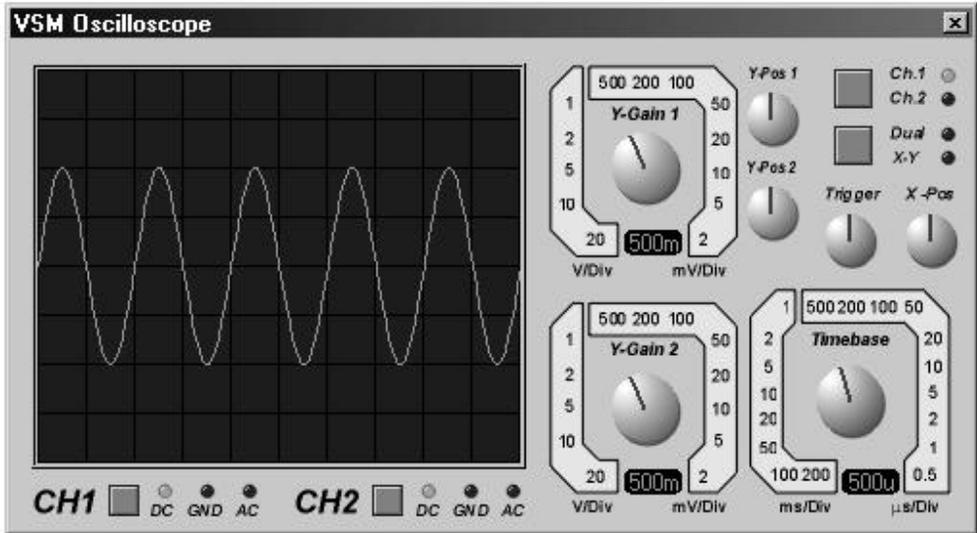
The AC voltmeters and ammeters display true RMS values integrated over a user definable time constant.

OSCILLOSCOPE

Overview

The VSM Oscilloscope models a basic dual beam analogue unit and is specified as follows:

- Dual channel or X-Y operation.
- Channel gain from 20V/div to 2mV/div
- Timebase from 200ms/div to 0.5us/div
- Automatic voltage level triggering locked to either channel.
- AC or DC coupled inputs.



Using the Oscilloscope

To display analogue waveforms:

1. Select the *Multimeter* icon, pick the OSCILLOSCOPE and place one on the schematic, wiring its inputs to the signals you want to record.
2. Start an interactive simulation by pressing the play button on the *Animation Control Panel*. The oscilloscope window should appear.
3. If displaying two signals, select *Dual* mode.
4. Set the *timebase* dial to a suitable value for the circuit. You will need to think about the waveform frequencies present in your circuit, and convert these to cycle times by taking their reciprocal.
5. If displaying signals with a DC offset, select AC mode on either or both input channels as appropriate.
6. Adjust the *Y-gain* and *Y-pos* dials so that the waveforms are of a suitable size and position. If a waveform consists of a small AC signal on top of a large DC voltage you may need to connect a capacitor between the test point and the oscilloscope, as the *Y-pos* controls can compensate for only a certain amount of DC.
7. Decide which channel you want to trigger off, and ensure that the *Ch1* or *Ch2* led is lit, as appropriate.

8. Rotate the *trigger* dial until the display locks to the required part of the input waveform. It will lock to rising slopes if the dial points up, and falling slopes if it points down.

 See *Rotary Dials* on page 51 for details of how to adjust the rotary controls featured in the oscilloscope.

Modes of Operation

The oscilloscope can operate in 3 modes, indicated as follows:

- Single Beam - neither *Dual* nor *X-Y* leds are lit In this mode, the *Ch1* and *Ch2* leds indicate which channel is being displayed.
- Dual Beam – the *Dual* led is lit. In this mode, the *Ch1* and *Ch2* leds indicate which channel is being used for triggering.
- X-Y mode – the *X-Y* led is lit.

The current mode may be cycled through these options by clicking the button next to the *Dual* and *X-Y* mode leds.

Triggering

The VSM oscilloscope provides an automatic triggering mechanism which enables it synchronize the timebase to the incoming waveform.

- Which input channel is used for triggering is indicated by the *Ch1* and *Ch2* leds.
- The *trigger* dial rotates continuously round 360 degrees and sets the voltage level and slope at which triggering occurs. When the mark is pointing upwards, the scope triggers on rising voltages; when the mark points downwards it triggers on a falling slope.
- If no triggering occurs for more than 1 timebase period, the timebase will free run.

Input Coupling

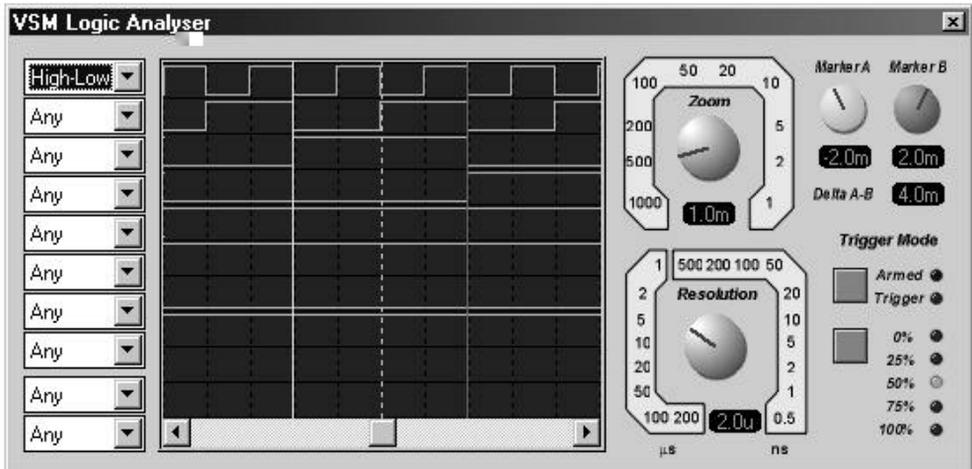
Each input channel can either be directly coupled (DC coupling) or coupled through a simulated capacitance (AC coupling). The latter mode is useful for displaying signals which carry a small AC signal on top of a much larger DC bias voltage.

The inputs may also be temporarily grounded which can be useful when aligning them to the graticule prior to taking measurements.

LOGIC ANALYSER

Overview

A logic analyser operates by continuously recording incoming digital data into a large *capture buffer*. This is a sampling process, so there is an adjustable *resolution* which defines the shortest pulse that can be recorded. A *triggering section* monitors the incoming data and causes the data capture process to stop a certain time after the triggering condition has arisen.; capturing is started by *arming* the instrument. The result is that the contents of the capture buffer both before and after the trigger time can be displayed. Since the capture buffer is very large (10000 samples, in this case) a means of zooming and panning the display is provided. Finally, movable *measurement markers* allow accurate timing measurements of pulse widths and so forth.



The VSM Logic Analyser models a basic 24 channel unit specified as follows:

- 8 x 1 bit traces and 2 x 8 bit bus traces.
- 10000 x 24 bit capture buffer.
- Capture resolution from 200us per sample to 0.5ns per sample with corresponding capture times from 2s to 5ms.
- Display zoom range from 1000 samples per division in to 1 sample per division.
- Triggering on ANDed combination of input states and/or edges, and bus values.
- Trigger positions at 0, 25, 50, 75 and 100% of the capture buffer.

- Two cursors provided for taking accurate timing measurements.

Using the Logic Analyser

To capture and display digital data:

1. Select the *Multimeter* icon, pick the LOGIC ANALYSER and place one on the schematic, wiring its inputs to the signals you want to record.
2. Start an interactive simulation by pressing the play button on the *Animation Control Panel*. The logic analyser popup window should appear.
3. Set the *resolution* dial to a value suitable for your application. This represents the smallest width of pulse that can be recorded. The finer the resolution, the shorter will be the time during which data is captured.
4. Set the combo-boxes on the left of the instrument to define the required *trigger condition*. For example, if you want to trigger the instrument when the signal connected to channel 1 is high and the signal connected to channel 3 is a rising edge, you would set the first combo-box to “High” and the third combo-box to “Low-High”.
5. Decide whether you want to view data mainly before or after the trigger condition occurs, and click the button next to the *percentage* led to selected the required trigger position.
6. When you are ready, click the button to the left of the *armed* led to arm the instrument.

The *armed* led will light and the *trigger* led will be extinguished. The logic analyser will now capture incoming data continuously whilst monitoring the inputs for the trigger condition. When this occurs, the *trigger* led will light. Data capture will then continue until the part of the capture buffer after the trigger position is full, at which point the *armed* led will extinguish and the captured data will appear in the display.

Panning and Zooming

Since the capture buffer holds 10000 samples, and the display is only 250 pixels wide, a means is needed to pan and zoom within the capture buffer. The *Zoom* dial determines the number of samples per division, whilst the scroll-bar attached to the display itself allows you to pan left and right.

Note that the readout located beneath the *Zoom* dial displays the current time per division in seconds, not the actual setting of the dial itself. The division time is calculated by multiplying the zoom setting by the resolution.

Taking Measurements

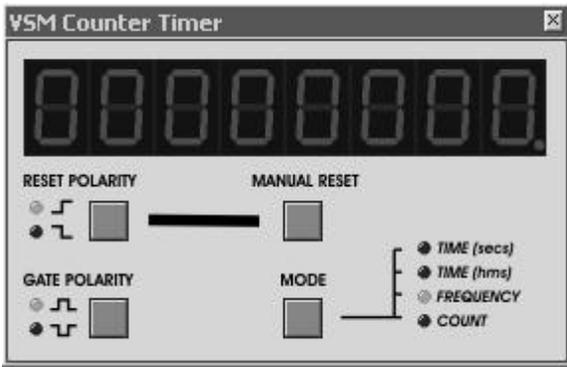
Two adjustable markers are provided which can be used for taking accurate time measurements. Each marker can be positioned using the correspondingly coloured dial. The readout below each dial displays the time point occupied by the marker relative to the trigger time, whilst the *Delta A-B* readout displays the time difference between the markers.

 See *Rotary Dials* on page 51 for details of how to adjust the rotary controls featured in the logic analyser.

COUNTER TIMER

Overview

The VSM Counter Timer is a multi-purpose digital instrument which can be used to measure time intervals, signal frequency and pulse counts.



The counter timer supports the following operating modes:

- Timer mode (seconds), resolution 1 μ s.
- Timer mode (hours, mins, secs), resolution 1ms.
- Frequency meter mode, resolution 1Hz.
- Counter mode - maximum count 99,999,999.

The time, frequency or count value is displayed both on the schematic component and also optionally on the counter timer popup as shown above. You can make the popup appear either by clicking left on the counter timer component or by selecting it from the Debug menu whilst the simulation is running.

Using the Timer Modes

To measure time intervals:

1. Select the *Multimeter* icon, pick the COUNTER TIMER and place one on the schematic.
2. If appropriate, wire the CE (clock enable) pin to a signal that will control when the time display advances. If you don't require a clock enable function you can leave this pin floating.
3. If appropriate, wire the RST (reset) pin to a signal that will switch state to reset the timer to zero. If you don't need to reset the timer from within the circuit you can leave this pin floating.
4. Point at the Counter Timer and press Ctrl-E to edit its settings.
5. Select the required timer mode (secs or hms) and logic polarities for the CE and RST functions.
6. Close the dialogue and start the simulation in the usual way.

Notes:

- The RST pin is edge triggered as opposed to level triggered. If you need to hold the timer at zero, you can use the CE and RST pins in combination.
- The Counter Timer popup provides a manual reset button which will reset the timer to zero at any point during the simulation run. This can be especially useful in embedded systems work where you can use the Counter Timer to establish the time taken to execute particular sections of a program.

Using the Frequency Mode

To measure the frequency of a digital signal:

1. Select the *Multimeter* icon, pick the COUNTER TIMER and place one on the schematic.
2. Wire the CLK (clock) pin to the signal you wish to measure. The CE and RST pins are not used in frequency mode.
3. Point at the Counter Timer and press Ctrl-E to edit its settings and then select *Frequency Mode*.
4. Close the dialogue and start the simulation in the usual way.

Notes:

- The frequency meter works by counting the number of positive edge transitions occurring in each second of simulation time. Consequently it needs a full second of steady input signal to display a valid reading. Also, if the simulation is not occurring at real time speed (due to heavy CPU load), the frequency meter will take longer in real time to produce a reading.
- The counter timer is a purely digital component. To measure the frequency of a low level analogue signal you will need to place an ADC primitive with appropriate logic thresholds in front of the counter timer CLK pin. Also, due to the fact that analogue simulation is approximately 1000 times slower than digital simulation, the Counter Timer is not really suitable for determining the frequency of analogue oscillator circuits operating much above 10kHz. In such cases you should use the virtual oscilloscope (or a graph) to measure the signal period.

Using the Counter Mode

To count digital pulses:

1. Select the *Multimeter* icon, pick the COUNTER TIMER and place one on the schematic, wiring the CLK (clock) pin to the source signal.
2. If appropriate, wire the CE (clock enable) pin to a signal that will control when the count advances. If you don't require a clock enable function you can leave this pin floating.
3. If appropriate, wire the RST (reset) pin to a signal that will switch state to reset the count to zero. If you don't need to reset the counter from within the circuit you can leave this pin floating.
4. Point at the Counter Timer and press Ctrl-E to edit its settings.
5. Select *Counter Mode* and appropriate logic polarities for the CE and RST functions.
6. Close the dialogue and start the simulation in the usual way.

Notes:

- The count advances on positive edge transitions of the CLK when CE is active.
- The RST pin is edge triggered as opposed to level triggered. If you need to hold the count at zero, you can use the CE and RST pins in combination.
- The Counter Timer popup provides a manual reset button which will reset the count to zero at any point during the simulation run.

VIRTUAL TERMINAL

Overview

The VSM Virtual Terminal enables you to use the keyboard and screen of your PC to send and receive RS232 asynchronous serial data to and from a simulated microprocessor system. It is especially useful in debugging where you can use it to display debug/trace messages generated by the software which you are developing.



The Virtual Terminal is specified as follows:

- Fully bi-directional - received serial data is displayed as ASCII characters whilst key presses are transmitted as serial ASCII data.
- Simple two wire serial data interface: RXD for received data and TXD for transmitted data.
- Simple two wire hardware handshake interface: RTS for ready-to-send and CTS for clear-to-send.
- Baud rate from 300 to 57,600 baud.
- 7 or 8 data bits.
- Odd, even or no parity.
- 0, 1 or 2 stop bits.
- XON/XOFF software handshaking in addition to hardware handshaking.

Using the Virtual Terminal

To attach a teletype terminal to your simulated target system:

1. Select the *Multimeter* icon, pick the VIRTUAL TERMINAL and place one on the schematic.
2. Wire the RX and TX pins to the transmitting and receiving lines of the system under test. RX is an input and TX is an output.
3. If you the target system uses hardware handshaking, wire the RTS and CTS pins to the appropriate flow control lines. RTS is an output, signalling that the virtual terminal is ready to receive data, whilst CTS is an input which must be high (or floating) before the virtual terminal will transmit.
4. Edit the virtual terminal component to select the require baud rate, word length, parity and flow control settings.
5. Start the simulation in the usual way. The terminal will display incoming data as soon as it is received; to transmit characters into the system, make sure the terminal popup has the focus and then type the required text on the PC keyboard.
6. Once the simulation is started, further functionality is available from a context menu which may be displayed by clicking right on the popup window. This menu enables you to pause the display, and to copy and paste text to and from the clipboard.

Notes:

- The Virtual Terminal supports the ASCII control codes CR (0Dh), BS (0x08h) and BEL (0x07h). All other codes including LF (0x0A) are ignored.
- The Virtual Terminal is a purely digital model and as such does not require any particular voltage levels on its pins. The assumption is that it will wired to other digital models rather than to the output side of level buffering devices such as the MAX232. This is also partly why we do not provide models for the MAX232 or similar; the other reason being that routing the signals through such models would incur computationally expensive analogue simulation for absolutely no benefit.
- The RX and TX pins are active high. Thus the idling state is high, the start bit is low and the stop bit high. Data bits appear as logic high for '1' and logic low for '0'. This is directly compatible with the on board UARTs present in many microcontrollers, and also with external UARTs such as the 6850 and 8250.
- The RTS and CTS pins are also active high. If you wish to connect them to inverted control lines (i.e. RTS/CTS bar), you should add an **INVERT** property to the virtual terminal component e.g.

INVERT=RTS , CTS

This will re-define the RTS and CTS to be active low.

- You can pre-define text to be transmitted into your circuit at start up using the **TEXT** property. e.g.

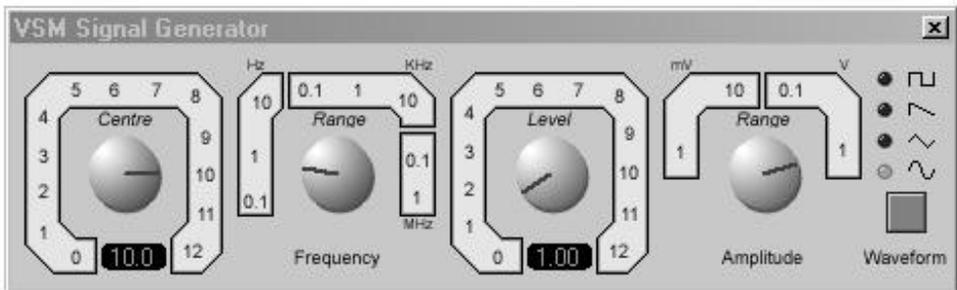
TEXT="Hello World"

will transmit the text "Hello World" into the circuit. The script text is transmitted one character time after start up, unless CTS is inactive.

SIGNAL GENERATOR

Overview

The VSM Signal Generator is provided as standard with all professional versions of Proteus .



The VSM signal generator models a simple audio functional generator with the following features:

- Square, saw-tooth, triangle and sine output waveforms.
- Output frequency range from 0-12MHz in 8 ranges
- Output amplitude from 0-12V in 4 ranges.
- Amplitude and frequency modulation inputs.

Using the Signal Generator

To set up a simple audio signal source

1. Select the *Multimeter* icon, pick the SIGNAL GENERATOR and place one on the schematic, wiring its outputs into the rest of the circuit. In most circumstances (i.e. unless the circuit you are driving requires a balance input source), you will want to

ground the -ve terminal of the generator. You can most easily achieve this using a ground terminal.

The amplitude and frequency modulation inputs can be left unconnected unless you are using these features.

2. Start an interactive simulation by pressing the play button on the *Animation Control Panel*. The signal generator popup window should appear.
3. Set the *frequency range* dial to a value suitable for your application. The range values indicate the frequency that is generated when the *centre* vernier control is set at 1.
4. Set the *amplitude range* dial to a value suitable for your application. The range values indicate the amplitude that is generated when the *level* vernier control is set at 1. The amplitude values represent peak output levels.
5. Push the *waveform* button until the LED next to the appropriate waveform icon is lit.

Using the AM & FM Modulation Inputs

The signal generator model supports both amplitude and frequency modulation of the output waveform. Both the amplitude and frequency inputs have the following features:

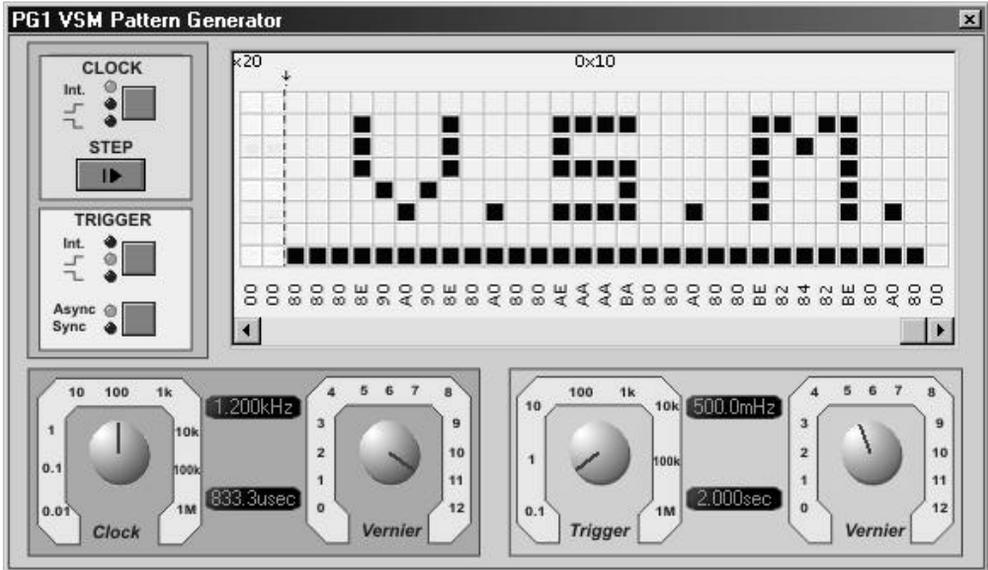
- The gain of the modulation input in terms of Hz/V or V/V is set by the *Frequency Range* and *Amplitude Range* controls respectively.
- The modulation input voltages are clamped at +/- 12V.
- The modulation inputs have infinite input impedance.
- The voltage at the modulation input is added to the setting of the appropriate vernier control before being multiplied by the range setting to determine the instantaneous output frequency of amplitude.

For example, if the frequency range is set at 1KHz, and the frequency vernier is set at 2.0, then a frequency modulation level of 2V will give an output frequency of 4kHz.

DIGITAL PATTERN GENERATOR

Overview

The VSM Pattern Generator is the digital equivalent of the analogue Signal Generator and is provided as standard with all professional versions of Proteus.



The VSM Pattern Generator allows for an 8-bit pattern of up to 1K bytes and supports the following main features:

- Will run in either graph based or interactive mode.
- Internal and External Clocking and Trigger modes.
- Vernier adjustment for both clock and trigger dials.
- Hexadecimal or Decimal grid display modes.
- Direct entry of specific values for greater accuracy.
- Loading and saving of pattern scripts.
- Manual specification of pattern period length.
- Single Step control allows you to advance the pattern incrementally.

- Tooltip Display allows you to see exactly where you are on the grid.
- Ability to externally hold the pattern in its current state.
- Block Editing commands on the grid for easier pattern configuration.

Using the Pattern Generator

Outputting a pattern with the Pattern Generator in an interactive simulation.

1. Select the *Multimeter* icon , pick the *PATTERN GENERATOR* and place one on the schematic, wiring its outputs up to the rest of the circuit.
2. Initialise an interactive simulation by pressing the pause button on the *Animation Control Panel*. The Pattern Generator window should appear.
3. Specify the pattern that you want to output by left clicking on grid squares to toggle their logic states.
4. Decide whether you are clocking internally or externally and set the clock mode button to reflect your choice.
5. If you are clocking internally adjust the clock dials to specify the desired clock frequency.
6. Decide whether you are triggering internally or externally and use the trigger buttons to set the mode accordingly. If you are triggering externally you need to think about whether you want to trigger asynchronously or synchronously with the clock.
7. If you are triggering internally adjust the trigger dials to specify the desired trigger frequency.
8. Press the play button on the *Animation Control Panel* to output the pattern.
9. To advance the pattern by a single clock cycle press the suspend button on the *Animation Control Panel* and then the step button to the left of the grid.



See *Rotary Dials* on page 51 for details of how to adjust the rotary controls featured in the Pattern Generator.



See Trigger Modes, below, for details of the different trigger modes available with the Pattern Generator.



See *Additional Functionality*, below, for more information on configuring and using the Pattern Generator.

Outputting a pattern with the Pattern Generator in a graph based simulation.

1. Set up the circuit in the normal way.

2. Insert probes on the schematic at points of interest and add those probes to the graph.
3. Right click and then left click on the Pattern Generator on the schematic to get the *Edit Component* dialogue form.
4. Configure the Trigger and Clock options.
5. Load the desired pattern file into the Pattern Generator Script field.
6. Exit the Pattern Generator Dialogue form and hit the space bar to run the simulation.

The Pattern Generator Component Pins

Data Output Pins (Tri-State Output)

The Pattern Generator can output either on a bus and/or on individual pins.

Clockout Pin (Output)

When the Pattern Generator is clocking internally you can configure this pin to mirror the internal clock pulses. This is set as a property and can be changed via the Edit Component dialogue form. By default this option is disabled as it incurs a performance hit, particularly for high frequency clocks.

Cascade Pin (Output)

The Cascade Pin is driven high when the first bit of the pattern is being driven and remains high until the next bit is driven (one clock cycle later). This means it is high for the first clock cycle on starting the simulation and again for the first clock cycle subsequent to a reset.

Trigger Pin (Input)

This input pin is used to feed an external trigger pulse into the Pattern Generator. There are four external trigger modes which are discussed in detail under *Reset Modes*.

Clock-In Pin (Input)

This input pin is used to apply external timing to the Pattern Generator. There are two external clocking modes which are discussed in greater depth under *Clocking Modes*.

Hold Pin (Input)

This pin, when driven high can be used to pause the Pattern Generator. The pattern will remain at the point it was at until the hold pin is released. For internal clocking and/or trigger the timing will resume relative to the point at which it was paused. For example, on a 1Hz internal clock, if the pattern is paused at time 3.6 seconds and resumed at 5.2 seconds then the next falling clock edge will be at 5.6 seconds.

Output Enable Pin (Input)

This pin must be driven high in order to enable the output pins. If this pin is not high then the Pattern Generator, while still running the specified pattern, will not drive the pattern onto the output pins.

Clocking Modes

Internal Clocking

Internal clocking is negative edge triggered. That is, the clock pulse will be low-high-low per clock cycle.

Internal clocking can be specified either prior to simulation via the Edit Component dialogue form for the Pattern Generator or during a pause in the simulation via the clock mode button.

The *Clockout* pin, when enabled, mirrors the internal clock. By default, this pin is disabled as it will cause some performance loss (particularly with high frequency clocks) but it can be enabled via the Edit Component dialogue form for the Pattern Generator.

External Clocking

There are two external clocking modes – negative edge (low-high-low) and positive edge (high-low-high).

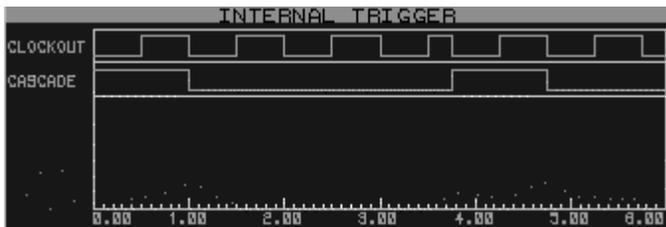
To clock externally wire the clock pulse to the *Clockin* pin and select one of the two external clocking modes.

As with Internal clocking you can change the external clocking mode either by editing the component prior to simulation or via the clock mode button during a pause in the simulation.

Trigger Modes

Internal Trigger

The Internal Trigger Mode in the Pattern Generator triggers the pattern at specified intervals. If the clocking is internal the clock pulse is reset at this point. This behaviour is shown graphically below.

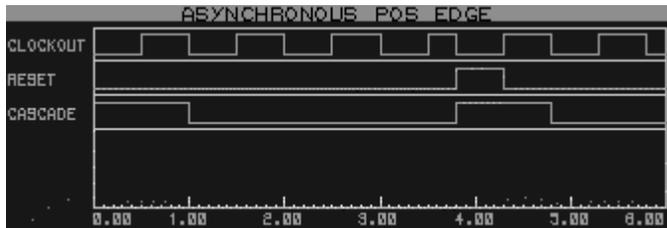


We are clocking internally at 1Hz and the internal trigger time is set to 3.75sec. The Cascade pin is high while the first bit of the pattern is being driven on the pins and low at all other times.

Note that at trigger time the internal clock is asynchronously reset. The first bit of the pattern is driven onto the output pins (as evidenced by the *Cascade Pin* being driven high).

External Asynchronous Positive Edge Trigger

The trigger is specified via a positive edge transition on the *Trigger Pin*. The Trigger actions immediately and the next clock edge will be a low-high transition at time bitclock/2 subsequent to the time of the reset as shown below.



We are clocking internally at 1Hz and the trigger pin goes high at time 3.75sec. Immediately on the positive edge of the trigger pin the clock is reset and the first bit of the pattern is driven onto the pins.

External Synchronous Positive Edge Trigger

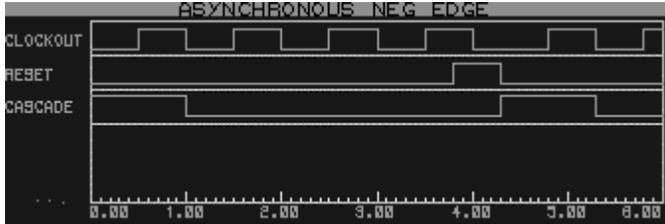
The trigger is specified via a positive edge transition on the *Trigger Pin*. The trigger is latched and will action synchronously with the next falling clock edge as shown below.



We are clocking internally at 1Hz. Note that the clock is unaffected by the trigger and that the trigger actions on the falling clock edge subsequent to the positive edge pulse.

External Asynchronous Negative Edge Trigger

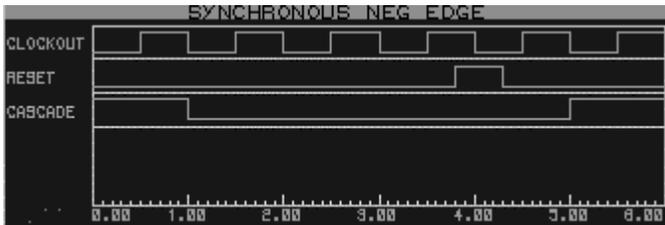
The trigger is specified via a negative edge transition on the *Trigger Pin*. The trigger actions immediately and the first bit of the pattern is driven onto the output pins.



We are clocking internally at 1Hz. We can see that clock resets on the negative edge of the trigger pulse and that the first bit of the pattern is driven at that time.

External Synchronous Negative Edge Trigger

The trigger is specified via a negative edge transition on the *Trigger Pin*. The trigger is latched and actions synchronously with the next falling edge clock transition as shown below.



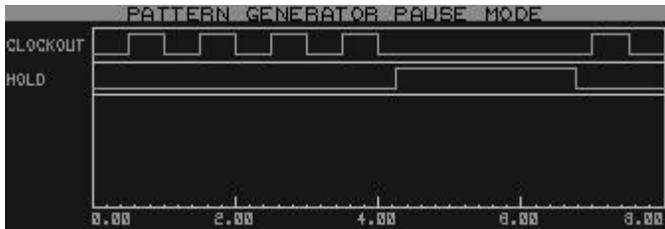
We are clocking internally at 1Hz. Note that the trigger takes place on the falling edge of the trigger pulse and that the pattern does not reset until the falling edge of the clock pulse subsequent to the trigger action.

External Hold

Holding the pattern in its current state.

If you want to hold the pattern for a period of time you can do so by arranging for the *hold* pin to be high during the period in which you want to pause. Releasing the hold pin will restart the pattern synchronously if you are using an internal clock. That is, should the hold

pin go high halfway through a clock cycle, then when you release the hold pin the next bit will be driven onto the output pins half a clock cycle later.



When the hold pin goes high the internal clock is paused. On release of the hold pin the clock resumes relative to the point in the clock cycle at which it was paused.

Additional Functionality

Loading and saving a pattern script.

Pattern Scripts can be loaded or saved by right clicking over the grid and selecting the relevant option from the popup menu. This is particularly useful if you want to use a particular pattern in several designs.

The pattern scripts are plain text and are simply a comma-separated list of bytes where each byte value represents a column on the grid. Any line beginning with a semi-colon is taken to be a comment line and is ignored by the parser. By default the byte format is hexadecimal though if you are creating your own script you can enter values in decimal, binary or hexadecimal.

Setting specific values for dials.

You can specify an exact value for both the bit and the trigger frequency by double clicking the mouse on the appropriate dials. This launches a floating edit box into which you can enter the value. By default the value entered is assumed to be a frequency but you can specify a value in seconds or fractions thereof by appending a suitable suffix to the value (sec, ms, etc.). Additionally, should you wish the trigger to be an exact multiple of the bit clock you can append the suffix 'bits' to the multiple desired (e.g. 5bits).

To confirm input press the *Enter* button or to cancel press the *Escape* key or click elsewhere on the Pattern Generator window.

These values can also be specified prior to simulation via the appropriate properties in the *Edit Component* dialogue form.

Setting specific values for the pattern grid.

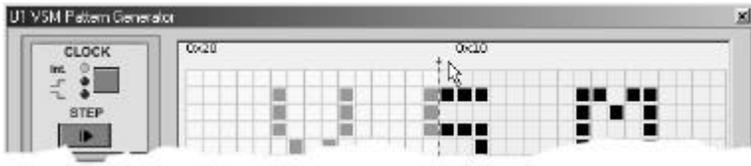
You can specify a specific value for any column on the grid by left clicking the mouse on the text displaying the current value for that column. A floating edit box is launched into which you can enter the desired value. You can specify the value in decimal (e.g. 135), hexadecimal (e.g. 0xA7) or binary (e.g. 0b10110101).

To confirm input press the *Enter* button or to cancel press the *Escape* key or click elsewhere on the Pattern Generator window.

For convenience you can set a column via the CTRL+1 keyboard shortcut and clear a column via the CTRL+SHIFT+1 keyboard shortcut with the mouse over the appropriate column.

Specifying the period length of the pattern manually.

You can specify a manual period by left clicking the mouse just above the grid itself at the column you want the pattern to finish on. To deselect the period right click the mouse once in the same area. This is shown below.



The mouse is pointing at the area where period selection can be specified. We can see that the period bar is indicating the pattern termination point and that everything to the left of the selection is faded out to indicate that the pattern will stop at the period bar.

Single stepping to advance the pattern.

The step button can be used to advance the simulation in time periods equal to that of the bitclock specified either internally or from an external clock. The simulation will run until the next clock cycle is completed and then suspend again.

Alternating the grid display mode.

The grid display can be switched between hexadecimal and decimal display modes. This can be done either by right clicking on the grid and selecting the desired option from the menu or alternatively by using the CTRL+X (Hexadecimal display) and CTRL+D (Decimal display) shortcut keys.

Specifying Output.

Edit the Schematic part for the Pattern Generator to produce the *Edit Component* dialogue form. The property at the bottom allows you to configure whether you want to output the pattern on both bus and pins, pins only or bus only.

Tooltip Display.

You can enable a tooltip which will follow the mouse indicating the current row and column information. This can be toggled on and off either via the context menu on the right mouse button or through the shortcut key CTRL+Q. Note that tooltip mode is disabled during a Block set or clear.

Block Editing.

You can use the *Block Set* and *Block Clear* commands to help you quickly configure the grid to the desired pattern. These are accessible via the context menu on the right mouse button or via their shortcut keys (CTRL+S for Block set and CTRL+C for Block clear). Note that the Block Editing commands are disabled when in tooltip mode.

VSM USER INTERFACE ELEMENTS

Rotary Dials

The VSM Virtual Instruments use mouse operated rotary dials (knobs) for adjusting some parameters. The procedure for adjusting these is as follows:

To set a rotary dial:

1. Point somewhere inside the dial.
2. Press the left mouse button and hold it down.
3. Move the mouse pointer away from the dial and then around the centre of the dial, tracing out a circular arc to rotate the dial to the required setting.
4. The dial will track the angle subtended by the mouse pointer from its centre. The further away you move the mouse, the finer the degree of control that you will have.
5. Release the mouse button to fix the new position of the dial.

WORKING WITH MICROPROCESSORS

INTRODUCTION

The combination of mixed mode simulation and circuit animation becomes most potent when used in the context of micro-controller based systems. Many such systems involve a user interface, or at least complex sequences of external events that cannot easily be simulated in a non-interactive environment, and PROTEUS VSM was created primarily to address this problem. Consequently, a substantial chunk of its functionality centres around micro-processor development.

In particular, the editing and compiling of source code, is integrated into the design environment so that you can edit source code and observe the effects of your changes with maximum ease. A source file can be brought up for editing with two keystrokes and the simulation re-run from there with just two more.

SOURCE CODE CONTROL SYSTEM

Overview

The source code control system provides two major functions:

- To register source code files within ISIS so that they can be brought up for editing without manually switching to another application.
- To define the rules for compiling source code into object code. Once set up, these rules are carried out every time a simulation is performed so that the object code is kept up to date.

Note that it is not necessary to use the source code control system in order to simulate microprocessor based designs. Indeed, if your chosen assembler or compiler has its own IDE, you may prefer to work directly in that, and only switch to Proteus when you have produced an executable program. If you are planning to work this way, then see the following section *Using a 3rd Party IDE* on page 56.

Attaching Source Code to the Design

To add a source file to the design:

1. Select the *Add/Remove Source Files* command from the *Source* menu.
2. Select the *Code Generation Tool* for the source file. If you are planning to use a new assembler or compiler for the first time, you will need register it using the *Define Code Generation Tools* command.
3. Click the *New* button and select or enter a name for the source code file with the file selector. You can enter a filename textually if it does not yet exist.
4. Enter the flags required specifically for processing this source file in the *Flags* field. Flags required every time a particular tool is used can be entered when the tool is registered.
5. Click OK to add the source file to the design.

Don't forget to edit the micro-processor and assign the name of the object code file (usually a HEX file) to its **PROGRAM** property. ISIS can't do this automatically as you might have more than one processor on the diagram!

Working on your Source Code

To edit a source file:

1. Press ALT-S
1. Press the ordinal number of the source file on the source menu.

If you prefer to use a more advanced text editor, see page 56.

To switch back to ISIS, build the object code and run the simulation:

1. From the text editor, press ALT-TAB to switch back to ISIS.
2. Press F12 to execute, or CTRL-12 to start debugging.

Either way, ISIS will instruct the text-editor to save its files, examine the time-stamps of the source code and object code files, and invoking the appropriate code generation tools to build the object code.

To rebuild all the object code:

1. Select the *Build All* command from the source menu.

ISIS will execute all the code generation tools required to build the object code, irrespective of the time/date stamps of the object code files. The command line output from the tools will be

displayed in a window. This provides an excellent way to check that everything has built without errors or warnings.

Installing 3rd Party Code Generation Tools

A number of shareware assemblers and compilers can be installed into the TOOLS directory from the system CD, and these will be set up automatically as code generation tools by the PROTEUS setup program. However, if you want to use other tools, you will need to use the *Define Code Generation Tools* command on the *Source* menu.

To register a new code generation tool:

1. Select the *Define Code Generation Tools* command from the *Source* menu.
2. Click *New* and use the file selector to locate the executable file for the tool. You can also register batch files as code generation tools.
3. Enter extensions for the source and object code files. These determine the file types that ISIS will look for when deciding whether to execute the tool for a particular source file, or not. If you check *Always Build* then the tool is always invoked, and an object code extension is not required.
4. Specify a proforma command line for the tool. Use %1 to represent the source code file and %2 for the object code file. You can also use %\$ for the path to the PROTEUS directory and %~ for the directory in which the DSN file is located.

This is a good place to put command line flags that are needed to make tools run quietly (i.e. without pausing for user input), and to specify paths to include directories of processor specific header files etc.

If you want to use the source level debugging features of PROTEUS VSM, you will need a *Debug Data Extractor* for your assembler or compiler. This is a small command line program, which extracts object code/source line cross reference information from the list file produced by the assembler or compiler. We hope to support all popular assemblers and compilers, so check our website for the latest info.

If you have a DDX program, enter the extension for the list file or symbolic debug file that is produced by the code generation tool, and click *Browse* to select the path and filename of the DDX program.

Using a MAKE Program

In some cases, the simple build rules implemented by ISIS may be insufficient to cater for your application – especially if you project involves multiple source and object code files, and a linking phase. In such cases, you will need to use an external ‘MAKE’ program, and you can proceed as follows:

To set up a project with an external make program:

1. Install your make program (usually supplied with the assembler/compiler environment) as a code generation tool. Set the source extension to MAK and check *Always Build*. For a typical make program, set the command line proforma as:

```
-f %1
```
2. Use the *Add/Remove Source Files* command to add the makefile (e.g. MYPROJECT.MAK) to the design.
3. Also add the source code files, but select the *Code Generation Tool* as <NONE>

Each time the project is built, ISIS will run the external MAKE program, with the project's makefile as a parameter. It is then up to the make program to decide what code generation tools are run. A good make program will provide enormous flexibility.

Using a 3rd Party Source Code Editor

PROTEUS VSM is supplied with a simple text editor – SRCEDIT which you can use to edit source code files. SRCEDIT is essentially a modified version of NOTEPAD, which can open multiple source files, and can respond to a DDE request to save any modified buffers.

If you have a more advanced text editor, such as *UltraEdit*, you can instruct ISIS to use this instead. Note that IDE type environments probably won't respond to DDE commands and so are unsuitable for integration in this way. However, you could always ask the vendor to add this support – it's not hard.

To set up an alternative source code editor:

1. Select the *Setup External Text Editor* command from the *Source* menu.
2. Click the *Browse* button and use the file selector to locate the executable file of your text editor.
3. ISIS instructs the text editor to open and save files using a DDE protocol. Refer to the text editor's documentation or supplier for details of the command syntax. If unsure of the service name, try the name of the product – e.g. ULTRAEDIT.

USING A 3RD PARTY IDE

Most professional compilers and assemblers come with their own integrated development environment or IDE. Examples include IAR's Embedded Workbench, Keil's uVision 2, Microchip's MP-LAB and Atmel's AVR studio. If you are developing your code with one of these tools, you may find it easier to carry out the editing and compilation steps within the IDE and then switch to Proteus VSM only when you have produced an executable image (e.g. a HEX or COD file) and are ready to simulate it.

Proteus VSM supports two ways of working with an external IDE

- Using Proteus as an external debugger - control of the debugging session is carried out from within ISIS, much as you would for a normal CAD simulation.
- Using Proteus as a plug-in simulator. - control of the debugging session is carried out from within the IDE's debugger. Proteus acts as a kind of virtual In Circuit Emulator, communicating with the IDE over TCP/IP. In this mode, you can run your IDE's debugger on one PC and the Proteus simulation on the other.

Using Proteus VSM as an External Debugger

To Proteus as an external debugger requires that the symbolic debug format produced by your compiler is supported by one of the *loaders* available in Proteus. A loader extracts both the addresses of each source line in the high level language program, and - where possible - the locations of the program variables.

Common debug formats are COD (used in the PIC world), UBROF - used by all IAR's compilers, OMF (used in 8051 circles) and COFF. We also provide loaders for other proprietary formats such as the list files produced Crownhill PICBasic. See our website under "*3rd Party Compilers*" for up to date information.

Assuming that there is a loader for your chosen compiler, the procedure for loading the program into the simulated micro-processor is extremely simple:

To load a program produced in an external IDE

1. Ensure that the program is compiled and linked with no errors.
2. Edit the **PROGRAM** property of the CPU model to be the name of the executable image file produced by the compiler or linker, e.g. MYPROG.COD.

Do not enter the name of the source file - Proteus VSM does not simulate 'C' or 'ASM' files; the CPU models load and execute binary machine code.

3. Press the PLAY button on the animation control panel to begin real-time simulation or press the STEP button to run up to the first source level instruction. In the latter case, the *Source Window* will appear and you can commence stepwise debugging of your code.

Using Proteus VSM as a Virtual In-Circuit Emulator (ICE)

At the time of writing, only Keil uVision 2 for the 8051 supports the virtual ICE approach although we are now very close to having a similar solution in place for IAR's Embedded Workbench. We are also working with Microchip and Atmel amongst others to integrate Proteus VSM with their tools. It is a fast moving area and we would recommend you check

our website under "*3rd Party Compilers*" for the latest information and documentation. Instructions for using setting up *uVision2* to work with Proteus will also be found there.

POPUP WINDOWS

Most micro-processor models written for PROTEUS VSM will create a number of popup windows which can be displayed and hidden from the *Debug* menu. These windows come in four major types:

- Status Windows – a processor model will generally use one of these to display its register values.
- Memory Windows – typically there will be one of these for each memory space in the processor architecture. Memory devices (RAMs and ROMs) also create these windows.
- Source Code Windows – one of these will be created for each processor on the schematic.
- Variable Windows – one of these will be created for each processor on the schematic, provided that the program loader supports program variable display.

To display a popup window:

1. Start debugging mode by pressing CTRL-F12, or if it is already running, click the *Pause* button on the *Animation Control Panel*.
2. Press ALT-D and then the ordinal number of the required window on the *Debug* menu.

These window types can only be displayed when the simulation is paused and are hidden automatically when it is running to give you better access to the active components on the schematic itself. When the simulation is paused (either manually, or due to a breakpoint) the windows that were showing will re-appear.

The debug windows all have right button context menus – if you point at the window and click the right mouse button, a menu will appear from which you can control the appearance and formatting of the data within the window.

The positions and visibility of the debug windows are saved automatically to a PWI file with the same filename as the current design. The PWI file also contains the locations of any breakpoints that have been set, and the contents of the watch window.

SOURCE LEVEL DEBUGGING WITHIN PROTEUS VSM

Overview

PROTEUS VSM supports source level debugging through the use of *debug loaders* for supported assemblers and compilers. The current set of debug loaders are contained within the system file LOADERS.DLL, and the number tools that are supported is increasing quite rapidly. The latest information is available from the support area of our website under "^{3rd} Party Compilers".

Assuming that you have used a supported assembler or compiler, PROTEUS VSM will create a source code window for each source file used in the project, and these windows will appear on the *Debug* menu.

The Source Code Window

The source code window has a number of features:

- A blue bar represents the current line number, at which a breakpoint will be set if you press F9 (*Toggle Breakpoint*), and to which the program will execute if you press CTRL-F10 (*Step To*).
- A red chevron indicates the current location of the processor's program counter.
- A red circuit marks a line on which a breakpoint has been set.

The right hand context menu provides a number of further options including: *Goto Line*, *Goto Address*, *Find Text* and toggles for displaying line numbers, addresses and object code bytes.

Single Stepping

A number of options for single stepping are provided, all available from the toolbar on the source window itself or from the *Debug* menu.

- Step Over – advances by one line, unless the instruction is a sub-routine call, in which case the entire subroutine is executed.
- Step Into – executes one source code instruction. If no source window is active, it executes one machine code instruction. These are usually the same thing anyway unless you are debugging in a high level language.
- Step Out – executes until the current sub-routine returns.
- Step To – executes until the program arrives at the current line. This option is only available when a source code window is active.

Note that apart from *Step To*, the single stepping commands will work without a source code window. It is possible – although not so easy – to debug code generated by a tool for which there is no loader support.

Using Breakpoints

Breakpoints provide a very powerful way to investigate problems in the software or software/hardware interaction in a design. Typically, you will set a breakpoint at the start of a subroutine that is causing trouble, start the simulation running, and then interact with the design such that the program flow arrives at the breakpoint. At this point, the simulation will be suspended. Thereafter you can single step the program code, observing register values, memory locations and other conditions in the circuit as you go. Turning on the *Show Logic State of Pins* effect can also be very instructive.

When a source code window is active, breakpoints can be set or cleared on the current line by pressing F9. You can only set a breakpoint on a line which has object code.

If the source code is changed, PROTEUS VSM will endeavour to re-located the breakpoints based on sub-routine addresses in the file, and by pattern matching the object code bytes. Obviously, if you change the code radically this can go awry but generally it works very well and you shouldn't need to think about it.

Variables Window

Most of the loaders provided with Proteus VSM are able to extract the location of the program variables as well as the source line number addresses. Whenever this is possible, Proteus displays a variables window as well as a source window.

There are a number of points to note about the variables window:

- When single stepping, any variables that change value are highlighted.
- The format in which each variable is displayed can be adjusted by clicking right on it and choosing an alternative format from the context menu.
- Although the variables window is hidden whilst the program is running, you can drag & drop variables to the watch window where they will remain visible.
- Depending on how the compiler re-uses memory, local variables which out of scope may display invalid values.

Multi CPU Debugging

Proteus VSM has no problem simulating designs containing more than one CPU on the schematic. Each CPU will produce its own set of popup windows including source code and variables, and these will be grouped together under sub-menus on the *Debug* menu.

When single stepping code, the processor whose source window has the focus will be treated as the master, and the other CPUs will free run, suspending their execution at whatever time-point the master CPU has completed an instruction. If you move the focus away from the source window, then the master CPU remains the one whose source window last had the focus; the master is changed by clicking on another CPU's source window.

THE WATCH WINDOW

Adding items to the Watch Window

Whereas the memory and register windows belonging to processor models are only displayed when the simulation is paused, the *Watch Window* provides a means to display values that are updated in real time. It also provides a means to assign names to individual memory locations which can be more manageable than trying to find them in a memory window.

To add an item to the Watch Window:

1. Press CTRL-F12 to start debugging, or pause the simulation if it is already running.
2. Display the memory window containing the item to be watched, and the watch window itself using the numbered options on the *Debug* menu.
3. Mark a memory location or range of memory locations using the left mouse button. The selected range should appear in inverted colours.
4. Drag the selected item(s) from the memory window to the watch window.

You can also add items to the *Watch Window* by using the *Add Item By Name* and *Add Item By Address* commands on its right mouse button context menu.

Modifying Items in the Watch Window

Having got an item or items in into the window, you can now select an item with the left mouse button and then:

- Rename it by pressing CTRL-R or F2.
- Change the data size to any of the options on the right mouse button context menu. For data sizes that comprise several bytes (e.g. 16 or 32 bit words, or strings) the second and subsequent bytes are assumed to follow on from the item address. Consequently, to display a multi-byte word or string you only need to drag in the first byte from the memory window.
- Change the number format to binary, octal, decimal or hex.

Watchpoint Conditions

The *Watch Window* provides a facility to suspend the simulation when a particular condition arises in the values of the items. At the simplest level, this can be when the value of a particular item changes but you can also specify more complex conditions if necessary.

To specify a Watchpoint Condition

1. Press CTRL-F12 to start debugging, or pause the simulation if it is already running.
2. Display the watch window itself using the option on the *Debug* menu.
3. Add the items on which the watchpoint condition will depend to the watch window as described in the previous section.
4. Select the *Watchpoint Condition* command from the right mouse button context menu.
5. Specify a *Global Break Condition*. This determines whether the simulation suspends when *any* item expression is true, or only when *all* item expressions are true.
6. Specify one or more *Item Break Expressions*. An item break expression consists of the watch-item on which it operates, a mask value (allowing you to activate the condition for particular bits of a register, for example), a conditional operator, and value.

Activating a watchpoint condition will impose a modest overhead on the simulator as ProSPICE must re-test the condition after every timestep in the simulation (i.e. at or above the machine cycle rate in a micro-controller application). However, the facility can be a god-send when looking for obscure bugs and the fact that the simulation may take a few extra minutes to reach the point of interest is not normally an issue.

BREAKPOINT TRIGGER OBJECTS

Overview

A number of component objects are provided which trigger a suspension of the simulation when a particular circuit condition arises. These are especially useful when combined with the single stepping facilities, since the circuit can be simulated normally until a particular condition arises, and then single stepped in order to see exactly what happens next.

The breakpoint trigger devices may be found in the REALTIME device library.

Voltage Breakpoint Trigger – RTVBREAK

This device is available in 1 and 2 pinned forms, and triggers a breakpoint when the voltage at its single pin, or the voltage between its two pins is greater than the specified value. You can couple this device to an arbitrary controlled voltage source (AVCS) to trigger a breakpoint on complex, formula based conditions involving multiple voltages, currents etc.

Once the trigger voltage has been exceeded, the device does not re-trigger until the voltage has dropped below the trigger threshold and risen again.

Current Breakpoint Trigger – RTIBREAK

This device has two pins and triggers a breakpoint when the current flowing through it is greater than the specified value.

Once the trigger current has been exceeded, the device does not re-trigger until the current has dropped below the trigger threshold and risen again.

Digital Breakpoint Trigger – RTDBREAK

This device is available with various numbers of pins, and you can make other sizes yourself if you need to. It triggers a breakpoint when the binary value at its inputs equals the value assigned to the component. For example, specifying the value of an RTDBREAK_8

0x80

will cause it to trigger when D7 is high and D0-D6 are low.

Once the trigger condition has arisen, the device does not re-trigger until the voltage a different input value has been seen.

GRAPH BASED SIMULATION

INTRODUCTION

Although interactive simulation has many advantages, there are still some situations in which it is advantageous to capture the entire simulation run to a graph and study the results at your leisure. In particular, it is possible to zoom in on particular events within the simulation run and take detailed measurements. Graph based simulation is also the only way to perform analyses which do not take place in real time at all, such as small-signal AC analysis, Noise Analysis and swept parameter measurements.

Graph based simulation is only available if you have purchased the *Advanced Simulation Features* option for your Proteus system.

SETTING UP A GRAPH BASED SIMULATION

Overview

Graph based simulation involves five main stages. These are summarized below, with detailed explanations of each stage being given in the subsequent sections:

- Entering the circuit to be simulated.
- Placing signal generators at points that require stimulus and test probes at points that you wish to inspect.
- Placing a graph corresponding with the type of the analysis you wish to perform - e.g. a frequency graph to display a frequency analysis.
- Adding the signal generators and test probes to the graph in order to display the data they generate/record.
- Setting up the simulation parameters (such as the run time) and executing the simulation.

Entering The Circuit

Entering the circuit you want to simulate is exactly the same as entering any other design in to ISIS; the techniques for this are covered in detail in the ISIS manual itself.

Placing Probes and Generators

The second stage of the simulation process is to set up signal generators at points that require stimuli and probes at points of interest. Setting up signal generators and probes is

extremely easy as they are treated like other ISIS objects such as components, terminals, or junction dots. All that is required is to select the appropriate icon, pick the type of generator or probe object from the selector, and place it on the schematic where you want it. This can be either by placing it directly on to an existing wire, or by placing it like a component and then wiring to it later. Defining the signal produced by a generator is then just a matter of editing the object and entering the required settings on its dialogue form.

At this stage you can also isolate a section of the design so that only certain components are involved in the simulation. This is achieved by checking the *Isolate Before* and *Isolate After* options in the probes and generators. The use of probes and generators in this way provides not only for faster simulations, but also means that errors do not creep in from removed wires being forgotten and never replaced.

Placing Graphs

The third stage of the simulation process is to define what analysis type or types you want performed. Analysis types include analogue and digital transient analyses, frequency analyses, sweep analyses, etc. Within ISIS, defining an analysis type is synonymous with placing a graph object of the required analysis type. Again, as graphs are just like most other objects within ISIS, placing one is simply a case of selecting the correct icon, selecting the required graph type, and placing the graph on the design, alongside the circuit. Not only does this allow you to view several types of analysis simultaneously, it fits very well with the 'drag-and-drop' methodology adopted throughout ISIS and has the added benefit that you can view (and generate hard-copy output with) the graphs alongside the circuit that generated them.

Adding Traces To Graphs

Having placed one or more graph objects, you must now specify which probe/generator data you want to see on which graphs. Each graph displays a number of *traces*. The data for a trace is generally derived from a single probe or generator. However, ISIS provides for a trace to display the data from up to four separate probes/generators combined by a mathematical *Trace Expression*. For example, a trace might be set up to display the product of the data from a voltage probe and current probe (both monitoring the same point) so effectively displaying the power at the monitored point.

Specifying the traces to be displayed on a graph can be done in a number of ways: you can tag-and-drag a probe onto a graph or you can tag several probes/generators and add them all to a graph in a single operation, or for traces requiring trace expressions, you can must use a dialogue form to select the probes and specify the expression.

The Simulation Process

Graph based simulations is *Demand Driven*. This means that the emphasis is on setting up generators, probes and graphs in order to determine what you want to measure, rather than on setting up the simulator and then running some kind of post processor in order to examine the results. Any parameters that are specific to a given simulation run are specified by either editing the explicit properties of the graph itself (e.g. the start and stop times for the simulation run) or by assigning additional properties to the graph (e.g. for a digital simulation, you can pass a 'randomise time delays' property to the simulator).

So what happens once you have initiated a simulation? In brief, the action proceeds through the following steps:

- Netlist generation - this is the usual process of tracing wires from pin to pin, and producing a list of components and a list of pin to pin connection groups, or *nets*. In addition, any components in the design that are to be simulated by *model files* are replaced by the components contained in these files.
- Partitioning - ISIS then looks at the points you where you have placed probes and traces back from these to where you have injected signals. This analysis results in the establishment of one or more partitions that may need to be simulated and the order in which they must be simulated. As each partition is simulated, the results are stored in a new *partition file*.
- Results Processing - Finally, ISIS accesses the partition files to build up the various traces on the graph. The graph is then re-displayed and can be maximized for measurement taking and so forth.

If errors occur during any of these stages, then the details are written to the simulation log file. Some errors are fatal, and others are just warnings. In the case of fatal errors, the log file is displayed for immediate viewing. If only warnings have occurred then the graph is re-displayed and you are offered the choice of viewing the log if you want to. Most errors relate either to badly drawn circuits (which cannot, for one reason or another, be mathematically solved), or to the omission or incorrect linking of model files.

GRAPH OBJECTS

Overview

A *graph* is an object that can be placed on to the design. The purpose of a graph is to control a particular simulation and display the results of that simulation. The type of analysis type performed by the simulation is determined by the type of the graph placed. The part of the design that is simulated and the data that is displayed on the graph is determined by the probe and generator objects that have been added to the graph.

The Current Graph

All graph-specific commands are on the *Graph* menu. The lower portion of this menu also contains a list of all the graphs in the design, with the *current* graph being marked by a small marker at the left of its name. The *current* graph is the last graph that was simulated or acted upon by a command.

Any command from the *Graph* menu can be invoked for a specific graph by pointing at the graph with the mouse and using the command's keyboard short-cut (shown on the menu to the right of the command). If the mouse pointer is not pointing at a graph, or if you select the command directly from the menu, the command is invoked for the *current* graph.

Placing Graphs

To place a graph:

1. Obtain a list of graph types by selecting the the *Graph* icon. The list of graph types is displayed in the selector on the right hand side of the display.
2. Select the type of graph you wish to place from the *Object Selector*.
3. Place the mouse in the *Editing Window* at the point you wish the top left corner of the graph to appear. Press down the left mouse button and drag the out a rectangle for the size of the graph you wish to place, then release the mouse button.

Editing Graphs

All graphs can be moved, resized or edited to change their properties using the standard ISIS editing techniques.

The properties of a graph are changed via its *Edit...* dialogue form invoked as for any object in ISIS by either first tagging it and then clicking the left mouse button on it, or by pointing at it with mouse and pressing CTRL+'E' on the keyboard.

Adding Traces To A Graph

Each graph displays one or more *traces*. Each trace normally displays the data associated with a single generator or probe. However, for analogue and mixed graph types it is possible to set up a single trace to display the data from between one and four probes combined by a mathematical formula which we call a *trace expression*.

Each trace has a label that is displayed alongside the y-axis to which the trace is assigned. Some graph types have only one y-axis and there is no option to assign the trace to a particular y-axis. By default, the name of a new trace is the same as the name of the probe (or the first probe in the trace's expression) - this can be changed by editing the trace.

Traces can be defined in one of three ways:

- Dragging and dropping an individual generator or probe on to the relevant graph.
- Tagging a selection of probes and using the *Quick Add* feature of the *Add Trace* command.
- Using the *Add Trace* command's dialogue form.

The first two methods are quick to use but limit you to adding a new trace for each generator or probe assigned to the graph. The third method is somewhat more involved but gives greater control over the type of trace added to the graph. In particular, traces that require *trace expressions* must be added to the graph via the *Add Trace* command's dialogue form.

To drag-and-drop a probe or generator on to a graph:

1. Tag the generator or probe you wish to add to a graph using the right mouse button.
2. Click and hold down the left mouse button on the generator or probe and drag the probe over to the graph and release the mouse button.

A new trace is created and added to the graph; the new trace displays the data associated with the individual probe/generator. Note that any existing traces may be shrunk in order to accommodate the new trace.

For graphs with two y-axes, releasing the probe or generator on left or right half of the graph assigns the new trace to the respective axis. Furthermore, for the mixed analogue and digital transient analysis graph type, releasing the generator or probe over existing digital or analogue traces creates a new trace of the respective type (the first probe or generator released on a mixed graph type is always creates an analogue trace).

To quick-add several generators or probes to a graph:

1. Ensure the graph you wish to add the generators or probes is the *current* graph. The current graph is the graph whose title is shown selected on the *Graph* menu.
2. Tag each generator or probe you want to add to the graph.
3. Select the *Add Trace* command on the *Graph* menu. As a result of the tagged probes and generators, the command first displays a *Quick add tagged probes?* prompt.
4. Select the *Yes* button to add the tagged generators and probes to the current graph.

A new trace is created for each tagged generator or probe and is added to the graph in alphabetical order; each new trace displays the data associated with its associated individual generator or probe. Traces are always assigned to the left y-axis for graph types that support two axes.

The Add Trace Command Dialogue Form

Assuming you are not performing *Quick Add*, the *Add Trace* command will display an *Add Trace* dialogue form (each graph type has slight variations). This form allows you to select the name of the new trace, its type (analogue, digital, etc.), the y-axis (left or right) to which it is to be added, up to four generator or probes whose data it is to use, and an expression that combines the data of the selected generators and probes.

To add a trace to a graph using the *Add Trace* command

1. Invoke the *Add Trace* command on the *Graph* menu against the graph to which you want to a new trace.

If there are any tagged probes or generators, a *Quick add tagged probes?* prompt will be presented. Respond by selecting the *No* button.

The *Add Transient Trace* dialogue form is displayed.

2. Select the type of trace you wish to add to the graph by selecting the appropriate *Trace Type* button. Only those trace types applicable to the graph are enabled.
3. Select the y-axis to which the new trace is to be added. Only those axes applicable to the graph and trace type selected are enabled.
4. Select one of the *Selected Probes* buttons *P1* through to *P4*, and then select a probe or generator from the *Probes* list box to be assigned to the selected trace probe. The name of the selected generator or probe will appear alongside the *Selected Probes* button and the *Selected Probe* name (*P1* through *P4*) will appear in the *Expression* field if it is not already present.

If a trace expression is not allowed, only the *P1* trace will be enabled - and the new trace will display the data associated with the *P1* probe selected.

5. Repeat steps [3] & [4] until you have selected all the generators or probes you require for the trace.
6. Enter the *trace expression* in the *Expression* field. Within the expression, the selected probes should be represented by the names *P1*, *P2*, *P3* and *P4* which correspond to the probes selected alongside the **P1**, **P2**, **P3** and **P4** buttons respectively.

If the trace type selected does not support a *trace expression* the contents of the *Expression* field will be ignored.

7. Select the *OK* button to add the new trace to the graph.

Editing Graph Traces

An individual trace on a graph can be edited to change its name or expression.

To tag, edit and untag a graph trace:

1. Ensure the graph displaying the trace to be edited is *not* tagged.
2. Tag the *trace* by clicking the right mouse button on the trace's name. A tagged trace displays its name highlighted.
3. Click the left mouse button on the trace's name. The trace displays its *Edit Graph Trace* dialogue form.
4. Edit the trace's name or expression as required. For a trace type that does not support trace expressions any changes to the *Expression* field will be ignored.
5. Select the OK button to accept the changes.
6. Untag the trace by clicking the right mouse button over the graph, but not over any trace names.

Changing the Order and/or Colour of Graph Traces

The order of the traces on a graph can be adjusted by dragging the labels with the left mouse button. The trace can be tagged or otherwise, as you wish.

- For digital traces the purpose of moving them is simply that of obtaining a particular stacking sequence on the graph.
- For an analogue graph, you can both drag the traces from the left to right axis and back. You can also change the colour assignments, which are made on the basis of the stack order, by shuffling the vertical stacking order.

The actual colours assigned to the sequence positions can be re-configured by maximising a graph (any one will do) and then using the *Set Graph Colours* command on the *Template* menu.

THE SIMULATION PROCESS

Demand Driven Simulation

When we designed Proteus, one of our major goals was to make the use of simulation software much more intuitive than it has previously been. Much of the problem with previous simulation packages stems from the fact that number crunching parts were written first and other aspects such as displaying the results graphically were added on later, almost as an afterthought. This tends to result in a fragmented way of working in which you run one

program to draw the schematic, another to actually simulate the circuit, and yet another to display the results.

Proteus is very different in that, having drawn the circuit, you start by answering the question of what you want to see by means of placing and setting up a graph. This graph then persists until you delete it, and each time you want to see the effect of a change to the circuit, you just update the graph by pointing at it and pressing the space-bar. We call this *Demand Driven Simulation* since ISIS must work out from the graph what actually requires simulating, rather than you having to do it textually. Furthermore, you can place several graphs which perform different experiments with each one 'remembering' a different setup for the simulators.

The most important benefit of all comes from the fact that a given graph defines a set of points of interest in the design by virtue of the probes that have been assigned to its traces. ISIS is then able to use this information to figure out which parts of the design actually need to be simulated, rather than you having to calve the design up manually. It follows that a *complete* design, ready for PCB layout, can be simulated without massive editing, even if you only want to simulate particular parts of it.

A further benefit arises from the fact that there is no risk of forgetting to undo modifications made for simulation purposes because there are none to undo; the probe and generator gadgets that you do place are ignored when generating netlists for PCB layout.

Executing Simulations

Once a graph has been placed with probes and generators assigned to it, you can initiate a simulation by pointing at the graph and pressing the space-bar. ISIS then determines which parts of the design need to be simulated in order to update the graph, runs the simulations, and displays the new data.

Throughout the simulation process, a *simulation log* is maintained. In general, the log contains little of interest. However, if simulation errors occur, or you have requested the simulation netlist to be logged (see Editing Graphs on page 68 on editing graphs for how to do this) or the analysis type results in data that is not amenable to graphical display (e.g. an analogue Noise analysis) then you *will* need to view the log at the end of the simulation run.

To update a graph and view the simulation log:

1. Either ensure the graph you want to update is the *current* graph and then select the *Simulate* command from the *Graph* menu, or, place the mouse pointer over the graph you wish to update and press the space bar.
2. At the end of the simulation, if errors have occurred, you will be prompted to *Load partial results?* If you reply *YES* simulation data is loaded up to the time of the error and displayed on the graph. If you reply *NO* the simulation log is displayed in a pop-up text viewer window.

If no errors occurred, or if you selected *YES* above, you can still view the simulation log by either selecting the *View Log* command from the *Graph* menu or by using its keyboard short-cut, CTRL+V.

What Happens When You Press the Space-Bar

When you update a graph, either using the *Simulate* command on the *Graph* menu or its space-bar keyboard short-cut - a great deal of analysis and processing occurs over and above the actual analogue or digital analyses themselves. We now outline in brief various steps in this process:

- Netlist compilation - this is the usual process of tracing wires from pin to pin, and producing a list of components and a list of pin to pin connection groups, or *nets*. The resulting netlist is held in memory at this stage.
- Netlist linking - some components are modelled using sub-netlists or *model files* which are generally kept in the directory specified by the *Module Path* field of the *Set Paths* command's dialogue form. Many models are supplied with Proteus and you can, of course, create your own as well. You can think of a model as being a child sheet of a hierarchical design in which the parent object is the component to be modelled.

Each time a component modelled in this way is encountered, the original part is removed from the netlist and replaced by the contents of the model file, with the inputs and outputs of the model connected in where the original component's pins were.

At the end of this process, every component left in the netlist should have a **PRIMITIVE** property, which means that it can be directly simulated by PROSPICE.

- Partitioning - ISIS then looks at the points where you have placed probes and traces back from these to where you have injected signals. A signal is deemed to be injected by any of: a power rail, a generator with its *Isolate Before* flag set or tape output. This analysis results in the establishment of one or more partitions that may need to be simulated.

Further analysis then works out the *order* in which they must be simulated. For example, if partition A has an output that connects to an input of partition B then clearly A must be simulated first. If it turns out that B has an output driving A as well, then all is lost - it is up to you to use tape objects in such a way that cyclic dependencies do not occur.

Yet more analysis then establishes whether there are any existing results in the directory specified by the *Results Path* field (as set by the *Set Paths* command on the *System* menu) that relate to identical simulation runs for any of the current set of partitions. If there are, and if the partition concerned is not driven by a partition which is itself being re-simulated, then ISIS does not bother re-simulating the partition, but uses the existing

results instead. It follows that if you are working on the back end of a design, there is no need to keep re-simulating the front end.

You will appreciate that this part of the system is extremely clever!

- Simulator Invocation - ISIS now invokes PROSPICE on each partition in turn to carry out the actual simulation.

Each simulator invocation results in a new partition file, and progress information is also added to the simulation log which is maintained throughout the entire simulation process.

- Results Processing - Finally, ISIS accesses the partition files to build up the various traces on the graph. The graph is then re-displayed and can be maximised for measurement taking and so forth.

If errors occur during any of these stages then the details are written to the simulation log. Some errors are fatal, and others are just warnings. In the case of fatal errors, the simulation log is displayed for immediate viewing. If only warnings have occurred then the graph is re-displayed and you are offered the choice of viewing the simulation log if you want to. Most errors relate either to badly drawn circuits (which cannot, for one reason or another, be mathematically solved), or to the omission or incorrect linking of model files.

ANALYSIS TYPES

INTRODUCTION

There are thirteen types of graph available; each one displays the results of a different type of circuit analysis supported by PROSPICE:

- | | |
|-------------------|---|
| Analogue | Plots voltages, and/or currents against time, much like an oscilloscope. Additionally, expressions involving several probed values can be plotted - for example, a current can be multiplied by a voltage to give a plot of instantaneous power.

This mode of analysis is often referred to as <i>Transient Analysis</i> . |
| Digital | Plots logic levels against time, much like a logic analyser. Traces can represent single data bits or the binary value of a bus.

Digital graphs are computed using <i>Event Driven Simulation</i> . |
| Mixed Mode | Combines both analogue and digital signals on the same graph. |
| Frequency | Plots small signal voltage or current gains against frequency. This is also known as a Bode plot, and both magnitude and phase can be displayed. In addition, with the use of <i>Trace Expressions</i> you can produce input and output impedance plots.

Note that plotted values are gains referenced to a specified input generator. |
| DC Sweep | Steady state operating point voltages or currents against a sweep variable. As with <i>Analogue</i> analysis, expressions combining several probed values can also be plotted. |
| AC Sweep | Creates a family of frequency plots with one response for each value of the sweep variable. |
| Transfer | Plots characteristic curves or curve families by sweeping the value of one or two input generators and plotting steady state voltages and currents. The value of the first generator variable is plotted on the x-axis; a separate curve is produced for each value of the second variable. |
| Noise | Input or Output Noise voltages against frequency. Also produces a listing of individual noise contributions. |

Distortion	Plots the level of 2 nd and 3 rd harmonic distortion against frequency. Can also be used to plot intermodulation distortion.
Fourier	Shows the harmonic content of a transient analysis. This is much like connecting a spectrum analyser in place of an oscilloscope.
Audio	Performs a transient analysis and then plays the result through your sound card. Can also generate Windows WAV files from circuit output.
Interactive	Performs an interactive simulation, and displays the results on a graph. This analysis type lets you combine the advantages of interactive and graph based simulation.
Conformance	Performs a digital simulation and then compares the results against a set of results stored from a previous run. This is especially relevant in creating software test suites for micro-controller based applications.

In addition, all types of analysis start by computing the operating point - i.e. the initial values of all node voltages, branch currents and state variables as at time 0.

ANALOGUE TRANSIENT ANALYSIS

Overview

This graph type represents what you might expect to see on an oscilloscope. The x-axis shows the advance of time, whilst the y-axis displays voltage or current. We often refer to this type of analysis as *Transient Analysis* because it takes place in the time domain.

Transient analysis is perhaps the most commonly used form of analysis. Everything that you would measure with an oscilloscope on a real prototype can be measured on an analogue graph. It can be used to check that the circuit operates in the expected manner, to take quick measurements of gain, to assess visually the way in which a signal is distorted, to measure the current flowing from the supply or through individual components, and so on.

Method of computation

Technically speaking, this type of simulation can be referred to as *Non-Linear Nodal Analysis*, and is the form of computation used by all SPICE simulators. Considering a single point in time, every component in the circuit is represented as a combination of current sources and/or resistors. This arrangement can then be described as a set simultaneous equations using Ohms law and Kirchoff's laws, and the equations solved by Gaussian Elimination. Each time the equations are solved, the values of the current sources and

resistors are adjusted by laws built into the component models and the process is repeated until a stable set of values results.

For a simulation involving the advance of time there are two separate stages. The first task is to compute the *operating point* of the circuit - that is the voltages around the circuit at the very start of the simulation. This is then followed by considering the effect of advancing time on the circuit, and re-calculating the voltages at every step (iterating to convergence as described above). The size of each time increment is crucial to the stability of the calculations, and so PROSPICE will adjust it automatically within user defined limits. Circuits that are changing quickly, such as high switching speed line drivers, need smaller time steps and so require more effort to simulate than circuits that change more smoothly. The use of time step control and iterative solutions all adds up to a great deal of calculations, which can make transient analysis seem comparatively slow.

Since the algorithm involves iteration, there is always a possibility that the solution will not converge. Most commonly this happens at the initial time-point, meaning that the simulator cannot establish a stable or unique set of values for the operating point. Occasionally, however, it can occur further into a simulation where the circuit behaviour at a particular time becomes highly unpredictable. PROSPICE implements a number of techniques to help avoid such problems, but it is not impossible to defeat. This said, being based on Berkeley SPICE3F5 it is as good as you are going to get.

Using Analogue Graphs

Analogue graphs may have either just the left y-axis, or both left and right y-axes. Probes can be placed on a particular axis in two ways:

- By tagging and dragging the probe to the appropriate side of the graph.
- By using the *Add Transient Trace* dialogue form.



See Adding Traces To A Graph on page 68 for more information about adding traces to graphs.

It is often convenient to use the left and right y-axes to separate traces with different units. For example, if both voltage and current probes are displayed, then the left side can be used for voltage and the right side for current. It is possible to place a digital probe on an *Analogue* graph, but a *Mixed* graph is usually more appropriate.

To perform a transient analysis of an analogue circuit:

1. Add generators as necessary around the circuit, to drive the circuit inputs.
 See Placing Generators on page 100 for information on placing generators, and GENERATORS AND PROBES on page 99 for a discussion of the different types of analogue generators.
2. Place probes around the circuit at points of interest. These may be at points within the circuit, as well as obvious outputs.
3. Place an *Analogue Graph*.
4. Add the probes (and generators, if desired) to the graph.
5. Edit the graph (point at it and press CTRL+'E', and set up the simulation stop time required, as well as labelling the graph and setting control properties if required.
6. Start the simulation by either selecting the *Simulate* command on the *Graph* menu or pressing the space bar.

Defining Analogue Trace Expressions

You will normally want to add voltage and current probes to your graphs, to see the operation of a circuit. However, in an analogue transient analysis it is possible to create traces that are mathematical *expressions* based on probe values. For example, suppose a circuit output has both a voltage and current probe placed on it. By multiplying these values together, the output power at this point will be plotted.

To plot the output power:

1. Add a current probe and a voltage probe to the circuit output.
2. Invoke the *Add Transient Trace* dialogue form (press CTRL+'A') for the graph.
3. Assign the voltage probe to **P1**.
4. Assign the current probe to **P2**.
5. Alter the expression to read **P1 * P2**.
6. Click on the **OK** button.
7. Press the space bar to invoke the simulation.
 See The Add Trace Command Dialogue Form on page 70 for a detailed description of the *Add Transient Trace* dialogue form.

DIGITAL TRANSIENT ANALYSIS

Overview

Digital graphs display what you would normally expect to see on a logic analyser. The x-axis shows the advance of time whilst y axis shows a vertical stack of signals. These can be either single data bits, or a representation of the binary values carried by a bus.

Method of Computation

Digital transient analysis is computed using a technique know as *Event Driven Simulation*. This is different from analogue transient analysis in that processing only occurs when some element of the circuit changes state. In addition, only discrete logic levels are considered and this enables component functionality to be represented at a far higher level. For example, we can think of a counter in terms of a register value that increments by one each time it is clocked, rather than in terms of several hundred transistors. These make event driven simulation several orders of magnitude faster than analogue simulation of the same circuit..

The Boot Pass

The purpose of the boot pass is to define the initial states of all nets in the circuit, prior to the simulation proper. The boot pass is carried out as follows:

- All input pins connected to the VCC and/or VDD nets are deemed to be high.
- All input pins connected to the GND and/or VSS nets are deemed to be low.
- All input pins connected to a net to which a generator is connected are to deemed to be at the same state as the **INIT** property of the generator.
- All remaining pins are deemed to be initially floating.
- All models are requested (in no set order) to evaluate their inputs and set their output pins accordingly. As each output pin is set, the state of the net to which the pin connects is re-evaluated.
- As nets change state, models connected to them are asked to re-evaluate their outputs. This process continues until a steady state is found.

The Event Processing Loop

Following the boot pass, DSIM begins the simulation process proper. The simulation is carried out in a loop which passes repeatedly through the following two steps:

- All the state change events for the current time are read off a queue and applied to the relevant nets. This process results in a new set of net states.

- Where a net changes state, all the models with input pins attached to the net are re-simulated. Where their outputs change state, this creates new events which are placed on the event queue.

Of course, different models will create events which fall due for processing at different times. The DSIM kernel thus has to order all the new events created at the end of each cycle round the loop.

It is also worth pointing out that our scheme quite happily supports models which have a zero time delay. In this context, events generated with the same time-code are processed in batches (one batch equals one trip round the loop), according to how they were generated.

Termination Conditions

Simulation stops when one of the following occurs:

- The specified stop time is reached.
- The event queue becomes empty. This means that the circuit has reached a permanently stable condition.
- A logic paradox with zero time delay occurs such that the current time ceases to advance, despite repeated cycles round the event processing loop.
- A system error such as running out of event queue memory arises. This is unlikely to occur in normal use unless there is something unstable about your design, perhaps leading to a high frequency (e.g. 100MHz) oscillation somewhere.

Using Digital Graphs

To perform a transient analysis of a digital circuit:

1. Add generators as necessary around the circuit, to drive the circuit inputs.
2. Place probes around the circuit at points of interest. These may be at points within the circuit, as well as obvious outputs.
3. Place a *Digital* graph.
4. Add the probes (and generators, if desired) to the graph.

Only *Digital* generators should be used to generate digital signals - note in particular that the *Pulse* generator is intended for generating *analogue* pulses.

Only voltage probes should be used for probing digital nets - using current probes will force PROSPICE to perform a mixed mode simulation.

5. Edit the graph and set up the simulation *Stop Time* required, as well as labelling the graph and setting control properties if required.

6. Invoke the *Simulate* command on the *Graph* menu against the graph, or press the space-bar.
-  See THE SIMULATION PROCESS on page 71 for general information on how to simulate a graph.
 -  See Placing Generators on page 100 for information on placing generators and Placing Probes on page 107 for information on placing probes.
 -  See Adding Traces To A Graph on page 68 for more information about adding probes to graphs.

How Digital Data is Displayed

Normal Traces

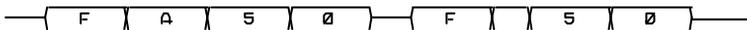
All output values from the DSIM simulator engine is in terms of six digital states:

State Type	Keyword	Graphical Appearance
Strong High	SHI	High level in cyan.
Weak High	WHI	High level in blue.
Floating	FLT	Mid level in white.
Contention	CON	Mid level in yellow.
Weak Low	WLO	Low level in blue.
Strong Low	SLO	Low level in cyan.

You can thus determine the state of a trace at a particular time, either by looking at the colour and position of the trace line, or by maximising the graph, positioning a cursor over the desired position, and reading the state off the status line. The logic levels at the cursor position are also displayed just to the right of the trace labels.

Bus Traces

Bus traces (resulting from the placement of voltage probes on bus wires) display the hex values of the bus bits between cross over lines:



Where one or more of the bus bits is neither high nor low, the bus line is drawn at the mid level. Also, if the edge transitions are too close together to display the hex value, then it is omitted. The value can still be established by positioning a cursor over it.

MIXED MODE TRANSIENT ANALYSIS

Overview

The mixed mode graph allows both digital traces and analogue waveforms to be displayed above the same x-axis which represents the advance of time. Mixed mode analysis is, in fact, used whenever a circuit contains both analogue and digital models, but the mixed mode graph is the only type capable of displaying both types of waveform simultaneously.

Method of Computation

Mixed mode transient analysis combines both the *Non-Linear Nodal Analysis* of SPICE3F5 with the *Event Driven Simulation* of DSIM. The detailed implementation of this is extremely complex, but the basic algorithm may be summarized as follows:

- Prior to simulation, each net is analysed to see whether analogue, digital or both types of model connect to it. Where analogue voltages drive digital inputs, PROSPICE inserts ADC interface objects, and where digital outputs drive analogue components, it inserts DAC interface objects.

Where several digital inputs are driven from the same net, multiple ADC objects are created so that different logic switching levels and loading characteristic can be modelled for each input. Similarly, in the rarer case of several digital outputs being connected together, multiple DAC objects are created.
- The operating point is established as described below.
- Simulation then proceeds as for a normal analogue analysis except that the ADC generate a digital event whenever their input voltages cross the switching threshold. As soon as this happens, the analogue simulation is suspended, and digital simulation is carried to establish the effect of these new events.
- When digital simulation results in a change of state for a DAC interface object, the analogue simulation is forced to simulate carefully around the switching time. In fact, the DAC outputs model a transition time (rise or fall) and a number of timepoints will be simulated during this period

Finding the Operating Point

Establishing the operating point is especially tortuous for mixed mode simulation, because the state of the analogue circuit affects the state of the digital components and vice versa. Essentially, PROSPICE proceeds as follows:

- Initial condition (IC) values are applied to both analogue and digital nets; other nets are assumed to start at zero volts.

- The nodal matrices are then constructed and solved as per normal SPICE simulation.
- For each such iteration, the digital circuit is re-processed to allow for changes in the input to the ADCs. Where such changes propagate through several digital models, the digital circuit is allowed to iterate to stability.
- DAC objects re-assign their outputs according to any changes of state in the digital circuit. If the digital circuit does not settle, this is ignored as it may be a transient citation.
- Iteration round this loop continues until a steady state is found, or until the iteration limit is reached.

Using Mixed Graphs

Mixed graphs are used in exactly the same way as *Analogue* graphs, except that you can add digital traces to them as well. To add the first digital trace to a mixed graph you should use the *Add Trace* dialogue form. Thereafter, dragging a probe on to the analogue part of the graph creates a new analogue trace, whilst dragging it onto the digital part creates a new digital trace.

FREQUENCY ANALYSIS

Overview

With *Frequency Analysis* you can see how the circuit behaves at different frequencies. Only one frequency is considered at a time, so the plots are *not* like those seen on a spectrum analyser, where all frequencies are considered together. Instead, the simulation is similar in effect to connecting a frequency generator to the input, and looking at the output with an AC voltmeter. As well as the *magnitude*, however, the *phase* of the probed signal can also be seen.

Frequency analysis produces frequency response or *Bode* plots. It is useful in checking that filters are behaving as expected, or that amplifier stages will work correctly across the required frequency range.

Frequency graphs can also be used to plot small signal input and output impedance against frequency.

Method of computation

Frequency analysis is performed by first finding the operating point of the circuit, and then replacing all the active components with linear models. The internal capacitances of the active devices are calculated at the operating point, and assumed not to vary much over the working voltage swings of the circuit. All the generators, apart from frequency reference generators

(see below) are replaced by their internal resistances. This causes power lines to be effectively connected together, as is normally the case during frequency computations.

The analysis is then performed with complex numbers in a linear fashion. The frequency of interest is gradually increased from the starting to the concluding frequency in even increments. The linear nature of this analysis makes it typically much faster than transient analysis, even though complex numbers are used.

It is important to remember that frequency analysis assumes a linear circuit. This means that a pure sine wave at the input will produce a pure sine wave at the output, over all the frequencies probed. Of course, no real active circuit is purely linear but many are close enough to allow this form of analysis. There are also circuits which are not at all linear, such as line repeaters with schmitt trigger inputs. For non-linear circuits the term 'frequency response' has no real mathematical meaning, and so any frequency simulation will not produce meaningful results. Should you be interested in the frequency domain behaviour of such circuits, then *Fourier* analysis is much more appropriate.

Using Frequency Graphs

To calculate the magnitude of sine waves at the *output*, we must inject a reference sine wave at the *input*. PROSPICE will do this automatically, but needs to know where the input of the circuit is. To do this, you must give each frequency graph a '*Reference Generator*' to tell the simulator where to inject the reference signal. There are three ways of doing this:

- Use the *Reference* field on the *Edit Frequency Graph* dialogue form to select an input generator. This object can be any ordinary single pinned generator object, or the FREQREF primitive from the ASIMMDLS library.
- Tag and drag any analogue generator onto the frequency graph.
- Use the *Add Trace* dialogue form to add a generator as a **REF**.

The FREQREF primitive is useful for defining models that drive the circuit - a microphone model for example. This could contain an explicit named generator, used as a reference when the effects of the rest of the microphone model (such as modelled frequency response) need to be taken into consideration.

The second and third techniques will be used more often. The action of dragging a generator onto a frequency graph is different to dragging it onto a transient graph. Instead of adding the generator's probe, the generator becomes the circuit reference point. The generator need not be a *Sine* generator - *DC*, *Pulse* and *Pwlin* will all work as well. If you want to add the generator as a probe, it can still be done using the *Add Trace* dialogue form.

The magnitude of the reference is always 1 volt, the phase is always 0 degrees. The internal resistance of the reference generator will follow whatever was defined for the generator in the

first place. This is used for the dB calculations, i.e. $0\text{dB} = 1$ volt. ISIS will limit very small values, to avoid $\log(0)$, to -200dB .

Frequency graphs always have both left and right y-axes. The left y-axis is used to display the magnitude of the probed signal, and the right y-axis is used to display the phase. If you drag a probe onto the left of the graph, it will display magnitude - if you drag it onto the right, the phase. The x-axis is used to display the frequency of the reference generator. A logarithmic scale is always used for the frequency. The left y-axis may be displayed either in dBs or normal units, and the right y-axis is always in degrees.

To see the frequency response of a circuit:

1. Place probes around the circuit at points of interest.
2. Place a *Frequency* graph.
3. Add the probes to it. Add to the left for magnitude, and to the right for phase. You may well want to add probes twice in order to see both phase and magnitude.
4. Edit the graph (point at it and press CTRL+'E') and set the required start and stop frequencies, and any *Simulation Control Properties* required.
5. Press the space bar to invoke PROSPICE.

The samples files ZIN.DSN and ZOUT.DSN show how to combine frequency graphs with trace expressions to plot input and output impedance.

DC SWEEP ANALYSIS

Overview

With a DC Sweep analysis you can see how *changing* a circuit will affect its operation. This is achieved using *Property Expression Evaluation* in the PROSPICE simulator engine. The sweep graph defines a variable that can be swept in even steps over a user-defined range. The sweep variable can appear in any property within the circuit, such as a resistor value, a transistor gain or even the circuit temperature.

A *DC Sweep* curve shows the steady state voltage (or current) levels of the probed points around the circuit, as the swept variable is increased. It can be used to plot the DC transfer characteristic of a circuit by assigning the swept variable to a generator value, as well as plotting the effect of changing component values on the operating point of the circuit.

Method of computation

In a *DC* sweep PROSPICE will repeatedly find the operating point of the circuit, incrementing the swept variable between calculations. PROSPICE will re-calculate all the variables used

between steps, so the swept variable can be used as often as is required, and variables based on the swept variable can also be used. Any circuit initialisation parameters will be honoured for every operating point calculated.

Using DC Sweep Graphs

In common with analogue transient analysis (see *Analogue Transient Analysis* on page 76), either the left, right or both y-axes may be used. The graph x-axis shows the swept variable. Trace expressions, described in *The Add Trace Command Dialogue Form* on page 70, may be used in DC sweeps. When choosing the number of steps, bear in mind that simulation time is roughly proportional to the number of steps used.

To plot the transfer function of a circuit:

1. Place a *DC* generator on the input of the circuit. Set its value to be X.
2. Place one or more probes at appropriate outputs of the circuit.
3. Place a *DC Sweep* graph, and add the probes to it - see *Adding Traces To A Graph* on page 68.
4. Edit the graph (CTRL+'E') and set the start and stop values to the extremes of the input sweep that you want. Check that the swept variable is X.
5. Press the space bar to invoke PROSPICE.
6. If the resulting traces look disjointed or angular when you zoom in on the graph, increase the number of steps in the *Edit DC Sweep Graph* dialogue form.

To see the effect of altering circuit values:

1. Set up the circuit as you would for a transient analysis. Add probes and generators at appropriate points in the circuit.
2. Edit the components whose values you are interested in. Set their values to be expressions containing X, the swept variable. You can edit just one, or several components.
3. Place a *DC Sweep* graph, and add the probes and generators to it - see *Adding Traces To A Graph* on page 68.
4. Edit the graph, and set the sweep parameters up accordingly.
7. Press the space bar to invoke PROSPICE.

AC SWEEP ANALYSIS

Overview

This type of analysis creates a family of frequency response curves for a different values of a swept variable.

The main use for this graph type is in seeing how a particular component value affects the frequency response of your circuit.

Method of Computation

This analysis is computed exactly as an ordinary frequency response except that multiple runs are performed, one run for each value of the sweep variable.

The restriction of a linear circuit applies to this analysis as it does for a normal frequency analysis - see Frequency Analysis on page 83.

Using AC Sweep Graphs

As with *Frequency* analysis, the left and right axes display gain and phase respectively. Also, an input generator must be specified as a reference point for gain calculations.

To see the effect of altering a circuit parameter on frequency response:

1. Set up the circuit as you would for a frequency analysis.
2. Edit the components of interest. Set their properties to be expressions containing **X**, the swept variable. You can edit just one, or several components.
3. Place an *AC Sweep* graph, and add the probes to it - see Adding Traces To A Graph on page 68.
4. If you do not have a generator on the input already, then place one.
5. Tag and drag a generator on the input of the circuit onto the graph to be the reference generator.
6. Edit the graph (CTRL+'E') and set the sweep parameters accordingly. Set the *Freq* parameter to the frequency of interest.
7. Press the space bar to invoke PROSPICE.

DC TRANSFER CURVE ANALYSIS

Overview

This graph type is specifically designed for producing characteristic curve families for semiconductor devices, although it occasionally finds other applications. Each curve consists of a plot of operating point voltage or current against the value of a nominated input generator which is swept from one DC value to another. An additional generator may also be stepped to produce a set of curves.

Method of Computation

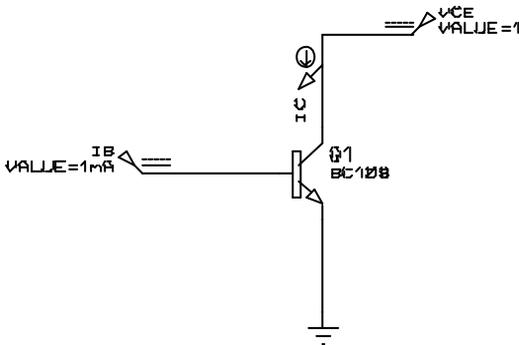
This is very similar to the computation for a DC Sweep analysis except that two values can be swept. The operating point is found for the starting value of each generator and the first generator is then stepped through the specified range. After each sweep of the first generator, the second generator's value is incremented; a new curve is plotted for each discrete value of the second generator.

Using Transfer Graphs

In common with analogue transient analysis (see Analogue Transient Analysis on page 76), either the left, right or both y-axes may be used. The graph x-axis shows the first swept variable, and a separate curve results for each value of the second generator. Trace expressions, described in *The Add Trace Command Dialogue Form* on page 70, may also be used as required.

To plot the transfer curves for an transistor:

1. Set up a circuit similar to the one below:



I_B is a current source and V_{CE} is a voltage source. I_C measures collector current.

2. Place a *Transfer* graph, and add the probe IC to it.
3. Point at the graph and press CTRL+E to edit it.
4. Assign *Source 1* to VCE and *Source 2* to IB.
5. Set the voltage and current ranges to something sensible for the transistor concerned - e.g. 0 -> 10V for VCE and 100uA -> 1mA for IB.
6. Adjust the number of steps for IB to give sensible intermediate values. Bear in mind that their will be one more curve than the number of steps, because one step implies two discrete values.
7. Close the dialogue form and select OK to simulate the graph.

NOISE ANALYSIS

Overview

The SPICE simulator engine can model the thermal noise generated in resistors and semiconductor devices. The individual contributions of noise are summed together at a *voltage* probe in the circuit for a range of frequencies. The noise voltage, normalised with respect to the square root of the noise bandwidth, can then be plotted against frequency. Traces on a noise graph always have units of $V/\sqrt{\text{Hz}}$.

Two types of noise figure are computed - *Output Noise* and *Equivalent Input Noise*. The computation of the latter enables comparison of the noise with the input signal, or input noise, since it represents the level of noise at the input that would be required to create the actually noise at the output taking into account the circuit gain at a particular frequency.

Placing a probe on the left axis displays output noise, whilst equivalent input noise can be displayed by dragging the probe onto the right axis.

Noise analysis tends to produce extremely small values (of the order of nanovolts). For this reason, there is the option to display them in dBs. The 0dB reference is $1V/\sqrt{\text{Hz}}$.

Correct modelling of noise for circuits involving IC macro models is not guaranteed in any way, because these models may well use linear controlled sources to model basic device behaviour only.

Method of calculation

The operating point of the circuit is computed in the normal way, and then the circuit model is modified to correctly sum the noise contributions. All generators except the input reference are ignored in noise analysis (except during computation of the operating point) and so do not have to be removed before the analysis. The PROSPICE engine will compute the thermal

noise for each voltage probe in the circuit, including those associated with generators and tapes, as it has no way of knowing which probes you wish to look at after the analysis. Noise currents are not supported, and the PROSPICE engine will ignore any current probes. A separate simulation is done for each probe in a noise analysis, so the simulation time is roughly proportional to the number of voltage probes placed.

Using Noise Graphs

When using noise analysis, it is important to remember that the effects of noise pick-up from external electric or magnetic fields are not modelled. In many situations where noise is critical the noise picked up in the input leads will dominate the noise performance of the system.

If you have partitioned the circuit using tapes (see TAPES AND PARTITIONING on page 128), you must be aware that only the current partition is considered by the PROSPICE simulator engine. A noise figure taken in isolation is still useful, but the tapes must be removed to see the equivalent noise of the entire circuit.

In order to determine the source of noise in the circuit, the individual noise voltages are entered in to the simulation log. Each probe is considered in turn, and the noise contribution for each component given. The contributions are computed, and printed, as squared values. This process is done at the starting and ending frequencies. If you have a large number of probes, there may well be a great deal of result data printed.

To analyse the noise in a circuit:

1. Place a *Noise* graph, and edit it to set the frequency range of interest, and the input reference generator.
2. Add the voltage probes at the circuit outputs, or other points of interest, to the graph. see Adding Traces To A Graph on page 68.
3. Press the space bar to invoke the PROSPICE simulator engine.
4. If the noise level needs to be reduced, examine the simulation log (CTRL+'V') to determine the source of the noise.

DISTORTION ANALYSIS

Overview

Distortion analysis determines the level of distortion harmonics produced by the circuit under test. These can be either the 2nd and 3rd harmonics of a single fundamental or the intermodulation products of two test signals.

Distortion is created by non-linearities in a circuit's transfer function - a circuit comprising linear components only (resistors, capacitors, inductors, linear controlled sources) will not

produce any distortion. SPICE distortion analysis models distortion for diodes, bipolar transistors, JFETs and MOSFETs.

Correct modelling of distortion for circuits involving IC macro models is not guaranteed in any way, because these models may well use linear controlled sources to model basic device behaviour only.

Similar information can be established using a *Fourier* analysis, but the *Distortion* analysis is also able to show how the distortion varies as the fundamental frequency is swept.

Method of Computation

This analysis is based on the small signal (AC) models for the devices in the circuit so the first step is to compute the operating point. Each non-linear device model then contributes a complex distortion value for the appropriate harmonics, dependent on how much the device is seeing of the input fundamental. The extent to which these harmonics appear at the output determines the values plotted. The process is repeated across the specified range of input frequencies. In fact, the mathematics of this analysis is extremely complicated and involves the construction and manipulation of Taylor series to represent the device non-linearities.

Note that complex values are used, so the analysis yields information about both the magnitude and phase of each harmonic.

For single frequency harmonic distortion, two curves are produced for each trace on the graph - one for 2nd harmonic and one for 3rd harmonic.

For intermodulation distortion, two input frequencies are used, specified in terms of a ratio between the 2nd frequency (F2) and the fundamental (F1). Three curves per trace are displayed showing the intermodulation artefacts at F1+F2, F1-F2 and 2F1-F2.

Some care is needed in choosing the ratio F2/F1 since mathematical oddities can otherwise occur. For example, if F2/F1 is 0.5, the value of F1-F2 is F2 and the value of the F1-F2 plot will then be meaningless as it coincides with the second fundamental. In general you should try to choose irrational numbers for this ratio. F2/F1=49/100 would be a much better choice.

Note that there is a constraint of $F2/F1 < 1$.

Using Distortion Graphs

Distortion graphs show the magnitude of harmonics on the left axis and the phase (normally of less interest) on the right axis. The test frequency (F1) is plotted on the x-axis.

For Harmonic Distortion analysis (one input frequency F1) two curves are plotted per trace, one each for the artefacts at 2F1 and 3F1.

For Intermodulation Distortion analysis (two input frequencies (F1 and F2) three curves are plotted per trace showing artefacts at F1+F2, F1-F2 and 2F1-F2.

In either case, which curve is which may be established by placing a cursor on the graph - the curve you are pointing at will be identified on the right hand side of the status bar.

FOURIER ANALYSIS

Overview

Fourier Analysis is the process of transforming time domain data into to the frequency domain and the result is similar to that obtained by connecting up a spectrum analyser instead of an oscilloscope. It is especially useful in analysing the harmonic content of signals, perhaps to look for particular types of distortion but has many other uses also.

Method of Calculation

Fourier analyses are created by first executing a *Transient Analysis* and then performing a Fast Fourier Transform on the resulting data. This process involves discrete time sampling of the time domain data with the result that the use Nyquist Sampling criterion applies. Put simply, this means that the highest frequency that can be observed is half that of the sampling frequency. However, other misleading effects can occur due to *aliasing* of the sampling frequency with harmonics of the input signal that are higher than half the sampling frequency. To minimize these effects, various types of *window* can be applied to the input data prior to the FFT.

Using Fourier Graphs

In the first instance, you need to set up your circuit as for a *Transient Analysis* except that you should use a *Fourier* graph instead of (or as well as!) an *Analogue* graph.

To analyse the frequency content of a signal:

1. Set up the circuit as for a *Transient Analysis*.
2. Add a *Fourier* graph to the design and drag probes connected to the points of interest onto it.
3. Choose start and stop times and frequency/resolution values to suit the signal you are analysing. If possible, choose a time interval and frequency resolution that correspond to the fundamental frequency of the signal being analysed.
4. Press the space bar to invoke PROSPICE.

If the start and stop times are the same for both Transient and Fourier graphs, PROSPICE will not need to resimulate the circuit between transient and fourier analyses. Instead, ISIS will just perform an FFT on the existing time domain data.

AUDIO ANALYSIS

Overview

PROTEUS VSM incorporates a number of features that enable you to *hear* the output from your circuits (providing you have a sound card, of course!). The major component of this is the *Audio* graph. This is essentially the same as an *Analogue* graph except that after simulation, a Windows WAV file is generated from the time domain data and played through your sound card.

The WAV files can also be exported for use in other applications.

Method of Calculation

Audio analyses are performed in exactly the same way as for *Transient Analysis* except that after simulation, the data is re-sampled at one of the standard PC sampling rates (11025, 22050 or 44100Hz) then written out in WAV format using the standard Windows functions provided for this purpose. Finally the command is sent to play the WAV file to your audio hardware.

Using Audio Graphs

In the first instance, you need to set up your circuit as for a *Transient Analysis* except that you should use an *Audio* graph instead of an *Analogue* graph.

To listen to the audio output of a circuit:

1. Set up the circuit as for a *Transient Analysis*.
2. Add an Audio graph to the design and drag a probe from the output of the circuit onto it.
3. Choose start, stop times and loop times to generate a waveform of reasonable length with the minimum of actual simulation. Creating one second of audio by analysing 1ms and looping it 1000 times is dramatically faster than analysing the circuit for the whole 1s.
4. Choose a sample resolution and rate to suit the nature of the signal you want to hear. Use 16 bit resolution unless you are creating big files and/or are short of disk space. Most PC sound speaker systems will not benefit much from 44.1KHz sampling.
5. Press the space bar to invoke PROSPICE.
6. Press CTRL-SPACE to replay the audio, without resimulating, if required.

INTERACTIVE ANALYSIS

Overview

The Interactive Analysis type combines the advantages of interactive and graph based simulation. The simulator is started in interactive mode, but the results are recorded and displayed on a graph as with Transient Analysis.

This analysis is especially useful in examining what happens when a particular control is operated in a design, and may be thought of as combining a storage oscilloscope and logic analyser into one device.

Method of Computation

The method of computation is identical to that for a mixed mode transient analysis, except that the simulator is run in interactive mode. Consequently, the operation of switches, keypads and other actuators in the circuit will have an effect on the results. Also, the simulation will proceed at a speed determined by the *Animation Timestep*, rather than at the maximum rate possible.

- Beware that very large amounts of data may be captured. A processor clocking at realistic speeds will generate millions of events per second and these will occupy many megabytes if captured and displayed on a graph – especially if you are probing data or address busses. You can quite easily crash your system if ISIS ends up loading 20 or 30Mb of results data.

You may do better to use the *Logic Analyser* if it is not possible to capture the required data in a relatively short simulation run.

- As with ordinary *Interactive Simulation*, multi-partitioning of the circuit is not supported and any tape objects not set to *Play* mode will be automatically removed from the circuit.

Using Interactive Graphs

The usual scenario for interactive analysis is that you are testing a design with interactive simulation, and find that something odd happens when particular controls are operated. In the first instance, you may try using the virtual instruments to see what is going on, but in some cases there is a need to capture the results to a graph and study them at your leisure.

To perform an interactive analysis:

1. Add probe objects to the points of interest.
2. Place an *Interactive* graph on a free part of the schematic and drag the probes onto it.
3. Edit the graph and choose appropriate start and stop times. Note that PROSPICE will not capture data to the probes prior to the start time – this is to reduce the amount of data that will be pulled into ISIS.
4. Set any interactive controls on the schematic to suitable initial states.
5. Get ready to perform any interactive operations and then press the SPACE bar. You have to operate the active components within the time you chose in step 4 if the effect of these actions is to be recorded. If this proves difficult, you can either increase the stop time, or reduce the *Timestep Per Frame* setting.

DIGITAL CONFORMANCE ANALYSIS

Overview

A conformance analysis compares one set of digital simulation results against another. The idea is that a design that has been previously accepted as working can be quickly re-tested after modification in order to prove that there have been no unwanted side effects from the change. This is particularly relevant in micro-controller based applications where the entire application may need to be re-tested after changes have been made to the firmware code.

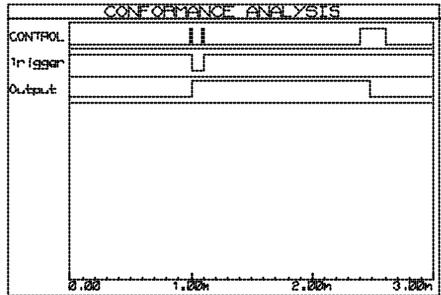
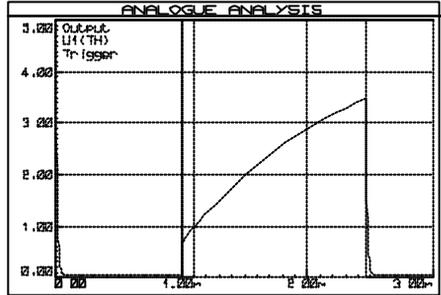
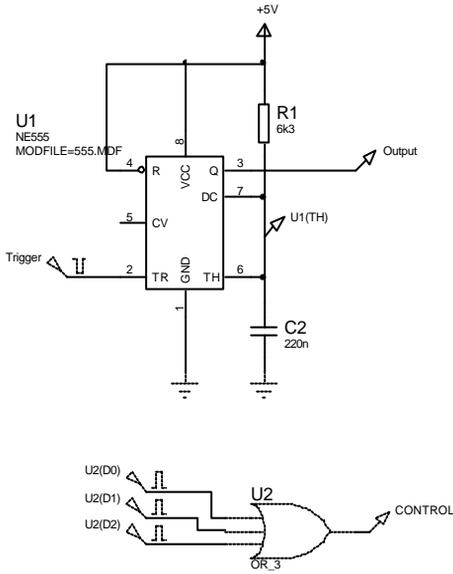
Method of Computation

The method of computation is identical to that for a digital transient analysis, except that two sets of results can be stored in the graph. We call the two sets of results the *test results* and the *reference results*.

Conformance or non-conformance is determined by comparing the test and reference results at each edge of the first trace. We call this trace the *control trace* and it is displayed in a different colour to distinguish it from the others. Very significantly, there is no requirement for the edges in the test and reference copies of the control trace to occur at the same times. This means that changes in the absolute timing of events within the result data do not necessarily imply non-conformance. This is particularly relevant in micro-controller applications where *any* changes to the code will be bound to affect the absolute timing of events within the system. In such cases, the control trace may be generated by the code itself, on entry and/or exit to the software routines under test.

Using Conformance Graphs

Typically, conformance analyses will be used as part of a test strategy, most usually in an embedded systems application although we also use them in-house for testing our simulator models. In the simple example, below, will show how to use a conformance graph to test the operation of 555 monostable.



The circuit under test consists of U1, R1 and C2 which are wired as a classic 555 monostable. The circuit is triggered at 1ms by a digital pulse generator, and the resulting output waveforms are displayed on the analogue analysis graph. To test the correct operation of the circuit, it would be reasonable to establish the following facts:

- That the output is low before the trigger pulse.
- That the output goes high soon after the trigger goes low.
- That the output remains high when the trigger returns high.
- That the output remains high for a period of around 1.5ms.
- That the output returns low after this time.

To achieve this using the conformance graph, we need sample the output data around each edge transition of the trigger signal, and also at either side of the output signal. We can determine a tolerance for the delay time of the monostable by choosing the spacing between the last two sampling points.

This is achieved using a series of digital pulse generators whose outputs are combined using an OR gate. The width setting on each pulse generator determines the timing tolerance for each expected event. For example, the third pulse generator is set to trigger between 2.4ms and 2.6ms, giving timing tolerance of +/- 100us on the width of the expected output pulse.

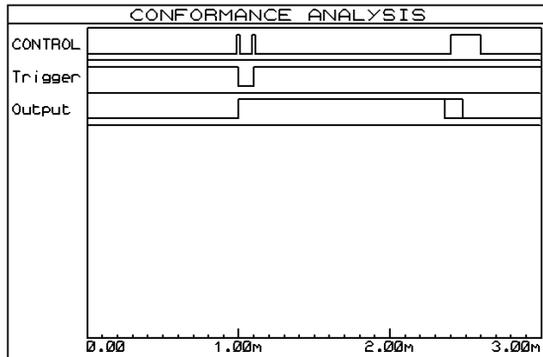
Note that you could have used a digital clock or pattern generator to generate the control signal; the use of a simple clock will result in verification at regular time intervals which will, in fact, suffice for many applications.

The general procedure may be summarized as follows:

To set up a conformance analysis:

1. Decide on what exactly you want to test and at what times.
2. Design a test case schematic which generates the outputs you wish to verify and a control signal that changes state at each time you wish to validate the results.
3. Place a conformance graph and set it up exactly as you would for a *Digital Analysis*, making sure that the control trace is at the top of the graph.
4. Run the simulation and verify that the results are as expected, and that the transitions on the control trace occur at the times that you wish to verify the results.
5. Edit the conformance graph and press the *Store Results* button. This will make the currently displayed results into the reference results. These then are displayed in a dimmed colour and the graph will compare any new results against them when it is re-simulated.
6. Save the design - it is now ready to be used for verification purposes at a future date.

Let us now suppose that due to component shortages, the values of R1 and C2 must be changed. If we make C2 100nf, can we make R1 15k, and will the design still work within spec? The results of this experiment are shown overleaf:



The answer is no. The output pulse is now slightly too short, and the following information appears in the simulation log for the graph:

```
Comparing Results...
Data mismatch at time 2.40m in trace 'Output'.
Data signatures were 'L' and 'H'.
Conformance analysis FAILED.
```

Note the graph cursor is positioned at the time of the first discrepancy i.e. 2.4ms

There are in fact three ways to invoke a conformance analysis:

- By re-simulating the graph in the usual way - e.g. by pressing the space bar or using the *Simulate Graph* command on the *Graph* menu.
- Using the *Conformance Analysis* command on the *Graph* menu. This re-simulates all the conformance graphs in the design and reports any errors.
- For advance users only - from the command line. Entering

```
ISIS <filename> /V
```

will perform a global conformance analysis as above. If any graph fails ISIS sets an error level of 1. This allows multiple test-cases to be automatically validated from a batch file.

GENERATORS AND PROBES

GENERATORS

Overview

A *generator* is an object that can be set up to produce a signal at the point at which it is connected. There are many types of generator, each of which generates a different sort of signal:

- *DC* - Constant DC voltage source.
- *Sine* - Sine wave generator with amplitude, frequency and phase control.
- *Pulse* - Analogue pulse generator with amplitude, period and rise/fall time control.
- *Exp* - Exponential pulse generator - produces pulses with the same shape as RC charge/discharge circuits.
- *SFFM* - Single Frequency FM generator - produces waveforms defined by the frequency modulation of one sine wave by another.
- *Pwlin* - Piece-wise linear generator for arbitrarily shaped pulses and signals.
- *File* - As above, but data is taken from an ASCII file.
- *Audio* - uses Windows WAV files as the input waveform. These are especially interesting when combined with Audio graphs as you can then listen to the effect of your circuit on audio signals.
- *DState* - Steady state logic level
- *DEdge* - Single logic level transition or edge.
- *DPulse* - Single digital clock pulse.
- *DClock* - Digital clock signal.
- *DPattern* - An arbitrary sequence of logic levels.

The first eight generator types are considered to be analogue components and are simulated by the SPICE simulator kernel whilst rest relate to digital circuitry and are handled by DSIM.

Placing Generators

To Place a Generator:

1. Obtain a list of generator types by selecting the *Generator* icon. The list of generator types is then displayed in the *Object Selector*.
2. Select the type of generator you wish to place from the selector. ISIS will show a preview of the generator in the *Overview Window*.
3. Use the *Rotation* and *Mirror* icons to orient the generator according to how you want to place it.
4. Place the mouse in the *Editing Window*, press down the left mouse button and drag the generator to the correct position. Release the mouse button.

You can place a generator directly on to an existing wire by placing it such that its connection point touches the wire. Alternatively you can place several generators in a free area of your design, and wire to them later.

When a generator is placed unconnected to any existing wires, it is given a default name of a question mark (?) to indicate that it is unannotated. When the generator is connected to a net (possibly when placing it if it is placed directly onto an existing wire) it is assigned the name of the net, or, if the net itself is unannotated, the component reference and pin name of the first pin connected to the net. The name of the generator will automatically be updated as it is unwired or as it is dragged from one net to another. You can assign your own name to the generator by editing the object, in which case the name becomes permanent and is not updated.

 See *NET NAMES* in the ISIS manual for an explanation of nets and net-naming.

Editing Generators

Any generator may be edited using any of the general ISIS editing techniques, the easiest of which are either to click right (to tag) and then left (to edit), or to point at the generator and press CTRL+E.

The *Edit Generator* dialogue form provides a number of common fields and then a further set of fields that change according to the generator's type. The common fields are explained below:

Name	The name of the generator.
-------------	----------------------------

You can change the name of a generator by typing in a new name into the name field. Once you have manually changed the name of a generator, ISIS will never replace it, even if you move the generator to a new net.

Type	<p>If you want a probe to revert to automatic naming, clear its name to the blank string by pressing ESC once, and then clicking OK.</p> <p>The type of generator. You can change the type of generator from the type placed by selecting the button for the new type required.</p> <p>This control determines the generator specific fields that are displayed on the right hand side of the dialogue.</p>
Current Source	<p>With the exception of the DIGITAL generator, all the other types are capable of operating either as voltage or current sources. Checking this box causes the generator to be a current source.</p>
Isolate Before	<p>This check box controls whether or not a generator placed in the middle of a wire acts to 'break' the wire to which it is connected, isolating the net to which the generator points from the net behind it.</p> <p>The check box has no affect on a generator connected directly to a net by a single wire.</p>
Manual Edits	<p>When checked, the generator's properties are displayed as a textual property list. This is provided mainly for backward compatibility with previous versions of the software. However, advanced users may wish to use property expressions within generators' properties, and this is only possible in the manual edit mode.</p>

DC Generators

The *DC* generator is used to generate a constant analogue DC voltage or current level. The generator has a single property which specifies the output level.

Sine Generators

The *Sine* generator is used to produce continuous sinusoidal waves at a fixed frequency. A number of parameters can be adjusted:

- The output level is specified as a peak amplitude (VA) with optional DC offset (VO). The amplitude may also be specified in term of RMS or peak-peak values.
- The frequency of oscillation can be given in Hz (FREQ), or as a period (PER), or in terms of the number of cycles over the entire graph.

- A phase shift can be specified in either degrees (PHASE) or as a time delay (TD). In the latter case, oscillation does not start until the specified time.
- Exponential decay of the waveform after the start of oscillations is specified by the damping factor, THETA.

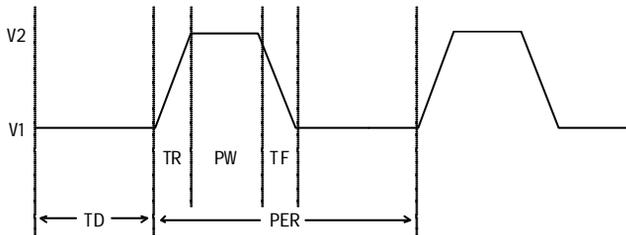
Mathematically, the output is given by:

$$V = VO + VAe^{-(t-TD)THETA} \sin(2pFREQ(t + TD))$$

for $t \geq TD$. For $t < TD$ the output is simply the offset voltage, VO.

Pulse Generators

The *Pulse* generator is used to produce a variety of repetitive input signals for analogue analyses. Square, saw-tooth and triangular waveforms can be produced, as well as single short pulses.. Note that the rise and fall times cannot be zero, so a truly square wave is not allowed. This is because instantaneous changes in general are not allowed in PROSPICE. The operation of the pulse generator is best described by the following diagram:



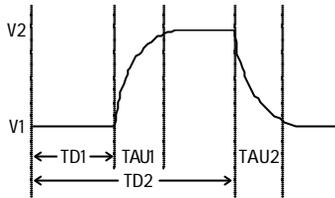
where

- PER** - The period of the waveform. If none is specified, then **FREQ** is used instead.
- FREQ** - The frequency of the waveform. This defaults to one period for a transient analysis.
- V1** - The low-level value of the output.
- V2** - The high-level of the output.
- PW** - The time for which the output is at V2 in each cycle. This does not include **TR** and **TF**.
- TR** - The rise time - time taken between **LOW** and **HIGH** in each cycle. This defaults to 1ns.
- TF** - The fall time - time taken between **HIGH** and **LOW** in each cycle. This defaults to **TR**.

- TD** - The delay time. The output of the generator starts at **V1**, and will remain at this level for **TD** seconds.

Exponential Generators

The *Exp* generator produces the waveforms as an RC circuit under charging and discharging conditions. The parameters are best described by the diagram overleaf.



where the parameters are

- V1** - The low-level value of the output.
- V2** - The high-level of the output.
- TD1** - The start time of the rise curve.
- TAU1** - The time constant of the rise curve. This is the time for the voltage to reach approximately 0.63 of its full potential.
- TD2** - The start time of the fall curve. Note that **TD2** is measured from zero.
- TAU2** - The time constant of the fall curve.

Mathematically, the waveform is defined in three sections:

$$0 \text{ to } TD1 \quad V1$$

$$TD1 \text{ to } TD2 \quad V1 + (V2 - V1) \left(1 - e^{-\frac{(t-TD1)}{TAU1}} \right)$$

$$TD2 \text{ to } TSTOP \quad V1 + (V2 - V1) \left(1 - e^{-\frac{(t-TD1)}{TAU1}} \right) + (V1 - V2) \left(1 - e^{-\frac{(t-TD2)}{TAU2}} \right)$$

Single Frequency FM Generators

The *SFFM* generator produces a waveform that represents the result of frequency modulating one sine-wave with another. Mathematically, this is represented as:

$$V = VO + VA \sin(2pFCt + MDI \sin(2pFS t))$$

where the parameters are

- VO** - The DC offset voltage.
- VA** - The carrier amplitude.
- FC** - The carrier frequency.
- FS** - The signal frequency.
- MDI** - The modulation index.

Piece-wise Linear Generators

The *Piece-wise Linear* generator is used to produce analogue signals which are too complicated for a pulse generator, or to re-produce a measured waveform. The output waveform is described using pairs of values for time and output magnitude. The output of the generator at intermediate times is then linearly interpolated between the given times.

The piece-wise linear generator dialogue form consists of a graph editor onto which you can drag and drop data points. Operation can be summarized as follows:

- To place a new data point click left.
- To move a data point, drag it using the left mouse button.
- To delete a data point click right.

The following constraints apply:

- There must always be a point at time zero, although its y-value can be changed - i.e. you can only drag it up or down.
- If you attempt create a vertical edge, the graph editor will separate the two data points by the minimum rise/fall time value.

If you have tabular data, you may find it easier to use manual edit mode. In this case, each data point is specified by a property of the form $V(t)$. The following example shows how this works.

$$\begin{aligned}V(0) &= 0 \\V(2n) &= 0 \\V(3n) &= 1 \\V(5n) &= 1 \\V(6n) &= 0\end{aligned}$$

If you have a large amount of data, you may find it easier to use a FILE generator instead.

File Generators

The *File* generator is used to drive a circuit from an analogue signal that is specified by series of time points and data values contained in an ASCII file. It is thus very similar to the piecewise linear generator except that the data values are held externally rather than being given as device properties.

The dialogue form has only one field, which specifies the name of the data file. There is no default extension for these files, and the file should be located in the same directory as the design file unless a full path is specified.

Data File Format

The ASCII data file should be formatted with one time/voltage pair on each line separated by white space (spaces or tabs, not commas). The time values must be in ascending order, and all values must be simple floating point numbers (no suffixes allowed).

Example

The following example data file produces three cycles of a saw-tooth waveform with rise time 0.9ms, fall time 0.1ms and amplitude 1V.

```

0          0
9E-4      1
1E-3      0
1.9E-3    1
2E-3      0
2.9E-3    1
3E-3      0
    
```

Audio Generators

The *Audio* generator is used to drive a circuit from a Windows WAV file. In conjunction with *Audio* graphs they make it possible to hear the results of processing a sound signal through a simulated circuit.

- The filename is assumed to have a default extension of WAV, and should be located in the same directory as the DSN file unless a full path is specified.
- The amplitude can be specified either in terms of the maximum absolute value for positive and negative excursions, or as a peak to peak value.
- A DC offset can be specified; if this is zero the output voltage will oscillate about zero.
- For stereo WAV files, you can select which channel is played, or whether to treat the data as mono.

Digital Generators

There are five sub-types of digital generator:

- Single Edge – a single transition from low to high or high to low.
- Single Pulse – a pair of transitions in opposite directions which together form a positive or negative pulse waveform. You can specify either the times of each edge (start time and stop time) or the start time and the pulse width.
- Clock – a continuous pulse train of even mark-space ratio. You can specify the starting value and the time of the first edge as well as the period or frequency. The period specifies the time for a whole cycle, not the width of a single mark or space.
- Pattern – this is the most flexible and can, in fact, generate any of the other types. The pattern generator is defined in terms of the following parameters:

Initial State This is the value at time zero, and also the value used when finding the operating point in a mixed mode circuit.

First Edge This is the time when the pattern actually starts; the output will remain at the initial state value until this time is reached.

Timing Each step of the pattern can take the same time (*Equal mark/space timing*) or can have a different time for high and low values. In this case, the *Pulse* width specifies the time for logic '1' values and the *Space Time* specifies the time for logic '0' values.

Transitions The output can be defined to run continuously till the end of the simulations, with the pattern repeating, or for a fixed number of edge transitions only.

Bit Pattern The default pattern is a simple alternative high-low sequence. Alternatively, a pattern string may be specified. The pattern string can contain the following characters:

- | | |
|------------|--|
| 0,L | - Output waveform is to go to a strong low level (note upper-case 'L'). |
| 1,H | - Output waveform is to go to a strong high level (note upper-case 'H'). |
| l | - Output waveform is to go to a weak low level (note lower-case 'l'). |
| h | - Output waveform is to go a weak high level (note lower-case 'h'). |
| F,f | - Output is to go to a floating level. |

- Script – the generator will be controlled from a DIGITAL BASIC script. The generator is accessed in the script by declaring a **PIN** variable with the same name as the generator's reference.

PROBES

Overview

A *probe* is an object that can be set up to record the state of the net to which it is connected. There are two types of probe:

- Voltage probes - these can be used in both analogue or digital simulations. In the former they record true voltage levels, whilst in the latter they record logic levels and their strengths.
- Current probes - these can only be used in analogue simulations, and must be placed on a wire such the wire passes through the probe. The direction of measurement is indicated by the current probe graphic.

You cannot use current probes in digital simulations, or on buses.

Probes are most normally used in *Graph Based Simulations*, but can also be used in interactive simulations to display operating point data and to partition the circuit.

Placing Probes

To Place a Probe:

1. First select either the *Voltage Probe* or *Current Probe* icon. ISIS will show a preview of the probe in the *Overview Window*.
2. Use the *Rotation* and *Mirror* icons to orient the probe according to how you want to place it.

Note that, for a *Current Probe* correct orientation is important. Current flow is measured in the direction indicated by the arrow enclosed in a circle which forms part of the probes symbol.

3. Place the mouse in the *Editing Window*, press down the left mouse button and drag the probe to the correct position. Release the mouse button.

You can place a probe directly on to an existing wire by placing it such that its connection point touches the wire. Alternatively you can place several probes in a free area of your design, and wire to them later.

When a probe is placed unconnected to any existing wires, it is given a default name of a question mark ('?') to indicate that it is unannotated. When the probe is connected to a net

(possibly when placing it if it is placed directly onto an existing wire) it is assigned the name of the net, or, if the net itself is unannotated, the component reference and pin name of the first pin connected to the net. The name of the probe will automatically be updated as it is unwired or as it is dragged from one net to another. You can assign your own name to the probe by editing the probe in which case the name becomes permanent and is not updated.

Probe Settings

The *Edit Probe* dialogue form allows two parameters to be adjusted:

LOAD Resistance

Voltage probes may be set to impose a load resistance on the schematic. This is useful where there would otherwise be no DC path to ground from the point being measured.

Record Filename

Both current and voltage probes can record data to a file which can then be played back using a *Tape Generator*. This feature allows you to create test waveforms using one circuit and then play them back into another.

 See *Tapes and Partitioning* on page 128 for more information.

USING SPICE MODELS

OVERVIEW

Many of the major component manufacturers now provide SPICE compatible simulator models for their product lines. Since PROSPICE is based on the original Berkeley SPICE code, you should have relatively few problems in using such models. However, there are a number of issues which you need to understand before attempting to use 3rd party models:

- A SPICE model is an ASCII netlist file. It contains no graphical information at all and cannot therefore be placed directly on a schematic. This means that you will need an ISIS library part for the device concerned as well as the SPICE model. Sadly, there is no standard file format whatsoever for schematic library parts, so you will usually need to draw this yourself.
- In a SPICE netlist, a model is called up by referring to a subcircuit using an X card. The binding of circuit nodes to model nodes is determined by the order in which circuit nodes are given on the X card. For example

```
XU1 46 43 32
```

means that node 46 of the circuit is connected to node 1 of the model. Similarly node 43 binds to node 2 and node 32 binds to node 3. Unfortunately, this scheme means that the model nodes are not named. Worse still, there is rarely any tie up between the model node numbers and the physical device pin numbers - for one thing, the physical package may have NC (not connected) pins, and these are hardly ever accounted for in numbering the nodes of a SPICE model. There can also be major complications with a multi-element device such as a TL074 - does the model implement one op-amp or four?

In practice, this means that some kind of explicit cross-referencing is needed to tell ISIS which SPICE model node number to use for each pin on the device, since neither the ISIS pin name nor the pin number can be used for the reasons given above. A special device property **SPICEPINS** has been introduced to make this as painless as possible.

- There are many variants of SPICE in existence. The lowest common denominator is SPICE 2 and most models appear to be written to this standard. However, a significant number of manufacturers declare their models to be compatible with PSPICE™, a proprietary version of SPICE 2 which has been developed extensively beyond the original standard. Whilst SPICE3F5 has many features not supported by PSPICE™, PSPICE™ has some primitive types that are different, and also uses a different syntax for some of the newer things that have been added to both products. The implication is, of course, that models specifically written for PSPICE™ may not work with PROSPICE.

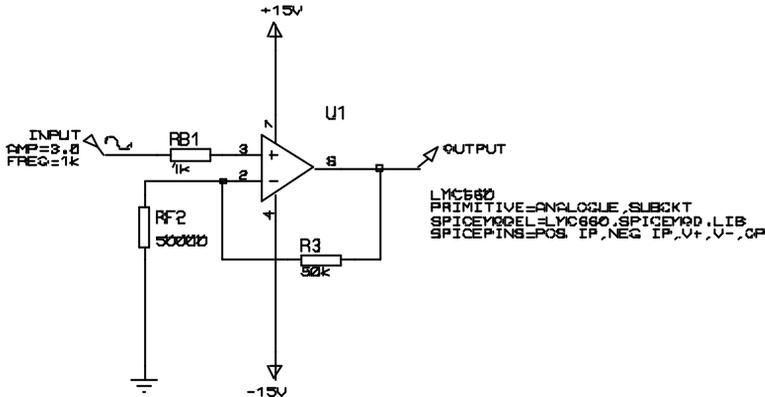
In short, if a 3rd party model does not work properly, the first thing to check is that it was actually written for SPICE 2 or SPICE 3.

The really good news is that we ourselves have gathered together a large number (well over 4000 at the time of writing) of manufacturers' models and created corresponding ISIS library parts for them.

You must, of course, appreciate that we can accept no responsibility for the accuracy or even functionality of these models, and that both ourselves and the manufacturers concerned disclaim any liability for losses of any kind whatsoever arising out of their use. In any case, there is never any guarantee that a circuit simulation will exactly reflect the performance of the real hardware and it is always advisable to build and test a physical prototype before entering into large scale production.

USING A SPICE SUBCIRCUIT (SUBCKT DEFINITION)

Our first example is an LMC660 OPAMP model held in the SPICE file SPICEMOD.LIB. This is provided as the file SPICE1.DSN in the samples directory and is shown below.



The op-amp component has been given three property assignments - **PRIMITIVE**, **SPICEMODEL** and **SPICEPINS**. Taking each of these in turn:

PRIMITIVE=ANALOGUE, SUBCKT

This assignment is the always the same for a component that is modelled by a SPICE subcircuit. It signifies that the component is to be simulated directly by SPICE3F5 and that it is represented by a SPICE subcircuit, rather than being an actual SPICE primitive.

SPICEMODEL=LMC660, SPICEMOD.LIB

This line indicates the name of the subcircuit to use, and the name of the ASCII file that contains the subcircuit definition. Alternatively you could use

```
SPICEMODEL=LMC660
SPICEFILE=SPICEMOD.LIB
```

Some manufacturers supply many models in one file, others supply one file per model. It matters not. The subcircuit name must match exactly that used in the model file, so if the subcircuit were called LMC660/NS you must include exactly this text in the **SPICEMODEL** property.

SPICE model files are searched for in the current directory, and on the *Module Path* as specified on the *Set Paths* dialogue form.

```
SPICEPINS=POS IP,NEG IP,V+,V-,OP
```

The **SPICEPINS** property deals with the issue of binding ISIS pin names to SPICE model node numbers. To understand how this property is used, you need to look at the comments in the original SPICE model file:

```
* ////////////////////////////////////////////////////////////////////
*LMC660AM/AI/C CMOS Quad OP-AMP MACRO-MODEL
* ////////////////////////////////////////////////////////////////////
*
* connections:          non-inverting input
*                       |   inverting input
*                       |   |   positive power supply
*                       |   |   |   negative power supply
*                       |   |   |   |   output
*                       |   |   |   |
*                       |   |   |   |
*                       |   |   |   |
* .SUBCKT LMC660        1   2   99  50  41
*
```

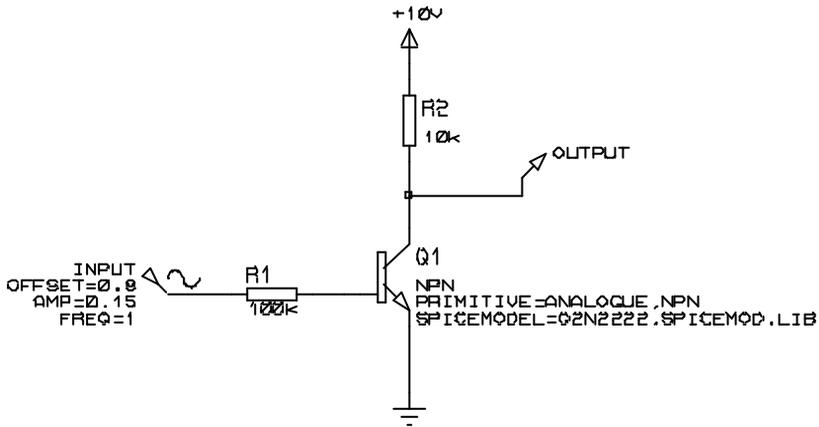
The ‘diagram’ lists the pin functions and their ordering on the SUBCKT line. It is important to realise that the *numbers* assigned relate to the internals of the model definition and are not in fact relevant to our purpose. The key information is that the non-inverting input is the first node, the inverting input the second and so on. This determines how you construct the value of the **SPICEPINS** property, in that you give the ISIS pin names in the corresponding order. Pin names with spaces are OK, but be careful not to leave spaces before the commas. Alternatively you can put each pin name in quotes - this is essential if you have a pin name that contains a comma.

You need, of course, to know the ISIS pin names which are often hidden. These can be established by moving the mouse over the pin ends of the component, and observing the status line.

USING A SPICE MODEL (MODEL CARD)

For semiconductor devices, specifically diodes, transistors, JFETs and MOSFETs, SPICE models are usually specified in terms of a single MODEL card. This lists the parameter values for the appropriate SPICE primitive. The procedure is similar to that for a SUBCKT model, but actually a little simpler because the issue of pin name translation does not arise. Additionally, ISIS library parts already exist for the various types of SPICE primitive – you will find them in ASIMMDLS.LIB.

Our second example, then, is a 2N2222 transistor, again held in the ASCII file SPICEMOD.LIB. This is provided as the file SPICE2.DSN in the samples directory.



The transistor has been given two property assignments - **PRIMITIVE** and **SPICEMODEL**.

Taking each of these in turn:

PRIMITIVE=ANALOGUE , NPN

This assignment indicates that the device is to be simulated directly by PROSPICE as an NPN type primitive. If you pick a diode, transistor, JFET or MOSFET from the ASIMMDLS library, it will have the appropriate **PRIMITIVE** assignment already in place.

SPICEMODEL=Q2N2222 , SPICEMOD.LIB

The SPICEMODEL assignment is essentially the same as for a SUBCKT model. It specifies the name of the model to use and the SPICE netlist file that contains it. Most manufacturers supply lots of MODEL definitions in a single file.

SPICE model files are searched for in the current directory, and on the *Module Path* as specified on the *Set Paths* dialogue form.

SPICE MODEL LIBRARIES

The 3rd party SPICE model libraries supplied with PROSPICE are contained in *binary* library files similar to ISIS device and symbol libraries. We have done this for two reasons:

- Many model files are extremely small and numerous and would waste a tremendous amount of space on hard disks with large cluster sizes if stored individually.
- When many models are contained in a single ASCII file, PROSPICE has to parse the whole file to use just one model and this is relatively slow.

Where a SPICE model is contained in a SPICE model library (SML), the syntax for binding it to an ISIS library part is slightly different. For example, the LMC660 in the first example would instead have the properties:

```
PRIMITIVE=ANALOGUE , SUBCKT           (as before)
SPICEMODEL=LMC660
SPICEPINS=POS IP , NEG IP , V+ , V- , OP   (as before)
SPICELIB=NATSEMI
```

The **SPICELIB** property gives the name of the SPICE model library file that contains the part. These files are searched for in the current directory, and on the *Module Path* as specified on the *Set Paths* dialogue form.

SPICE model libraries can be manipulated with the command line tools PUTSPICE.EXE and GETSPICE.EXE although we do not expect that ordinary users to create or gather sufficient numbers of models to warrant their use. Running either program without parameters will give usage information.

***SPICE SCRIPTS**

It is actually possible to type SPICE model definitions directly into ISIS and then call up the models from the relevant components on the schematic. For example, the transistor model for the 2N2222 could be entered into an ISIS script as follows:

```
*SCRIPT SPICE
.MODEL Q2N2222 NPN(IS=3.108E-15 XTI=3 EG=1.11 VAF=131.5 BF=300
NE=1.541
+ISE=190.7E-15 IKF=1.296 XTB=1.5 BR=6.18 NC=2 ISC=0 IKR=0 RC=1
+CJC=14.57E-12 VJC=.75 MJC=.3333 FC=.5 CJE=26.08E-12 VJE=.75
+MJE=.3333 TR=51.35E-9 TF=451E-12 ITF=.1 VTF=10 XTF=2)
*ENDSCRIPT
```

The transistor's **SPICEMODEL** property would then be changed to

```
SPICEMODEL=Q2N2222
```

i.e. no filename specified.

This feature might be most useful where a model is obtained on paper, or where you want to experiment interactively with the parameter values.

HANDLING MODEL SIMULATION FAILURES

Subcircuit models vary greatly in complexity and design. As a result, some models are easier to simulate than others. As a general rule, if PROSPICE could not simulate your circuit with its translated model, then not much else could either.

We have often found that problems are greatly worsened by poor circuit design. If you have an application note for the device, then compare it with your circuit to see if, for example, you are not allowing enough input current to bias the input stage successfully. The model designer almost certainly made the model work with circuits similar to those in an application note, rather than in a general case.

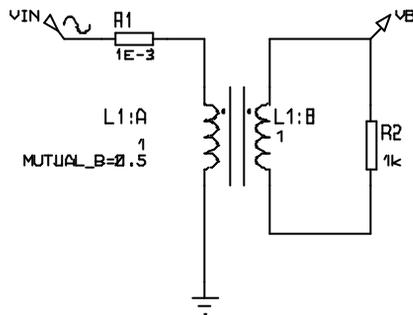
ProSPICE can be made more stable (but less accurate) by increasing the value of **GMIN**. The default is 1E-14, so good values to try are 1E-13 and 1E-12. At values like 1E-6, it is likely that any simulation results will bear no relation to reality at all, so pick the lowest value you can.

ADVANCED TOPICS

GROUND AND POWER RAILS

Why You Need a Ground

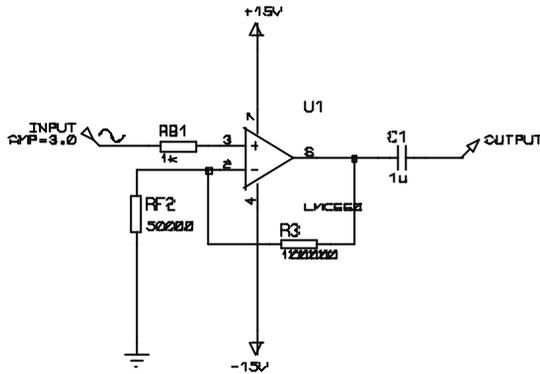
All simulations require a ground net be defined - otherwise, placing a probe on a particular net has no meaning as the voltage on that net must be defined in terms of a fixed reference point. In fact, there is a further requirement that all parts of the circuit must have a DC path to ground, as probing a point in the circuit that is attached to a floating network is equally meaningless. For example, in the circuit below, asking for the voltage at VB is meaningless



because the secondary side is floating. In theory, we could have provided two pinned probes (as with a real multimeter) but there are serious mathematical difficulties in solving circuits without a ground, and more to the point, Berkeley University have not addressed them in SPICE3F5.

The key point, then, is that all sections of your circuit must have a DC path to ground. The good news is that ProSPICE checks this for you, and will report as warnings any nets which it considers do not meet this criterion. In most cases, the simulation will then fail.

Another circuit configuration which has caused trouble in the past is shown overleaf:



Here, the presence of coupling capacitor C1 means that the output probe has no DC path to ground. Consequently, the simulator cannot resolve the operating point for the output, because the operating point is computed with all capacitors open circuit. We have chosen to resolve this by making capacitors very slightly leaky. Therefore, in the absence of any other DC path, the operating point at the output is computed with C1 fully discharged.

The leakyness of capacitors is determined by the simulator control property GLEAK, which is an admittance with default value 1E-12Mho. Setting this value to zero makes capacitors non leaky, as with traditional SPICE simulators.

Apart from the simulation of the innards of DRAM memory circuitry we cannot foresee any problems with this scheme, and it will save relative beginners from many strange error messages. In any case, real capacitors generally have leakage considerably more than a million megohms.

The ground net in a circuit can be defined either explicitly or implicitly. An explicit ground is defined by placing an unlabelled *Ground* terminal, as on the bottom end of RF2, above. You can also label a wire with the text **GND** if space is tight.

An implicit ground can be created through the use of a power rail, single pinned generator, loaded probe or a digital output, as well as by using any model that contains an internal grounded node.

Power Rails

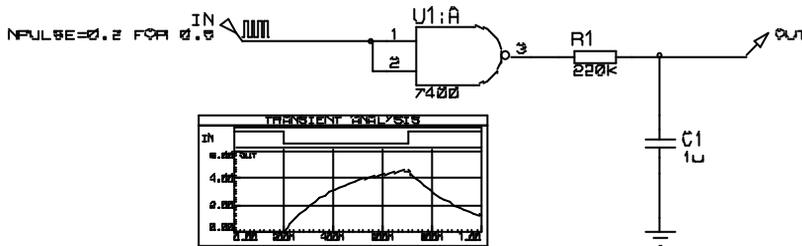
PROSPICE recognises a number of entities as power rails; the rules are as follows:

- Any net named GND or VSS is considered to be the ground reference. So far as PROSPICE is concerned, GND and VSS are aliases for the same thing.
- The net names VCC and VDD are considered to be the logic power rail, and will be treated as a logic '1' by DSIM. As with GND and VSS, PROTEUS VSM assumes these two names refer to the same net - if you really want split logic power supplies you must choose different net names.
- Power terminals with names of the form +5, -5 or +10V, -10V are taken as defining fixed power rails with reference to ground. The '+' and '-' symbols are crucial; a label such as 5.0V will not do.

Connecting two such labels with different values to the same net is an error.

The creation of a power rail also implicitly creates a ground.

An additional feature, added as part of the scheme for mixed mode simulation is that Logic IC models can be thought of as self powering. This enables you to draw circuits such as the one below and get sensible results, without having to draw in power supplies explicitly.



This works because the *Interface Model* defined for the 7400 includes a **VOLTAGE** property. This causes the creation of a 5V battery between the 7400's hidden power pins (VCC and GND) such that the VCC net acquires a potential of 5V. Other bits of circuitry connected to VCC, including the DAC object at the 7400 output then see this voltage.

 For a further discussion of Interface Models, see page 125.

INITIAL CONDITIONS

Overview

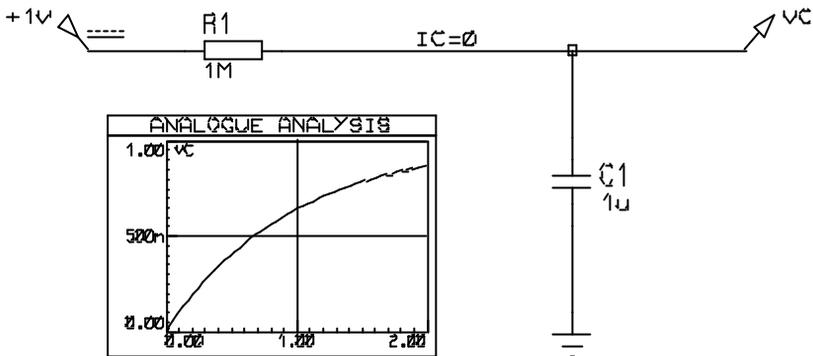
Whenever any type of simulation is performed, the first thing that PROSPICE does is to compute the operating point of the circuit - that is the steady state conditions that apply prior to any input signals being applied. There are two distinct modes of operation associated with computing these values:

- The operating point is found with all capacitors charged and inductors fluxed. In this case, a *Transient Analysis* will show the behaviour of the circuit as though power were applied sufficiently prior to time zero that by the start of the simulation everything had settled down. This is the default mode of operation, and given the ability to specify initial conditions for particular components and/or node voltages, serves for most purposes.
- The operating point is found with all capacitors short circuited and inductors open circuit. In this case, a *Transient Analysis* will show the behaviour of the circuit as though power were applied at time 0. This mode of operation can be selected by unchecking the *Compute Operating Point* option on the graph.

In either case, it is possible to specify the initial conditions for particular components or node voltages. This is especially useful for circuits which are oscillators, or for which circuit operation depends on particular capacitor being discharged at time zero. Indeed, the concept of a steady state operating point is meaningless for an oscillator, and the simulation may fail completely if some initial conditions are not given.

Specifying Initial Conditions for a Net

The easiest way to specify initial conditions is often to indicate the starting voltage for a particular net. In the circuit overleaf, this is achieved by adding a wire label with the text $IC=0$ to the probed net. Without this assignment, PROSPICE would compute the steady state value of the voltage on C1 i.e. 1 Volt and the graph would show VC as a horizontal straight line.



For nets which interconnect only digital components, you should use logic states for initial conditions - e.g. **1,0,H,L,HIGH,LOW,SHI,WHI,SLO,WLO** or **FLT** and assign them to the **BS** (Boot State) property. For mixed nets, you should specify an initial voltage - this will be automatically propagated as a logic level to the digital components.

Specifying Initial Conditions for Components

This option is only available when PROSPICE does not compute the initial operating point - i.e. when the *Initial DC Solution* option on the graph is not checked. In this case, all node voltages will be zero at time zero, except for nets with initial condition properties as described above. Under these conditions, it is then possible to specify initial conditions for particular components. For example, you could add the property

IC=1

to C1 in the previous example such that C1 started from its steady state value of 1 Volt.

Details of the IC properties supported by the various SPICE primitives are given in the chapter on modelling.

In some ways it is a shame that this option is not available when the operating point *is* computed, but this is how SPICE3F5 has been coded by Berkeley. However, we have added the **PRECHARGE** property as a remedy to this problem.

NS (NODESET) Properties

Occasionally, when a simulation fails to find the operating point, it can be useful to give SPICE a hint as to the initial values to use for particular nets. This is different from setting the initial conditions in that the value given is only used for the first iteration, and the net then

'floats' to whatever value the matrix solution converges at. It is, then, an aid to convergence and should not affect the actual operating point solution.

Such convergence hints can be specified using the **NS** net property, so placing a wire label with the text NS=10 will suggest a starting value of 10 Volts for that net.

PRECHARGE Properties

A further option for specifying the initial conditions is the **PRECHARGE** property. This may be assigned to any capacitor or inductor in the circuit and specifies either voltage across the device or the current flowing through it respectively.

The **PRECHARGE** property is a Labcenter specific addition to SPICE, and differs from the IC property in that it is applied irrespective of whether the *Initial DC Solution* option is selected or otherwise.

TEMPERATURE MODELLING

The SPICE3F5 kernel provides extensive support for modelling the effects of temperature. The scheme operates as follows:

- There is a global property **TEMP**, which will, if nothing else is done, apply to all components in the circuit.
- The temperature of individual components can be specified by giving them their own **TEMP** property.
- Where parameters for a device model vary according to the temperature at which they were measured, this temperature can be specified globally and/or individually by **TNOM** properties.

Temperature modelling is provided for resistors, diodes, JFETs, MESFETs, BJTs, and level 1, 2, and 3 MOSFETs. BSIM models are expected to have been created for specific temperatures. No temperature dependency is implemented for digital primitives.

In addition, it is very important to realize that most equivalent circuit models for ICs will not model temperature effects correctly. This is because such models usually use ideal controlled sources and other macro modelling primitives, rather than containing an exact replica of the device internals. Once such primitives are used, it is unlikely that the model will show correct temperature dependent behaviour unless someone has taken the trouble to make it do so.

THE DIGITAL SIMULATION PARADIGM

Nine State Model

You might think that a digital simulator would model just high and low states, but in fact DSIM models a total of nine distinct states:

State Type	Keyword	Description
Power High	PHI	Logic 1 power rail.
Strong High	SHI	Logic 1 active output.
Weak High	WHI	Logic 1 passive output.
Floating	FLT	Floating output - high-impedance.
Undefined	WUD	Mid voltage from analogue source.
Contention	CON	Mid voltage from digital conflict.
Weak Low	WLO	Logic 0 passive output.
Strong Low	SLO	Logic 0 active output.
Power Low	PLO	Logic 0 power rail.

Essentially, a given state contains information about its polarity - high, low or mid-way -and its strength. Strength is a measure of the amount of current the output can source or sink and becomes relevant if two or more outputs are connected to the same net.

For example, if an open-collector output is wired through a resistor to VCC, then when the output is pulling low, both a *Weak High* and a *Strong Low* are applied to the net. The *Strong Low* wins, and the net goes low. On the other hand, if two tristate outputs both go active onto a net, and drive in opposite directions, neither output wins and the result is a *Contention* state.

This scheme permits DSIM to simulate circuits with open-collector or open-emitter outputs and pull up resistors, and also circuits in which tristate outputs oppose each other through resistors - a kind of poor man's multiplexer if you like. However, it is important to remember that DSIM is a digital simulator only and cannot model behaviour which becomes decidedly analogue. For example, connecting overly large resistors up to TTL inputs would work OK in DSIM but would fail in practice due to insufficient current being drawn from the inputs.

The Undefined State

Where an input to a digital model is undefined, this is propagated through the model according to what might be described as common sense rules. For example, if an AND gate has an input low, then the output will be low, but if all but one input is high, and that input is undefined then the output will be undefined.

- Edge triggered devices require a transition from a positively defined logic 0 to logic 1 (or vice versa) for an edge to be detected. Transitions from 0 or 1 to undefined and back are not counted as edges.
- The more complex sequential logic devices (counters, latches etc.) will often undefined inputs to a logic 0 or logic 1, according to the coding of their internal logic. This is not unlike real life!

Floating Input Behaviour

It is common, if not altogether sound practice to rely on the fact that unconnected TTL inputs behave as though connected to a logic 1. This situation can arise both as result of omitted wiring, and also if an input is connected to an inactive tristate output. DSIM has to do *something* in these situations since the internal models assume true logical behaviour with inputs expected to be either high or low.

This is catered for through the **FLOAT** property. This may be assigned either directly to a component, or an an *Interface Model*. In particular, the interface model for TTL parts has the assignment:

```
FLOAT=HIGH
```

which causes floating inputs to be interpreted as logic 1 levels.

To specify that floating inputs be taken as logic 0, use

```
FLOAT=LOW
```

Otherwise, floating inputs will be taken to be at the *Undefined* state. See page 121 for more information.

Glitch Handling

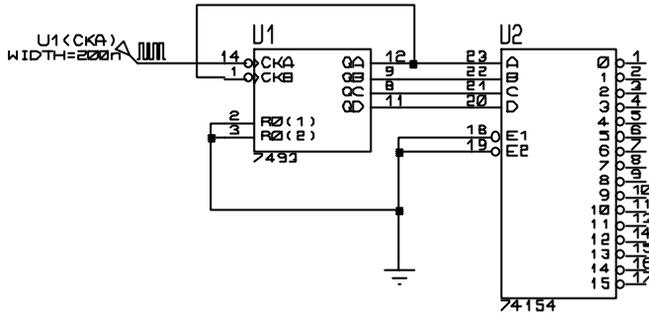
In designing DSIM we debated at great length how it should handle the simulation of models subjected to very short pulses. The fundamental problem is that, under these conditions, a major assumption of the DSIM paradigm - that the models behave purely digitally - starts to break down. For example, a real 7400 subjected to a 5ns input pulse will generate some sort of pulse on its output, but not one that meets the logic level specifications for TTL. Whether such an output pulse would clock a following counter is then a matter dependent on very much analogue phenomena.

The best one can do is to consider the extremes, namely:

- A 1ns input pulse will not propagate at all.
- A 20ns pulse will come through nicely.

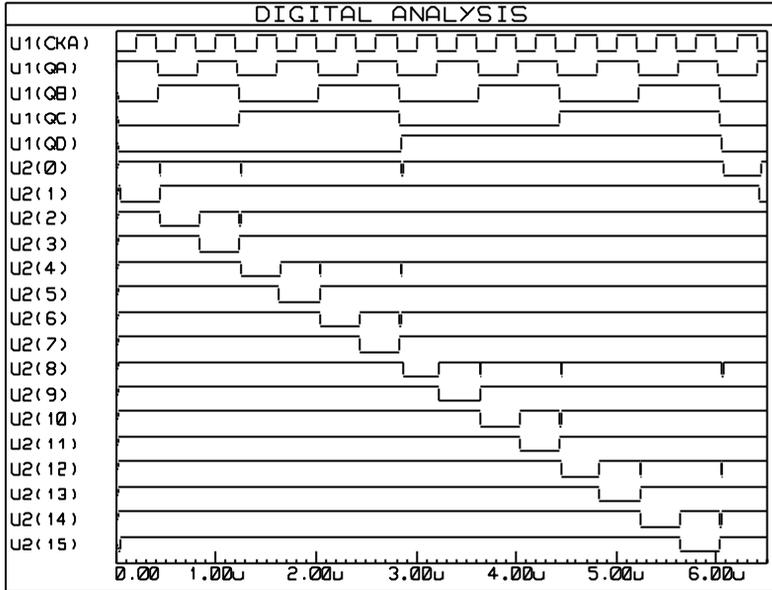
Somewhere in between, the gate will cease to propagate pulses properly and could be said to *suppress* glitches. This gives us the concept of a *Glitch Threshold Time*, which can be an additional property of the model along with the usual **TDLH** and **TDHL**.

Another subtlety concerns whether the glitch is suppressed at the input or the output of a model. To resolve this, consider a 4-16 decoder driven from a ripple counter as shown below:



The outputs of the ripple counter are staggered, and thus the possibility arises of the decoder generating spurious pulses as the inputs pass through intermediate states. This situation is shown on the graph, overleaf.

Taking the first glitch an example of the phenomenon, as U1(QA) falls for the first time, it beats the rise of U1(QB) and an intermediate input state of 0 is passed to the decoder for approximately 10ns. The question is whether the decoder can actually respond to this or not, and even more to the point, what would happen if the input stagger was only 1ns or 1ps? Clearly, in the last two cases the real device would not respond, and this tells us that we must handle glitches on the outputs rather than the inputs. This is because, in the above example, the input *pulses* are all relatively long and would *not* be considered glitches by any sensible criteria. Certain rival products make a terrible mess of this, and will predict a response even in the 1ps case!



The above graph was produced with TGQ=0 for the 74154

The really interesting part of this tale is that, if you build the above circuit, it will probably not glitch. It is very bad design certainly, but the **TDLH** and **TDHL** of the '154 are around 22ns and this makes it a tall order for it to respond to a 10ns input condition. With the individual components we tried, no output pulses, other than perhaps the slightest twitches off the supply rails, were measurable.

To provide control over glitch handling, all the DSIM primitives offer user definable *Glitch Threshold Time* properties named **TGxx**, where **xx** is the name of the relevant output. Our TTL models are defined such that these properties can be overridden on the TTL components, and the values are then defaulted such that the *Glitch Threshold Times* are the average of the main low-high and high-low propagation delays. Setting the *Glitch Threshold Times* to zero will allow all glitches through, should you prefer this behaviour. The graph, above, was thus created by attaching the property assignment **TGQ=0** to the 74154.

Finally, it is important to point out that if the *Glitch Threshold Time* is greater than either of the low-high or high-low propagation delays, then the *Glitch Threshold Time* will be ignored. This is because, after an input edge, and once the relevant time delay has elapsed, the gate output *must* change its output - it cannot look into the future and see whether another input event (that might cancel the output) is coming. Consider a symmetric gate with a propagation delay of 10ns and a *Glitch Threshold Time* of 20ns. At t=0ns the input goes high and t=15ns the input goes low. You might expect this to propagate, with the output going high at t=10ns

and low again at $t=25\text{ns}$, so producing a pulse of width 15ns which would be suppressed, since it is less than the *Glitch Threshold Time*. The reason the pulse is not suppressed is that, at $t=10\text{ns}$, the output *must* go high - it cannot remain low for a further 20ns on the off chance (as in our example) a second edge comes along so producing an output pulse it would need to suppress! Once the output has gone high at $t=10\text{ns}$ then the second edge (at $t=25\text{ns}$) is free to reset it. You will need to think carefully about this to understand it.

MIXED MODE INTERFACE MODELS (ITFMOD)

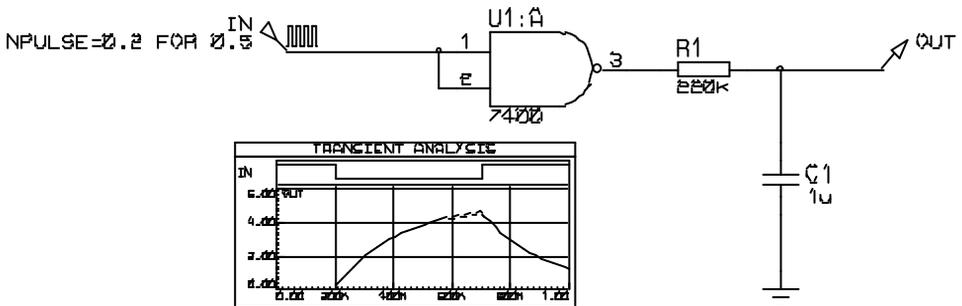
Overview

In designing our scheme for mixed mode simulation within PROSPICE, we gave considerable thought to the problem of how to specify the analogue characteristics of a digital device family. These characteristics include:

- The input and output impedances of the devices.
- The logic thresholds of device inputs.
- The voltage levels for high and low outputs.
- The rise and fall times of device outputs.
- The default logic state for floating inputs.

A scheme which involved specifying all these parameters for every device in the TTL libraries, say, would be extremely unwieldy.

In addition, a significant problem arises (for beginners, at least) in the specification of power supplies - there is a tendency to plonk down a circuit such as the one below and expect sensible results. The problem here, of course, is an implicit assumption that the 7400 has a 5V power rail obtained from its hidden power pins which connect to VCC/GND.



All these problems are solved by the introduction of the **ITFMOD** component property. This is very similar to the **MODEL** property in that it provides a reference to a set of property values but it also activates a special mechanism within the netlist compiler. Essentially this works as follows:

- For any device that has an **ITFMOD** property an additional model definition is called up during netlisting that will specify control parameters for ADC and DAC objects, and also the pin names of the positive and negative power supplies. In the above circuit, U1:A will have `ITFMOD=TTL`.
- Having obtained the names of the power supply pins (VCC, GND in this case), ISIS creates a special primitive and connects it across the power supply pins. ISIS names this object similarly to an object on the child sheet or model, so that in the above circuit, the power supply object will be called U1:A_#P.
- When PROSPICE simulates a mixed mode circuit, it creates ADC and DAC objects and considers them to 'belong' to the objects to which they connect. In the case of the circuit above, a DAC object will be created with the name U1:A_DAC#0000 because it forms the interface from U1:A's output.

The clever part is that on doing this, it also looks for a power supply interface object with the same name stem i.e. U1:A, and finds U1:A_#P. It then instructs U1:A_DAC#0000 to take its properties from U1:A_#P which in turn has inherited its properties from the model specified in the original ITFMOD assignment. Thus the DAC object operates with parameters defined for the TTL logic family.

- Each power interface object also contains a battery which will be assigned to the **VOLTAGE** property given in the interface model definition. The TTL interface model definition specifies `VOLTAGE=5V`.

This means that in the above circuit, a 5V battery gets inserted between VCC and GND, because these are the nets indicated by the power pins of the 7400 device.

- The batteries have a small internal impedance (1miliohm). This means that if you assign a real power rail to VCC/VDD (by placing a power terminal or voltage source) then this will override the level defined by the internal batteries - in the world of simulation, a large current flow through the batteries does not matter!

Using ITFMOD Properties

Existing interface models have been defined as follows:

TTL	Standard TTL (74 series)
TTLSS	Low power Schottky TTL (74LS series)
TTLS	Standard Schottky TTL(74S series)
TTLHC	High Speed CMOS TTL (74HC series)
TTLHCT	High Speed CMOS TTL with TTL outputs (74HCT series)
CMOS	4000 series CMOS.
NMOS	Microprocessor type MOS circuits.
PLD	PLD type MOS circuits.

It follows that any new digital model can be assigned a device family by adding a property such as

```
ITFMOD=TTL
```

The family definitions are held in the file ITFMOD.MDF which is kept in the models directory.

Each definition can contain any or all of the properties defined for the ADC and DAC interface primitives. In addition, the following may be given:

V+	-	Name of the positive power supply pin.
V-	-	Name of negative power supply pin.
VOLTAGE	5V	Specifies the default operating voltage.
RINT	1mΩ	Specifies the impedance of the internal battery. A value of zero will disable the battery.
FLOAT	-	Specifies HIGH or LOW value for floating inputs.

Finally, it is worth pointing out that any specific property e.g. TRISE, can be overridden on the parent device, so if you want simulate a 4000 series IC with a slow rise time, you could add TRISE=10u directly to its property list.

PERSISTENT MODEL DATA

Certain simulator models, particular those for EPROMs, and micro-processors containing EEPROM or flash memory are able to ‘remember’ data between simulation runs, just like their real life counterparts. This action is facilitated by the **MODDATA** property. Persistent model data blocks are stored in the DSN file, and so do not persist between PROTEUS sessions unless you save the design.

To reset the persistent model data to its initial condition, use the *Reset Persistent Model Data* command on the *Debug* menu.

TAPES AND PARTITIONING

Overview

A unique feature of the VSM system is its ability to divide a large design into one or more sections or *partitions* and simulate each individually.

There are two principal advantages associated with this:

- In a fully integrated CAD exercise, the circuit for the whole design of the product will contain some sections which you may not wish to or cannot simulate. To prevent the need for carving up the design, some mechanism is required for determining which components in the design are actually relevant to a given simulation experiment.

ISIS does this by considering the points being measured and, further back down the design, the points being driven by test sources and/or power rails.

- If the design consists of several stages, it will be a common requirement to see how later stages perform when driven by the outputs of earlier stages. Whilst this could be achieved by simulating all the stages together, this would tend to be unduly slow. Partitioning allows the results from simulating previous stages to be captured in tapes and played back as the input for subsequent stages.

ISIS contains logic to do this either under manual control or automatically. In the later case, it detects when the circuitry within given partitions has changed and re-simulates only those which have changed, or which are affected by those which have changed.

Single Partition Operation

Many simulation experiments will consist of testing a single section of a design in isolation from the rest. This will generally involve injecting a signal or signals at the input(s) of the section, and then monitoring its operation at points throughout it.

It should be self evident that signals can be injected by placing generator gadgets on appropriate input wires, and that circuit operation can be monitored by placing probes. However, these actions by themselves do not isolate the section under test from the rest of the design.

To do this, you need to set the *Isolate Before* checkboxes on the generators and the *Isolate After* checkboxes on the probes. Once this is done, only the section between the isolation points will be netlisted and simulated.

To work out which section to simulate, ISIS looks at the probes that have been assigned to the graph, and propagates through components and wires (including inter-terminal connections) outwards until one of the following conditions is met:

- There are no more components to process.
- An isolating probe or generator is reached.
- A tape input or output is reached.
- A power rail is reached. A net is made into a power rail by connected a POWER or GROUND terminal to it. A GND or VCC wire label will NOT do here.

Tape Objects

Tape objects have two distinct uses with the system:

- To define points at which it is reasonable to ignore circuitry to the right of when simulating circuitry to left. Such points are generally where low impedance outputs drive high impedance inputs. This can also be achieved by placing isolating probes.
- To provide the means to capture the output of one stage of a design, and use it to drive the next stage but without the overhead of simulating the first stage.

To Place a Tape:

1. Select the *Gadget Mode* icon and then the *Tape* icon.
2. Use the *Rotation* and *Mirror* icons to orient the tape according to how you want to place it. The little arrow symbol within the tape body designates the direction of the tape; in its normal orientation, the input is on the left and the output is on the right.
3. Place the mouse in the *Editing Window*, press down the left mouse button and drag the tape to the correct position. Release the mouse button.

You can place a tape directly on to an existing wire by placing it such that its connection point touches the wire. Alternatively you can place several tapes in a free area of your design, and wire to them later.

It is vital to place tapes only in sensible locations - that is where low impedance drives high. If you place tapes in other locations, you will change the operation of your circuit and invalidate any results that may then be produced.

Tape Modes

To give the maximum degree of user control, tapes have three modes:- AUTO, PLAY and RECORD. In the following discussions, we use the terms left and right in terms of the logical dependency of the partition tree, reading the design input⇒output, left⇒right.

AUTO mode

This is the default scheme and defines a mode of operation in which ISIS decides which partitions need re-simulating, and which can use previously stored data. For most situations that require the use of tapes, AUTO mode will prove to be the most useful.

This automatic detection is based on considering all the text that occurs within the sub-netlist related to the partition. It follows that if any part names, values, properties, wiring etc. in that partition is changed, a re-simulation will be performed unless a partition file already exists for that set of information.

Note also that ALL models, scripts, design global properties and so forth are included in the sub-netlist, so changing any such object will cause a re-simulation of all automatic partitions. ISIS does not delve into the question of whether particular models, scripts etc. are used by components in a particular partition. If you need to overcome this, you can use the manual RECORD and PLAY modes.

Note that a TAPE object in auto mode will be removed from the circuit if an interactive simulation is performed.

PLAY mode

This mode enables you to play a named file that you have previously recorded, either with a tape or by adding a **RECORD** property to a probe. The filename of the data to be played should be entered into the *Filename* field of the tape; you cannot use PLAY mode unless a filename has been entered.

When a tape is in PLAY mode, the circuitry to its left is disconnected and ignored, unless it includes probes which are also included on the current graph.

Do bear in mind that you can only play data that has been recorded for the type of analysis currently being performed. Attempts to do otherwise will result in an error.

RECORD mode

This mode causes the data present at the input of the tape to be recorded into the file named into its *Filename* field; you cannot use RECORD mode unless a filename has been entered.

Another effect of record mode is to force the partition to the tape's left to be re-simulated, irrespective of whether it has changed or not. If there is a partition to the right of the tape that is probed, then this will also be re-simulated since it will be deemed to depend on the one to its left.

The RECORD mode for tapes is probably most useful in situations in which you want to record a waveform once, and then lock it as the input for further experiments on the right hand side of the tape.

SIMULATOR CONTROL PROPERTIES

Overview

There are a large number of parameters that affect the details of how a simulation is performed. These include things such as the maximum number of iterations allowed to find the operating point, the tolerances that determine when a solution has converged, the integration method used and so forth.

These options are common to all types of analysis and can be adjusted separately for each graph by editing it and clicking the *SPICE Options* button.

Tolerance Properties

This group of parameters determine how accurately SPICE will compute the solution. Higher accuracy is generally achieved at the expense of longer simulation times, and in some circumstance the circuit may fail to converge at all if you ask for too high a set of tolerances.

The most useful value here is the *Truncation Error Estimation* factor, or **TRTOL** in traditional SPICE nomenclature. If you have results which appear overly spiky, or suffer from overshoots you should try a lower value here.

The minimum conductance value, **GMIN**, defines the leakage of reverse biased semiconductor junctions and other theoretical points of infinite impedance. Reducing this value may help achieve convergence for circuits which fail to simulate although this may be at the expense of simulation accuracy. See page 139 for more information about convergence issues.

The leakage conductance, **GLEAK** is something we have introduced to enable solution of circuits with DC blocking capacitors. It defines the DC leakage of capacitors, and can generally be left alone unless you are simulating something unusual such as CMOS memory cells or the like.

Mosfet Properties

SPICE simulation of MOSFETs is based on the assumption that you are doing IC design, and consequently implements a scheme for scaleable geometries. In practice this means that there are a number of parameters which determine the default physical sizes for elements of the MOSFET devices. These are values are defined here.

In addition, some models have been created which rely on the behaviour of older versions of SPICE, and this MOSFET behaviour can be switched on or off here if you are using older MOSFET models.

Iteration Properties

The properties on the *Iteration* tab determine how SPICE deals with circuits that are hard to converge.

The integration method can be either *Gear* or *Trapezoidal*. The latter is provided mainly for backward compatibility with previous versions of SPICE since *Gear* integration generally gives more accurate results for a given number of time-steps. In *Gear* integration, high orders than 2 are possible; this involves SPICE using more of the past history of a time-point in predicting what happens at the next time-point.

When SPICE fails to get a convergent solution at the operating point, it tries two approaches: *Gmin Stepping* and *Source Stepping*. The number of steps tried in each method can be set here.

Three further options determine that maximum number of iterations that will be tried at each of the operating point, a step in a *Transfer* analysis, and a time-point in a *Transient Analysis*. Increasing these determines may help in getting a result out of a hard or near unstable circuit.

Finally, two options are given that may result in faster simulations. LTRA compaction applies only for circuits incorporating lossy transmission lines (LOSSYLINE model). The idea is that near identical values in the data pipeline get discarded so that this data points are processed. Bypassing unchanging elements is a general optimization that saves SPICE from re-computing the values of semiconductor devices whose node voltages have not changed since the last evaluation.

Temperature Properties

There are two global temperature properties: **TEMP** - the default operating temperature, and **TNOM** the parameter measurement temperature. **TEMP** defines the actual temperature of the circuit, whilst **TNOM** is the value at which temperature dependent device parameters are taken to have been measurement. For further discussion of temperature modelling in PROSPICE see page 120.

Digital Simulator Properties

TDSCALE, TDSEED, TDLOWER and TDUPPER

The **TDSCALE** variable is used to control the scaling of all timing properties used by models in the simulation run that have not been explicitly referenced by the model as non-scaleable. The **TDSCALE** variable can be assigned either a constant floating-point value or the keyword **RANDOM**.

If a constant value is specified, then all timing properties specified in models used in the simulation are multiplied by the value; a value less than 1.0 reduces the timing properties whilst a value greater than 1.0 increases them. For example, the variable assignment:

```
TDSCALE = 1.1
```

has the affect of extending all timing properties used in the simulation by 10%. The default value for **TDSCALE** is the constant value 1.0; this causes all timing properties to be used unmodified.

If the **TDSCALE** variable is assigned the keyword **RANDOM**, the DSIM engine will randomly scale each timing property by multiplying it by a random floating-point time-scaling value. The range of time-scaling values chosen by the simulation engine can be limited by assigning the **TDLOWER** and **TDUPPER** variables; these define the lowest and highest random floating-point scale values allowed respectively. The default values for **TDLOWER** and **TDUPPER** are 0.9 and 1.1 respectively which has the affect of limiting random scaling to within $\pm 10\%$.

The sequence of random values generated is said to be *pseudo-random* - that is, each successive value generated, whilst appearing to be random, is in fact determined by the previous value (and a complex formula). It follows that any sequence of random values is determined by a *seed* value and that for a given *seed* value the sequence of random values generated is always the same. The **TDSEED** property allows you to specify a seed value for the generation of random time-scaling values and thus guarantee that the sequence chosen from one simulation run to another is always the same. This avoids the problem of a timing design error based on random propagation delays appearing and then disappearing on successive simulation runs.

The **TDSEED** variable should only be assigned positive integer values in the range 1-32767. The default value for **TDSEED** is itself a random value (based on the date and time) and this provides a means for randomising the sequence of values generated from one simulation run to the next.

For example, the assignments:

```
TDSCALE = RANDOM
TDLOWER = 2.00
TDUPPER = 3.00
TDSEED  = 723
```

have the affect of randomly increasing all timing properties by approximately 200-300% with a random sequence of scaling values. The seed value of 723 causes the same sequence of pseudo-random values to be generated for each simulation run.

INITSEED

The **INITSEED** property is used to seed the random initialisation value generator, used by DSIM primitive models whose initialisation properties have been assigned the **RANDOM** keyword.

As with the **TDSCALE** and **TDSEED** properties described above, whilst the sequence of values generated by the random initialisation value generator are random, the sequence as a

whole is finite and determinate. The **INITSEED** property provides a means of selecting which sequence of random values is used by the DSIM simulator.

The **INITSEED** property should only be assigned positive integer value in the range 1-32767. The default value for **INITSEED** is itself a random value (based on the date and time) and this provides a means for randomising the sequence of values generated from one simulation run to the next.

TYPES OF SIMULATOR MODEL

How to tell if a Component Has a Model

PROTEUS is supplied with over 8000 library parts of which about 6000 have simulator models. The devices which do not have models are perfectly relevant for use in PCB design, and the notion that every part should have a model is quite untenable. It would imply, amongst other things, that we should create a model for the 68020 processor which is not exactly a trivial job.

So, for the purposes of circuit simulation, you need to be able to tell if a component you are using has a simulator model. In the first instance, an attempt at simulation will fail if it doesn't, generally with a message something like:

```
ERROR [PSM] : No model specified for 'U1'.
```

The error is detected in the partitioning (PSM) phase, because up until this point, it may be that the un-modelled component is not actually in the part of the circuit being simulated.

Occasionally you will get:

```
ERROR [U1] : Value '74F00' of VALUE not found in  
parameter mapping table.
```

This means that there is a model file for the device, but that you have changed the value to a part type that is not modelled by the MDF file. In the above example, we have changed the value of a 74LS00 gate (which is modelled) to a 74F00 gate, which isn't.

The type of simulator model (if any) available for a component is displayed at the top left of the preview window in the device library browser. For example, if you open the library browser and select the 74LS00 device in the 74LS library, you will see

```
Schematic Model [74NAND.MDF]
```

Further information about the various types of models is given in the following sections.

Primitive Models

A significant number of basic component types are built directly into PROSPICE. These device types are called *Primitives* and include resistors, capacitors, diodes, transistors, gates, counters, latches, memories and many more.

Primitive models require no extra files to be simulated, and are identified by the presence of a single PRIMITIVE property. Additional properties are specified directly within the component and are passed to PROSPICE via the netlist. For example, a resistor will have:

```
PRIMITIVE=ANALOG,RESISTOR
```

This identifies the part as a SPICE Resistor primitive.

The standard set of simulator primitives may be found in the ASIMMDLS and DSIMMDLS libraries. These parts provide context sensitive help for their properties, and examples of their use may be found in the VSM SDK Documentation.

Schematic Models

Where a more complex device is to be simulated, a common approach is to draw a circuit that mimics its action using simulator primitives. This circuit can be the actual internal electronics of the device, but more commonly will utilize ideal current sources, voltage sources and switches to speed things along.

A schematic model is specified with the **MODFILE** property, which by convention we make a read only property. For example, the 741 op-amp has the assignment:

```
MODFILE=OA_BIP
```

You will note from this, that the model file is able to model several devices – a feat which it achieves using a *Parameter Mapping Table*.

Schematic models are created by drawing a schematic in ISIS, and then compiling it with the *Model Compiler* to produce an MDF file. Further details of this process are provided in the VSM SDK documentation.

VSM Models

VSM models are really primitive models which are implemented in external DLLs as opposed to inside PROSPICE itself. They provide the means to simulate device functionality using the programming language of your choice, although it is generally easiest to use C++.

A VSM model will have both a PRIMITIVE property (because both ISIS and PROSPICE treat it as a primitive) and a MODDLL property which specifies the name of the DLL file in which the model's code resides.

For example, the 8052 model has

```
PRIMITIVE=DIGITAL,8052
MODDLL=MCS8051
```

Note that a model DLL can implement more than one primitive type – MCS8051.DLL implements a number of 8051 variants.

VSM models can also implement functionality that relates to animation, such that the electrical and graphical aspects of a component's operation can be combined in fairly astounding ways. The LCD display model is a good example of this.

Creating VSM models revolves around a number of C++ Interface classes (similar to COM). These are documented in the VSM SDK manual.

SPICE Models

Since PROSPICE is based on Berkeley SPICE3F5 it is directly compatible with standard SPICE models, and many of the components in the PROTEUS libraries are modelled using SPICE files obtained from component manufacturers. SPICE models can be specified either by SUBCKT blocks or by sets of parameters in a MODEL record. SUBCKT models will have the property assignments such as

```
PRIMITIVE=ANALOG,SUBCKT
SPICEMODEL=CA3140
```

whereas SPICE primitive model for a transistor might have the properties:

```
PRIMITIVE=ANALOG,NPN
SPICEMODEL=BC108
```

The model itself can be stored either in an ASCII file, or in a SPICE Model Library. The name of these files is specified by a either **SPICEFILE** or **SPICELIB** property.

Extensive detail on how to make use of manufacturers SPICE models is given in the chapter entitled USING SPICEMODELS on page 109.

TROUBLESHOOTING

THE SIMULATION LOG

Whenever a simulation is performed a *Simulation Log* file is created. The simulation log file contains all information, warning and error messages from both the simulator itself and from individual models. Where a message originates from a model, it is prefixed with the models component reference in square brackets, so you might see:

```
[U1] Loaded 26 files from PROGRAM.HEX
```

During an interactive simulation, the contents of this file can be displayed in a popup window from the *Debug* menu whilst for a graph based simulation, the file can be viewed by pointing at the graph and pressing CTRL+V.

In the case where the simulation fails completely, the log file will be displayed automatically in order that you can see the cause of the problem immediately.

NETLISTING ERRORS

Netlisting errors occur as a result of a problem when ISIS tries to create a netlist from the schematic - you will also encounter these if you try to export the schematic to ARES for PCB layout. Common ones are:

- Having two components with the same name, or unnamed components e.g. two resistors labelled R?.
- Badly formed script files such as **MAP ON** tables etc. Refer to the syntax definitions in the ISIS manual if you can't spot the problem immediately.

LINKING ERRORS

Model linking is the process whereby ISIS calls up MDF files for components which are modelled by equivalent circuits. By far the most common problem is where the specified model file does not exist. The model file must be in the current directory, or in the *Module Path* as set on the *Set Paths* dialogue form.

Other common linker errors include:

- Value not found in parameter mapping table. This means that the part type - e.g. CA3140 is not listed in the mapping table of the specified model file. Model files such as OA_MOS.MDF are design to model several different components using the same circuit but with different values. This error means that the model file specified does not have

parameters for the device type you are using - you can edit the MDF file with a text editor to see which devices are modelled.

- Unresolved module pin. This is a warning rather than an error, and means that the parent component body has a pin which is not present in the model. This is often OK - for example, most op-amp models do not model the offset null pins, but it can be a mistake, and if a new model doesn't work this is a common cause - especially if there are also 'No DC path to Ground' warnings.

PARTITIONING ERRORS

Partitioning is the mechanism by which ISIS decides which part(s) of the circuit need to be simulated. Problems that can occur here are:

- Cyclic dependencies detected. This means that an arrangement of tapes is such that the partitions behind and in front of a tape are mutually dependent. Assuming that you have correctly set any *Isolate Before* and *Isolate After* flags on probes and generators your simplest action here will probably be to delete the tape object(s) and simulate the whole circuit in one go.
- No model specified - this means that the component indicated does not have a `PRIMITIVE`, `MODFILE` property, and has appeared in that part of the circuit which needs to be simulated. If the device is irrelevant (e.g. a connector) then you must specify `PRIMITIVE=NULLL`, otherwise you need a model!

See page 134 for more information about simulator model types.

SIMULATION ERRORS

Simulation errors are generated by PROSPICE rather than ISIS, and therefore occur after a netlist file has been successfully generated. Common problems that get detected here include:

- Device type not recognized. This means you have specified a primitive type that is not supported, or that a model file has used one.
- No DC path to ground. This is discussed further under `GROUND AND POWER RAILS` on page 115.
- Could not find probe - you have tried to reference a probe or voltage generator that does not exist. Remember that you *must* use an `Iprobe` object from `ASIMMDLS.LIB` - you cannot reference a current probe gadget.
- Cannot open SPICE source file. The source file specified in a `SPICEMODEL` property cannot be located. It should be in the current directory or in the *Module Path* as set on the *Set Paths* dialog box.

- Cannot find library model. The SUBCKT or MODEL you have specified does not exist in the specified library or on disk.
- Model DLL not found. The specified VSM model DLL cannot be located. It should be in the current directory or in the *Module Path* as set on the *Set Paths* dialogue form.

CONVERGENCE PROBLEMS

This final set of errors relate to what happens if SPICE itself fails to simulate the circuit. There are basically three error messages that indicate this:

- Singular matrix. This is akin to have more unknowns than equations and usually relates to circuits which are mis-drawn, or in which some initial conditions need to be given in order to define the starting state.

This error will often be preceded by “No DC path to ground” warnings, and you need to investigate the wiring around the pins listed after this message. If part of your circuit is not grounded, the simulator cannot resolve its voltage relative to ground - it’s as simple as that.

- To many iterations without convergence. This means that circuit solution is unstable. Circuits with VSWITCH or CSWITCH primitives can create this condition easily, but any circuit whose transfer function is discontinuous can give SPICE serious problems.
- Timestep too small. This means that the circuit has switched in such a way that advancing the time even by very small amounts (typically 1E-18s) still does not produce an acceptably small change in circuit voltages.

Often, this is caused by a badly designed model, or by not supplying sufficient parameters to a diode or transistor model. In a particular, if the junction capacitance values are not chosen correctly, these devices will exhibit zero switching times which can lead directly to this error message.

Most convergence errors are due to badly drawn circuits or incorrect models - time after time we have had circuits sent in that ‘won’t simulate’ only to find that something isn’t connected. Please check the simulation log for clues, and re-check your circuit before jumping to the conclusion that PROSPICE is at fault. *Where 3rd party SPICE or VSM models have been used, we cannot spend time debugging them unless can supply a simple circuit demonstrating the problem.*

Beware also of using 3rd party SPICE models which use features not supported in standard SPICE 2 or SPICE 3. Models developed for PSPICE™ can include all manner of elements and syntax constructs that are not standard SPICE.

Oscillators cause special problems because the initial solution for the operating point will fail. After all, an oscillator has no steady state! Use **IC**, **NS** or **OFF** properties to define a starting state as discussed under *Initial Conditions* on page 118.

If the problem really is numerical convergence, you can try the following tactics:

- Increase the value of **GMIN**. This is a leakage resistance for reverse biased semiconductor junctions, and lower values make the circuit look more and more like a network of resistors (which will always solve). But this is at the expense of accuracy. The default is 1E-12; values above 1E-9 will give fairly meaningless results.

Note in any case, that SPICE3F5 will try what is called *GMIN stepping* if at first the circuit will not converge. This means that a large GMIN is used to find an initial solution, and the value is then ramped back to its original value in order to maintain accuracy.

- Increase the value of **ABSTOL** and/or **RELTOL**. These values control the accuracy that is required for the simulation to be deemed to have converged. However, the larger you make these tolerances, the less accurate the results will actually be.
- If the circuit uses op-amps, try specifying **MODFILE=OA_IDEAL** instead of a specific device type - this model is much easier to simulate.
- Lower the value of **TRTOL**. This will make SPICE use smaller timesteps so it will be less likely to 'lose' a convergent solution, but at the expense of longer run times. This will only work if the simulation has failed part way through a transient analysis.

You should also try reducing **TRTOL** if plotted traces look 'spiky', or contain mathematical noise. This often manifests itself as oscillation of a value after a rapid level transition.

A

Aborting a Simulation	15
AC Sweep Analysis	85
Active Components	22
Actuators	22
Add Trace Command	13, 67, 68
Add/Remove Source Files command	52
Ammeter	29
Analogue Analysis	74
Analogue Traces	76
Analyses	
AC Sweep	85
Analogue	74
Audio	91
Conformance	93
DC Sweep	83
DC Transfer Curve	86
Digital	77
Distortion	88
Fourier	90
Frequency	81
Interactive	92
Mixed Mode	80
Noise	87
Power	76
Transient	74
Animation Control Panel	21
Animation timestep control	25, 26
Assemblers	53
Audio Analysis	91
Audio Generator	103

B

Bode Plot	81
Breakpoints	58, 60, 61
Build All command	52
Bus Traces	79

C

Capacitors	
initial conditions	118
leakage value	113

Code Generation Tools	53
Coloured Wires	23
Compilers	53
Components	
Linking To SPICE Models	107
PRIMITIVE Property	71
Conformance Analysis	93
Convergence	75, 117, 129, 137
Counter Timer	34
CPU load	25
Current	
displaying wire arrows	23
Current Arrows	23
Current Breakpoint	61
Current Generators	99
Current Graph, The	15, 66
Current Probes	17, 24, 105
Cursors	
on graphs	16
on Logic Analyser	34

D

Data Driven Generator	103
DC Generator	99
DC Operating Point	24
DC Sweep Analysis	19, 83
DC Transfer Curve Analysis	86
DDE	54
DDX program	53, 57
Debug Data Extractor	53, 57
Define Code Generation Tools command	53
Demand Drive Simulation	69
Digital Analysis	77
Digital Breakpoint	61
Digital Generators	104
Digital Simulation	
Floating Input Behaviour	120
Glitch Handling	120
Glitch Threshold Time Properties	122
Initial Conditions	77
State Contention Behaviour	119
Termination Conditions	78
Undefined State	119

LABCENTER ELECTRONICS

Digital Traces	79	Using To Partition A Design	126
Distortion Analysis	88	WAV File Based	103
DSIM	77	GETSPICE.EXE	111
control properties	130	GLEAK property	113
E		Glitch Threshold Times	121, 122
Earthing	26, 27	Global break condition	60
Edit Generator Dialogue Form	98	GMIN Stepping	138
Edit Graph Trace Dialogue Form	69	GND netname	77, 113, 115
Editing		Graph Based Simulation	
Generators	12, 98	Aborting	15
Graph Traces	69	How It Works	71
Graphs	14, 66	Report Log	15, 20, 88
Probes	12	Simulation Run Time	14
EEPROMs	125	Graphs	65
Embedded Workbench	54	AC Sweep	85
Error Handling	72, 135, 136	Adding Generators	13, 64
Event Driven Simulation	77	Adding Probes	13, 64
Exponential Pulse Generator	101	Adding Traces	13, 66
		Analogue	75
		Audio	91
		Colour of Traces	69
		Conformance	94
		Current Graph	15, 66
		DC Sweep	84
		DC Transfer Curve	86
		Digital	78
		Distortion	89
		Editing	14
		Fourier	90
		Frequency Graphs	82
		Interactive	92
		Maximising	15
		Minimising	15
		Mixed Mode	81
		Moving Traces	14, 15
		Noise	88
		Parts Of	15
		Placing	13, 64
		Removing Traces	14
		Resizing and Moving Graphs	13
		Simulating	70
		Status Bar	16
		Tagging Traces	69
		Taking Measurements From	15
		Using Cursors With	16
		Ground	26, 27, 113
F			
Fast Fourier Transform	90		
File Generator	103		
Fourier Analysis	90		
Frames per second	25, 26		
Frequency Analysis	17, 81		
Frequency Meter	34		
Frequency Reference	18, 82		
Function Generator	39		
G			
Generators	97		
Adding To Graphs	13, 64, 67		
Analogue Pulse	100		
Audio Signal	103		
Automatic References	12		
DC	99		
Digital	104		
Editing	12		
Exponential Pulse	101		
File Based	103		
Frequency Reference	18, 82		
Piecewise Linear	102		
Placing	11, 63		
SFFM	101		
Sine	99		

H

Harmonic Distortion	88
High Impedance Points	27

I

IAR Embedded Workbench	54
IC net property	116
IDE	54
Indicators	22
Inductors	
initial conditions	118
Initial Conditions	116
Initial DC Solution Checkbox	117
INITSEED control property	131
Input Impedance Plots	83
Interactive Analysis	92
Interactive Simulation	22
Interface Models	115, 123
Intermodulation Distortion	88
Isolate Before Checkbox	99
Item break expression	60
ITFMOD property	123

K

Keil uVision2	54
Knobs	49

L

Leaky Capacitors	113
Logic states	23
LST files	53, 57

M

Magnitude Axis	83
MAKE program	53
Manufacturers' Model Files	107
Mathematical Noise	138
Maximising Graphs	15
Measurements	
using Graphs	15
using Logic Analyser	34
using Oscilloscope	29
using Voltmeters and Ammeters	29
Memory windows	56, 59
Meters	29

Microprocessors	51
Minimising Graphs	15
Mixed Mode Simulation	80, 123
MODDATA property	125
MODDLL property	133
MODEL card	110, 134
Model Files	71
SPICE	107
Model types	132, 133, 134
Modelling	
Using SPICE Models	107
MODFILE property	133
Module Path Directory	71
MOSFET default dimensional properties	129
Moving	
Graphs	13
Traces On Graphs	14, 15
MP-LAB	54
Multiple CPUs	58

N

Netlists	
Compilation Of	71
Error Handling	135
Linking Of	71
No DC Path to Ground	113
No model specified	132
Nodal Analysis	74
NODESET option	117
Noise Analysis	19, 87
NS property	117

O

Object Code	52
Object Selector	13, 66, 98
Operating Point	24, 75, 116
In Mixed Mode Simulation	80
Oscillators	116
Oscilloscope	29
Output Impedance Plots	83

P

Parameter mapping table	132
Partitions	71, 126, 136
Pattern Generator	41
Persistent Model Data	125

Phase Axis	83
Piecewise Linear Generator	102
Pin logic states	23
Pin name translation	109
Placing	
Generators	11, 63, 98
Graphs	13, 64, 66
Probes	12, 63, 105
Tapes	127
Popup Windows	56
Power dissipation	76
Power Supplies	115
in Mixed Mode Simulation	123
Role In Partitioning	127
Precharge Property	118
Primary cursor	16
PRIMITIVE Component Property	71
Primitive models	132
PRIMITIVE property	132
Probes	24, 105
Adding To Graphs	13, 64, 67
Current Probes	17
Editing	12
Placing	12, 63
Using To Partition A Design	126
Properties	
Glitch Threshold Time Properties	122
ITFMOD Component Property	124
PRIMITIVE Component Property	71
Pulse Generator	100
PUTSPICE.EXE	111
PWI File	56

R

Reference Generators	
Frequency	18
On Frequency Graphs	82
Register Windows	56
Reset Persistent Model Data command	125
Resizing Graphs	13
Results Path Directory	71
Rotary Dials	49
RS232 Terminal	37
RTDBREAK object	61
RTIBREAK object	61
RTVBREAK object	60

S

Schematic models	133
Setup External Text Editor command	54
SFFM Generator	101
Signal Generator	39
Simulate Command	15
Simulation Log	15, 20, 70, 72, 88, 135
Simulator Control Properties	128
Simulator models	132
Sine Generator	99
Single Frequency FM Generator	101
Single Step Time	26
Single stepping	57
Singular Matrix	137
SML files	111
Source Code Control System	51
Source code editor	54
Source code windows	56, 57
Source Files	52
Source Level Debugging	53, 57
SPICE model libraries	111
SPICE models	107, 134
SPICEFILE property	134
SPICELIB property	111, 134
SPICEMODEL property	109, 111, 134
SPICEPINS property	109
SRCEDIT	54
Status windows	56
Step commands	57
Stopping a Simulation	15
SUBCKT definition	108, 134
Swept Variable Analysis	19

T

Tapes	127
TDLOWER control property	130
TDSCALE control property	130
TDSEED control property	131
TDUPPER control property	130
Teletype Terminal	37
TEMP property	118, 130
Temperature	118, 130
Terminal (RS232)	37
Text Editor	54
Time Domain Analysis	74, 77
Timer	34

Timestep per frame	25, 26	VSM models	133
Timestep too small	137	VSS netname	77, 115
TNOM property	118, 130		
Toggle breakpoint command	58	W	
Total Harmonic Distortion	88	Watch window	59
Trace Expressions	68, 76	Watchpoint	60
Traces		WAV Files	91, 103
Adding To Graphs	66	Windows	56
Analogue Traces	16	Wire arrows	23
Bus Traces	79	Wire colour	23
Deleting	14, 15		
Digital Traces	79	Z	
Editing	69	Zero time delay in models	78
Moving On Graphs	14, 15		
Tagging	14, 15, 69		
Untagging	14, 69		
Using Expressions With	64		
Transient Analysis	74, 77		
Translating pin names for SPICE	109		
TRTOL property	138		
Tutorials			
Graph Based Simulation	10		
Interactive Simulation	3		
Types of model	132		

U

UIC option	116
UltraEdit	54

V

Value not found	132
Variables windows	56, 58
VCC netname	77, 115
VCR controls	21
VDD netname	77, 115
View Log command	15
Viewing Simulation Logs	15, 20
Virtual Instruments	29
Virtual Terminal	37
Voltage	
displaying as wire colour	23
Voltage Breakpoint	60
Voltage probes	24
Voltage Probes	105
VOLTAGE Property	115
Voltmeter	29



*Advanced Routing
and Editing Software*

User Manual

Issue 6.3 - November 2003

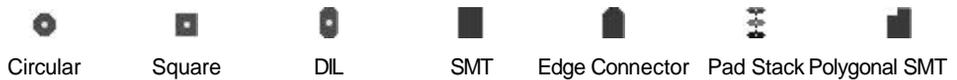
© Labcenter Electronics

ICON REFERENCE CHART

Placing and Routing



Pad Style Icons



2D Graphics Icons



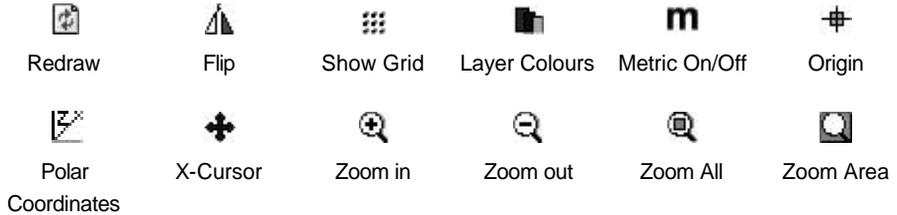
Editing Commands



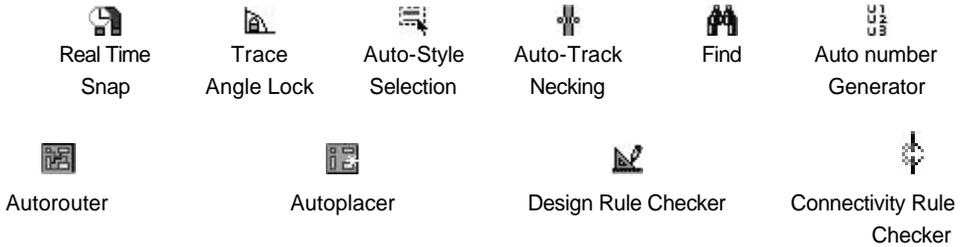
File and Print Icons



Display Commands



Layout tools



Rotate And Mirror Icons



TABLE OF CONTENTS

INTRODUCTION	1
WHAT IS ARES?.....	1
Layout Editor Features	1
Auto-Placer	2
Auto-Routing.....	2
Power Plane Support	2
HOW TO USE THIS MANUAL.....	3
GETTING STARTED	5
INSTALLATION	5
START UP OPTIONS	5
SCREEN AND HARD COPY DRIVERS	5
Graphics Driver.....	5
Printer & Plotter Drivers.....	5
SYSTEM INFORMATION.....	6
TUTORIAL	7
INTRODUCTION	7
OVERVIEW OF THE LAYOUT EDITOR.....	7
BASIC PLACEMENT & ROUTING TECHNIQUES	9
Package Placement.....	9
Routing.....	10
Annotation	10
Board Outline.....	11
BLOCK EDITING FACILITIES.....	13
ROUTE EDITING.....	13
Via Placement	13
Changing a Route's Width.....	14
Auto Track Necking.....	14
Tagging a Route.....	14
Moving/Dragging a Tagged route.....	15
Deleting a Tagged Route.....	15
The Context Menu	15
Re-routing.....	16
Connectivity Highlight	16
HARD COPY GENERATION.....	17
THE PACKAGE LIBRARY	18

THE SYMBOL LIBRARY.....	18
ROUTING A PCB FROM A NETLIST	19
Preparing a Schematic for PCB Design	19
Placing the Components.....	20
Editing a Placed Component	21
The Ratsnest	21
Manual Routing.....	22
Auto Routing.....	23
Mixed Manual and Auto-Routing.....	23
Router Strategies	24
Power Planes	24
REPORT GENERATION.....	25
Connectivity Rule Checker	25
Design Rule Check	25

GENERAL CONCEPTS 27

SCREEN LAYOUT	27
The Menu Bar	27
The Toolbars	27
The Editing Window	28
The Overview Window.....	30
The Object Selector	30
The Layer Selector	31
Co-ordinate Display	31
CO-ORDINATE SYSTEMS.....	31
Dimension Entry Fields	32
Output Origin.....	32
False Origin	34
Grid Dots	34
Snapping Grids.....	34
Real Time Snap	34
OBJECT PLACEMENT.....	35
Placing Components.....	35
Packages	38
Pads.....	38
2D Graphics	39
Zones.....	40
OBJECT EDITING.....	42
Tagging a Single Object.....	42
Tagging a Group of Objects	42
Untagging All Objects	42
The Tag Filter.....	43

Deleting an Object	43
Dragging an Object	43
Editing an Object	43
Highlighting a Component by Name.....	44
ROUTE PLACEMENT & EDITING.....	44
Trace Placement - No Netlist Loaded	45
Trace Placement - Netlist Loaded	45
Curved Track Segments.....	46
Auto Track Necking.....	46
Trace Angle Lock	47
Tagging a Route.....	48
Changing the Width of a Route.....	48
Changing the Layer of a Route.....	49
Modifying a Route.....	49
Copying a Route.....	49
Deleting a Route.....	50
Tidying the Routes.....	50
BLOCK EDITING COMMANDS	50
Block Copy	50
Block Move.....	51
Block Rotate.....	51
Block Delete.....	51
FILING COMMANDS	52
Starting a New Layout	52
Loading a Layout.....	53
Saving a Layout	53
Import / Export	53
Auto-Save.....	54
Back-up and Last Loaded Files	54
PAD & TRACE STYLES	55
Pad Styles	55
Polygonal Pads.....	55
Pad Stacks	56
Pad Styles and the Solder Resist Layer.....	58
Pad Styles and the Solder Paste Mask Layer.....	59
Trace Styles.....	59
Via Styles	59
Style Management & DEFAULT.STY.....	59
2D Graphics Styles	60
KEYBOARD CONFIGURATION.....	61

LIBRARY FACILITIES	63
GENERAL POINTS ABOUT LIBRARIES.....	63
Library Discipline	63
The Pick Command.....	64
The Tidy Command.....	64
THE PACKAGE LIBRARY.....	64
Making a Package.....	65
Editing a Package.....	67
THE SYMBOL LIBRARY.....	68
NETLIST MANAGEMENT.....	69
NETLIST FEATURES	69
The Load Netlist Command.....	69
Using the Netlist Loader on an Empty Layout.....	69
The Netlist Loader & Existing Components.....	69
The Netlist Loader & Existing Tracking.....	70
Problems with Pin Numbers	70
Packaging Considerations.....	71
Connectivity Highlight	72
RATSNEST FEATURES	72
Automatic Ratsnest Recalculation	72
Force Vectors	73
Ratsnest Mode.....	73
Manual Ratsnest Entry	73
PIN_SWAP/GATE SWAP.....	75
Manual Pin-Swap/Gate-Swap.....	75
Automatic Gate-Swap Optimization.....	75
Synchronization with the Schematic.....	76
ROUTING STRATEGIES.....	77
Strategies and the Netlist.....	77
Special Strategy Names	78
Editing a Strategy	78
BACK ANNOTATION.....	80
Manual Re-Annotation	80
Automatic Re-Annotation.....	80
Back-Annotation to ISIS.....	80
REVERSE NETLISTING.....	81
SDFGEN - CONVERTING 3RD PARTY NETLISTS	81
AUTO-PLACEMENT	83
INTRODUCTION	83
USING THE AUTO-PLACER	83

THE AUTO-PLACER DIALOGUE.....	84
The Component Selector.....	84
Design Rules.....	84
Trial Placement and Cost Weightings.....	84
Options	86
OCCUPANCY DEFINITIONS.....	86
LIMITATIONS	87
AUTO-ROUTING.....	89
INTRODUCTION	89
THE AUTO-ROUTER COMMAND.....	90
HINTS AND TIPS ABOUT AUTO-ROUTING.....	92
Single Sided Boards	92
Avoiding Using Component Pads as Through Holes.....	92
Off Grid Pads.....	93
Surface Mount Components	93
Routing to Internal Power Planes.....	94
TIDY PASS.....	95
POWER PLANES.....	97
INTRODUCTION & BACKGROUND.....	97
Grid Based Power Planes	97
Negative Image Power Planes.....	97
Polygonal Gridless Power Planes.....	98
Ground Planes Without a Netlist.....	100
USING POLYGONAL POWER PLANES	100
The Power Plane Generator Command.....	100
Zone Placement Mode	101
Editing a Power Plane.....	101
Deleting a Power Plane.....	103
Automatic Regeneration of Power Planes	103
Quick Redraw Mode.....	104
Auto-Routing & Power Planes	104
REPORT GENERATION	107
CONNECTIVITY RULE CHECK.....	107
Basic CRC Functions.....	107
Advanced CRC Functions.....	107
DESIGN RULE CHECK.....	108
HARD COPY GENERATION	109

PRINTER OUTPUT	109
PLOTTER OUTPUT	110
Plotter Pen Colours	111
Plotter Tips.....	111
POSTSCRIPT OUTPUT.....	112
CLIPBOARD AND GRAPHICS FILE GENERATION.....	112
Bitmap Generation.....	112
Metafile Generation	112
DXF File Generation.....	113
EPS File Generation	113
Overlay Bitmap Generation.....	113
CADCAM OUTPUT	115
THE CADCAM OUTPUT COMMAND.....	115
GERBER OUTPUT.....	115
NC DRILL OUTPUT.....	116
MECHANICAL ROUTING AND SLOTS	117
PICK AND PLACE FILE.....	117
GERBER VIEWING.....	119
PANELIZATION.....	119
DXF IMPORT TOOL	121
INTRODUCTION	121
SETTING UP	122
Generating The DXF File.....	122
Layer Assignments.....	122
PERFORMING A CONVERSION	122
Basic Conversions	122
Conversion Errors	123
Conversion Warnings	123
LIMITATIONS	124
GERBER IMPORT TOOL.....	127
INTRODUCTION	127
REQUIREMENTS FOR CONVERTIBILITY.....	127
TUTORIAL.....	128
Preparing for Conversion.	128
THE FILE/LAYER VIEW	134
THE TOOL VIEW	135
THE QUICK VIEW WINDOW	135
ADVANCED CONSIDERATIONS	136
Memory Usage.....	136

Pin Numbering.....137
Converting a Batch of Layouts137
Alignment138
Systems Which Output a Single Pad File.....138

INDEX139

INTRODUCTION

WHAT IS ARES?

ARES (Advanced Routing and Editing Software) forms the PCB layout module of the PROTEUS system and offers netlist based PCB design complete with a suite of high performance design automation tools.

The latest version, is compatible with Windows 98/Me/2k/XP and later. It includes a brand new Auto-Placer, improved Auto-Routing, automatic Gate-Swap optimization and even more powerful support for power planes.

Layout Editor Features

Major features of ARES include:

- 32 bit high-precision database giving a linear resolution of 10nm, an angular resolution of 0.1° and a maximum board size of +/- 10m. ARES supports 16 copper layers, two silk screens, four mechanical layers plus solder resist and paste mask layers.
- Netlist based integration with ISIS schematic capture, including the ability to specify routing information on the schematic.
- Automatic Back-Annotation of component renumbering, pin-swap and gate-swap changes.
- Physical and Connectivity Rule Check reports.
- Powerful route editing features including topological route editing, auto track necking and curved trace support.
- 2D Drawing with Symbol Library.
- Comprehensive package libraries for both through hole and surface mount parts including SM782 standard SMT footprints. There are now over 1000 parts in total in the package library.
- Unlimited Pad/Trace/Via Styles.
- Full metric and SMT support. This includes all dialogue form fields as well as the coordinate display and grid settings.
- Output to a wide range of printers and plotters. Also output in DXF, EPS, WMF and BMP graphics formats - to file or clipboard where appropriate.
- Built in Gerber Viewer - this enables you to check your Gerber output files before spending money on bureau fees or board manufacture.

- DXF import for import of mechanical data such as board outlines.
- Gerber import for bringing in PCB layouts from other EDA packages.

Auto-Placer

ARES includes an autoplacer module, an addition which in combination with the auto-router makes it possible to create PCBs almost entirely automatically. Alternatively, you can use it interactively, either by pre-placing critical components manually and auto-placing the rest, or else by using it to place small sections of the design in turn with manual adjustments being made after each section is placed.

The auto-placer is highly configurable and can be set up to handle a wide variety of board types.

Auto-Routing

The most tedious, error prone and time consuming part of the traditional electronics development process is undoubtedly routing the PCB. As a result, it is in this area that the greatest benefits of Electronic Design Automation are to be found.

Our autorouter is the result of intensive (and continuing) research into techniques for getting the highest completion rates and you will find it capable of near 100% completion of designs that could take days by hand. On the harder jobs where some routes are left undone you will find that *Topological Route Editing* is ideally suited to moving existing routes so that the rest can be completed.

The latest version incorporates a rip-up & retry algorithm to ensure maximum possible completion rate. At the same time, new algorithms have been added to generate fan-out patterns for rows of off-grid surface mount pads.

The auto-router also includes a tidy pass which reduces track length and via count as well as improving the aesthetic quality of the layout.

Power Plane Support

ARES supports Polygonal Gridless Power Planes which overcome most, if not all of the disadvantages associated with other methods of implementing copper fills. The essence of the approach is to generate polygonal boundaries around all the objects within the target area, and then to merge them together. The resultant multi-edged hole boundaries are subtracted from the original (user placed boundary) in order to establish whether or not there is complete or partial connectivity.

Our implementation of this is fast - to the point that real time update is feasible for a modest board on a fast PC, and takes care of both 'slivers' - where two holes nearly touch, but not

quite - and also of the issues arising from rendering the computed shapes with a pen of some minimum thickness.

HOW TO USE THIS MANUAL

The Graphical User Interface and the general intelligence of the software itself will enable many users to be productive almost from the outset. However, as with ISIS there is a great deal of functionality 'under the hood' and you cannot expect to master all aspects of the package immediately.

For those who need some initial tuition and guidance, we have followed the adage that the best way to learn is by doing - once you have installed the package by following the instructions in the next chapter, we suggest that you proceed to work through the extensive tutorial. This takes you right the way through the PCB design process from loading a netlist to performing the final CRC and DRC checks.

The remainder of the chapters provided background detail on all aspects of the system, and for quick reference, the final chapter deals with all the commands and any associated dialogue forms.

An index is provided as a further aid to reference.

GETTING STARTED

INSTALLATION

ARES is installed by the PROTEUS installation program and the procedures for using this are documented in the common installation instructions at the front of the binder. This installation process will also copy files for ISIS and ProSpice if you have purchased them.

After you have completed the PROTEUS installation, you may also want to configure ARES for your system and it is the procedures for this that are documented here.

START UP OPTIONS

The installation process will create a group called “Proteus 6 Professional” and ARES may be started by double-clicking the ARES icon. In addition, the setup program installs appropriate file-associations such that double-clicking on PCB layout files will also launch a copy of ARES.

SCREEN AND HARD COPY DRIVERS

Graphics Driver

ARES simply draws itself through whatever display driver you have installed for Windows. Thus the resolution, number of colours and so forth is determined by this choice, and not by ARES.

Printer & Plotter Drivers

Again, ARES will (in theory at least) generate output through any Windows printer or plotter device driver. In addition to printing through standard Windows drivers, ARES can also generate Gerber and Excellon drill output files as well as WMF, BMP, DXF and EPS files for graphics export. All this activity is controlled from the *Output* menu.

If you have problems with output, do feel free to contact us for technical support. However, the quality of some Windows device drivers leaves a lot to be desired and some problems may be beyond our control. We cannot be responsible for bugs in code that we have not written!

SYSTEM INFORMATION

The *System Info* command on the *System* menu brings up details of the Release Number of the system and to whom it is registered. Please have this information to hand when you call for technical support.

The command also shows how much free RAM you have available - this includes both free physical and virtual memory. Provided that you have a reasonable amount of free disk space, you should not encounter memory shortages with ARES, although more memory may well improve performance.

In addition, statistics about the number of components, pads, tracks etc. are displayed on this form. Also displayed is the number of 'missing' connections, which equates to the number of ratsnest lines displayed in the work area.

INTRODUCTION

The purpose of this tutorial 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.

The tutorial proceeds by taking you through worked examples involving all the important aspects of the package including:

- Basic techniques for placement and routing.
- Netlist based design including both manual and automatic routing .
- More advanced editing techniques such as block editing and route editing.
- Report generation - the CRC and DRC tools.
- Hard copy generation.
- Library part creation.

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.

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 the shortcuts mentioned may not be valid. Information on configuring your own keyboard shortcuts can be found on page 61

OVERVIEW OF THE LAYOUT EDITOR

We shall assume at this point that you have Proteus installed on your PC, and launched ARES from the *Start Menu*.

Menus are activated by pointing at the menu name; options are selected by clicking the left mouse button (henceforth referred to as "clicking left") on the required option. Some options have associated keys which can be used instead of the menus - these are displayed at the right of the menu option. Options which enable or disable some program feature show that

feature's status with a small tick symbol to the left of the option. Presence of the tick symbol implies that the option is selected. .

The largest area of the screen is called the *Editing Window* and it acts as a partial view on the layout. You can adjust the scale at which the layout is displayed using the zoom options on the *View* menu, or with the associated function keys (by default F6 to Zoom in and F7 to Zoom out). In this case, the position of the mouse pointer is taken as the new centre for the *Editing Window*. You can also pan to adjacent areas by holding down the SHIFT key and 'bumping' the mouse pointer against the appropriate edge of the *Editing Window*.

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.

The smaller area at the top left of the screen is called the *Overview Window*. Not entirely illogically, this is used to display an overview of the entire layout. You can move the work area to a chosen part of the layout by pointing to where you want to go on the *Overview Window*, and clicking left. The green box shows the area currently displayed in the *Editing Window*.

Just below the *Editing Window* is the *Layer Selector* which determines the current layer or layer set. The current layer applies both to the placement and selection of PCB objects.

Which layers are *displayed* can be adjusted by use of the *Layers* command on the *View* menu.

Also at the bottom the screen 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. Two things affect this:

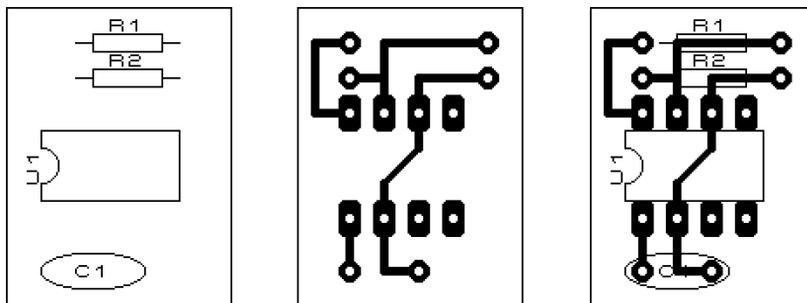
- The currently selected snap grid. The options available appear on the *View* menu and also via keyboard shortcuts (these are configured by default as CTRL-F1 and F2-F4 but can be customised as detailed on page 61). You can re-define the snap values through the *Set Grids* command on the *System* menu.
- Real Time Snap. When this feature is enabled, the cursor will lock onto pads and/or tracks, even if they were not placed on the currently selected snap grid. RTS always snaps to pads and vias, and will also snap to tracks when the Routing mode icon is selected. RTS can be turned on and off using the *Real-Time Snap* command on the *Tools* menu or by accessing the keyboard shortcut for this command (by default this is set to be CTRL+'S').

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 shortcut key 'X'.

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

BASIC PLACEMENT & ROUTING TECHNIQUES

Before going on to look at designing a board from a netlist we shall first cover the basics of placing and routing using the extremely simple board shown overleaf.



Package Placement

The most direct way to build up a rough layout of a board is to drive ARES in Package mode. In this mode, you can pick component footprints or *Packages* directly from the library and place them onto the work area.

In our simple example, three packages are used

```
CAP20
DIL08
RES40
```

and you can start by picking them from the package library. To do this, first click left on the *Package* icon (see the icon sheet at the front of the manual). Next, click left on the 'P' toggle at the top left of the *Object Selector* which is now displaying the word 'PACKAGES'. The *Library Pick* form will appear and you can pick the required packages from it. When you have done this, close down the form by clicking the close button (Windows) at the top left.

The three package names should now have appeared in the *Object Selector*, with the last one you picked highlighted. Ensure that DIL08 is selected by clicking left on it, point somewhere in the middle of the *Editing Window* and hold down the left mouse button. A green outline of an 8 pin IC will appear which you can move about with the mouse. Position it roughly central and release the mouse button. Now select the RES40 package and place the two resistors, 0.1" apart (2 grid squares) and just above pin 8 of the IC. Similarly, place the capacitor outline just below pin 1.

Unless you are quite adept, you may not have the components positioned quite correctly, so we will now take a look at how to move things around. Point at a component outline (rather than its pads) and click right. This will 'tag' it, and cause it to be highlighted. Now, still keeping the pointer over it, hold the left button down and drag the mouse. This is one of the ways to move objects. Also, you can delete a tagged object by pointing at it and clicking right. A *Multi Level Undo* system is also available and accessible from either the *Edit* Menu or the configured keyboard shortcuts.

All the objects can be untagged by pointing at no object and clicking right.

Routing

Routing mode is commenced by clicking left on the *Trace* icon. The *Object Selector* will change to display a list of 'Trace Styles' - the default selection of track widths. Select T20 for a 20 thou track.

Traces are placed by clicking left at each point along the required route, and clicking right to finish.

Other points to note about routing mode:

- Clicking twice at the same point places a via and changes the current layer as defined by the *Set Layer Pairs* command on the *System* menu. The selection of via types can be displayed by clicking on the *Via* icon.
- Whilst routing, you can change the current layer by pressing the PGUP and PGDN keys. In addition CTRL-PGUP selects the top layer and CTRL-PGDN selects the bottom layer.
- Holding the CTRL key down allows you to place a curved track segment. The progress of the arc (horizontal then vertical or vice versa) is determined by how you move the mouse away from the fixed point. It is best to press and hold the CTRL key, then move the mouse, then click left, then release the CTRL key.

Annotation

When components are placed in Package mode, they have no annotation information associated with them - later on you will see how components are automatically annotated when a netlist is used.

To annotate the components, select the *Instant Edit* icon and then click left on each component in turn. Each time you do this, a form will appear with fields for the part ID and value. You might find it easier to use only the keyboard when annotating: the cursor keys will move the mouse pointer one grid step at a time and the ENTER key will do for the left mouse button and the OK button on the dialogue forms.

Alternatively, the *Auto Name Generator* command may be used to generate numerical sequences for component numbering.

Part IDs and values can be moved by tagging the parent object and then pointing specifically at the ID or value before dragging using the left mouse button. Bear in mind also that you can set different snapping grids from the *View* menu or by using the snap command keys (by default these are mapped to CTRL-F1 and F2-F4 – more information on keyboard configuration can be found on page 61).

The default size for these labels can be determined using the *Set Template* command on the *System* menu.

Board Outline

ARES has a special layer, the EDGE layer, which is intended for holding 2D graphics which represent the board outline. Objects placed on the EDGE layer will appear on artwork generated for any of the other layers.

In this case, the board outline is just a box.

To place a rectangular board edge:

1. Select the *Box* icon.
2. Select the EDGE layer from the *Layer Selector*.
3. Point at where you want the top left corner of the box, press the left mouse button and drag out the box to enclose the layout.
4. If you need to resize the box, tag it by pointing at it and clicking right, and then drag the sizing handles as required using the left mouse button.

Curved or irregularly shaped boards are fully supported; the boundary should be formed from lines and arcs, or by selecting the *Path* icon and drawing a single path object.

LABCENTER ELECTRONICS

When you have finished experimenting with this exercise, save it if you wish and then start a new layout for the next exercise by using the *New Layout* command on the file menu.

BLOCK EDITING FACILITIES

We have seen already that an object can be tagged by pointing at it and clicking right, and that once tagged it can be dragged (left button) or deleted (right button). There is, however, another way to tag objects. If you point at no object, hold down the right button and drag the mouse, a green box will appear. When you release the button, any objects inside the box will be tagged. Re-load 'PPSU.LYT' and then give this a try by dragging a tag box round the entire layout. Once you have a tag box, the *Copy*, *Move*, *Rotate* and *Delete* icons come into play - try them out, bearing in mind the following:

- The types of objects which are selected by the tag box can be changed through the use of the *Tag Filter* command. This also allows you to choose which layers are affected.
- You can undo a delete using the *Undo* command (default key mapping 'CTRL+Z').
- When you perform a block rotation, you will be prompted to mark an origin point about which the rotation/reflection will occur.

Try completing a few routes and then tagging just one or two components and moving them - track segments with one end inside the tag box and the other end outside are stretched.

ROUTE EDITING

Powerful route editing facilities are a major feature of ARES and we shall now take a good look at the features available in ARES.

Unlike many other PCB design packages, route editing in ARES is based around the topology of the current tracking rather than depending in any way on how the sections of track were laid down. In addition, modifications can be made to any part of a route, not just to sections between 'nodes'.

Via Placement

In almost all circumstances, vias are placed automatically for you. To see this in operation, select the *Trace* icon and place a route segment by clicking left at two points. Now click left over the second point a second time, then at a third point and finally click right to terminate the route. Clicking (left) at the same point twice causes ARES to place a via and change the current layer so that (in this case) the next segment will be placed on the top copper layer. At the same time, a via is placed at the point where the route changes layer.

Which layer is selected is determined by the *Set Layer Pairs* command on the *System* menu. It is possible to define layer pairs, triples or whatever as required. You can also change layer

manually whilst placing a route by using the PGDN and PGUP keys, but note that a via is not placed in this case.

The type of via used can be changed by selecting the *Via* icon and choosing one of the via styles from the *Object Selector*. You can also place, replace, tag, move and delete vias manually in this mode. The left and right mouse buttons operate as for manipulating components.

For multi-layer boards, you can select whether normal, blind or buried vias are placed by selecting the *Via* icon and then adjusting the *Layer Selector*.

Changing a Route's Width

If you need to change the width of a section of tracking, there are two approaches:

- Simply select the required trace style and place new tracking over old.
- Tag the tracking with the right mouse button, and then click right a second time and select the required trace style from the context menu.

Remember that to change track widths globally, you can always edit the appropriate trace style by selecting it and clicking on the *Object Selector* 'E' toggle.

Auto Track Necking

In many cases, the reason for necking down a track is to that it can pass between two pads or other obstacles without violating the design rules. The Auto Track Necking feature allows ARES to do this for you.

The function is controlled by the *Set Design Rules* command on the *System* menu. The dialogue form allows you to enter the clearances for pad-pad, pad-trace and trace-trace and also the trace style to neck to. The default neck style is T10 - a 10 thou trace.

Tagging a Route

To re-route, delete or copy a section of tracking requires that you first tag it. In a similar fashion to the object editing facilities, you do this by clicking right on it, though there are one or two subtleties...

- ARES will only 'sense' tracking on the current layer so you must set the *Layer Selector* to make the appropriate layer current. The space bar or the middle mouse button (if your mouse has one) will cause the selection of the next layer in the layer-pair sequence as defined by the *Set Layer Pairs* command. Alternatively, you can move up and down the layers with PGUP and PGDN.

- If you click on a via, or a point at which several tracks are joined, all tracks meeting at that point are tagged.

As with object editing, pointing at nothing and clicking right untags the tagged route.

We suggest that you practice tagging and untagging the routes on the layout before proceeding further - it is important, though not at all difficult, to grasp how this feature works.

Moving/Dragging a Tagged route

Once you have a section of tracking highlighted, you can drag any of its segments by pointing and dragging with the left mouse button.

- If you point at a horizontal or vertical segment, then it will move vertically or horizontally respectively, and any adjacent segments will be 'fixed up' to maintain orthogonal routing.
- If you point at a node (corner) then that single point will be dragged, and the adjacent segments will stretch diagonally.
- If you point at a diagonal segment, then a new node will be created.

Deleting a Tagged Route

There are two ways to delete tracking.

- The *Block Delete* icon will delete all highlight objects on the layout. During route editing, these are typically just the tracks that you have tagged, so this works as a quick way to delete the tagged route.
- You can also delete a single, interconnected section of tracking using the *Context Menu* - see below.

The Context Menu

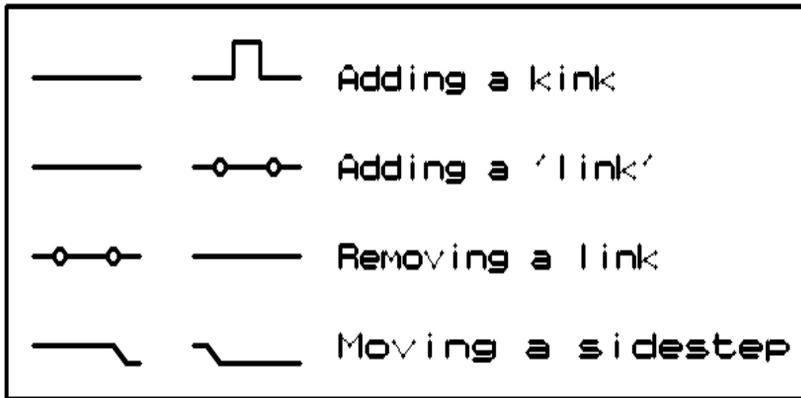
If you click right on a tagged track, a popup menu will appear with options to:

- Delete, Copy and Move the route.
- Change the layer and trace style.
- Change the via style

The *Copy* command provides the means to do memory buses and similar repeated patterns of tracking. You can make as many copies as are required by repeatedly clicking left. Click right to finish.

Re-routing

Finally, there is a very nice way of modifying the actual path taken by a route. Once you have a tagged route, you can alter its path by simply placing a new section of tracking (in the usual way) that starts and ends on the old one. ARES automatically computes which section of the tagged route is shorted out, and removes it. This is a very natural way to work - all of the route editing operations shown in the diagram below can easily be carried out with this feature.



Note that vias which are made redundant (i.e. secondary vias) will also be removed, along with the shorted out section.

You will find this feature especially useful for moving routes that turn out to be blocking other ones towards the end of the routing process.

Connectivity Highlight

Although not strictly part of the route editing facilities, Connectivity Highlight does come in most handy at the end of the routing process when you find yourself needing to check which connections have and (more importantly) have not been made. For example, you might want to know if a clock signal has been connected to all the chips requiring it.

To see the feature in operation, select the *Connectivity Highlight* icon. Click left on a component pad with attached tracking and you will see that everything connected to that pad will be highlighted in bright white. All such highlighted objects will remain highlighted, irrespective of pan and zoom operations, until you invoke the *Redraw* command (default key

mapping 'R' - more information on configuring your own keyboard mappings can be found on page 61).

Also in this mode, clicking on the net selector toggle will highlight all objects assigned/connected to the currently selected net.

Clicking on the Delete icon will delete traces or vias highlighted by either of the above techniques thus providing the means to delete all or part of a net.

HARD COPY GENERATION

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.

We will deal here solely 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 HARD COPY GENERATION on page 109.

You may also wish to compensate for any inaccuracy of scaling in your output device using the layout CALTEST.LYT (in the "Samples\Schematic and PCB Design" directory of your Proteus installation) and the *XCompensation/YCompensation* fields which are located on the *Print* dialogue form.

CADCAM output and Gerber Viewing are described in CADCAM OUTPUT on page 115.

THE PACKAGE LIBRARY

Packages are made by placing pads and silk screen graphics in the work area, tagging them, and then invoking the *Make Package* command on the *Library* menu. As a quick example, try the following:

1. Select the *Circular Pad* icon and then style C-80-30 from the Object Selector.
2. Place two circular pads 0.5" apart
3. Then select the *Box* icon and draw a box round the two pads.
4. Tag the whole ensemble by dragging a box round it with the right mouse button.
5. Invoke the *Make Package* command on the *Library* menu and key in THING for the name

If you now select the *Package* icon, you will see that THING has appeared in the Object Selector, and can be placed like any of the packages you have used so far. The reference or anchor point for the package is always the first pad placed, unless one was explicitly defined using an ORIGIN marker when the package was created.

There is rather more to it than this - you can, for example, have pads which are on one side only, and silk screen legends on both sides of the board. For further details, see LIBRARY FACILITIES on page 63.

To edit an existing package, pick it from the library, tag it, and then invoke the *Decompose* command. This will break the package into its constant elements (pads and 2D graphics). It is not recommended to do this to a component that is part of a layout.

THE SYMBOL LIBRARY

Symbols are simply groups of 2D graphics objects which are stored in a library for general use. Typical applications include things like drilling targets, graphics for non-electrical components like brackets and heat-sinks, and possibly your company logo.

You make a symbol by tagging the 2D graphics objects which form it (including, if you like, other symbols), and then invoking the *Make Symbol* command. Note that layer information is ignored when making a symbol.

A symbol may also be edited using the *Decompose* command as described for packages.

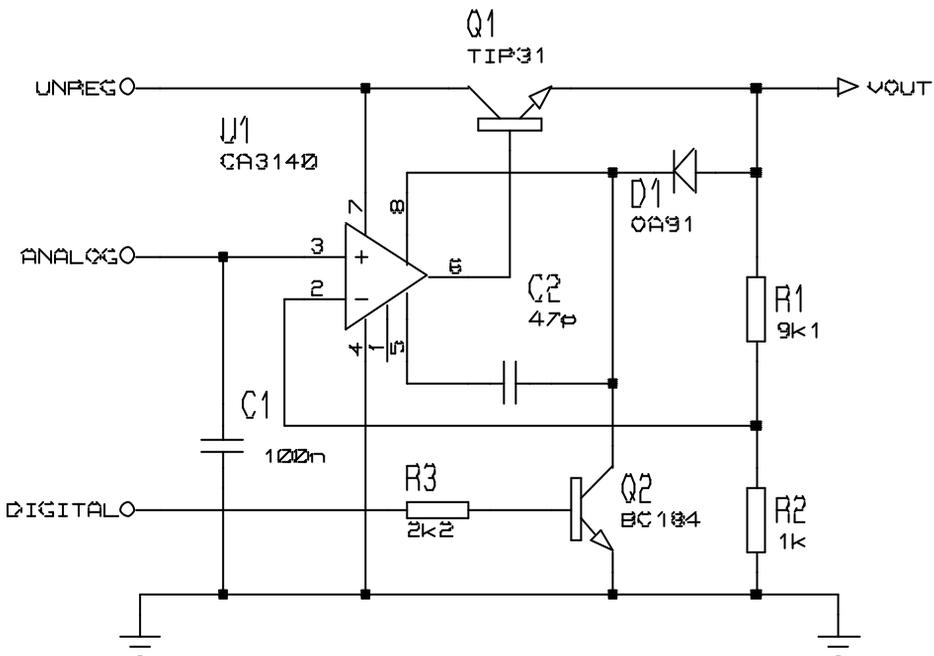
ROUTING A PCB FROM A NETLIST

In the previous exercise, you used ARES as no more than a computerized drawing board. However, ARES is really intended to be used in conjunction with ISIS. Since it is much easier to check that a schematic is correct, the overall result is an improvement in the quality of your design process. Also, since ARES can use the netlist to show you what is connected to what, there is no longer the need for constantly referring to data books in order to check pinouts and so forth.

Preparing a Schematic for PCB Design

For the purposes of this exercise, you need to load up the sample design PPSU.DSN. You will find it in the "Samples\Tutorials" directory within your Proteus installation.

This design is shown below:



The ISIS library parts already contain packaging information for the PCB, by virtue of their **PACKAGE** properties. Thus, you can go straight into PCB design by selecting the *Netlist to ARES* command from the *Tools* menu in ISIS.

 For a fuller discussion of preparing schematics for PCB design, see the chapter entitled *ISIS & ARES* in the ISIS manual.

Placing the Components

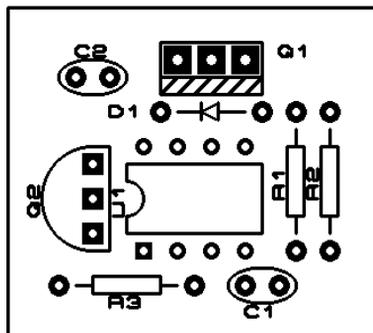
The sample file PPSU.LYT is actually ready placed so the first thing you need to do is remove the components from it. To do this, drag a tag-box round the components (i.e. just inside the board outline) using the right mouse button and then click the *Delete* icon.

Once this is done, selecting the *Component* icon will show all the components in the *Object Selector*. Object placement is the same as in the previous exercise though this time you will need to use the *Rotation* icons to select the orientation for some of the parts. This can be done either before placement, or else if you tag an object and then operate the icon, the object will rotate in sympathy.

You can also rotate a component whilst placing it using the '+' and '-' keys on the numeric keypad.

We suggest that you start with the op-amp and then place the small components around it. In a real situation, it is normal practice to sketch out a floor plan on paper before you start though in this case you can use the diagram below as a guide.

As you place components, they disappear from the selector - this gives you a clear indication of how many components remain to be placed. If placed components are deleted they are restored to the selector. Note though that this does *not* apply if components were placed in Package mode and then annotated - only components specified in a netlist are treated this way.



Editing a Placed Component

There are several things you can do to a placed component:

- Move it wholesale by tagging it and dragging with the mouse over the body or pads but not the labels. Whilst dragging, the '+' and '-' keys on the numeric pad will rotate it.
- Move its labels by tagging the whole object and then dragging.
- Move any of its pads by tagging the component, selecting the *Instant Edit* icon, and then dragging the pad with the left mouse button.
- Rotate or reflect it by tagging it and then clicking the *Rotate* icons and/or the *Mirror* icon.

You can set it to non-orthogonal angles by typing into the angle edit-box on the toolbar.

- Edit its labels by tagging it and then clicking left without moving the mouse.
- Change any of its pads by placing new ones over them in Pad mode.
- Change its package style by selecting package mode, choosing the new package and then placing it such that its pin 1 touches the old package's pin 1. It is not recommended to replace a package with one with a different number of pins as netlist information may be lost. This situation can be retrieved by re-loading the netlist.

The Ratsnest

As you place the components, you will see that green 'ratsnest lines' or 'connections' appear while the object is being placed. It should be fairly intuitive that the longer the ratsnest lines, the less optimum the components position and this is indeed the case - optimizing component placement is equivalent to minimizing the total connection length. Unfortunately, solving this problem is more an art than a science!

- Remember that the '+' and '-' keys will rotate each component whilst you are placing or dragging it. Combined with the ratsnest display, this gives you a rapid means to optimize the orientation of each part as you place it.
- Also worthy of note is the fact that this time there is no need to annotate the parts - when you place U1 it comes out as a DIL08 and there is nothing else to do.

ARES re-calculates the ratsnest not only as components are placed, but also while they are being dragged. This can mean that ratsnest lines may 'jump about' as the component's pads move between various possible connection points. It should be obvious, that with something

like a decoupling capacitor, the nearest power pins to it will depend on the region of the board in which it is located.

In addition, ARES shows *Force Vectors* for each placed component. These appear as yellow arrows which point to the optimum location for the component. The shorter the force vectors, the better the overall placement. The force vector for a component being dragged is also updated in real time, although its effect on the force vectors of the other components is not shown until it is placed. Once a component is tracked to, its force vector will automatically disappear.

One final detail is to place the board edge. To do this, select the *Graphics* icon, the *Box* icon and the *EDGE* layer on the *Layer Selector*. Then drag out a box for the board edge around all the components.

You are now ready to begin routing.

Manual Routing

At this point, we suggest that you load the file PPSU.LYT which represents where you should have got to so far. You will find this file in the "Samples\Tutorials" directory within your Proteus installation..

You should now see that all *Ratsnest* interconnections are displayed. Select the *Trace* icon. and click left on pin 4 of the op-amp. At this point several things will happen:

- A prompt message will appear in the status bar indicating that you are routing part of the ground net.
- All three connections connecting to pin 4 will highlight, indicating which pins it is legal to route to.
- The trace selector will automatically display trace style T25. This is associated with the net-strategies feature, but for now simply take it that this has been predefined as the default thickness for power nets.

Note that this only occurs if the *Auto-Trace Selection* option on the *Tools* menu is on.

- A green 'rubber' line will appear from pin 4 to the current mouse pointer position - this shows you where a track segment would be placed if you clicked left.
- Point at the left hand pin of C1 and click left a second time. ARES will sense that you have completed the route and will replace the connection line with a segment of 25 thou tracking. Click left on U1 pin 4 again, move up 1 grid square, click left, move over to 1 square above the lower pin of R2 and then down onto the pin.

Now make the connection for U1 pin 4 to Q2's emitter. In this case, note that you need not take the route pin to pin - the Advanced Netlist Management features enable ARES to tell which ratsnest line to remove, even if you take the route from Q2's emitter to the route corner above U1 pin 4.

The next connection to do is the one from U1 pin 2 to R1. Assuming that we want to route this on the top side of the board running above the lower row of U1's pins, press the space bar or else the middle button of your mouse if it has one. This selects the other layer in the current layer pair - in this case 'Top Copper'. Route the connection as before, noting that the trace type selected is now DEFAULT as defined in the SIGNAL strategy. Also note that the trace comes out in red, indicating (with the default colours) that it is on layer C1. Now complete this net by making the connection to R2

It is in fact possible to route this board with no vias at all. However, for the purposes of tutorial we shall route the connection from U1 pin 3 to C1 using two vias. Start by selecting 'Bottom Copper', click left on U1 pin 3 and then click left twice at the point two squares below. Clicking left twice at a point places a via and also swaps the primary and secondary layers. The via will take on a layer range determined by the current strategy's via mode (normal, buried or blind). Having got one via, move across to the point two squares above the target pin, click left twice and complete the route on the underside of the board.

Auto Routing

Using the auto-router is extremely simple - after all the whole point is for the computer to do the work. To see it in action, start by re-loading PPSU.LYT and then invoke the *Auto-Router* command from the *Tools* menu. The default settings will do for this example board so click on OK, sit back and watch. The status display at the bottom of the screen shows what is happening and how things are going. The yellow route is the one being considered for routing. On modern PCs, this board will route to completion in the twinkling of an eye!

Mixed Manual and Auto-Routing

Although the above exercise routed the board entirely automatically you can, if you wish, exert a lot more control over the proceedings. Reload PPSU.LYT and then select the *Main Mode* icon followed by the *Ratsnest* icon. The selector will then display a list of the nets in the design. Select the GND net and click on the selector 'T' toggle. This tags all the connections in the net.

In this mode you can also:

- Tag connections by clicking right over them.
- Tag a connection by clicking right over it.

- Untag all connections by clicking right over nothing.

Given that the auto-router can be set to route all connections or just either the tagged or untagged connections, this gives you total control if you want to mix manual and auto-routing.

Router Strategies

ARES handles the problem of routing different nets with different trace/via widths and so forth in a very sophisticated and convenient manner. Each net in the design is assigned (either implicitly or explicitly) a named strategy which defines how it is to be routed. In practice this means that a net called 12VRAIL can be assigned the POWER strategy on the schematic but that the details of POWER routing can be left undefined until the PCB begins to take shape. At the same time, the need to allocate all sorts of separate properties to each net in the design is avoided.

For a taste of what is possible, invoke the *Set Strategies* command from the *System* menu, select the POWER strategy and then click on the selector toggle. The *Edit Strategy* form allows you to determine how nets assigned to this strategy should be routed.

You will see that there are fields for the trace and via styles, the algorithm to use (i.e. POWER, BUS or SIGNAL), controls for whether diagonals are allowed and whether corners should be optimized (i.e. cut at 45 degrees) - each option on the dialogue form has context sensitive help associated with it for quick reference. Up to 4 passes are allowed per strategy and each pass can use different layers. If the H and V layers for a pass are the same then single sided routing will be attempted.

You can also set individual design rules for each strategy. This can be useful where some tracks must carry high voltages and need to be more widely spaced than those carrying only low voltage signals.

Further details on all these features are to be found under NETLIST MANAGEMENT on page 69 and AUTO-ROUTING on page 89.

Power Planes

As an example of the sophisticated power plane capabilities, we will create a simple ground plane for the PPSU board. Start as follows:

1. Load the file PPSU.LYT.
2. Route it with the autorouter.
3. Select the *Connectivity Highlight* icon.
4. Select the GND=POWER net in the *Object Selector*.

5. Click the 'T' for tag toggle on the *Object Selector*.
6. Click the *Delete* icon.

This sequence of events routes the board, and then removes the ground tracking placed by the router.

Then

7. Select the *Power Plane Generator* command from the *Tools* menu.
8. Select the GND net (this may appear as “GND=POWER”).
9. Click OK.

ARES will then generate the ground plane.

REPORT GENERATION

Aside from Gerber Photoplot and Excellon NC Drill tool information which is really part of the printing/plotting sub-system, ARES can generate two reports detailing how well the current layout meets its specification:

- Connectivity Rule Check - Checks for electrical errors (extra/missing connections)
- Design Rule Check - Checks for physical errors (overly small copper clearances)

Connectivity Rule Checker

This tool establishes which pins are connected to each other (by tracking and vias) and compares this with which pins have been assigned to the same net in the netlist. A report indicating the 'net-groups' within each net is produced and presented in a pop-up window. If you click on the items in the list, the net or nets affected by the error will be highlighted.

If you design from a netlist in the first place, and especially if you use the auto-router, you are unlikely to make mistakes other than missing connections out. However, a zero errors CRC report does give you extra confidence that your design is correct.

Design Rule Check

In PCB layout, (physical) design rules are Pad-Pad, Pad-Trace and Trace-Trace clearance. The auto-router will, of course, obey the correct design rules in choosing where to place routes but the *Design Rule Checker* will also verify that manually placed routes are in order. Where violations occur, these are marked on screen with a red circle and a line showing which objects are too near and a window appears listing all the violations that were found. You can click on the items in list list to highlight each violation in turn; if you double click, ARES will zoom right in to show the violation in detail.

LABCENTER ELECTRONICS

DRC errors will also be flagged if two objects are physically touching but not 'properly' connected - an example of this would be two pads which overlap but do not have a track running between them. This type of error will also show up in the CRC report as a missing connection.

Further details of these reports can be found in **REPORT GENERATION** on page 107.

GENERAL CONCEPTS

SCREEN LAYOUT

The Menu Bar

File Output View Edit Library Tools System Help

The *Menu Bar* runs across the top row of the screen and its main purpose (the selection of commands from the menus) is the same as with any other Windows application. In addition, the title bar area above the menu names is used to display certain prompt messages which indicate when the program has entered a particular editing or processing mode.

The Toolbars

As with other modern Windows applications, ARES provides access to a number of its commands and modes through the use of toolbars. The toolbars can be dragged to any of the four edges of the ARES application window.

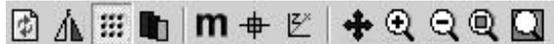
Command Toolbars

The tools located along the top of the screen (by default) provide alternative access to the menu commands, as follows:

File/Print commands



Display Commands



Editing Commands



Layout Tools

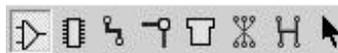


If you are working on a relatively small monitor, you can hide any or all of the command toolbars using the *Toolbars* command on the *View* menu.

Mode Selector Toolbar

The toolbar located down the left hand edge of the screen select the editor mode, i.e. what happens when you click the mouse on the *Editing Window*.

Placing & Routing



Pad Placement



2D Graphics



Note that the mode toolbar cannot be hidden, as its functions are not duplicated on the menus.

Orientation Toolbar

The orientation toolbar displays and controls the rotation and reflection for objects placed onto the layout.

Rotation



Reflection



The edit box allows you to type a rotation angle in directly; this is the only way to enter non-orthogonal angles.

When an existing object is tagged, the rotation and reflections icons highlight in red to show that they will modify the orientation of an object on the layout. When the icons are not highlighted, they serve to determine the orientation for new objects.

The Editing Window

The *Editing Window* displays the part of the board that you are currently working on.

The contents of the *Editing Window* may be redrawn using the *Redraw* command which also redraws the *Overview Window*.

Panning

You can reposition the work area over different parts of the layout in several ways:

- By clicking left at a point on the *Overview Window* - this re-centres the work area about the marked point.
- By moving the mouse over the *Editing Window*, holding down the SHIFT key, and 'bumping' the pointer against one of its edges. This pans the display in the appropriate direction.
- By pointing in the *Editing Window* and pressing the F5 key. This re-centres the display about the cursor position.

- By using the *Pan* icon on the toolbar.

Zoom In / Zoom Out

You can magnify or reduce the display of the board using the *Zoom In* and *Zoom Out* commands which can also be invoked via keyboard shortcuts (by default, the F6 and F7 keys). Additionally, you can use the shortcut key for *Zoom All* (F8 by default) to display a view of the entire board. You can also use the corresponding icons on the toolbar.

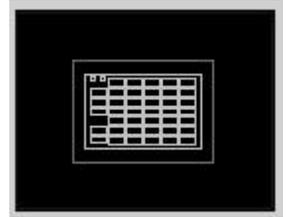
If the keyboard is used to invoke the command, then the *Editing Window* will be redrawn to display a region centred around where the mouse cursor was pointing before. This also provides a way to effect a pan by pressing the zoom key for the current level and simultaneously pointing with the mouse at where you want centre of the new display to be.

Variable Zoom

An arbitrary degree of magnification can be achieved using the Shift-Zoom feature. A given area of the board can be selected to fill the *Editing Window* by holding down the SHIFT key, pressing the left mouse button and dragging out a box around the desired area. The area can be marked on either the *Editing Window* or the *Overview Window*.

The Overview Window

This window shows a simplified representation of the whole drawing. The blue box marks the outline of the work area as set by the *Set Work Area* command whilst the green box indicates the region on view in the *Editing Window*.



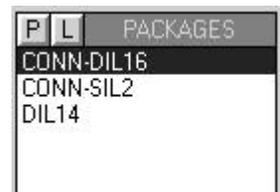
Clicking left at a point on the grid re-centres the *Editing Window* around this point.

The *Overview Window* is also used to display previews of objects that are selected for placement. This features helps you to orient (rotate and mirror) an object correctly before placing it on the board.

The width and height of the *Overview Window* can be adjusted by dragging its borders. If you drag the vertical border right over to the other side of the application Window, ARES will re-organize the display so that the *Overview Window* and *Object Selector* are located at the right hand side.

The Object Selector

The Object Selector is used for picking components, packages, pad & trace styles and so on from those that are currently available. It always carries a label indicating what it is listing and this serves as a prompt additional to the state of the Icon Panel



as to which mode is current.

The width and position of the *Object Selector* can be adjusted in conjunction with the width and height of the *Overview Window*, as described above.

The Layer Selector

The *Layer Selector* is used to display and select the current layer or layer set. A layer is a single layer, whereas a layer set is a combination of single layers, such as 'ALL'.



The layer selector can be operated with either the mouse or the keyboard. The keyboard controls are as follows:

SPACE	Selects the next layer in the current layer pair sequence.
PGDN	Selects the next layer down the list.
PGUP	Selects the previous layer up the list
CTRL-PGDN	Selects the last layer in the list.
CTRL-PGUP	Selects the first layer in the list.

Which layers are displayed in the list is determined by the current editor mode, such that PCB objects cannot be placed on inappropriate layers.

Which layers are displayed is controlled with the *Layers* command on the *View* menu whilst the layer pair sequences can be redefined using the *Set Layer Pairs* command on the *System* menu.

This latter command defines the layer that the SPACE key will select for each layer. Thus you can make pairs, triplets quadruplets or whatever.

Co-ordinate Display

The current co-ordinates of the mouse pointer are displayed down at the bottom right of the screen by default. The read-out can be in imperial or metric units and a false origin may be set. Further details are given under CO-ORDINATE SYSTEMS on page 31.

The *X-Cursor* command will display a small or large cross, in addition to the mouse arrow, at the exact location of the current co-ordinates. This is particularly helpful when used in conjunction with the Real Time Snap feature, since it gives you an immediate indication of what ARES thinks you are pointing at.

CO-ORDINATE SYSTEMS

The fundamental unit of linear measurement in ARES is 10nm (ten nanometers). Given 32 bit representation of the co-ordinates this allows a board size up to +/- 10m (ten meters) with

some headroom for calculations. The 10nm unit divides exactly into both 1um (micron) and 0.1 thou (one ten thousandth of an inch) giving exact representations of both imperial and metric dimensions to these resolutions.

Rotational angles are stored to 0.1degree.

The co-ordinate display can be switched to reading metric units by invoking the *Metric* command or via the keyboard shortcut (by default this is the 'M' key). Restoring imperial mode is achieved by invoking the *Metric* command again.

Dimension Entry Fields

A number of the dialogue forms contain fields that deal with dimensions such as pad style sizes, track widths, design rule clearances and so forth. All these fields are handled in a such a way that both imperial and imperial units can be used at will. The operation of these fields is in accordance with the following rules:

- Values are displayed in units chosen by a heuristic algorithm which detects whether a dimension is a whole number of imperial or metric units. Thus 25.4mm (however originally entered) will always display as 1in, when the field first appears.
- Displayed values always carry a unit. If you delete the existing value, the original displayed unit will be used.
- If you prefer, you can enter a new value with an explicit unit. Valid units are

th	thou	(10e-3 inch)
in	inch	
um	micron	(10e-6 meter)
mm	millimetre	(10e-3 meter)
cm	centimetre	(10e-2 meter)
m	meter	

- Whatever units are chosen, values must be less than +/- 10m, and are held at a resolution of 10nm.

Output Origin

For CAD/CAM output in particular, but also when working on a board which has to fit into a pre-designed mechanical casing, it is useful to be able to define a reference point on the board which relates to the mechanical design's co-ordinate system. The output origin defines this point within ARES, and is drawn in blue on the *Editing Window* as a target symbol.

- You can re-position the output origin using the *Output Origin* command.

- The output origin does not affect the co-ordinates used in Region files.

False Origin

Although the *Origin* command appears on the *View* menu, it should only be used via its keyboard short cut (by default this command is mapped to the 'O' key – information on customising your own keyboard shortcuts can be found on page 61). Its function is to zero the co-ordinate display at the current mouse position, a feature that is most useful when laying out a complex pattern of pads given a set of dimensions on a drawing of the component.

When a false origin is set, the co-ordinate display changes colour from black to magenta as a reminder.

Cancelling a false origin is done by invoking the *Origin* command a second time.

Grid Dots

A grid of dots is displayed in the *Editing Window* - this can be toggled on and off using the *Display-Grid* command. The spacing of the dots normally equals the current snap setting (see below), but if you are zoomed out a good way, it will be multiplied by a factor of 2,4,8 etc. The threshold at which this occurs can be adjusted on the *Set Grids* command on the *System* menu.

Snapping Grids

Although the motion of the mouse arrow itself is smooth, you will see that the co-ordinates normally jump in fixed multiples of, for example, 50 thou. The size of these multiples is set by the *Snap* commands and the purpose of this is to facilitate the placement of objects on a fixed grid.

Under normal circumstances, boards tend to be routed on a 50 or 25 thou grid.

Three snap grids for each of imperial and metric modes are available at any one time, though you can redefine their values using the *Set Grids* command on the *System* menu. The current snap options are selectable from the *Display* menu or by using shortcut keys (the preconfigured keys for the snap commands are F2-F4).

The snapping points are computed starting from the current origin as set by the *Set Origin* command. This provides the means to measure out pad placement for a new component, since you can set the origin at the first pin, set one of the snap values to the pin spacing, and then advance the cursor along the pad positions using the cursor keys.

Real Time Snap

As well as snapping to the grid dots, ARES will also snap the cursor to pads and/or tracks which lie off the currently selected snap grid. This process takes place in real time - as the pointer is moved - hence the name Real Time Snap.

The rules for what is snapped to are as follows:

- Pads are always snapped to. Only pads whose layer range overlaps the current layer range are scanned.
- Tracks are snapped to when either the *Trace* icon or *Via* icon is selected. Only tracks on the current layer are scanned.
- Ratsnest lines are snapped to when the *Ratsnest* icon is selected.
- The range within which the pointer will 'see' an object is either half the current snap spacing or, if grid snap is disabled, the actual bounds of the object concerned.

RTS is extremely useful when routing between components whose pins are on different grids, since it alleviates the need to constantly switch between routing grids. In this context, the *Trace Angle Lock* feature is also very important.

With a large board or a slow computer, RTS can result in a noticeable lag between the motion of the pointer and the cursor. In this case, you may find it helpful to disable RTS except when you really need it using the *Real-Time-Snap* command on *Tools* menu.

OBJECT PLACEMENT

There are five basic types of object in ARES:

- Components
- Packages
- Pads
- Graphics
- Zones

All are placed in much the same way - select the appropriate object type from the toolbar and then click left on the *Editing Window* to place the object. If you hold the left button down, you can move the object around before actually placing it. Also, when appropriate, the *Rotation* and *Mirror* icons can be used to pre-set the orientation of placed objects.

Placing Components

ARES makes a distinction between *Components* and *Packages* in that a component is a part specified in the netlist whereas a package is not linked to the netlist, has no part ID, and plays no part in netlist driven design verification / modification. Assuming you are using a netlist,

LABCENTER ELECTRONICS

99% of the parts you place would usually be components. You would only use packages for parts which, for whatever reason, were not entered on the schematic. Heatsinks are the most common example of this. If a netlist is in use, parts which are not specified in it must not be given names.

To place a component

1. Select the *Component* icon from the *Mode Selector* toolbar. In this mode, the *Object Selector* lists all components not yet placed.
2. Choose the component you want to place from the *Object Selector*.
3. Set the *Rotation* and *Mirror* icons to determine the required orientation.

You can set the rotation to non-orthogonal angles by typing into the text edit-box.

4. Set the *Layer Selector* to determine the required layer. Note that the *Layer Selector* and the *Mirror* icon are interlocked, since a reflected component must be on the underside of the board.
5. Point at the desired position for the component on the board and press the left mouse button.
6. If you hold the button down, you can drag the component around before it is placed; at the same time, ARES will show ratsnest lines and/or a force vector to indicate how the component connects to any others already placed.

Further, the '+' and '-' keys on the numeric keypad will rotate the component anticlockwise and clockwise respectively in steps of 90 degrees.

Components on the underside of the board

When a component is placed on the solder side of the board, it is effectively 'turned over'. This means that it suffers both a reflection in 2D space and an inversion of the layers of its pads. More specifically:

- The silk screen legend moves from the Top Silk Screen to the Bottom Silk Screen.
- Pads on the Top Layer only, move to the Bottom Layer and vice versa. This caters for underside placement of surface mount components.
- Pads on multiple layers - most commonly normal through hole pads - and padstacks are left alone. On the assumption that a padstack has been defined in a particular way to reflect the usage (signal, ground plane, etc.) of a particular layer, it would be unhelpful to invert pad stacks for solder side placements.

Packages

As discussed above, the term *Package* is used to refer to a placed library part which has pins but is *not* linked to the netlist. In ARES, they are useful in the following circumstances:

- If no netlist is available, or you are doing a simple 'quick and dirty' job, you can lay out a board by placing packages and wiring them up using traces and vias. In doing this, you are using ARES as a computerised light box, rather than as a true EDA tool.
- Sometimes you need to place objects which have pads and graphics but are not actually components in the electrical sense of the word. Heatsinks with mounting holes are a common example of this. These can be placed as unlabeled packages, and as such, will play no part in the netlist driven functions of the design work.
- In order to edit a library-package, it is necessary to place it as a package, tag it and then decompose it using the *Decompose* command. The individual elements can then be edited as required prior to re-making the library-package using the *Make Package* command.

To place a package

1. Select the *Package* icon from the *Mode Selector* toolbar. In this mode, the *Object Selector* lists all the packages picked from the libraries.
2. Assuming that the package you want has not yet been picked from the libraries, click the 'P' for pick toggle on the *Object Selector*.
3. Pick the package or packages you want from the *Library Pick Form*.
4. When you have finished picking packages, close the form in the usual way.
5. Highlight the package you want to place in the *Object Selector*.
6. Proceed from step [3] in *To place a component* above.

 Some further information on creating library-packages is available under THE PACKAGE LIBRARY on page 64.

Pads

The main reason for placing lone pads is when defining a new library-package but they are also useful for test points, drilling alignment targets and such like. ARES offers five shapes of pad, namely: Circle, Square, DIL, SMT and Edge Connector.

In addition, ARES supports pad stacks, in which a different pad shape can appear on each layer of the board.

The various pad shapes may be selected from the *Mode Selector* toolbar.

Given the selection of a pad shape, an actual pad style of that shape may be selected from the *Object Selector*. You'll see that all the pad styles have names - this is a unique feature of ARES and has two advantages over other systems:

- There is less chance of confusion or error when choosing a style (provided that you use sensible names!).
- There is no limit to the number of styles you can have.

Pads are placed in much the same way as components and packages. The *Rotation* icon affects the placement of DIL, SMT and Edge Connector pads whilst the *Layer Selector* again determines the layer-set of the pads.

You can edit a pad style by selecting it in the *Object Selector* and then clicking on the toggle. Any changes to the pad dimensions take effect as soon as the display is redrawn - you can force a redraw with the *Redraw* command. This feature is particularly powerful should you find the need to tweak pad styles globally in order, for instance, to change the pad sizes used in the standard package library.

New pad styles may be created using the *New Pad Style* command. See *Style Management & DEFAULT.STY* on page 59 for more information on styles.

2D Graphics

The 2D drawing facilities are intended, in the main, for adding text and graphics to the silk screen layers though they can also be placed on other layers. Objects placed on the *Edge* layer appear on artwork produced from any of the other layers making it a suitable location to draw the board outline.

There are 8 types of 2D graphic object, namely:

- Line
- Box
- Circle
- Arc
- Path
- Text
- Symbol
- Marker

The corresponding icons are contained in the *Mode Selector* toolbar.

- Lines are drawn by clicking left at each end.

- Boxes and circles are dragged out using the left button.
- Arcs are placed dragged out into a quadrant using the left button. They can be further adjusted by tagging and dragging the four handles. ARES arcs are actually Bezier curves.
- Paths are closed figures with boundaries defined as multiple line and arc segments. Click left at each required vertex and hold the CTRL key down to place arc sections. Further adjustments may be made by tagging the path and moving the ‘handles’ about.

We strongly recommend that paths are not used to create copper fills as they are not-recognized by the connectivity database. Use zones instead.

- Text is placed by clicking at the bottom left corner of the text region. A form then appears which allows input of the text itself and also allows control of the text dimensions. Text orientation can be set using the Rotation and Mirror Icons.
- Selecting the *Symbol* icon displays symbols in the *Object Selector*. Symbol selection is identical to package selection as described earlier. More information on the symbol library facility is contained in THE SYMBOL LIBRARY on page 68.
- Markers are used in the creation of library parts to define the origin, and also the position for component labels.

Zones

Zones represent the means by which ARES handles power planes. In this section we will just describe briefly how to place a simple power plane; further documentation on power planes is given under POWER PLANES on page 97.

To place a simple power plane

1. Select the *Zone* icon from the *Mode Selector* toolbar.
2. Select the required boundary trace style for the zone from the *Object Selector*.
3. To define the zone boundary, either:
 - Drag out a box with the left mouse button held down.
 - Click at several points to form a polygonal boundary.

If a zone boundary is placed over existing tracking, it will pick up its net from that tracking. This makes it quite easy to place lots of small zones in order to ‘fill in’ between tracks.

4. Select the required net and hatch/fill style from the *Edit Zone* dialogue form, then click OK.
5. ARES will generate the ground plane.

OBJECT EDITING

Due to its object oriented design, all objects in ARES are treated in much the same way and we can discuss generally their dragging, copying, movement, rotation, deletion and editing. You should note that traces and vias are not regarded as objects for this purpose; their editing is dealt with further under Manual Routing on page 22. However, there are operations where traces attached to objects are affected by operations performed on the object - such cases are discussed here.

Tagging a Single Object

Any object may be tagged by pointing at it and clicking right. This action highlights the object and selects it for further editing operations.

Objects are regarded as occupying a layer-set, and will be found if the current layer-set as indicated by the *Layer Selector* overlaps that occupied by the object. This is necessary since two objects may occupy the same area on different layers, as in the case of component and solder side pads being placed for an edge connector. However, it does mean that you must remember to check/set the *Layer Selector* appropriately before tagging.

- When you tag an object, the layout editor will change to the mode that is used to place that type of object. This is done so that the *Layer Selector* can then offer the correct set of layers for that object.
- Component outlines occupy the silk screen layer, but component pads occupy the copper layers.

Tagging a Group of Objects

A group of tagged objects may be assembled by either tagging each one in turn as described above, or by dragging a box around them using the right mouse button. Only objects wholly enclosed in the box will be tagged and, unlike tagging a single object, all layers are scanned.

The box is called a tag-box, and will remain on the screen until all the objects are untagged as described in the next section. The tag-box is of particular significance as regards the block editing commands and their effect on any routing within it.

Only one tag-box can exist at any one time - drawing a new one will delete the old one, and untag the objects within it.

Untagging All Objects

To untag all the objects, you need simply to point where there is no object and click right. This will also remove the tag-box if one has been defined.

Performing an explicit *Redraw* command (default key mapping 'R') will also untag all objects - this can be handy if the board is very densely packed and there is nowhere free of objects on the screen.

The Tag Filter

The tag filter determines the type(s) of objects that are tagged when a tag box is drawn. This makes it possible to tag only components, only tracks and so forth. In addition, you can select the layers upon which objects are tagged.

In conjunction with the *Block Delete* command, this makes it possible to delete particular objects on particular layers.

If you change the tag-filter whilst a tag-box is present, the effect of the tag-box will be re-applied using the new filter settings.

The tag-filter dialogue form can be displayed using the *Tag-Filter* command on the *Edit* menu, or via the keyboard shortcut (by default this is CTRL-X). More information on configuring your own keyboard shortcuts can be found on page 61

Deleting an Object

Any tagged object can be deleted by pointing at it and clicking right. Any traces connected to its pads will remain in place, allowing a different pad or package to be placed over them.

Dragging an Object

You can drag (i.e. re-position) any tagged object by pointing at it and then dragging the mouse with the left button depressed.

Any traces attached to the object will be rubber banded (stretched) unless both ends connect to it, in which case the route will be moved wholesale.

Editing an Object

Some objects, especially packages, components and 2D Graphics have properties that can be edited with a dialogue form. There are two ways to do this:

- A tagged object can be edited by pointing at it and then clicking left as if to drag it but without moving it.
- If you have several objects to edit it may be more convenient to select the *Instant Edit* icon. Clicking left on any editable object will then bring up its dialogue form.

Highlighting a Component by Name

When the *Instant Edit* icon is selected, the *Object Selector* lists all the components in the design. Clicking on the selector toggle tags the selected component causing it to be highlighted, and displayed in the centre of the screen.

Alternatively, you can locate a component using the *Goto Component* command on the *View* menu.

ROUTE PLACEMENT & EDITING

ARES provides a variety of methods for the manual placement and editing of tracking and vias. In particular, existing routes may be modified by any of the following methods:

- Over placement - place a new section of route over an old one and the new will replace the old. This provides an easy way to change the thickness of sections of tracking.
- Segment dragging - the path of a tagged route can be easily changed by dragging its segments. Neighbouring segments are automatically stretched so as to maintain an orthogonal path. This is a very intuitive method of working for users accustomed to other Windows drawing packages.
- Re-routing - if a new route is placed such that it replaces part of the path of a tagged route, then the bypass section of the existing route is automatically removed. This method is the most flexible, especially if the new section needs to change layers or have a significantly different number of segments. It is particularly effective when trying to clear space on the board towards the end of the routing process.

In all cases, ARES will validate newly placed or modified route sections against the connectivity data specified in the netlist. Sections of tracking which violate the connectivity rules are marked as 'dirty' and will flash in yellow to indicate that they are in illegal positions.

Trace Placement - No Netlist Loaded

Placing a new trace on the primary layer is done by first selecting the *Trace* icon and the required trace style from the *Object Selector* and then clicking left at each node along the required path before clicking right to finish.

Clicking left twice at the same location will cause a via to be placed and the next layer in the layer pair sequence to be selected. The type of via used can be changed by selecting the *Via* icon and choosing one of the via styles from the *Object Selector*. You can also place, replace, tag, drag and delete vias manually in this mode. The mouse buttons work in the same way as for ordinary objects.

You can change layers whilst routing using the *Layer Selector* keyboard controls: PGUP, PGDN, CTRL-PGUP, CTRL-PGDN.

Should you start to enter a trace, and then decide you wish to abort it entirely, simply press the ESC key.

Trace Placement - Netlist Loaded

If a netlist is loaded and you commence routing from a pin assigned to a given net, ARES will:

- Indicate the net name of the given net.
- Indicate to which other pins the first one connects to.
- Select the appropriate trace and via styles for the given net according to the strategy to which it is assigned. This function can be disabled by toggling the *Auto Trace Selection* option on the *Tools* menu. This can be useful if the odd part of a net needs to be a different thickness. See ROUTING STRATEGIES on page 77 for more information on strategies.

You can then place route segments and vias as described in the previous section. Assuming that you take the route to one of the indicated pins, ARES will automatically terminate routing mode when you click left on it.

Should you wish to connect to a point other than a selected pin, this is possible by clicking right to terminate routing as described in *Trace Placement - No Netlist Loaded* on page 45. ARES will still detect any ratsnest lines that have been connected and will remove them from the display though this takes a little longer than if you route directly from pin to pin. In fact, ARES will remove connected ratsnest lines even if neither end of the placed route starts on a pin - *Advanced Netlist Management* can check a whole net's connection status very rapidly.

ARES will also check whether the placed route connects to anything belonging to another net. This could be a pin, via, or part of another trace. This prevents major blunders where a

route is placed across several others which you failed to see. If such a violation is detected ARES displays an error message and the tracking will be marked as dirty. Dirty tracking flashes between bright yellow and its normal colours, except when it is tagged.

When *Auto Track Necking* is enabled, ARES also checks for physical design rule violations. In the first instance, it will neck the track to avoid them, but if there is still a violation then it displays a warning message and beeps. The route is, however, placed. This process can take some time, and if you find that trace placement is becoming unduly tardy, you can try disabling ATN from the *Tools* menu.

Curved Track Segments

ARES fully supports curved tracks and supports them as true arcs for output purposes. The segments also play a full role in connectivity and design rule checking.

To place a curved segment, first commence routing as usual (e.g. click left on a pad) but then, before moving the mouse, hold the CTRL key down. Move the mouse roughly to trace the arc, click left to fix the arc endpoint and then release the CTRL key. You can place a curved segment at any stage of laying a route - it does not have to be the first or last segment.

Two important points about curved tracks:

- Because ARES renders a curved track as an arc, the mechanism that 'backs off' a trace end from any pad to which it connects does not operate for curved track segments. It follows that if curved segment connects directly to a pad, the centre hole of the pad may be obscured.
- It is possible, using the route editing facilities described in the following sections, to break a curved segment into sub-sections. ARES will allow this but the sub-sections will then be treated as runs of short linear segments. As such, they may not render satisfactorily on pen plotters.

Auto Track Necking

In many cases, the reason for necking down a track is so that it can pass between two pads or other obstacles without violating the design rules. The *Auto Track Necking* feature allows ARES to do this for you.

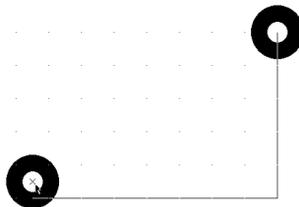
The function is controlled by the *Set Design Rules* command on the *System* menu. The dialogue form allows you to enter the clearances for pad-pad, pad-trace and trace-trace and also the trace style to neck to. The default neck style is T10 - a 10 thou track.

To disable ATN totally, use the toggle command on the *Tools* menu. You may find this speeds up route placement considerably, since the analysis involved in performing ATN can be quite complex, even if the end result is that the route is OK.

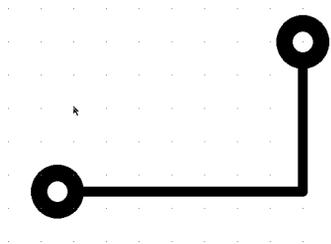
Trace Angle Lock

Many people like to keep all tracking running at either 90 or 45 degrees as this gives a tidy, professional look to a layout. To make this easier, ARES has a *Trace Angle Lock* command which restricts route segments to 90 or 45 degree angles only.

TAL also has a more subtle effect when combined with *Real Time Snap*. Consider the following routing problem... You are trying to route from a pad on the grid to one that is off it. You have moved down, clicked left and moved the pointer over to the destination pad. *Real Time Snap* has detected the pad, and the cursor has locked onto it. However, because your first click left was on a grid dot, the horizontal part of the track doesn't line up with the pad centre. The diagram below shows the position at this stage:



When *Trace Angle Lock* is enabled, clicking over a pad or track in this circumstance will activate *Trace Angle Fixup*. The result is that the previous node - in this case the route corner, is moved so that trace angle lock rules can be maintained. The result is a tidy connection as shown below overleaf.



Trace Angle Lock & Fixup are active by default. You can disable or re-enable them using the *Trace Angle Lock* command on the *Tools* menu.

Tagging a Route

To re-route, delete or copy a section of tracking requires that you first tag the route containing it.

To tag a route

1. Select the *Trace* icon from the *Mode Selector* toolbar.
2. Set the *Layer Selector* to the layer on which the trace lies.
3. Point at a part of the route on the selected layer, and click right.

If you click on a via, or pin, or a point at which three or tracks meet, all the tracking originating from that point will be selected.

Changing the Width of a Route

ARES provides two methods of changing track width. The first relies simply on the general ability of ARES to cope with new objects being placed over old.

To change the width of tracking by overplacement

1. Ensure that there is no tagged route, and that *Auto Trace Selection* is disabled on the *Tools* menu.
2. Select the new trace style in the *Object Selector*.
3. Place the new tracking directly over the old. ARES will work out that there is overplacement, and will replace the old tracking with the new in the database.

The second follows the principle of allowing various editing operations to be performed on the currently tagged object.

To change the width of a tagged section of tracking

1. Tag the required section of tracking as described under *Tagging a Route* on page 48. The *Object Selector* will display the current trace width. If the tagged route has several widths, then the width at the point under the cursor is displayed.
2. Click right on the tracking to display the context menu.
3. Select a new trace style from the menu.

Changing the Layer of a Route

To change the layer of a tagged section of tracking

1. Tag the required section of tracking as described under Tagging a Route on page 48.
2. Click right on the tracking to display the context menu.
3. Select a new layer for the tracking from the menu.

Modifying a Route

To rubber band a section of tracking

1. Tag the route as described under Tagging a Route on page 48.
2. Drag the segments and/or corners of the route using the left mouse button.

If you drag horizontal or vertical segments, they will move vertically or horizontally respectively with adjacent segments between stretched to accommodate this. On the other hand, if you drag on the route corners, then the adjacent segments will simply stretch diagonally. Finally, if you drag in the middle of diagonal segments, a new corner will be created.

To re-route a section of tracking

1. Tag the route as described under Tagging a Route on page 48.
2. Place a new section of tracking starting and finishing on the tagged route. ARES will then remove the section of tracking that the new route shorts out.

Note that vias which are made redundant (i.e. secondary vias) will also be removed, along with the shorted out section.

Copying a Route

To duplicate a route

1. Tag the route as described under Tagging a Route on page 48.
2. Click right on the tracking to display the context menu.
3. Select the *Copy* command.
4. Click left at each position for the duplicate routes; click right to finish.

This feature provides an excellent way do memory buses and other repeated patterns of tracking.

Deleting a Route

To delete a route

1. Tag the route as described under Tagging a Route on page 48.
2. Click right to display the context menu.
3. Select the *Delete* command.

Any interconnections specified in the netlist that were formed by the deleted tracking will automatically appear as ratsnest lines.

Note that you can also delete *all* tagged tracking by using the *Block Delete* icon. This can be quicker than the above procedure if no other tracking is tagged.

Tidying the Routes

Extensive route editing can lead to the creation of unwanted nodes between tracks segments that join at 180 degrees. These nodes waste memory and also degrade the quality of plotted output. One of the processes carried out by the *Tidy* command is to scan for and remove such nodes.

The *Tidy* command may be found on the *Edit* menu.

BLOCK EDITING COMMANDS

Four block operations are provided: *Copy*, *Move*, *Rotate* and *Delete* and each operates on the currently tagged objects and the traces & vias within the tag-box. If there is no tag-box, then a default one is assumed comprising the total area occupied by the currently tagged objects. If there are no tagged objects, nothing at all will happen.

Block Copy

After creating a tag-box as described in Tagging a Group of Objects on page 42, clicking on the *Copy* icon will cause a second box to appear over the tag-box which you can then re-position using the mouse. Clicking left will cause a copy to be made of the objects within the tag-box whilst clicking right will abort the operation.

Traces are only copied if both ends of them are inside the tag-box.

Beware that copying components will generally leave the netlist management out of kilter as there will be multiple placements of the same component references. If, having routed a module that is to be repeated several times, your intention is to copy it out to save re-routing,

you should use the *Auto Name Generator* to renumber the components in the copied section(s) and then re-load the netlist.

Block Move

After creating a tag-box as described in *Tagging a Group of Objects* on page 42, clicking on the *Move* icon will cause a second box to appear over the tag-box which you can then re-position using the mouse. Clicking left will cause the tag-box and its contents inside it to be moved whilst clicking right will abort the operation.

Trace segments with one end only inside the tag-box will be rubber-banded.

Block Rotate

After creating a tag-box as described in *Tagging a Group of Objects* on page 42, clicking on the *Rotate* icon will allow you to rotate and/or reflect the block by any angle.

- Trace segments with one end only inside the tag-box will be rubber-banded although this generally leaves something of a mess - it's best to rotate only completely enclosed sections of a layout.
- Rotation of whole groups of components to non-orthogonal angles (even 45°) creates large numbers of off grid pads. Routing such boards can be very tricky, even with features such as real time snap and trace angle fixup.

The autorouter, being grid based, will also perform poorly under such circumstances.

Block Delete

After creating a tag-box as described in *Tagging a Group of Objects* on page 42, clicking on the *Delete* icon will delete all tagged objects.

Remember that you can tag objects and tracking in a number of ways:

- Using the right mouse button to select objects individually.
- Using the tag-box and tag-filter to select specified objects within an area.
- Using connectivity highlight to select tracking on particular nets, or running from particular pins - see page 72 for more information.

FILING COMMANDS

ARES makes use of the following file types:

Layout Files	(.LYT)
Backup Files	(.LBK)
Region Files	(.RGN)
Library Files	(.LIB)
Netlist Files	(.SDF)

Layout files contain all the information about one board, and have the file extension 'LYT'. They contain within them copies of all packages and styles used on the board, so the complete design can be given to someone else solely by giving them a copy of the layout file. Backup copies of layout files made when saving over an existing file are given the extension "LBK".

A region of a board can be exported to a region file and subsequently read into another layout. Region files have the extension 'RGN' and are read and written by the *Import* and *Export* commands on the *File* menu. They are analogous to section files in ISIS. ARES region files are in an ASCII format and for the advanced user, this opens the possibility of manually editing the layout database, or else writing utility software to perform specialist operations on it. Import of data from other PCB design packages is also a possibility. Contact our technical support department or look on our Web Site if you want a copy of the format specification.

Package and symbol libraries have the extension 'LIB'. 6 libraries are supplied:

PACKAGE.LIB	Standard through hole footprints
SMTDISC.LIB	Discrete SMT footprints
SMTCHIP.LIB	IC SMT footprints
USERPKG.LIB	For your own footprints
SYSTEM.LIB	Standard symbols
USERSYM.LIB	For your own symbols.

Full details of library facilities are given under LIBRARY FACILITIES on page 63.

Native format netlist files have the extension 'SDF' standing for Schematic Description Format. Netlisting features are covered in detail in NETLIST MANAGEMENT on page 69.

Starting a New Layout

To start over on an empty work area, use the *New Layout* command on the *File* menu. The layout filestem is set to 'UNTITLED' until such time as you invoke the *Save As* command.

Loading a Layout

A layout may be loaded in various ways:

- From the command line as in:

```
ARES my_board
```

- By using the *Load Layout* option once ARES is running.
- By double clicking the file in *Windows Explorer*.

Saving a Layout

You can save a board when quitting ARES via the *Exit* option, or at any other time, using the *Save Layout* command.

- In both cases it is saved to the same file from which it was loaded; the old file being re-named. On the first occasion that the file is saved, it is renamed to "Last Loaded <filename>.LBK" whilst on subsequent occasions it is renamed to "Backup of <filename>.LBK". This means that a copy of the file as it was at the start of an editing session is always preserved.
- If no filename was given at load time or the *New* command was issued, the name 'UNTITLED.LYT' is used.

The *Save As* command allows you to save the layout to a file with a different name.

Import / Export

The *Export Region* command creates a region file out of all currently tagged objects. This file can then be read into another layout using the *Import Region* command. After you have chosen the region file, operation is identical to the block-copy function.

Some difficulty may arise if an attempt is made to import a region saved from a layout in which the pad/trace/via styles in use are different from the current one.

A common difficulty arises from exporting a region file from a design where you have made local changes to pad/trace styles. Region files do not save this information and so when you import the region file all pad/trace styles will revert back to their default values. The solution of course is to create new styles in the first instance reflecting the modifications that you want to make. You can then safely create your board and export/import it via region files into other designs.

Region files may also be loaded using the *Load* command, and this is done to facilitate the loading of data produced by the external converters DXFCVT and GERBIT.

Note that region files are not good way to panelize multiple boards for manufacture because netlist information is lost and imported ground planes will lose connectivity as a result. The *Gerber Viewer* provides a much better way to perform panelization.

Auto-Save

ARES has an auto-save facility whereby your work will be backed up automatically at regular intervals. The interval defaults to 15 minutes and can be changed using the *Set Environment* command on the *System* menu.

Should ARES terminate unexpectedly, the next time it starts it will look for the auto-save file from the previous session and prompt to reload it.

The auto-save file will be stored in your Windows temporary directory, normally addressed by the environment variable TEMP. PROTEUS auto-save files have the extension 'ASV'.

This feature is not an excuse for not saving and backing up regularly - there is no guarantee that the auto-save file will be recovered in all cases.

Back-up and Last Loaded Files

In addition to the auto-save ARES provides extra security for your layouts via a *Backup* and *Last Loaded* mechanisms.

Whenever a layout is saved any existing layout file on the disk is backed up to either a last-loaded or back-up file. For a layout with the file name MYFILE.LYT the last-loaded file has a file name of "Last Loaded MYFILE.LBK" and the back-up a file name of "Backup Of MYFILE.LBK". Note that a different file extension (LBK) is for back-up and last-loaded files so they don't clutter the *Load Layout* dialogue form. To see these files when loading a layout you must change the *Files Of Type* field (at the bottom of the *Load Layout* file selector dialogue form) to "*Backup Layout Files*".

For a new layout, the back-up process works as follows. The first save operation saves the layout file. For each subsequent save, any existing back-up file is deleted, the existing layout file is renamed to the back-up file name and the layout is then saved.

For an existing layout, the back-up process is slightly different. The first save operation after loading the layout deletes any last-loaded file and renames the current layout file as the last-loaded file. For each subsequent save (during the editing session) the normal back-up scheme applies – that is, for each save, any existing back-up file is deleted, the existing layout file is renamed as the back-up file and the layout is then saved.

In summary, the layout file (MYFILE.LYT) is always the last saved version of the layout and the back-up file (Backup of MYFILE.LBK) is usually the version of the file prior to the last

save. The last-loaded file (Last Loaded MYFILE.LBK) is a version of the layout that is known to load successfully and represents the version of a layout at the start of an editing session regardless of how many times you save the layout during that editing session.

PAD & TRACE STYLES

ARES features a sophisticated and flexible system for defining shapes and sizes for pads and tracks. Each type of pad and track is given a style name, and then this name is then used to refer to the style whenever it is used. The dimensions and other characteristics for each style are held in a layout-global table, and this makes it possible to change the dimensions of all the pads or tracks in a particular style extremely easily. At the same time, the use of names means that there is no serious limit on the number of pad and trace types that you can have per layout.

Pad Styles

ARES supports 7 types of pad, namely:

- Circular PTH
- Square PTH
- DIL PTH
- Circular SMT
- Rectangular SMT
- Polygonal SMT
- Edge Connector

and the list of styles defined for each type displays in the *Object Selector* whenever the appropriate pad icon is highlighted.

When one of these lists is displayed, you can edit an existing pad style by highlighting it in the selector and then clicking on the selector toggle (the 'E' symbol). All fields on the resulting dialogue form have context sensitive help associated with them.

Polygonal Pads

ARES supports a user defined, polygonal pad style in order to cater for parts requiring unusual pad shapes.

To define a new polygonal pad style

1. Using the 2D graphics *Line*, *Arc* or *Path* icons, draw a closed path to define the shape of the new pad style. It does not matter which layer you draw on for this purpose.

2. Place an *Origin* marker within the closed path to specify the origin of the pad style. The origin is used as the connection point for tracking, and also serves as the point of alignment if the pad style is used within a pad stack.

Note that the origin must lie within the closed boundary i.e. within the copper area of the pad.

3. Tag the 2D graphics and origin marker by drawing a tag-box with the right mouse button.
4. Invoke the *New Pad Style* command from the *Edit* menu.
5. Enter a name for the new pad style.
6. Set the *Polygonal* pad style type under *SMT*.
7. Click OK.
8. Specify a *Guard Gap* for the pad style, if required.
9. If you wish to create the style only in the current layout, and not in DEFAULT.STY, then select *Local Edit* instead of *Update Defaults*.
10. Click OK to create the new pad style.

Note that there is no method to change the outline of a polygonal pad once it has been made - you must re-define it from scratch.

Pad Stacks

The ordinary through hole pad styles can be placed on a single layer or on all the copper layers, but this does not provide a very convenient or satisfactory method for defining component pins which have different pad shapes on different layers. In particular, the ARES connectivity system does not recognize connectivity between single layer pads on different layers.

Instead, ARES provides the facility to define *Pad-Stacks*. There are several points about a pad stack:

- A pad stack can have a circular hole, a rectangular slot or be surface only. The latter type of pad stack is used where you wish to defined explicit styles for the solder resist and/or solder paste mask apertures.
- For each layer, you can assign a different pad style, or no pad.
- For obvious reasons the hole or slot diameter of a pad stack is the same for all layers.

- You must use a pad stack to create a pad shape with a slotted hole. You cannot specify a slotted hole in an ordinary pad style.

To define a new pad stack

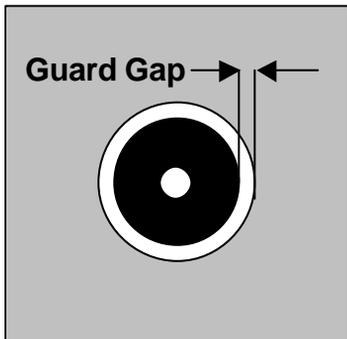
1. Select the *Create New Pad Stack* command from the *Edit* menu.
2. Enter a name for the pad stack, and a default pad style. All layers will start out with this style of pad.
3. Click OK to proceed to the *Edit Pad Stack* dialogue.
4. On this form, you can adjust the pad-layer assignments, and also assign the layer-global drill mark and drill hole/slot size.

To edit an existing pad stack

1. Select the *Pad Mode* and *Pad Stack* icons.
2. Select the pad stack you want to edit from the *Object Selector*.
3. Click the 'E' for edit toggle on the *Object Selector* to bring up the *Edit Pad Stack* dialogue form.

Pad Styles and the Solder Resist Layer

The solder resist artworks are generated automatically from the pad shapes present on the top and bottom of the board. Each pads' radius is enlarged by the amount specified in the *Guard Gap* field of its style, as shown overleaf:



Note that in the case of a pad stack, you can define the pad style used for the top and bottom resist layers explicitly, using the selectors in the *Edit Pad Stack* dialogue form. Note that the guard gap of this style is still applied in determining the final size of the aperture.

You can also specify a pad stack which has no solder resist - i.e. the pad will be covered by the resist coating.

Pad Styles and the Solder Paste Mask Layer

The solder paste mask artworks are also generated automatically from the SMT pads present on the board. The paste mask aperture is exactly the same as that used for the pad itself. Once again, in the case of pad stacks, different pad style may be specified explicitly for this purpose.

Trace Styles

Currently there is just one trace style, though in the future we may implement additional types for micro-strip etc. The available trace styles are displayed when the *Trace* icon is selected. You can then edit any the styles by highlighting its name and clicking on the selector toggle.

Trace styles have a single attribute which defines their width.

New trace styles can be created using the *New Trace Styles* command on the *Edit* menu. There is no way to delete trace styles once they have been created.

Via Styles

Via styles are much the same as pad styles except that only circular or square vias are permitted. The list of via styles is displayed when the *Via* icon is selected.

New via styles can be created using the *New Via Styles* command on the *Edit* menu. There is no way to delete via styles once they have been created.

Style Management & DEFAULT.STY

When you start a new layout, an initial set of styles appears and these come from a system file called DEFAULT.STY which is kept in the "Library" directory of your Proteus installation. This file contains a selection of common pad styles that we feel you may find useful.

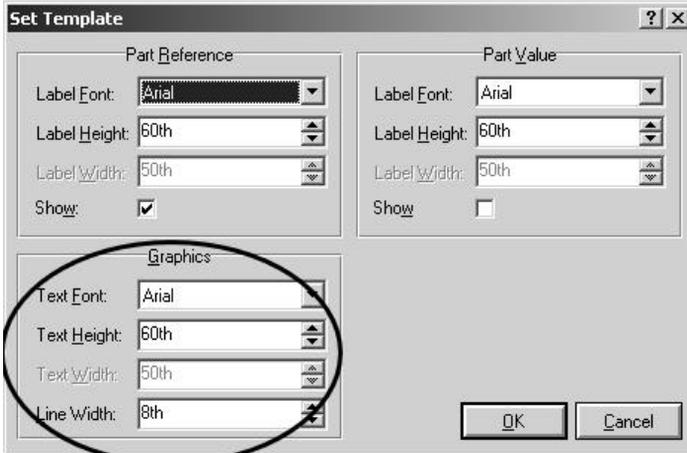
When you create or edit pad style you will see that the dialogue form contains a *Changes* box. If you select *Local Edit* then only the copy of the style maintained within the layout will be modified. If you select *Update Defaults* then DEFAULT.STY will also be updated.

Package library parts contain their own copies of styles used by their pads. When the package is brought into a layout for the first time, the style definitions will be taken from the library part if they do not already exist within the layout.

2D Graphics Styles

Global Defaults

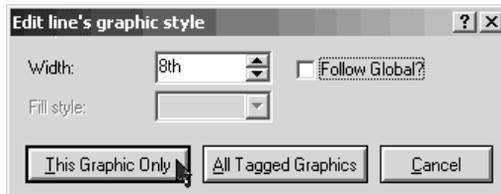
Default values for 2D Graphics are set via the *System Menu – Set Template* command as shown in the dialogue below.



After making changes to this dialogue remember to use *System Menu – Save Preferences* to commit changes to the registry. This will ensure that the global changes you have made persist across future designs (as well as being applied to the current layout).

Individual Configuration

You can edit specific graphics in the normal way (right click to highlight, left click to edit). An example dialogue from editing a 2D Graphics Box is shown below.



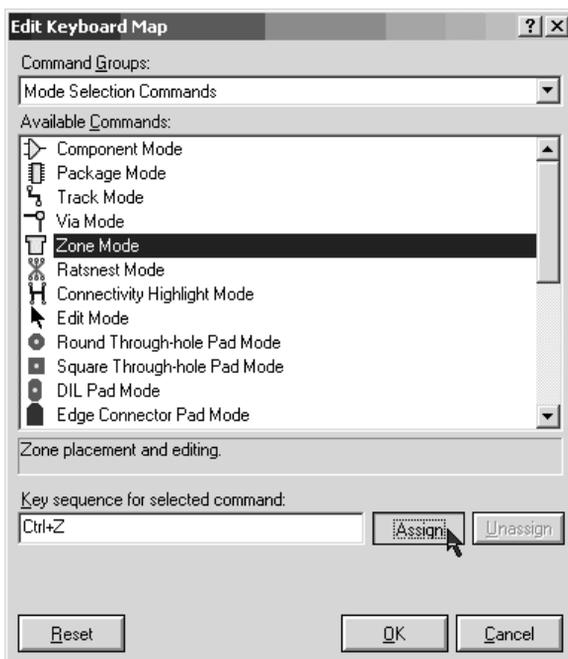
In order to change the width of the outline pen simply uncheck the *Follow Global* checkbox and enter the desired value in the edit field.

The fill style can also be edited for any 2D Graphics object which encompasses an area of free space. For example, you may well want to have a solid fill for a 2D graphics box but it has no meaning to have a fill style for a 2D graphics line.

Finally, you can tag multiple objects and then apply any changes to all tagged objects using the *All Tagged Objects* button. If you have multiple objects of different types tagged (for example, some graphics lines and graphic boxes) and exit with this button then ARES will update applicable changes to each object.

KEYBOARD CONFIGURATION

ARES provides an interface for you to map the keyboard to a command set. This allows you to 'bring out' the functionality you commonly use according to personal preference. The configuration dialogue, accessible via the *System Menu – Set Keyboard Mapping* command is shown below :



Notes :

- Accelerators can use any combination of CTRL, SHIFT and ALT keys. For example, CTRL+X, CTRL+ALT+T,SHIFT+CTRL+1, etc.
- A *Reset* option is provided to reload all Proteus standard shortcuts. This will overwrite any configuration you currently had.
- Some European keyboards have an ALT GR key (normally found directly to the right of the space bar). Where present this translates to be CTRL+SHIFT for the purposes of assigning keyboard shortcuts.
- If you assign an ALT accelerator that is used as a menu accelerator then you will no longer be able to accelerate the menu from the shortcut. For example, ALT+T by default expands the *Tools Menu* within ARES. If you assign ALT+T to the *Tidy command* then you would have to have to access the *Tools Menu* using the mouse.
- Some commands are locked and their keyboard shortcuts cannot be assigned or re-assigned to another command. The assignment box on the dialogue box is grayed out for these commands.
- When the NUM LOCK is on the keys 0-9 on the numeric keypad are distinct from the 0-9 keys on the main keyboard. For example, you could assign CTRL+0 up to CTRL+9 on the main keyboard as shortcuts to the Most Recently Used file list (MRU list) and on the numeric keypad you can assign CTRL+NUM-0 up to CTRL+NUM-9 for different commands.
- When the NUM LOCK is off keys 09 on the numeric keypad action their secondary functions and these functions are considered identical to those on the main keypad. The key for the number 5 on the numeric keypad has no secondary function when NUM LOCK is off.
- You can save your custom keyboard configuration via the '*Export Keyboard Map*' command on the *Options* tab. This allows you, for example, to port your keyboard configuration to another computer. Similarly, the '*Import Keyboard Map*' command allows you to reload a custom keyboard configuration that has previously been saved.

LIBRARY FACILITIES

GENERAL POINTS ABOUT LIBRARIES

As supplied there are 2 symbol libraries (which are shared with ISIS):

SYSTEM.LIB
USERSYM.LIB

and 6 package libraries:

PACKAGE.LIB
CONNECTORS.LIB
SMTCHIP.LIB
SMTDISC.LIB
SMTBGA.LIB
USERPKG.LIB

Detailed information - including pictures of all the footprints - can be found in the file LIBRARY.PDF. You will need to install the Adobe Acrobat reader if you have not already done so.

You can, of course, create further libraries of your own using the Library Manager

Library Discipline

USERSYM.LIB and USERPKG.LIB are set to read/write; the rest are set to read-only. The idea is that you should only add things to USERSYM.LIB (new symbols) and USERPKG.LIB (new packages). This means that we can supply you with updates to the parts we have defined without risk of overwriting similarly named objects in your own libraries.

If you must change things in the read-only libraries, you can set them to read/write using the *Properties* command in *File Manager* under Windows or with the DOS ATTRIB program. The command

```
ATTRIB filename -R
```

sets a file to read/write whilst

```
ATTRIB filename +R
```

sets a file to read-only. Wildcards may also be used.

The *Library Manager* also includes an option to toggle the read/write status of libraries.

The Pick Command

The *Pick* command serves as an alternative to the library browser, primarily for when you know the name of the package or symbol you are looking for. You can search for an exact match, or else use the various pattern matching options to search for a part when you have a rough idea of its name.

If you pick a package or symbol that is already loaded into the layout, ARES will update it from the libraries on disk.

Note that where there are two or more packages or symbols with the same name spread across several libraries, the *Pick* command will load newest one - i.e. the one most recently created. This is generally a helpful behaviour in that if you have re-defined one of our parts and put it in your USERPKG.LIB, your version will be newer than ours.

The Tidy Command

The *Tidy* command on the *Edit* menu performs several functions, one of which is to remove all packages and symbols from the selectors that are not actually in use on the layout.

This saves memory and also simplifies picking the ones you are actually using.

THE PACKAGE LIBRARY

A package is a collection of pads and silk screen graphics used to site a component on the board. A library of many common package styles is provided and this will save you a great deal of time compared with manual methods of PCB layout. However there will be occasions when you will have to create your own packages, and the steps to follow are set out overleaf.

Making a Package

To make a new library package:

1. Select the appropriate pad shape icons and place the pads of the package as required. Use the *Layer Selector* to place the pads on the appropriate layers. See below for more information.
2. By default, ARES will give the pins numbers from '1' upwards, numbering them in the order they were placed. If this is not what is required, you must either renumber them with the *Auto Name Generator*, or else edit them manually. See below for more information.
2. Select the appropriate 2D graphics icons and place silk screen graphics as required. A package can have legends on both the component side and solder side screens; these will be swapped if the package is placed with the mirror icon active.
3. Tag all the objects by dragging a box round them.
4. Invoke the *Make Package* command on the *Edit* menu, type in a name and select the library to store it in. The package will then be stored in the chosen library and added to the contents of the *Object Selector*, ready for immediate placement.
5. The reference or 'anchor point' for a package can be determined by placing the ORIGIN marker. If there is no ORIGIN marker, the reference will be the centre of first pin placed. Markers are accessed by selecting the *Marker* icon.
6. It is also possible to define non-default positions for the reference and/or value labels. To do this, place the REFERENCE or VALUE markers at the required position(s).

Choosing the Correct Layers for the Pins

Ordinary pad styles can be placed on single copper layers, or on all the copper layers as defined by the current setting of the *Layer Selector*.

- For ordinary through hole parts, you should place the pads on all the layers.
- For SMT parts, you should place the pads on the top layer only. If such a part is placed on the Solder Side, the pads will be automatically transferred to the bottom copper layer.
- For doubled sided edge connectors, you should place separate fingers on both the top and bottom copper layers.
- If you need different shaped pads on different layers for a through hole pin, you should use pad stacks as defined under *Pad Stacks* on page 56.

The Replicate Command

ARES provides a command to replicate tagged objects at a specified spacings in X and/or Y. This can be extremely useful when creating new packages as you can use it to lay out rows of pads with the spacings entered directly from the keyboard.

To place a row of pads by replication

1. Place a single pad of the require style at the first position of the row.
2. Ensure no other objects are tagged, and then tag the pad with the right mouse button.
3. Invoke the *Replicate* command from the *Edit* menu.
4. Enter the required X-Step (for a row) or Y-Step (for a column). Remember to enter a dimension suffix for these values - e.g. 20th or 1.5mm.
5. Enter the number of copies - this should be one less that the total number of pads required as you have placed one already.
6. Click OK - ARES will replicate the original pad as specified.

☞ You can create grids of pads by replicating rows.

Pin Numbering

When a netlist is loaded, ARES attempts to match pin numbers appearing in the netlist with the pin numbers assigned to the pins of the component packages. Where a match cannot be found, this is an error and either the ISIS library part or the ARES library part must be changed.

When you are defining an ARES library part, there are three ways to control the pin numbering:

- By default, the pins will be numbering from '1' upwards in the order that they were placed.
- If you invoke the *Auto Name Generator* command on the *Tools* menu, you can then click on the pads in any order you like to assign incrementing pin numbers.
- If you select the *Instant Edit* icon, you can click on each pin and key in the pin numbers manually.

Note that trying to use a mix of default and manual pin numbering is likely to lead you to confusion and mistakes.

Other Points

ARES normally chooses locations for the part ID and value; these labels can always be moved after a part has been placed by tagging it, pointing at the label you wish to move, and dragging with the left mouse button depressed. The locations chosen will generally be good enough as is, unless many components are closely packed together.

However, if you wish to force the label to a particular default position, you can use the REFERENCE and VALUE markers, as described above.

The size of the labels is set by the *Set Template* command on the *System* menu. This will apply globally except for labels whose size has been manually edited.

Editing a Package

To edit an existing library package,

1. Select the *Main Mode* and *Package* icons.
2. Pick the package to be edited from the library (using the toggle on the *Object Selector*) and place it on the work area.
3. Tag the package and then invoke the *Decompose* command from the *Edit* menu. This will break the package into its constituent pads and 2D graphics and also place an ORIGIN marker at the location of the part's placement reference.
4. Edit the pads and graphics as required.
5. When you have finished you can use the *Make Package* command to re-store the package either with the same name as before, or as a new part.

Of course, if the original package came from PACKAGE, which is normally read only, you will have to store it back to USERPKG. We recommend against using *Library Manager* to re-copy the part to PACKAGE as this can cause problems when we issue a new PACKAGE.LIB - how are you going to sort out which parts have been edited by you in order to preserve them?

Note that the pins will carry the pin numbers from the original package so decomposing a DIL14, sticking two pins on the end, and then re-making is not the way to make a DIL16 unless you are prepared to re-number all the pins manually.

THE SYMBOL LIBRARY

A symbol is a group of graphic objects which are treated as a single object. For instance, using 3 lines and 2 arcs you can form an AND gate symbol.



A symbol may be created by tagging the objects you wish to form it and then invoking the *Make Symbol* command on the *Edit* menu. Layers are ignored for this purpose. A form will appear allowing you to name the symbol which will then be stored in the Symbol Library and made available for immediate placement from the *Object Selector*. If there is already a symbol with the same name, it will be replaced.

ARES quite happily allows a symbol to contain other symbols and/or other graphic objects. This allows you to make, for instance, a NAND gate out of the previously defined AND gate plus a circle.

As with packages, the placement reference for a symbol may be determined by placing an ORIGIN symbol.

ISIS and ARES symbols are essentially exchangeable, and all symbol libraries have the type 'PROTEUS SYMBOL LIBRARY'. However, ARES does not support all of the drawing appearance features of ISIS, so correct rendering of ISIS symbols with complex fill patterns is not guaranteed, especially for CAD/CAM output where we can say with some certainty that *it will not work!*

NETLIST MANAGEMENT

NETLIST FEATURES

A netlist produced using ISIS or some other schematic capture package contains as a minimum, a list of the components used in the design and a specification of how their pins are to be connected. The ability of ARES to use this kind of information distinguishes it from budget packages which provide what amounts to a computerized version of rub-down transfers and drafting film.

Our own netlist format is called SDF which stands for Schematic Description Format. As well as component name and connectivity information, an SDF file can also detail information concerning the package to be used for each component and the routing strategy to be used for each net. As a result, it is possible to completely specify a PCB with an SDF file (and therefore from within ISIS), save for the physical positions of the components and the detailed routing of the interconnections.

The Load Netlist Command

The *Netlist Loader* is the means by which you import the data held in an SDF file into the current PCB layout.

A netlist can be loaded with the work area completely empty, with a set of placed and pre-annotated components on the work area or, perhaps in order to effect a modification to an existing design, with a completely placed and routed layout loaded.

Using the Netlist Loader on an Empty Layout

With no components placed, all parts specified in the netlist are simply noted and displayed in the *Object Selector* ready for placement as described in *Placing Components* on page 35.

The Netlist Loader & Existing Components

Where components or annotated packages are already on the board, any that have been given the same names as those in the netlist are linked into the netlist database. They will then play a full part in subsequent netlisting activities such as ratsnest compilation. Components or annotated packages which are not specified in the netlist are tagged. This is a half way house between ignoring them and deleting them - one click on the *Delete* icon will remove them if they really are superfluous.

ARES checks to see if the library package specified for a component in the netlist matches that of the placed component on the layout. If the specified package differs, one of two things will happen:

- If the new package has the same number of pins as the existing one, then a replacement will occur such that the new package is placed with its pin 1 lying over the old package's pin 1. The component is tagged to highlight the fact that something has happened to it.
- If the new package has a different number of pins, the component is removed from the layout and added to the *Object Selector* for manual replacement.

Unannotated packages are ignored by the netlist loader. This makes them a suitable way to place footprints for parts which, for whatever reason, you do not want subject to full rigors of netlist based design. You should also note that because their pins are not specified in the netlist, they can be connected to anything without CRC violations occurring.

The Netlist Loader & Existing Tracking

In the case where there is tracking as well as components on the board, ARES checks the connections made by the tracking to see whether they agree with the netlist. Where tracking is found that joins two nets (presumably as a result of a change to the schematic) the traces and vias involved are assigned to the VOID net.

VOID tracking and vias appear in flashing yellow, are ignored by the connectivity scanner and are not printed/plotted. They can be removed in one of two ways:

- By invoking the *Tidy* command on the *Edit* menu. This does other things too and can take some time on a large board.
- By selecting the *Connectivity Highlight* icon, then selecting the VOID net in the *Object Selector* and finally clicking its toggle to highlight the VOID tracking. Clicking on the *Delete* icon will then remove them.

VOID tracking shows you which bits of tracking need to be removed after a design change. If, having seen this, you decide against the design change, the procedure is to quit ARES without saving and then restore the schematic to its original state.

New connections specified in the netlist will automatically appear as extra ratsnest lines.

Problems with Pin Numbers

As the netlist is loaded, ARES pulls in the specified package for each component and then attempts to match the pin numbers used for that component in the netlist against the pin numbering in the library-package. If the netlist references a pin number that does not appear

in the library package then it displays an error message box and you must click OK to acknowledge. Problems in this area usually arise from one of the following causes:

- Specifying the wrong package - e.g. a bipolar transistor outline for a regulator.
- Failing to manually number pads which require non numeric 'numbers'. A typical example might be a DIN connector which has pin numbers like A1, A2 etc. Remember that by default, ARES always numbers the pads in the order you placed them.
- Leaving spaces on the ends of ISIS device pin names/numbers or in ARES pin numbers. This can be checked by editing the relevant object and examining closely the state of the baseline of the Data Entry Field on which the text sits. If there are any gaps in it, these designate the presence of spaces.

If you use the *Packaging Tool* in ISIS when creating new library parts, you should avoid any of the above problems.

Packaging Considerations

At some point during the design process, it is necessary to specify the library-package to be used for each component. With the ISIS/ARES system, this can be done:

- At the ISIS end by storing the package name in the **PACKAGE** property. This can be done either manually by editing each part in turn, or automatically using the Property Assignment Tool and/or ASCII Data Import facilities

This approach is by far the preferred one as it avoids having to re-enter data each time the netlist is loaded.

- At the ARES end as the netlist is loaded.

In the latter case, ARES will prompt you for the package name to use for each component in the netlist.

A limited facility is provided to automate this process for commonly used parts - the file DEVICE.XLT can contain lines specifying device/package pairs such as

```
7400 , DILL14
```

To use this feature, you must check the *Enable Device->Package Lookup* checkbox on the *Environment Settings* dialogue form. This command resides on the *System* menu.

You can then add entries to the DEVICE.XLT file either with a text editor or by selecting the *Store* button on the package input dialogue form.

Connectivity Highlight

Connectivity Highlight mode is selected by clicking left on the *Connectivity Highlight* icon. You can then:

- Highlight/tag a group of pads, tracks and vias connected to a pad by clicking left on the pad.
- Highlight all pads, tracks and vias connected to a net by selecting the required net from the *Object Selector* and then clicking left on the 'T' toggle.
- Delete all tagged pads, tracks and vias by clicking left on the *Delete* icon. Combined with the above this provides the means to rip up all tracking associated with a given net.
- Untag all pads/tracks/vias by clicking right at a point where there is no object or else by invoking the *Redraw* command.

RATSNEST FEATURES

The term *Ratsnest* is used to describe the pattern you get on the screen when the pin to pin connections specified in the netlist are shown as single straight lines rather than copper tracking. Once a netlist has been loaded and the components placed (or vice versa for that matter) the ratsnest gives a good visual impression of the complexity of the routing task that is before you or the autorouter. Furthermore, it gives an indication of the quality of component placement since the presence of lots of long ratsnest lines suggests that some components would be better placed closer together.

Automatic Ratsnest Recalculation

ARES keeps the ratsnest up to date and optimized at all times. Add a component or delete a track and new ratsnest lines will appear. Make connections and ratsnest lines will disappear. It's as simple as that.

The term 'optimization' refers to that fact that ARES always displays the 'Minimum Spanning Tree' for each net. This means that the displayed ratsnest lines always represent the shortest interconnection pattern between the pads.

The ratsnest can be turned off wholesale by using the checkbox on the *Layers* dialogue on the *Display* menu. Ratsnest lines related to a particular strategy can also be turned on and off using the *Set Strategies* command on the *System* menu.

Force Vectors

Force vectors are provided as a further aid to component placement. They appear as yellow arrows which point from the centre of each component to the point where it's ratsnest length (as defined by its current ratsnest lines) would be shortest. Another way of thinking about them is to consider each ratsnest line as a rubber band. The force vector then points to the place where the component would move to if released (strictly speaking this would also depend on the elastic linearity of the rubber bands, but we won't get into that - it's a good analogy!)

One oddity that can arise is that if you move the component to the position marked by the tip of its force vector, the ratsnest itself may change because a better minimum spanning tree may then be found. This can mean that the force vector then points to somewhere slightly different. We do not regard this as solvable; most PCB packages do not re-optimize the ratsnest after each edit, so such effects would not be so obvious - but they are still there.

The force vectors can be turned off by using the checkbox on the *Layers* dialogue on the *Display* menu.

Ratsnest Mode

Ratsnest mode is selected by clicking the *Ratsnest* icon. You can then

- Specify connections manually - useful if you are working without a schematic but still wish to use the auto-router.
- Perform pin and gate-swaps (where legal) by dragging tagged ratsnest lines from one pad to another.
- Tag a connection by pointing at it and clicking right.
- Tag all the connections for one net by selecting the net in the *Object Selector* and clicking left on the 'T' toggle.

In conjunction with the autorouter's ability to route all, tagged or untagged connections, ratsnest mode gives you the means to autoroute the board selectively.

Manual Ratsnest Entry

If you are working without a schematic, it is possible to enter ratsnest connections manually. Connections are specified by clicking left the pads to be interconnected. ARES will automatically create net-names for connections specified in this way.

LABCENTER ELECTRONICS

Note that where component pads are on single layers, you must use the layer selector to select the correct layer for each pad. If a ratsnest line must change layer, then you can press the SPACE bar to toggle between top and bottom whilst placing the ratsnest line.

PIN_SWAP/GATE_SWAP

When used in conjunction with ISIS, ARES supports pin-swap/gate-swap changes to the connectivity whilst the layout is being routed. This means that you can choose to interchange the wiring to like pins, and/or interchange the use of like elements of multi-element parts. A full discussion of how to prepare your library parts to exploit this feature is given in the ISIS manual, but from the ARES side there are two ways of using this feature:

Manual Pin-Swap/Gate-Swap

To perform a manual pin or gate swap

1. Select the *Ratsnest* icon.
2. Click right on the source pin. For a gate-swap, this can be any member of the gate. The ratsnest lines connected to the pin will highlight. In addition, legal destination pins will also highlight.
3. Hold down the left mouse button and drag the ratsnest lines to the required destination pin.
4. Release the left button. ARES will make the change, updating the ratsnest and force vectors as appropriate. In the case of a gate-swap, ARES will move other ratsnest lines automatically.

It is possible to combine a pin-swap and a gate-swap in one operation - for example, swapping input A gate 1 (pin 1) with input B gate 2 (pin 5) on a 7400 will do just this.

WARNING

Pin-swaps and Gate-swaps constitute changes to the connectivity of your design. ARES uses the pin-swap and gate-swap data specified in the ISIS libraries to decide what is, and is not, a valid swap. If there are errors in this data, then ARES may well suggest illegal swaps. We will not, under any circumstances, be held liable for any costs incurred or losses arising as result of such mishaps, whether the error be in your library parts or ours or in the software itself. We strongly recommend that you check that the swaps you make are really legal, and that your prototype your PCB prior to the manufacture of large quantities.

Automatic Gate-Swap Optimization

In the case of a board with a large number of possible gate-swaps (the SHIFTPCB sample is a spectacular example) in can be very hard to find the best arrangement. For these cases we have provided an automatic gate-swap optimizer.

To perform automatic gate swap optimization

1. Invoke the *Gate-swap Optimizer* command from the *Tools* menu.
2. ARES will make repeated passes trying the current set of possible swaps. The process repeats until no reduction in ratsnest length is achieved.

WARNING

The Gate-Swap Optimizer relies entirely on the gate-swap data specified in the ISIS component libraries to decide what is, and is not, a valid swap. If there are errors in this data, then the swap-optimizer is likely to make erroneous changes to the connectivity of your design. We will not, under any circumstances, be held liable for any costs incurred or losses arising as result of such mishaps, whether the error be in your library parts or ours or in the software itself. We strongly recommend that this command be used only if you are going to prototype your PCB prior to manufacture.

Synchronization with the Schematic

Whether manual or automatic swaps are to be performed, it is a requirement that the changes can be reflected or ‘back-annotated’ into the schematic. For this to occur successfully, the schematic must not have been changed as well.

PROTEUS manages this by using the netlist file as a kind of token. If ARES cannot find an up to date netlist, it will not allow changes, and if ISIS makes changes it deletes the netlist.

When ARES does make changes it writes out a back annotation file (extension ‘BAF’) the next time the PCB is saved. ISIS picks this up the next time it comes to the foreground. If ARES has changes that are not saved, ISIS will not allow changes.

This scheme will prevent the making of simultaneous changes to both schematic and PCB in normal use. It is, of course, possible to circumvent the token mechanism by renaming files, editing and copying back, editing on other machines etc. If you deliberately contrive to modify the schematic and PCB at the same time, then you must live with the consequences! The only cure for such situations is to check carefully that the schematic and PCB are the same and then re-load the netlist into ARES.

Further discussion of pin-swap/gate-swap is appears in the chapter entitled *ISIS and ARES* in the ISIS manual.

ROUTING STRATEGIES

In ARES a strategy defines everything about how the nets assigned to it are to be routed. This information includes:

- The trace and via styles to use.
- The via type to use - normal, buried or blind.
- Assorted controls for the auto-router.
- The design rules for nets routed with this strategy.

The beauty of encapsulating all these properties in a single entity and then assigning nets to it as opposed to assigning the properties separately to each net is that it greatly reduces the amount of information that need be entered on the schematic.

Strategies and the Netlist

ISIS can associate a named property - in this case a strategy name -with a net by placing a net label such as `STRAT=POWER`. This will then be processed by the ISIS netlist compiler and will appear as a net property just after the net name.

For example:

```
VDD , 2 , STRAT=POWER
U1 , 14
U2 , 14
```

If you are using some other schematics package, much depends on its ability to process net properties in some manner similar to the above. If so, then SDFGEN will include them in its output in the same manner as above. If not, then you have three choices:

- Use net names of the form such as `VDD=POWER`. ARES will parse this into a net called VDD assigned to a strategy called POWER.
- Edit the SDF netlist with a text editor and add in the strategy information by hand. By judicious use of the macro facilities of your text editor, this can be a very convenient and flexible approach.
- Ignore the strategy capabilities altogether as far as the schematic side of things is concerned. Given the automatic strategy assignment described in the next section, this may be adequate and is certainly the simplest option.

Special Strategy Names

The most common reason for needing different strategies is to distinguish power connections from signal connections in order that the power routes can be made out of thicker tracking. Also, especially when using the autorouter it is often an idea if memory bus connections are given special treatment as they really need to be routed in an orderly, regular fashion.

In order to help with achieving the above distinctions, the strategy names **POWER**, **BUS** and **SIGNAL** are given special meaning and certain forms of net name will automatically be assigned to these strategies.

- The net names **GND** and **VCC** default to the **POWER** strategy unless they are explicitly assigned to a different one in the netlist.
- Net names of the form D[0] (it is the square brackets that are important) default to the **BUS** strategy. This is now somewhat obsolete (at least as far as ISIS is concerned) since you can place a bus label such as STRAT=BUS directly on a bus segment. Nevertheless, this syntax may be of use with other schematics packages which do not have net properties.
- All other net names default to the **SIGNAL** strategy.

This scheme will enable many boards to be handled without need to explicitly specify strategies on the schematic or in the netlist.

Editing a Strategy

Strategies are created automatically by the *Netlist Loader* as their names are encountered. However, you will normally then need to edit them in order to set the various fields to whatever you require.

Strategies may be edited using the *Set Strategies* command on the *System* menu.

The meaning of the various fields is explained below:

Name

Selects the strategy being edited. Selecting a different strategies saves the changes made to the current one.

Priority

This is used by the auto-router - connections assigned to the highest priority (smallest number) strategy will be routed first. Strategies with the same priority will be routed simultaneously. This will probably result in the router spending much more time 'mapping',

but may lead to higher completion rates than routing them in turn, especially on single sided boards.

Trace Style

The name of the trace style to be used for connections assigned to this strategy.

Via Style

The name of the via style to be used for connections assigned to this strategy.

Power/Bus/Signal Switch

This is used by the auto-router as a guide to the type of connections to be made.

Corner Opt

This button determines whether the auto-router is to optimize corners by cutting them at 45 degrees.

Diagonal

This button determines whether the auto-router is allowed to use diagonal routes.

Pass H / Pass V

Each strategy can involve up to 8 layers on which connections assigned to it will be routed by the auto-router.

The (H) layer is for predominantly horizontal routes and the (V) layer for predominantly vertical routes. Single sided routing can be achieved by specifying the same layer for both H and V routes.

Neck Style

If not blank, this field enables *Auto Track Necking* for the autorouter, and defines the style which it will neck to.

Design Rules

These fields specify the minimum clearances allowable for nets that are associated with this strategy. In the case that tracking on two nets with different strategies is in proximity, the smaller clearances are used .

Hide Ratsnest

Check this box if you wish to hide the ratsnest lines for nets that are associated with this strategy.

BACK ANNOTATION

Back annotation is the process whereby changes to the component annotation on the PCB can be fed back into the schematic. This would normally be done if it was felt that a particular order of annotation on the PCB would aid the production department in making the board. In fact, the most natural order for this purpose is if the placement path for placing the components in successive order is a minimum.

Manual Re-Annotation

It is perfectly possible to re-annotate the board manually by editing each component reference label as required. PROTEUS uses internal unique identifiers to track 'was-is' data so it does not matter what changes you make, in what order, or when they are eventually read back into ISIS.

The only restriction is that it is not advisable to renumber connectors made from *Physical Terminals* in ISIS. This is because ISIS cannot re-annotate them - it would involve it finding not only the terminals but also the references to the part name in the *FIELD blocks. The work around is to manually re-annotate in both ISIS *and* ARES. A warning to this effect is displayed if you attempt such a change.

Automatic Re-Annotation

Should your sole aim be to renumber the components to give a geometrically sensible placement order, then the *Component Re-Annotator* command on the *Tools* menu will do this for you.

The process starts at the top left of the board and proceeds for all components picking the next nearest one not yet processed.

Back-Annotation to ISIS

Both changes to the component numbering and also changes to the connectivity caused by pin-swap/gate-swap operations will be automatically picked up by ISIS. This works whether ARES is running or not.

You have a choice of allowing these updates to occur entirely automatically (in which case, your PCB will be saved when you switch to ISIS) or to do it manually, in which case ISIS will prevent you from editing the schematic until the PCB is saved. This choice appears on the *Set Environment* command on the *System* menu in ISIS.

Further discussion of this mechanism is provided in the ISIS manual in the chapter entitled *ISIS AND ARES*.

REVERSE NETLISTING

ARES can create a netlist from a placed and routed PCB. This has a number of uses:

- As a way to check the wiring where no schematic is available.
- In cases where you wish to re-route a board, without risking changing its connectivity, and do not have the schematic.
- As an aid in reverse engineering designs brought in from Gerbit or elsewhere. (Some day we may allow you to load the netlists back into ISIS!).
- As a debugging aid where there is some doubt about whether the PCB corresponds with the schematic.

To create a netlist from a PCB

1. Load the PCB in the usual way.
2. Invoke the *Save Netlist* command from the *File* menu.
3. Choose a filename for the netlist.
4. Click OK

In cases where the layout file already contains a netlist, the net-names used in this netlist will be used. Otherwise numeric net names will be generated automatically.

SDFGEN - CONVERTING 3RD PARTY NETLISTS

Although we strongly recommend that you use ISIS to prepare your netlists for loading into ARES, we have included a utility which can convert some of the more popular netlist formats to SDF. The SDFGEN program is self-documenting and typing

```
SDFGEN
```

will display basic usage information.

The main problem with taking this route is that not all schematics packages support net properties (which are required to get best effect from the ARES strategy management features) and, probably more seriously, 3rd party device libraries will not use the same naming conventions as ISIS. This means that you will be on your own as far as compiling a device to package translation file is concerned.

AUTO-PLACEMENT

INTRODUCTION

As well as an auto-router, ARES incorporates an algorithm for automatic component placement. As with auto-routing, this may not perform as well as a human operator in all circumstances but on the other hand it can save an enormous amount of time and effort. In addition, the resulting placement can usually be 'hand tweaked' - perhaps more so than can be said of auto-routed tracking. At any rate, we would recommend that most users at least try the auto-placer as a way of quickly achieving a starting point.

The overall system is now sufficiently developed that small-medium complexity boards such as our CPU sample can be converted from schematic to PCB with no human interaction whatsoever!

USING THE AUTO-PLACER

In operation, the placer requires only that the board outline be defined (by placing lines and arcs on the *Edge* layer) and that a netlist be loaded to specify the components to be placed. A more detailed sequence of events is set out below:

To auto-place a PCB layout

1. Create a schematic in ISIS assigning group properties to those components which you wish to be located physically close to each other in the final layout.
2. Invoke the *Netlist to ARES* command on *Tools* menu in ISIS.
3. Draw a board outline of the correct dimensions using the 2D graphics tools on the board *Edge* layer. Ensure that the boundary forms a closed polygon. The best way to do this is to use a *Path* object.
4. Place any components which are immovable on the board. Hand placed components are viewed as fixed, these will be treated as obstacles by the auto-placer.
5. Select *Auto-Placer* command from the *Tools* menu in ARES and adjust the various options as you see fit. See below for more information about this.
6. Select those components from the list which you wish to be placed automatically.
7. Modify the weightings and design rules to suit the board.
8. Click OK.

9. Move and rotate the components as required and start the auto-placer again.
10. Once you have finished with the auto-placer manually place any remaining components and start routing.

THE AUTO-PLACER DIALOGUE

The auto-placer is highly configurable and as a result the dialogue form may at first appear somewhat daunting. However, it will operate quite happily with the default settings for most purposes. The function of the various fields is discussed below.

The Component Selector

The left hand pane show a list of all the components still to be placed, when the dialog form is first displayed the components are listed in alphabetical order with all the components selected for placement.

The function of the *All* and *None* buttons should be fairly obvious; the *Schedule* requires a little more explanation. By the default, the components are presented in alphanumeric order but by pressing the *Schedule* button they are then presented in the order that they will be placed by the auto-placer. In this case, components not-selected for placement always appear at the bottom of the list.

Design Rules

The *Placement Grid* defines the step size used by the auto-placer when trying to find a position for each component. This should always be set to a multiple of the grid size you are going to use for auto-routing. Values other than 100th, 50th or 25th would be highly unusual.

The *Edge Boundary* defines the minimum distance to be allowed between any component and the edge of the board as drawn on the edge layer.

Trial Placement and Cost Weightings

Auto-placement is achieved by trying the next component at various positions on the board in an attempt to minimise a number of key factors. The relative importance of these factors is set by you, the user, by moving the sliders to the right (more important) or left (less important) for each of the seven categories.

Grouping

It is often necessary to force two (or more) components to be placed together (such as decoupling capacitors) and this can be achieved in one of two ways. Firstly you can manually edit a placement to move the components to the desired location, however this rather negates

the use of auto-placement. Secondly you can give the auto-placer a helping hand by telling it which components are to be grouped together.

A particular problem arises with decoupling capacitors since the auto-placer will, other things being equal, tend to place all of them in one place since this will give the most routable solution! To prevent this you can use the **GROUP** property in **ISIS**. If the decoupling capacitor C1 is intended to decouple the chip U1, then you should add the property

```
GROUP=U1
```

to the capacitor. Then, when the placer comes to place C1, it will give high costs to positions which are far from U1 and a placement close to the device will result.

If a number of components are to be grouped together they may all be given a common group name, rather than a component name, i.e. to group U1, U2, U3, C1 & C5 together you could add the property "GROUP=TIMER" to each component.

Hint: To add properties to a large number of components use the Property Assignment Tool in **ISIS**.

Ratsnest Length

Arguably the most important weighting factor in terms of routability, this assigns importance to the minimisation of the length of inter-connectivity of components before routing. Note however that in our experience this should be balanced with some knowledge of the number of ratsnest crossings. A board with short total ratsnest length and a large number of crossings can often be unroutable.

Ratsnest Crossings

This weighting defines the importance of minimising the number of ratsnest lines which cross. As a component is moved around the board the ratsnest length and number of crossing changes, both of these factors will affect the overall routability of the final placement. It is important to get a good balance between this and ratsnest length.

Congestion

Small components such as resistors and capacitors may be encouraged to avoid highly congested areas of the board by increasing the importance of the congestion weighting. Note that when a small component has been associated with another component via the **GROUP** property congestion is ignored for that component.

DIL packages

Components which are allocated DIL packages are treated as a special case. It is seen as desirable to align DIL packages such that they all have the same orientation. To this end a number of controls on the dialogue are tailored to the placement of DIL packages. *Preferred DIL rotation* is by default horizontal but the shape of your board may dictate that vertically placed DIL packages may make better use of the available space. Two further controls in the cost weightings group on the dialogue form define how heavily penalised a component will be if the package is placed with either 90 or 180 degree rotation from the default. If you are not concerned about the orientation of your DIL packages then set these costs to zero.

Alignment

A successful PCB layout is judged not only on routability but also on aesthetics. One key factor in whether a PCB looks right is whether components are aligned. This weighting factor stresses the importance of aligning component edges. Increased alignment can sometimes also aid routing since if like components are aligned then it follows that their pins are also aligned.

Options

Push & Shove

The auto-placer recognises two types of component, those initially placed by the user which are deemed to be fixed and immovable and those placed by the auto-placer which may or may not have been moved by the user. The latter will be pushed and shoved around the board where possible in an attempt to best place the remaining components. This shuffling can become restrictive and counter-productive on boards where the pre-placed components are in the centre of the board. In such cases *Push & Shove* should be disabled.

Swap Pass

Since the components are placed sequentially, it may well be that further improvements can be made to the placement by interchanging like package styles once all the parts are down. This process can be enabled by selecting the *Swap Pass* checkbox.

OCCUPANCY DEFINITIONS

Each component that is placed on the board is treated as an obstacle for the remaining components to be placed. By default the area which is occupied by a component is assumed to be the smallest box which contains all silk screen graphics and component pins. Any two such boxes may not touch or overlap.

It is quite feasible that you will wish to modify the area taken by an object such that, for example, a DIL package has additional space on the sides occupied by the pins, or that a connector has increased occupancy on one side only. To achieve this ARES provides an *Occupancy* layer onto which 2D graphics may be drawn in much the same way as silk screen graphics are added.

Occupancy graphics are not normally seen but may be examined and edited by first decomposing the package, amending or adding graphics as necessary and then invoking the *Make Package* command from the *Tools* menu.

To add an occupancy definition to a package

1. Place a copy of the package on the layout.
2. Tag it and then invoke the *Decompose* command on the *Edit* menu.
3. Select the *Graphics Mode* icon.
4. Select the *Occupancy* layer in the *Layer Selector*.
5. Place either a box or a circle to define the occupancy.
6. Tag all the elements of the package.
7. Use the *Make Package* command to store your handiwork back the library.

The current implementation does *not* support the use of path objects in the context. Multiple boxes and circles are allowed at present, but this may be dropped when we code support for path objects as testing for multiple objects carries a speed penalty.

LIMITATIONS

The auto-placer is another very useful tool in the armoury provided by PROTEUS but there are a number of limitations of which you should be aware:

- It can place components on the top of the board only. Boards requiring double sided placement are beyond its scope at present.
- Whilst the auto-placer can operate on boards on which some components have been pre-placed, our experience is that the more of these there are, the less well it performs. If possible, it is better to let it place everything and then move things about afterwards.

Pre-placing components in the middle of the board will cripple its performance because this interferes with its ability to shuffle the components about as placement proceeds. If you must have pre-placed components in the middle you may find it does a little better if the *Push & Shove* option is disabled but in general it will do badly.

- Boards which are decidedly non-rectangular also interfere with the *Push & Shove* mechanism and again the placer will perform badly in such cases. Boards with small cut-outs etc. should not present a problem, however.

Naturally we hope to code around these limitations in a future version. Our current assessment is that double sided placement is not too difficult an extension; the other issues are more intractable.

AUTO-ROUTING

INTRODUCTION

The ARES auto-router is the one of the most powerful tools in the PROTEUS system and is capable of saving you an enormous amount of time and effort. However, you'll be glad to know that despite its considerable internal sophistication, actually using it is very simple.

Features

- Grid based operation. The ARES auto-router divides the board into a grid of cells of user selectable size and decides where it can and cannot place tracks and vias according to whether the cells are 'free' or occupied by existing pads & tracking.

This approach limits the router to placing tracks at coordinates on the specified grid, but special routines are included to deal with both single off grid pads and also rows of off grid pads. The latter involves generating a 'fan out' from the pads to the nearest grid squares and is especially effective for routing to surface mount chips.

Special code is also included to allow routing at 45 degrees, something which a number of the recent 'shape based' routers are incapable of doing.

- Multi-strategy routing. There are a number of techniques by which a router can search for a viable route for a connection and ARES tries different techniques in turn to find the best connections. In broad terms, we use first a line search algorithm to establish good horizontal & vertical discipline and then use a costed maze search to complete the routing. A special route scheduling technique ensures that all connections are routed at minimum cost to ensure best manufacturability.
- True multi-layer routing. The router can route on up to 8 layers simultaneously enabling it to tackle the most densely packed boards and to make best use of multi-layer routing space when available.
- Rip-up & retry operation. On the harder boards, the router may not manage to route all the connections at the first attempt. In this case, the board is analysed for blockages and having removed them, it then attempts to reroute the remaining connections using the freed space. Clearly this is an iterative process but our code contains special logic to prevent cyclical behaviour. The result is that the router can make headway from as low as 80% initial completion to 100% completion given sufficient run time.

The operation of the auto-router is closely tied up with the netlisting and strategy management features which are covered in the previous chapter.

THE AUTO-ROUTER COMMAND

The auto-router dialogue form has 5 main controls:

- *All Routes/Tagged Routes/Untagged Routes* - This selects which connections will be attempted. *All Routes* would be the normal choice but in conjunction with the tag/untag connection capabilities of ratsnest mode the others to enable you to auto-route selected parts of the layout.
- *Grid* - this sets the grid spacing that the auto-router will lay down tracks on. Smaller grid sizes allow higher density routing provided that the design rules and trace style selected allow this - it is no use selecting a 25 thou grid with 50 thou tracking and a 20 thou trace-trace gap. Note also that memory requirements and routing time increase in inverse proportion to the square of the grid size. Whilst 50 thou routing generally works in seconds or minutes, 10 thou routing needs several megabytes of memory and can take hours. Fortunately, there are very few situations where a grid smaller than 25 thou is needed, especially as SMT components are handled by the fan out mechanism.
- *Edit Strategies* - this button takes you to the *Edit Strategies* dialogue form where you can specify the track widths, via styles and design rules for different groups of nets in the design. See page 78 for more information.
- *Rip-Up & Retry* - if checked, the router will engage its rip-up & retry passes once basic routing has run as far as it can. Otherwise, routing stops once no more routes can be placed.
- *Infinite Retry* - normally, the rip up and retry algorithm will stop if it detects a stalemate condition - that is where the same tracks are repeatedly ripped up and put down. Select infinite retry causes the router to go on forever, with the added twist that the board is tidied when a stalemate is detected. The tidy process often 'frees' the stalemate and a few more tracks may be routed.
- *Tidy Pass* - selecting this option runs the tidy pass, a process which reroutes each track in such a way as to reduce track length and via count whilst also improving the aesthetic quality of the layout. The tidy process can be very long for large boards, as the process runs for all tracks and then repeats until no more tidying is possible. A good one to run overnight!
- *Protect Manual Tracks* - when the rip-up & retry pass engages, it needs to know whether it is allowed to remove existing manual tracking or otherwise. If you have pre-routed existing tracking and do not want it moved, then check this field.

This option also prevents manually laid tracks being tidied.

- *Enable Map Cacheing* - this basically makes the router run faster at the expense of using more memory. If you have 8Mb or less and get out of memory messages or disk 'thrashing' whilst the router is running, try deselecting this option.

Once underway, the auto-router displays a status line indicating how routing is proceeding. As well as the strategy of the connection being processed, the status line also displays:

- What the router is doing - this can be one of MAP, NOVIA, MAZE, NECK, RIPUP, RETRY or TIDY. MAP means that the router is re-generating its copper utilization map, NOVIA and MAZE are routing algorithms.

NECK is a further MAZE routing pass but using the NECK style (if any) for the current strategy. In this mode, the router 'pretends' that it is routing with tracks in the neck style, and then places them with the normal style. The *Auto Track Necking* feature then ensures that the tracks are necked down where they pass between thin gaps.

RIPUP means that the router is analysing for routes blocking the track displayed in red, and RETRY appears when the router is trying to replace tracks it has just ripped up.

- TR - the number of connections To Route.
- RC - the current number of Routes Completed.
- PK - the PeaK number of routes completed. Once rip-up & retry has been engaged, the RC value can actually go down temporarily if several tracks are removed to get one new one in. The PK value shows the best position achieved; this position is saved for later retrieval if the routing process is interrupted.
- RF - the number of Routes Failed by the current pass.
- PC - the Percentage of connections Completed.

The connection currently being considered is drawn in yellow.

The auto-router can be stopped by pressing ESC. Where rip-up and retry mode is in progress, you will get an option to retrieve the 'best position' which corresponds to the state of the board when the Peak Routed value was last achieved.

HINTS AND TIPS ABOUT AUTO-ROUTING

Most auto-routing exercises proceed along the following lines:

- Load the netlist and place some or all of the components.
- Set up routing strategies to define track and via styles for routing.
- Run the auto-router with rip-up & retry disabled.
- Unless complete routing occurs, examine why/where the routing has become congested and move components around accordingly. To remove tracking placed by the router, you can use the *Delete* icon in *Route Placement* mode.
- When you are satisfied with the component placement, enable *Rip-Up & Retry* mode and set the router going again. You should observe a gradual rise in the *Peak Routed* (PK) status box. Progress tends to be exponential - that is, after basic routing, the first 10 further tracks take as long as the next 5. On large, difficult boards overnight runs may prove worthwhile.
- Sometimes, the router may get stuck on the last few tracks, in which case you can break out and try to complete the board manually from the best position found by the router.

Apart from the above, the following sections form answers to commonly asked questions about using the auto-router.

Single Sided Boards

To route single sided boards, you need to edit the routing strategies so that all passes specify the single layer on which you want to route. This will force ARES to route both horizontal and vertical traces on the underside of the board.

The completion rate figure for single sided boards will obviously be much lower than for double sided. However, results of around 80% are still possible with good component placement.

Routes which are not completed will be left displayed as ratsnest lines; some manual route editing will be required to convert them to be sensible wire links.

Avoiding Using Component Pads as Through Holes

For small scale prototype, hobby or educational production it can be useful to produce boards in which the pins of some components (e.g. ICs, electrolytic capacitors) are *not* used as through holes.

The way to achieve this is to redefine the component library parts to have pads on the *Bottom Copper* layer only. The router will then only route to these pads from the underside.

Off Grid Pads

The ARES routing algorithm uses a map or 'grid' of the board which is generated at the chosen resolution. It then uses this to decide where it can and cannot place tracks and vias. However, because the grid squares are at fixed co-ordinates, it does mean that the router can only place tracking 'on the grid' - i.e. at co-ordinates of multiples of the grid setting.

This is not really a problem provided that the component pads are also on the grid - in fact it leads naturally to a tidy result and is much faster than gridless techniques. Where component pads are not on the grid (as may happen if the pin spacing is metric, for example) the router routes to the nearest grid square and then a special routine adds an extra track segment to complete the connection to the pad centre.

What is important, however, is that you try to avoid placing components off grid without good reason. Placing normal imperial spaced components off grid can seriously degrade the router performance. For example, if a part with 0.1" pad spacing is placed at (125,120) and routing is attempted at 50 thou grid, then pins 1 and 2 will totally block the grid square at (150,120) with the result that the router will not be able to route tracks between the pads.

Surface Mount Components

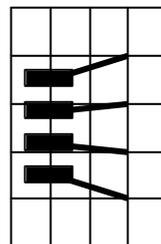
Surface mount components, especially ICs, present something of a problem to a traditional grid based router because the pin spaces are rarely 25 or 50 thou. Thus tracks running on the grid cannot connect directly to the pads because the centres of the grid squares do not coincide with the pad centres.

To overcome this, whilst retaining the advantages of grid based routing (speed and technology based on extensive research) we have implemented special logic to deal with rows of off grid pads. What this logic does is to create an exclusion area around the row of pads, and then to generate within that a *Fanout* from the pad centres to the nearest suitable row of grid cells.

The fanout logic also allows fan-ins so that the area inside the chip can be utilized for routing too.

There are a number of things to bear in mind about fanouts:

- The trace style use for the fanout segments – that is the tracking from the SMT pad to the on grid routing point is created in the FANOUT trace style. The width of this trace



style must be set such that a track can leave the SMT pad at 45 degrees without violating the design rules. Since this width depends on the SMT pad spacing, the design rules and the autorouter grid it is not possible for ARES to set it automatically – instead, you should reduce it from the default value of 8th if you get DRC violations.

- The fanout logic operates outside of the autorouter code, and is therefore unable to ‘see’ objects placed without the fanout regions. Consequently, you must not place objects (e.g. decoupling capacitors) in the fanout areas, as ARES may well just route over them. This will, of course be picked up by a subsequent DRC check, but would result in the need to re-work the board.
- For similar reasons to the above, it is not advisable to pre-route to SMT pads that will be the subject of fanout processing. It is acceptable to pre-route a whole SMT chip, but not to pre-route to only some of its pads. Again, the consequence will likely be that ARES will place fanout segments over the manually laid tracking.

Routing to Internal Power Planes

ARES includes a router pass that deals with the connection of SMT pads to inner power planes. This is a special situation because it requires the placement of routes that goes from pad to via, rather than from pad to pad. Such routes do not correspond to particular ratsnest lines, either.

This routing pass (called PPGVIA) is activated automatically, whenever there are zones which have their *Route to this Zone* checkbox enabled. This is so by default for all zones created by the *Power Plane Generator*. Consequently, the zones should be placed prior to commencing autorouting.

The PPGVIA pass assumes that routing is allowed on the both the power plane layer(s), and on the layer occupied by the SMT pad; there is no need to allow this specifically in the strategy. However, by doing so, you will also allow the router to complete power connections using the normal routing strategies. In particular, the router will then share vias for adjacent SMT pads.

For boards with partial or split power planes, the router will complete the connectivity by conventional tracking, provided that the power strategy allows routing on the top and bottom layers, as discussed above.

TIDY PASS

ARES also incorporates a mode of operation in the auto-router which we call the *Tidy Pass*. This is provided primarily as a means of tidying layouts produced by the auto-router but there is nothing to stop you using it on manually auto-routed boards also.

To automatically tidy a PCB

1. Invoke the *Auto-Router* command from the *Tools* menu.
2. Select the *Tidy Pass* checkbox.
3. If tidying a manually routed board, deselect the *Protect Manual Tracks* checkbox.
4. Click OK.

The tidy pass can be run as part of the main auto-routing process or at a later time. It makes no difference whatsoever.

The tidy process operates by ripping up each track in turn and then re-laying it using costing heuristics that discourage vias but which also allow diagonal movement more cheaply. In addition costs associated with anticipated board usage are removed such that kinks and pointless detours in tracks are also removed. Of course, once a particular track is moved, it may reveal a new and better route for another track that has already been tidied. The tidy process thus iterates until no more tidying can occur. This does mean that the tidy process can become equivalent to routing the board 5 or more times. If it is a large board, this may take hours rather than minutes. In such cases, the best thing to do is leave it till it is convenient to run the process overnight.

POWER PLANES

INTRODUCTION & BACKGROUND

The way in which a PCB design program handles power planes has become much more important in recent years, mainly as a result of the new legislation on electromagnetic compatibility. There are at least three ways in which PCB software can implement power planes:

Grid Based Power Planes

Many mid-to-high end PCB design programs perform some kind of flood fill operation to produce power planes. Essentially, the target area is divided into squares and tests are made to see which squares can be filled with copper, and which are occupied by other objects.

This approach, although relatively straightforward to implement, suffers from a number of problems:

- Because a rectangular grid is used, any diagonal boundaries invariably come out in a staircase pattern, which is non-ideal for RF boards in particular. In addition, the power plane cannot 'squeeze through' narrow or non-orthogonal gaps and so optimum connectivity is not always achieved.
- If any of the pads to be connected to the ground plane are off grid, and if the ground plane is hatched, it is difficult to ensure that they are properly connected.
- In order to maintain the connectivity database, it is usual to represent the ground plane hatching as ordinary tracks. Not only does this use large amounts of memory and slow down the software, but it also makes it difficult to edit or remove the ground plane after it has been generated.

The last two problems could probably be overcome by sufficient coding but the first point is very intractable.

Negative Image Power Planes

Another approach is simply to draw enlarged copies of all the pads and tracks which are not to connect to the ground plane, and say that the result is a negative image of the power plane. Although used in some quite expensive software, this approach is badly flawed:

- The software cannot check if full connectivity is actually achieved - it only needs one track running right across the ground plane area to break it into two unconnected

regions. If you are not allowed tracking on ground plane layers, then this usually forces the use of a multi-layer board.

- Even if connectivity is achieved, it is possible for 'slivers' to exist where the enlarged obstacle images nearly touch - perhaps leaving just 1 thou of copper between them.

Polygonal Gridless Power Planes

Although by far the hardest approach to implement, this scheme has none of the disadvantages of the other two and, so far as we know, no significant ones of its own either.

In essence, the idea builds on the negative image approach:

- The process starts by generating polygonal boundaries surrounding all the pads and tracks within the target area. We call these polygons *Holes*.
- The holes are 'added' together in the sense that where two or more overlap they are replaced by a single hole which encloses them both. This process continues until a much smaller number of many-edged holes remain.
- These holes are subtracted from the original boundary of the target area. This boundary is also a polygon, so you can have any shape of area you like filled with copper.

It is at this stage that the software can find out if the connectivity is complete or otherwise. If a hole cuts right across the boundary, then the boundary is split into two separate regions of copper.

- Where holes are created by pads which are on the same net as the ground plane, the software checks to see if and how a thermal relief can be placed, and amends the connectivity database accordingly.

With this much implemented, there still remain two problems:

- Where two holes nearly touch, 'slivers' of copper can exist which give theoretical but unmanufacturable or unsatisfactory connectivity.
- If the power plane is to be drawn with a Gerber photoplotter or a pen-plotter, only a pen of some minimum thickness can be used to draw it. If such a pen is used to draw the boundary of a polygon, then it will actually end up enlarged by half a pen-width, and this could violate the design rules. In addition it is impossible to draw very pointed corners with a 'fat' pen.

Both these problems can be solved by starting out from the premise that all the boundaries will be drawn with a pen of specified thickness. Then, all the holes are computed as enlarged by half this pen width over their nominal size. Thus the second point (above) is taken care of

at the hole generation stage. But in addition, *no polygon boundary can be less than a pen width in thickness* and so no slivers are created either.

Ground Planes Without a Netlist

One issue that cropped up with annoying regularity in technical support for ARES II was a requirement to produce a ground plane for a PCB which had been produced without a netlist. The problem with this is that of knowing which pads are actually connected to ground. In ARES II - which performed a flood fill from the pads on the specified net to generate a power plane - this problem appears more or less insurmountable.

Note that there is a selector checkbox on the *Edit Pin* dialogue form so that you can select individual pads to have thermal reliefs, or else connect solidly to the power plane.

USING POLYGONAL POWER PLANES

There are two alternative ways of using the polygonal power plane functionality:

- As supported in ARES II, the *Power Plane Generator* command.
- Using the *Zone* icon you can place rectangular or polygonal regions of copper which can form arbitrarily shaped ground planes occupying any chosen region of the board.

In both cases, the zones can be assigned to be hatched or solid, and computed with boundary tracks of specified thickness.

The Power Plane Generator Command

This command provides the simplest way to create a power plane and causes the generation of a single zone object occupying the entire area of a given layer of the board.

To use the Power Plane Generator command:

1. Choose a net for the power plane. If no net is specified, the power plane will connect to any pads which are marked for *Thermal* or *Solid* connection.
2. Choose a layer for the power plane.
3. Choose a boundary style for the power plane.

ARES will then generate the power plane, and update the ratsnest display to show the effect on connectivity.

Zone Placement Mode

The most flexible and powerful way to use the polygonal power plane functionality is to use the zone placement mode:

To manually place a power plane:

1. Select the *Zone* icon.
2. Select the required layer using the *Layer Selector*.
3. Select the boundary trace style for the zone from the *Object Selector*.
4. *Either:*
 - Drag out a rectangular power plane by clicking and dragging from one corner to another with the left mouse button depressed.

Or:

- Mark a polygonal power plane by clicking left at each vertex. In this mode, you can also hold down the CTRL key to place curved sections of boundary.
5. ARES will then pop up the *Edit Zone* dialogue enabling you to choose a net and fill style for the zone.
 6. When you click OK, ARES will generate the power plane and update the ratsnest display to show its effect on connectivity.

Editing a Power Plane

Power planes are held as zone objects which behave similarly to other objects in ARES.

To edit a zone:

1. Select the *Zone* icon.
2. Select the zone's layer using the *Layer Selector*.
3. Click right then left somewhere on the zone *boundary* to tag then edit the zone.

The *Edit Zone* dialogue form contains the following fields:

Net

The net to which the zone connects. Pads on this net will connect to the zone with short track segments in the style selected by the *Relief* field. If no net is selected, then the zone will connect to pads whose *Relief* checkboxes are set.

Layer

The layer on which the zone is placed. This can be any copper layer.

Boundary

The trace style in which the inner and outer boundaries of the zone are drawn. This 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.

If the zone is hatched, the boundary style is also used for the internal hatching.

Relief

The trace style with which thermal relief connections to component pins are made. Connection to vias are by direct contact.

Do not use a relief track style that is larger than the boundary thickness or the reliefs may 'stick out' of the boundary.

Type

This can be set to one of *Solid*, *Outline*, *Hatched* or *Empty*.

Solid and *Hatched* zones should be self-explanatory - the step for the latter being definable by the *Step* field.

Outline zones draw only their outer boundaries. This option is provided mainly so that you can turn off the redrawing of large zones which might otherwise obscure the view of something else you are working on. However, you may find other applications...

Empty zones are intended to be used for creating holes in other zones - see *Overlapping Zones*, below.

Clearance

The distance by which the power plane is separated from other copper objects.

Relieve Pins

If this option is selected, then pins which connect to the zone, and which are not individually marked for *Solid* connection will be connected using thermal reliefs. If the option is unchecked, then pads which connect to the zone, and which are not explicitly marked for *Thermal* connection will be connected by solid copper.

It follows that you can achieve any desired mix of solid and thermal relief connections.

Exclude Tracking

If 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.

Suppress Islands

If checked, the zone regeneration logic will not draw regions of copper that do not connect to any pads, even if they touch the zones boundary box.

Route to this Zone

If checked, the auto-router will invoke its PPGVIA pass to attempt to complete connections on the Zone's net by routing to vias placed within the zone's boundary.

Deleting a Power Plane

This is very simple since the zone is a self contained object 'owning' all the copper regions within its boundary:

To delete a zone:

1. Select the *Zone* icon.
2. Select the zone's layer using the *Layer Selector*.
3. Click right twice *somewhere on the zone boundary* to delete the zone.

Automatic Regeneration of Power Planes

If you place, move or delete tracking or vias onto a board containing one or more power planes, ARES detects whether the power plane needs to be regenerated and then either:

- Proceeds to recompute the internal boundaries of the zone immediately. On a fast PC (e.g. Pentium) and a board of low to medium complexity this real time update mode of working can be quite viable with regen times of the order of 1 or 2 seconds.

The regeneration of complex zones is run in the background such that you can carry on editing; the zone will redraw as soon as the regen process completes. For long and complex regenerations, ARES draws the zone in hatched form whilst its regeneration is pending.

The *Background Regen Threshold* field on the *Set Zones* dialogue form determines the number of holes required in a zone for it to be deemed complex and thus processed using background regens.

or:

- Redraws the zone in hatched form to show that it is invalid. In this case, all such invalid zones can be regenerated by invoking the *Regen* command, key 'R'. This mode saves much frustration if the zone regen times are rather long, as will be the case on slower PCs or with complex boards.

You can toggle auto regeneration on and off using the *Auto Zone Regen* command on the *Tools* menu or via the keyboard shortcut for this command . This is mapped to the key 'Z' by default but can be configured according to your preference. More information on custom keyboard configurations can be found on page 61

Quick Redraw Mode

When a zone is drawn 'properly', all the external and internal polygon boundaries are drawn in tracking of the specified style. However, this can be quite time consuming and so a *Quick Redraw* option is provided in which

- The polygon boundaries are not drawn at all.
- Thermal relief segments are drawn as single pixel lines.
- If the zone is hatched, then the hatching lines are drawn as single pixel lines.

The quick redraw mode, and also the zone auto-regen feature are editable from the *Set Zones* command on the *System* menu. Both settings can be saved in the configuration file by invoking the *Save Preferences* command.

Quick redraw is disabled by default.

Auto-Routing & Power Planes

In general, it is best to place power planes prior to auto-routing. There are two principal reasons:

- The power plane will make connections to any through hole pads within itself, and this will remove the need for the autorouter to do so.
- In the case of SMT components, the automatic via placement pass (PPGVIA) is activated by the presence of the zone(s), and so it is essential that they be placed prior to routing, and that their *Route to this Zone* checkboxes are enabled. For more information see page 94.

In cases where the power plane is on a layer shared with other tracking, the power plane may become split by tracking placed by the router. In this case, some ratsnest lines may be left at

the end of the routing pass, and the best thing to do is to re-run the auto-router in order to complete these by conventional tracking.

REPORT GENERATION

CONNECTIVITY RULE CHECK

When you have completed routing a board it is useful to check whether it really corresponds with the netlist from which it was created. It is important to understand that ARES relies on exact correspondence between track ends and pad centres to establish connectivity, and that tracks which run onto pads, but do not terminate properly at their centres will show both as missing connections (because no connection is detected) and physical design rule errors (because a tracks is in contact with a pad, but not connected to it).

Basic CRC Functions

The CRC check shows:

- Each pair of pins joined by a ratsnest line is listed..
- Any extra (non-specified) connections to each net are highlighted.

If you click on the entries in the listing, ARES will tag the associated tracking in a similar fashion to the *Connectivity Highlight* function.

Advanced CRC Functions

The CRC check also:

- Completely re-initializes the netlist management part of the layout database. Any floating tracking - that is tracking not connected to anything - is detached from netlist management whilst any detached tracking that has become connected to a net is brought under netlist management.
- Traces and vias involved in any extra connections are assigned to the VOID net. Further information on VOID tracking is given in The Netlist Loader & Existing Tracking on page 70.

DESIGN RULE CHECK

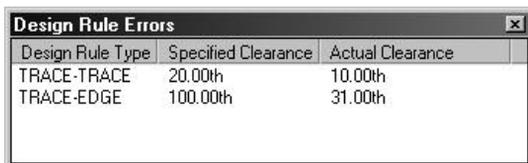
In PCB design, the physical design rules are Pad-Pad, Pad-Track and Track-Track clearance. ARES can check any layout, irrespective of whether it was created from a netlist or not, to see whether it meets a given set of design rules.

To perform a global design rule check:

1. Invoke the *Design Rule Checker* command on the *Tools* menu.
2. Enter the required design rules into the dialogue form and click on OK.

Where errors are found they are marked on screen with a red circle and a white line joining the two objects that are in conflict. These 'error flags' are actually a kind of object and you can 'edit' them to display more information about the DRC violation. The DRC flags can be cleared by invoking the *Redraw* command.

ARES also displays a popup windows listing each violation.



Design Rule Type	Specified Clearance	Actual Clearance
TRACE-TRACE	20.00th	10.00th
TRACE-EDGE	100.00th	31.00th

If you click on an entry, ARES will highlight the associated DRC flag, and if you double click an entry, ARES will both highlight it and zoom to its location on the board.

One thing to beware of is objects which are touching but not (as far as ARES is concerned) connected. This can happen if two large pads overlap or else a track does not terminate at the centre of a pad. Such situations will be marked as DRC violations since ARES sees them as two unconnected objects which are too close to each other. To eliminate such errors, make sure that such objects are connected by tracking running from pad centre to pad centre.

HARD COPY GENERATION

PRINTER OUTPUT

Output through standard Windows device drivers is performed using the *Print* command on the *File* menu whilst the device to print to may be selected using the *Printer Setup* command. Context sensitive help is available on all fields within these dialogue forms and is accessed with a 'point and shoot' interface via the '?' button at the top right of the dialogue box as shown below



Specifying a Printer Device

You can specify the printer device in one of two ways :

- Use the *Printer Setup* command from the *Output* Menu. All printer settings including the printer device specified will be remembered across ARES sessions and the printer device selected will be the default for future ARES sessions.
- Use the *Printer* button from the *Print* command on the *Output* Menu. Under these circumstances no printer settings will be retained. This is useful if you want to print to a different printer for a specific design.

Print Preview

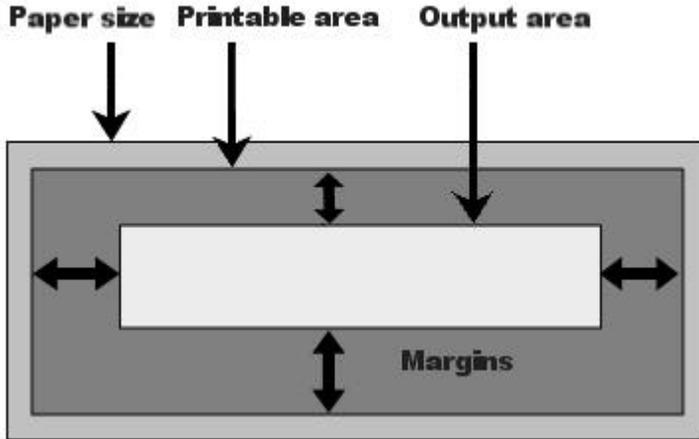
The Print Preview appears on the far right of the *Print* dialogue form and displays the position and size of the schematic relative to the specified paper size. It will 'live update' as you configure the dialogue (scaling, colour, etc..) to your requirements.

You can adjust the position of the printed area by left depressing the mouse on the print preview and dragging to the desired position. Alternatively, you can choose a preconfigured position by right clicking the mouse on the preview pane and selecting the desired position from the context menu.

Configuring Margins

When not printing all sheets you can manually configure the position of the output within the printable area. Right click on the Print Preview and select *Position Output Numerically* and enter the margins that you want between the edge of the printable area and the edge of the output area. This is, in effect, an absolute method of positioning the output area within the

printable area. In general, however, we would anticipate most users to position the output area via the mouse drag method detailed above. The graphical illustration below depicts the use of margins.



If you turn off the option to *Use Printer Margins* on the context menu the margins you specify will be from the edge of the paper itself. In most cases this is not recommended as, depending on your printer driver, you are likely to lose part of your layout if you position it absolutely at the edge of the paper (outside of the printable area).

Printer Information

The *Printer Information* option on the *Output Menu* will dump a log of your printer details and capabilities. This is primarily for support purposes. Whilst our experience of printing support issues is that they are almost invariably the result of a defect in the printer driver we can make a better determination of the problem with the information detailed via this command.

PLOTTER OUTPUT

Windows support for Pen plotters is, unfortunately, very poor. Although drivers are supplied for HPGL and other plotters, the implementation of these drivers is very sketchy. Better drivers may be available for particular plotters, but we have not relied on this in designing the plotter support within ARES for Windows. Instead, we rely on the Windows plotter driver only to draw straight lines and then ARES itself does the rest. To do this, it uses the plotter driver module from the DOS version. This module allows control of the following options which are adjusted using the *Set Plotter* command on the *System* menu.

Plotter Pen Colours

The plotter drivers are able to generate a multi-coloured plot when ALL layers are selected on the *Print* dialogue form.

The *Set Plotter* command on the *System* menu is used to determine the colours for each layer. Where through hole pads are encountered, the pen number is determined by ORing the colours for the outer layers of the pad in the same way as is done for screen colours.

Note that this only applies to an ALL layers plot - single layer plots always use the black pen.

Unfortunately, the Windows plotter drivers do not (as far as we have been able to find out) support the direct selection of plotter pens by number. Therefore it is up to you to find out how your plotter driver maps colours onto pens and to select appropriate colours on the *Set Plotter Pens* dialogue.

Plotter Tips

On the *Set Plotter* command form are fields for *Pen Width* and *Circle Step*.

Pen width

This is a value in thou which should correspond (at least initially) to the width of pen you are using.

Circle step

This value is the size of step (in thou) that ARES will use when plotting the outer radius of a 1 inch diameter pad. A smaller number results in faster but more jagged circular pads whilst a value of 0 will instruct ARES to use the plotters internal circle drawing command. Whilst this is generally fast and smooth, it does mean that the pen is picked up and put down for each pass round the pad.

Getting high quality plots will require experiment with pens, film, ink, and the settings described above. You may also find it necessary to tweak the sizes of some of the pad, via and trace styles - the problem is that there are too many variables outside our control to guarantee that the dimensions of plotted objects will accurately tie up with their nominal size. As a starting point, we recommend a 0.25mm or 0.30mm tungsten carbide tipped pen, professional drafting film (obtainable from a good stationers) and Marsplot 747 ink.

Finally, the *Line Width* command on the *Set Template* command form is ignored when plotting - a single pass of the pen is used for all silk screen graphics for reasons of speed. If you need a different line thickness, use a thicker pen for your silk screen plots!

POSTSCRIPT OUTPUT

Postscript, invented by Adobe Inc, is most commonly used in professional DTP and typesetting. However, it is an extremely flexible language well suited to accurately specifying a PCB artwork. More importantly, the £10K photo-typesetters which are used by professional publishers are capable of producing >1200dpi images on film. Access to these machines is facilitated by the existence of bureaux and our experience is that they charge less than Gerber Photoplotting bureaux for an equivalent plotted area. Try calling a few local printers from the Yellow Pages to track down a local bureau.

Those of you with a Postscript laser printer will be able to proof your boards on it before generating a file for photo-typesetting.

With ARES for Windows, you just select and install the appropriate Postscript printer driver, and use the *Print* command in the normal way. If you use a local phototypesetting bureau, they should be able to supply you with the correct driver for their phototypesetter.

CLIPBOARD AND GRAPHICS FILE GENERATION

As well as printing directly to Windows print devices, ARES for Windows can generate output for use by other graphics applications. You have the choice of generating this output as either a Bitmap or a Windows Metafile, and you can transfer the output to the other applications either through the clipboard, or by saving it to a disk file.

Bitmap Generation

The *Export Bitmap* command on the *Output* menu will create a bitmap of the board and place it either on the clipboard or in a disk file. You have the following additional options:

- Resolution - choose from 100 to 600 DPI. Memory usage increases in proportion to the square of the resolution.
- Number of colours - mono resolution is simple black & white, display resolution uses the same bitmap format as your display adaptor.
- Other options are as for PRINTER OUTPUT on page 109.

Metafile Generation

The Windows Metafile format has the advantage of being truly scalable where a bitmap is not. However, not all Windows applications (e.g. *Paintbrush*) can read a metafile.

The *Export Metafile* command on the *Output* menu will create a metafile of the board and place it either on the clipboard or in a disk file. You have the following additional options:

- Colour - choose mono or colour output.
- Placeable header - if you are generating a disk file, there are two formats - one with a placeable header and one without. Some applications need the header (e.g. Word) and others will not load a metafile that has one!. All you can do is experiment.
- Black background - if checked, this option causes ARES to draw a black rectangle before drawing anything else in the metafile.
- Other options are as for PRINTER OUTPUT on page 109.

DXF File Generation

The DXF format can be used to transfer output to DOS based mechanical CAD applications (it is better to use a clipboard metafile to transfer to Windows based CAD programs). The file is generated by a Labcenter output formatter, rather than by Windows.

Our current experience is of considerable incompatibility and disagreement between applications on what constitutes a valid DXF file. To put this another way, given 6 applications supporting DXF, only about 30% of file exchange pairings seem to work! For Windows work, the Clipboard provides a much more reliable transfer medium.

DXF file generation shares the same output options as PRINTER OUTPUT on page 109.

EPS File Generation

An EPS file is a form of Postscript file that can be embedded in another document. Although popular in the world of DTP, for Windows based DTP work you are much better off transferring graphics using a clipboard metafile.

EPS file generation shares the same output options as PRINTER OUTPUT on page 109.

Overlay Bitmap Generation

The *Export Overlay* command on the *Output* menu will create an overlay diagram and place it as a bitmap either on the clipboard or in a disk file.

The overlay image is created by first rendering the copper layers in a 'tint' and then superposing one or more silk screen layers on top of this. The effect is to show the positions of the components with the tracking faintly visible underneath.

You have the following additional options:

- Resolution - choose from 100 to 600 DPI. Memory usage increases in proportion to the square of the resolution.
- Number of colours - mono resolution is simple black & white, display resolution uses the same bitmap format as your display adaptor.
- Heavy - this checkbox causes the 'tint' for the copper layers to be darker - 75% instead of 50%.
- Other options are as for **PRINTER OUTPUT** on page 109.

CADCAM OUTPUT

THE CADCAM OUTPUT COMMAND

All CADCAM output - Gerber files, NC Drill and Tool Information is generated from a single command. The dialogue form allows you to select:

- A common filestem for all files produced. This defaults to the layout filestem but a neat trick is to change it to something like A:MYBOARD such that all files produced will go directly onto a floppy disk in drive A.
- Whether the files are to contain normal or mirror image co-ordinates.
- Whether the screen X co-ordinate is directed to the CADCAM x co-ordinate (X-Horizontal) or to the CADCAM y co-ordinate (X-Vertical).
- Whether the newer RS274X or the older RS274D Gerber format is used. RS274X has the major advantage that aperture information is embedded within the files and does not have to be manually entered by the photoplotting bureau.
- Which layer is used for mechanical routing data - see page 117 for more information.
- Which layers are output.

Any files generated from a previous run but now corresponding to a disabled layer are deleted - the new tool information table may be wrong for them.

Finally, an 'INF' file is produced which lists all files generated together with the tool settings for the photoplotter and the drilling machine.

GERBER OUTPUT

The Gerber format, named after Gerber Scientific Instruments Inc., is almost universal in the PCB industry as regards specifying PCB artwork although we suspect that it may gradually be superseded by Postscript.

A photoplotter is essentially the same as a pen plotter except that it writes with a light beam on photographic film rather than with an ink pen on paper. The aperture through which the beam shines can be varied, allowing it to expose a pad in a single flash and a track with just one movement of the plotting head.

In order for a photoplot to be produced (usually by a bureau, as photoplotters are not cheap) it is first necessary to set the machine up with a set of apertures corresponding to the various pad shapes and track widths used on the board. Each different aperture is referenced by a so

called D-Code in the Gerber file and a table listing D-Codes against aperture shape and size must thus be compiled. This table is compiled automatically and then written out as part of the tool information file.

ARES is clever enough to tell if two pad styles result in the same aperture, and it will only use one D-code in this situation. Also, styles which are present in the selectors but not used on the board do not waste table space.

Other points to bear in mind:

- An aperture cannot render the image of a pad with a centre hole - the piece of metal for the hole would fall out! Photoplots thus come out with completely solid pads and boards produced from them are very difficult to drill manually. This, among other things is why Postscript is a much better format.
- Rectangular pads that are rotated to non-orthogonal angles have to be rendered by hatching them with a fine line. Once again, Postscript overcomes this problem.
- Polygonal ground planes also have to be hatched (even if you have selected them as solid) and the resulting output files can be massive. However, the boundaries will be drawn exactly as computed - using an aperture corresponding to the boundary thickness for the zone - so there is no risk of design rule violations.
- The origin for the Gerber co-ordinate system is defined by the *Set Output Origin* command on the *Output* menu.

NC DRILL OUTPUT

Many PCB manufacturers now have Numerically Controlled drilling machines which provided with the positions and drill tools to use for each pad, can automatically drill a circuit board. Nearly all these machines use the Excellon format, invented by Excellon Industries.

The format is very simple, consisting of the (x,y) locations of the holes and T-Codes specifying which tools to use. As with Gerber Format, a table must be supplied to the manufacturer listing the hole diameter corresponding with each T-Code. ARES handles this in the same way as it handles each Gerber D-Code - it assigns a new T-Code for each new hole size it encounters, and this information is also put in the tool information file.

ARES uses the same co-ordinate systems for Gerber and NC Drill output - based on the position of the *Output Origin* - so your manufacturer should have no problem in aligning the drilling machine with the artwork.

The drill sizes used are taken from the Drill Hole attributes of the pad styles. Please make sure that the default sizes we have allocated are suitable for your application before having large numbers of boards manufactured.

MECHANICAL ROUTING AND SLOTS

There are two sets of circumstances in which it may be necessary to form make a non-circular hole within a board:

- Some components have solder lugs rather than round pins, and these are best mounted on a pad which has a slotted rather than a drilled hole.

This can be achieved by defining a slotted pad-stack. See page 56 for more information.

- Sometimes there is a need to make a large cut-out in a board, usually to accommodate some aspect of the mechanical design of the product.

This can be achieved by drawing 2D graphics on the appropriate MECH layer - see below.

Unfortunately, there is no widely accepted mechanism within the industry for specifying the location of slots and cut-outs within the CAD/CAM data. Therefore, the best we can do is to output the coordinates of each routing stroke in Gerber format, and then leave it to individual board manufacturers to convert the data for their own particular routing machines.

To do this, we make use of one of the MECH layers within ARES. The *CAD/CAM Output* dialogue form allows you to specify which MECH layer is to be used, and any slotted pads which then generate appropriate routing strokes on that layer. 2D graphics placed on that layer will then also be interpreted as defining routing strokes.

It is important to ensure that your board manufacturer understands that the specified mech layer contains routing co-ordinates in Gerber format. You will need to tell them this explicitly.

We will continue to monitor the industry to see if a more standardized method for specifying mechanical routing operations will emerge.

PICK AND PLACE FILE

ARES incorporates the ability to produce a file for use as a starting point in setting up automatic insertion machines. Essentially this file lists the component layers, positions and rotations in standard quote/comma delimited format.

An example file is shown below:

LABCENTER ELECTRONICS

LABCENTER PROTEUS PICK AND PLACE FILE
=====

Component positions for K:\Prodev\Ares\ppsu.LYT

Fields: Part ID, Value, Package, Layer, Rotation, X, Y

Units: Rotation - degrees, X, Y - thou

Notes: The X, Y value is the centre of package as drawn in ARES.

The origin for these values is the Output Origin.

The values are a guide only and must be checked manually
when
setting up automatic insertion equipment.

```
"U1", "", "DIL08", TOP, 0, 6000, 5000
"Q1", "", "TO220", TOP, 180, 6050, 5375
"D1", "", "DIODE30", TOP, 180, 6050, 5250
"R1", "", "RES40", TOP, 270, 6300, 5050
"R2", "", "RES40", TOP, 270, 6400, 5050
"Q2", "", "TO92", TOP, 90, 5650, 5050
"R3", "", "RES40", TOP, 180, 5800, 4750
"C1", "", "CAP10", TOP, 180, 6200, 4750
"C2", "", "CAP10", TOP, 0, 5650, 5350
```

There are a number of points that must be appreciated about this file:

- The origin for the co-ordinates is the *Output Origin* - this is the same origin as is used for the Gerber and Excellon outputs.
- The (x,y) coordinates are quantified in units of 1 thou and represent the centre of the component's package. This position may or may not correspond to the origin of the component for auto-insertion, but it will be in approximately the correct place. Our understanding is that this is helpful in that it provides a starting point for manual alignment of the placement head.
- The rotations are in anti-clockwise values in degrees relative to the orientation of the package when it was defined. Since there is no standard for default orientations of packages these values may be of limited use unless they can be combined with a translation table that is specific to ARES packages including ones you have defined yourself. This is a matter between yourself and whoever is providing the auto-insertion facility.

As far as we are concerned, this is an evolving and experimental area of the software; a number of users have requested an output file of this nature but there is as yet no industry standard for us to base it on. Any feedback on this issue will be greatly appreciated.

GERBER VIEWING

Since photoplotting charges can be quite substantial, it is useful to be able to view Gerber files to ensure that all is well prior to sending them to the bureau.

To facilitate this, ARES provides a *Gerber View* command which will load and display selected files produced by the *CADCAM* command.

To view Gerber output files

1. Invoke the *Gerber View* command from the *Output* menu. If you have modified but not saved the current design you will be prompted to do so, as the current design data is lost when the Gerber files are read.
2. Choose an 'INF' file from the file selector. The *Gerber View* can only read Gerber files produced by ARES for which an INF file exists.
3. ARES will then parse the file and display a dialogue form for selecting which layers to view. By default, all available layers are loaded.

Although it is possible to perform editing operations on the Gerber View data, we do not recommend or support this mode of operation, aside for panelization which is discussed in the next section.

Do not confuse this feature with GERBIT (the Gerber Import Tool), which is available as an optional extra. See GERBER IMPORT TOOL on page 127 for more information on GERBIT.

PANELIZATION

Panelization refers to the processes of combining several board artworks onto one panel in order to reduce manufacturing costs. It is common practice to panelize both multiple copies of the same board and/or several different boards.

ARES provides facilities to achieve this through the Gerber Viewer.

To create a panelized artwork

1. Use the *CADCAM Output* command to produce Gerber and Excellon files for each board to be included on the panel.
2. Invoke the *Gerber View* command and choose the INF file of the first board to go on the panel.
3. Accept the default settings for the layers (these should correspond with the layers you chose to generate at step 1) and check the *Panelization Mode* checkbox.

4. Click OK. ARES will import the CAD/CAM data for the board and display it as a tagged set of objects.
5. Use the *Set Work Area* command on the *System* menu to specify the dimensions for the panel.
6. Use the *Block Move* command to re-locate the board image to where you want it on the panel.
7. If the panel is to contain multiple images of a single board only, you can now use the *Block Copy* command to lay them out.
8. If the panel is to contain images of additional boards, you can import them into the panel by returning to step 2 above.
9. When the panel is complete, use the *CAD/CAM Output* command to generate final artwork files for the panel. It is these files that you need to send to your board manufacturer.

There are a number of points to note about the panelization process:

- You can add additional text and graphics to the panel as required to designate board IDs and other manufacturing information.
- When importing multiple board images, ARES will assign new pad and track style codes (D-Codes) for each image. However, when the CAD/CAM files are re-generated, ARES will re-combine styles that are actually the same, so that the total usage of aperture and tool codes will not be excessive.
- The resulting set of files may be quite large - especially if the board contains ground planes or non-orthogonal components. However, that is an inevitable consequence of producing a file that contains multiple board images. Fortunately, the files are ASCII, and will compress quite well with WINZIP or similar, should you need to transmit them by email.
- The process is not suitable for panelizing boards containing buried or blind vias, because the layer ranges of the drill holes are not preserved. If you need to do this, we recommended you make use of a 3rd party Gerber viewer/editor package.

DXF IMPORT TOOL

INTRODUCTION

The DXFCVT converter is fully integrated into ARES and is driven through the *Import DXF* tool on the *File* menu. It facilitates the import of mechanical data in DXF format, and we envisage it being used in situations where a board needs to be fitted into an existing mechanical design. It could also be used to import other graphics such as company logos.

The conversion process is as follows:

- Generate the DXF file from your mechanical CAD application. This should, if possible, be a 2D drawing. ARES cannot handle 3D data and the converter thus ignores all z co-ordinates in the DXF file so 'flattening' 3D drawings to two dimensions.
- Determine the mechanical layers in the DXF file that are to be combined and placed on particular ARES layers. The DXF Import dialogue form allows you to specify one or more layer assignments with each assignment specifying which of the mechanical layers in the DXF file are to be combined and placed on the specified ARES layer.

Note that it is particularly important that the correct units are used for DXF import. As per the original DXF specification all co-ordinates are interpreted as being in inches. If after importing your file you find that the graphics appear wildly out of scale then you probably have a DXF file whose units are in millimetres and you will need to specify a scaling factor of 0.03937. This causes the importer to divide all co-ordinates by 25.4 (1" = 25.4mm).

You should appreciate that Autocad (the application around which the DXF format was based) supports much more powerful mechanical drafting than that offered by ARES. Thus some entities in a DXF file are ignored by DXFCVT, and others are approximated. However, for the purpose of displaying simple cabinet internals and the like inside ARES, DXFCVT is quite adequate to the task.

- ☞ See the *Limitations* section for specific information on the limitations of the DXFCVT converter.

SETTING UP

Generating The DXF File

The DXF file to be imported should be generated by the mechanical CAD application you use. Consult the documentation or on-line help associated with the CAD application to determine how best to do this.

As far our import converter is concerned, the only requirements are that:

- The file exported by the application be in plain ASCII and *not* in the alternative binary (DXB) format.
- Co-ordinates in the DXF file are in floating-point inches with the value 1.0000 being assumed as one inch - this is the norm for DXF files.

Layer Assignments

Having generated the DXF file you must now decide on what *layer assignments* you require. Each layer assignment tells the converter which layers in the DXF file are to be combined and placed on a single ARES layer. The controlling dialogue form is laid out in a fashion that should make this process fairly transparent and Context Sensitive Help is provided for all fields on the form should you encounter any problems.

Note that, for a given conversion, an ARES layer may only be assigned once. An attempt to assign a layer twice will result in an error.

PERFORMING A CONVERSION

Basic Conversions

To perform a DXF file conversion you need to load the converter with the name of the DXF file to be converted. This process is implicit with ARES V in that immediately on selecting the *Import DXF* tool you will be asked to open the appropriate DXF file. A successful loading of the file will result in the appearance of the *DXF Import* dialogue form. All available options on the dialogue form are covered via Context Sensitive Help on the appropriate field.

Conversion Errors

With regard to DXF file parsing errors, the converter has been written to be as fault-tolerant as is consistent with reading a DXF file. Parsing errors are only generated when:

1. The DXF file doesn't contain an ENTITIES section. A DXF file without entities is deemed 'empty', which makes conversion impossible.
2. The DXF file contains a section (e.g. HEADER, ENTITIES) or a primitive (e.g. LINE, CIRCLE, etc.) that doesn't contain a single record. All sections should at least contain an end-of-section record and all primitives should contain at least one record. This implies the DXF file is corrupted.
3. The end of a BLOCK entity in a non-BLOCKS section or the end of the BLOCKS section before the end of a BLOCK entity. This implies the DXF file is corrupted.
4. The failure to extract (parse) data of a type consistent with the group record code. For example a record with a group code of 10 implies an integer value and a record with a group code of 41 implies a floating point value. An error is generated if (regardless of formatting) a value cannot be parsed.

Whenever an error occurs the converter reports the error in the form:

```
ERROR (000413): Entity Group Expected - EOF found?
```

and the conversion is aborted.

For file parsing errors, the number of the last line read from the DXF file is shown in brackets after the ERROR keyword (note that the converter reads whole groups - two lines - from the DXF file at a time, and so any error may be on the line indicated or the preceding line). For non-parsing errors, the line number is displayed as dashes.

Conversion Warnings

Apart from errors (described above) all other reports of unusual or possibly erroneous conversion behaviour are treated as warnings. Warnings are not fatal (they do not stop the operation and completion of the converter) but are useful if the resulting region file is not what was expected. In order to be complete, the converter issues warnings whenever it encounters a situation that may or may not lead to unusual region files and this includes any record it reads and discards as being either of unknown purpose or unconvertible. In order not to slow the conversion down (with screens of scrolling text), warnings are only displayed if requested (through the *Generate Warnings* checkbox on the dialogue form).

Warnings are displayed in a similar format to errors (described above) except that the DXF file line number line and message text is preceded by the word WARNING.

LIMITATIONS

The principle limitations of the DXFCVT converter are summarised as follows:

- DXF files must be in plain ASCII. The converter does not support the DXB binary file format.
- Three-dimensional DXF drawings are flattened to two-dimensions ('plan' view) by the converter as ARES only supports two dimensions.
- The only entities (alone or as part of a BLOCK entity) supported are the BLOCK, LINE, POLYLINE, CIRCLE, ARC, TEXT, SOLID, TRACE, and INSERT entities. All other entities are ignored and do not affect file conversion. The SOLID and TRACE entities are converted as polygons. In particular, the DIMENSION entity is *not* supported.
- Only those attributes of an entity that can be honoured by ARES are converted. Thus text attributes such as font face, weight (boldness) and italic slope angle are discarded and do not affect the file conversion.

Other minor limitations include:

- Line types (dots, dashes, thicknesses, etc.) are lost. All lines are single-pixel.
- Colours are lost - all converted entities are displayed in a colour appropriate to the ARES layer they are on.
- Text styles (from the DXF TABLES section) are ignored. The converter relies (for those attributes it converts, such as rotation, mirroring, etc.) on these attributes being specified as part of the TEXT entity - this is the norm.
- The conversion of DXF TEXT entities drawn in a proportional text style will convert to a fixed-pitch font in ARES where the width of the characters may result in the overall length of the string in ARES being different to the overall length of the string in the DXF drawing. This may then lead to alignment problems unless the user has made judicious use of the DXF text alignment flags (e.g. text that is specified as fitting between two points).

As the ARES text primitive is limited to fixed-pitch characters which only allows for the specification of a character height and character width for the text string, the only way around this problem would be to adjust the ARES character width such that the number of characters in the string multiplied by the adjusted character width is the same as the overall length of the string in the DXF file. Unfortunately, except for certain justifications, the DXF file does not contain the overall string length and so this isn't possible.

The actual behaviour of the converter is to calculate the character height as being equal to the height of the DXF TEXT entity and the character width as being equal to the TEXT entity height scaled by the TEXT entity's x-scaling factor (the latter defaults to 1.0).

- The following text attributes and properties are **not** converted:
 - ? Vertical fonts (i.e. where characters are stacked on top of each other rather than alongside each other).
 - ? Obliquing angle (for italics).
 - ? Styles - all text is in ARES's fixed-pitch vector font.
 - ? Control characters. These occur in the DXF file as a caret followed by an ASCII character - all such characters in the string are preserved.

The following text attributes and properties **are** converted:

- ? Rotation.
- ? X and Y mirroring.
- ? Justification, except attribute '5' ("Text is fit [sic] between two points (width varies)") as it is ambiguous and hasn't been tested.
- Only the following entities are converted as noted.:
 - ? LINE and 3DLINE entities.
 - ? CIRCLE entities.
 - ? ARC entities (by default, by linearisation to multiple chords).
 - ? SOLID, TRACE and 3DFACE entities are converted to unfilled three/four sided polygons. The 'hidden' attributes of 3DFACEs is not honoured.
 - ? TEXT entities is converted as described above.
 - ? BLOCK entities.
 - ? INSERT entities are converted but any associated ATTRIB entities are discarded - this leads to text declared in the INSERT via a ATTDEF entity being lost.
 - ? POLYLINE and associated VERTEX entities. All polylines are assumed to be continuous (i.e. **not** a mesh) and all VERTEX entities are assumed to be single point (i.e. not Bezier or spline).

In addition to the above some other omissions are:

- ? DIMENSION entities - these are usually also exported as LINE, SOLID and TEXT entities as well, so there should be no loss of visible information.
- ? ATTRIB and ATTDEF entities, as outlined above.
- ? SHAPE entities - these will (if a problem) be added in a future release.
- All DXF file co-ordinates are assumed to be mathematical, in inches, and with an implied origin at (0,0). It is the user's responsibility to specify alternate scaling parameters (via the appropriate options on the dialogue form) to comply with this assumption.

Positive angles are assumed to be measured in degrees counter-clockwise from an implied zero axis corresponding to the positive x-axis. The necessary DXF file header variables are checked and a warning issued if this is not the case.

- Layer names are honoured throughout the conversion, with the layer name "0" (the single character zero) being taken to have the layer name of its parent. For example, for a BLOCK object with entities on layers 0, BLUE and GREEN and an INSERT of the BLOCK on layer RED, the BLOCK entities on layer 0 are deemed to be on layer RED.

How the converter determines which entities with a BLOCK are to be output is determined by which layers are enabled and the settings chosen under the *Symbols* group on the dialogue form.

GERBER IMPORT TOOL

INTRODUCTION

GERBIT is available as an optional extra - it is not part of the standard Proteus package. It facilitates the conversion of PCBs designed with other CAD programs to ARES format. The conversion process involves the following steps:

- Generating the required Gerber files with the source CAD system. At the same time, you need to generate an aperture list so that you know which 'D-Codes' specify which pad and track types.
- Creating new pad and trace styles in ARES to correspond with those used in the source layout(s).
- Processing the sets of Gerber files with GERBIT to create a single ARES region file for each layout.
- Where several boards are to be converted, it is often worthwhile generating a dummy board in the source CAD system containing all the library parts used in the real ones. This can then be converted and the library parts captured from it into the ARES libraries.
- Reading each region file into ARES, and reconstituting its components by over-placing library packages over their pads. It is for this reason that the previous step can often be useful.

It is vital to appreciate that the Gerber format specifies the board at the level of individual pads and tracks, and that all information pertaining to which pads belong to which components is lost. The extent to which this matters depends on how much editing you want to do on the imported boards, but ARES provides specific features & tools to help with the reconstitution process. Nevertheless, reconstituting a board requires a considerable degree of manual effort.

REQUIREMENTS FOR CONVERTIBILITY

The scheme we have implemented for Gerber import makes certain demands on and assumptions about the source CAD system. These are:

- The ability to generate standard Gerber photoplotting files. GERBIT can accept any M.N format (e.g. 2.3 or 2.4) in imperial or metric units. GERBIT assumes 2.3 imperial by default, so it is simplest if you can set the source system to generate this. The end of block character must be '*' and the character set ASCII.

- The ability to create a human readable aperture list corresponding to a batch of Gerber files.
- At minimum, the system must be able to generate separate Gerber files for each of the copper layers. Additionally, it will prove of great assistance if it can generate files for the component and solder side silk screen layers. If it can generate a file with either pads and vias only, or better still, component pads only, this will further facilitate the conversion and capture of library parts.
- The system must be set so as *not* to create pads out of multiple flashes or, worse still, draw strokes. The conversion process assumes that a flash corresponds to a single pad, and a draw stroke corresponds to a track segment.
- If the system can produce an Excellon format NC drill file, this can be used to assist with the combination of vertically aligned pads on each layer into single through hole pads in ARES. If no drill file is available, GERBIT will perform this task by matching coordinates, but this may lead to occasional problems if there are pads which are coincident, but not through hole (e.g. SMT pads).

WARNING:

Note that the Gerber Import Tool will only work with files in the RS247D Gerber Format. Files in the new RS274X format cannot be imported using this utility.

TUTORIAL

Before we discuss some of the more advanced issues involved in Gerber import, we will first take you through an example conversion session.

This section presents a walk through using the ARES door bell sample. Note that this is rather contrived since we have generated the GERBER files from ARES in the first place.

Preparing for Conversion.

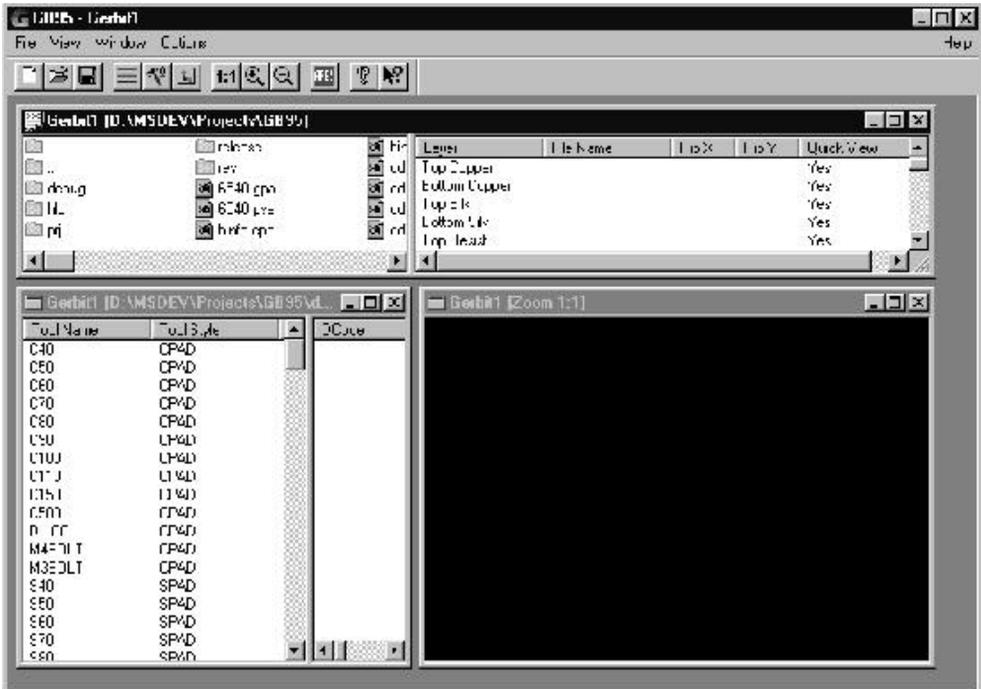
The exercise starts with the GERBER files having already been generated from the CAD system, these are

DBELL.TOP	Component side copper
DBELL.BOT	Solder side copper
DBELL.TS	Component side silk screen
DBELL.DRL	NC drill file
DBELL.INF	Aperture information file

Start Gerbit now and select *New* from the *File Menu* (File | New). Since this is the first time that you have used Gerbit you will be asked to locate your DEFAULT.STY file, this is in the

same directory as your ARES program and contains definitions for all the pad and track styles that ARES knows about.

You are now presented with three views, the uppermost view is the file/layer view, the lower left is the style/tool view and the lower right is the preview window.



In the *File View* window double click on the file entitled DBELL.INF. This will launch a text viewer which allows examination of any ASCII file.

The contents of the information file is shown below:

```
LABCENTER ARES III TOOL INFORMATION FILE
=====
Tool set up for C:\PROTEUS\SAMPLES\DBELL.LYT
```

File List

```
Top Copper      c:\Ares\dbell.TOP
Bottom Copper   c:\Ares\dbell.BOT
Top Silk        c:\Ares\dbell.TS
Drill           c:\Ares\dbell.DRL
```

Photoplotter Setup

Format: ASCII, 2.4, imperial, absolute, eob=*, LZO

Notes: D=Diam, S=Side, W=Width, H=Height

```
D10  T25
D11  T15
D12  T10
D13  C50
D14  C80
D15  S50
D16  C60
D17  S60
D18  C100
D19  CIRCLE D=8th          DRAW
D70  CIRCLE D=50th        DRAW
```

NC Drill Setup

Format: ASCII, 2.4, imperial, absolute, eob=<CR><LF>,
no zero suppression.

Notes: Tool sizes are diameters.

```
T01  20th
T02  30th
```

It is important to study this file before converting as it provides us with key information regarding the conversion process. Firstly we can note that there are four GERBER files to convert and secondly that they are in 2.4 imperial format.

As is happens imperial 2.4 is the default so we do not need to change the resolution for this project. We now have to place the GERBER files in the Layer view and this is achieved using drag and drop. Left click on the file called DBELL.TOP and keep the left mouse button pressed drag the file to the right until the top copper layer in the layer view window is highlighted in blue. Now release the mouse button.

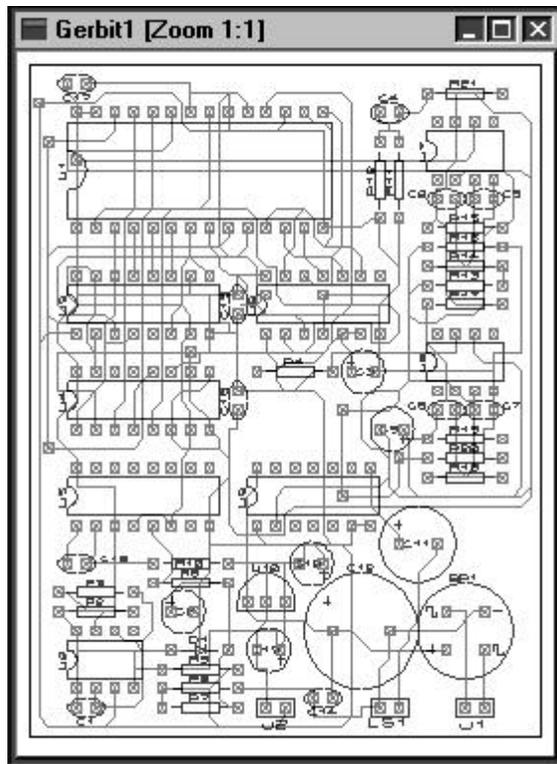
A number of things will happen at this point.

- The name of the file will appear in the layer view.
- A list of D-Codes used in the GERBER file will appear in the Tool View
- A preview of the board graphic will appear in the preview window.

Now drag DBELL.BOT onto the bottom copper layer and DBELL.TS onto the top silk layer.

DO NOT drag the drill file across yet (we will demonstrate its use later).

The preview window as shown below does not accurately represent the board since the tracks are all the same thickness and the pads are all assumed to be a checked box.



To complete the process we must use the style and tool views to allocate the correct apertures as defined in the DBELL.INF file. This is once more a drag and drop process. Select the tool view window and drag the apertures from default to their appropriate tool codes such that you have:

D10 T25
D11 T15
D12 T10
D13 C50
D14 C80
D15 S50
D16 C60
D17 S60
D18 C100
D70 T50

you will notice that DCODE 19 is defined as

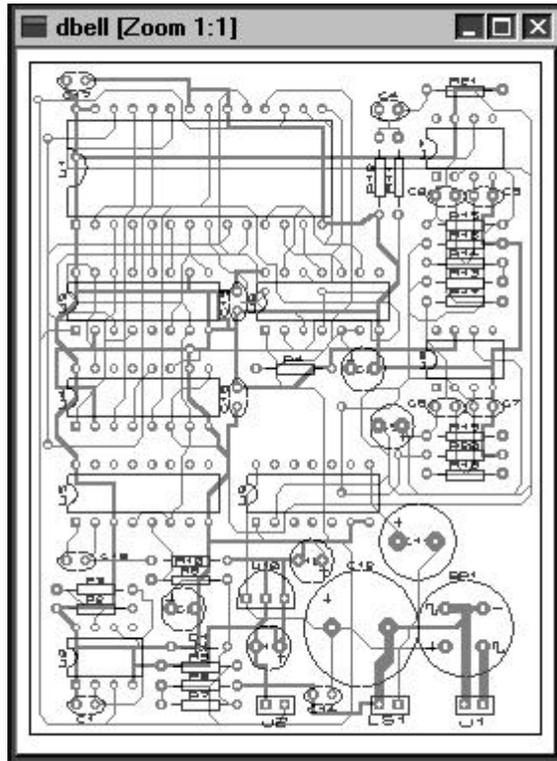
D19 CIRCLE D=8th DRAW

this means a trace (DRAW) of thickness 8th and that no such style exists in ARES. This will be a common problem and you will need to be aware of the range of pads and trace styles which are used in your GERBER files. You can then define the required styles in ARES.

Alternatively you can create a pad or trace style specifically for this board. Simply select Create New Trace Style (or Pad Style) from the options menu. You are then asked to name the style and supply the required dimensions. When defining trace styles you only need to define the track thickness. When defining pads the dialog box changes to reflect the dimensions required for the pad style selected.

Once the new trace style has been defined it appears in the tool view along with tools defined in the DEFAULT.STY file.

As you drag each of these styles onto the tool view the preview will update to reflect the changes. With all the tools in place you should see the following.



All that now remains is to generate a region file to import into ARES and this is achieved by selecting *Options* and then the *Make Region File* command .

THE FILE/LAYER VIEW

The File/Layer View is split into two sections with a file navigator window on the left and a PCB layer selector on the right. Files are dragged from the file navigation window onto the layer view to define the structure of the board. As files are dragged across the layer to which the file is to be applied is highlighted.

Right clicking in the file view window displays a pop up menu with three options, display disk drives, Change directory and view file. By default the file browser window does not display the disk drives available on your machine (or network), selecting this option displays the disk drives and allows you to navigate to them by double clicking the desired disk drive icon. In a similar manner you may change directory by double clicking the directory folder icons or you may alternatively select change directory from the pop up menu. You are then asked to enter a directory name. The third and final menu option is view file and is used to allow you to

examine text files (usually information files) in a read only text window. Simply right click on the file you wish to view and select view file, alternatively double click on the file to view.

The layer view also has a right hand mouse button menu, which has the following options.

- Flip X flip the currently highlighted layers left to right.
- Flip Y flip the currently highlighted layers top to bottom.
- View Include/Exclude highlighted layers from graphic preview.
- Clear Reset highlighted layer information to default settings and remove file name.
- Colour Define the graphic display colour of the currently selected layer.

THE TOOL VIEW

The tool view is again split into two windows with a list of all pre-defined tools in the left hand window and a list of the tools needed by the Gerber files in the right hand window. As files are dragged onto the layer view the right hand pane is updated to show the tool codes used by the Gerber file which has been added to the project. There is no way for the Gerbit program to know what pad and trace styles these codes represent, but it will indicate the tool type as either a Flash (Pad) or Draw (Trace). You must then drag the tool types from the left hand pane into the tool views right hand pane with reference being made to the Gerber information document.

New pad and trace styles may be added to the list of defined types using *Create New Pad Style* or *Create New Trace Style* . Once defined these styles may be used as any other predefined style. Note however that they are only available to the current design. If a style is to be repeatedly used add the style to the list of Ares defined pads and traces.

THE QUICK VIEW WINDOW

The quick view window is used to provide a graphic representation of the Gerber board at its current state. Before tools are defined the board is displayed with thin lines representing all tracks and a hollow box representing all pads. As the tools are added to the design the preview is updated to reflect the changes made. You may zoom in or out of the drawing using the three zoom options under the view menu. Normal provides and approximate one to one mapping of the PCB if the board appears far too small or far too large at one to one check the resolution settings for the Gerbit project. If the tracks appear as a tangled mess check that the positioning is set to either relative or absolute as required.

When zooming out a maximum zoom ratio of 1:10 is applied. When zooming in the maximum board size is 32 in or a maximum ratio of 10:1 whichever is smaller. Hence a board nominally

2inx2in may be zoomed to a ration of 10:1 whilst a board 4inx4in may only be zoomed to 7:1. A board which is nominally larger than 32 in will not be previewed.

The primary usage of the preview window is to ensure correct orientation of the layers since some cad systems flip tracking on the bottom copper. If you wish to see a more accurate view of the board load the region file generated by Gerbit into Ares.

ADVANCED CONSIDERATIONS

The following topics may be relevant to you once you have mastered the basics of Gerber import, or if you encounter difficulties with converting particular types of board.

Memory Usage

In general, ARES will need considerably more memory to handle a board during Gerber import, than it would if the same board were created in the usual way. This is because all the silk screen graphics are held in a much lower level way than they would be if they part of library packages.

Another thing to bear in mind is that captured library parts will often be considerably larger than they would be if they created from 2D graphics. This is particularly true for silk screen outlines involving curves, as these will be rendered as lots of short straight lines in Gerber format. Some CAD systems use an awful lot of lines too.

Finally, the *Convert Vias* command should be used on small numbers of vias at a time since when a via is converted to a lone pad, the lone pad form uses about three times as much memory.

Pin Numbering

ARES uses the pin numbers assigned to library parts when it is loading a netlist in order to tie up package pins with device pins in ISIS. Now it is perhaps unlikely that this will matter much for imported boards, since you are unlikely to have an ISIS schematic. However, if you intend using Gerber Import to capture library parts for future use with netlists then pin numbers do matter.

If you simply convert a group of vias to pads, tweak the silk screen and invoke the *Make Package* command, the pins of the package will be numbered in the order that they appeared in the Gerber file. Unfortunately, this is in no way guaranteed to be the correct even for simple packages like DIL outlines. The only way to find out is to use *Make Package* and then check the assigned pin numbers by zooming in – the pin numbers will be drawn on the pads once there is sufficient space for them to be legible.

On the (more than likely) assumption that the pin numbering is wrong, you will need to adopt a slightly more involved approach to capturing packages. Having converted a set of vias to pads, the additional steps required are as follows:

4. Select the *Instant Edit* icon.
5. Invoke the *Auto Name Generator*, set the prefix to the empty string and the count to '1'.
6. Click on the pads in the order you want them numbered.

ARES will indicate the number it has assigned to each pin as you mark it.

Converting a Batch of Layouts

If you have several boards to convert, rather than just one, it may be worth capturing all the library parts first. This is because it can be rather messy trying to capture them in situ from existing boards.

The trick is to create a 'dummy' board in the source CAD system with all the library parts you wish to capture nicely spaced out on it. You can then convert this board and capture the parts off it without having to mess about deleting stuff off the silk screen and so forth.

Also, don't forget the *Write Style Set* and *Read Style Set* features can save you having to re-enter the aperture list on each run, provided that it is actually the same for each set of files.

Alignment

Due to the number of stages through which design passes during Gerber import, the alignment of its pads onto the absolute grid sometimes goes astray. For example, the relative spacing of the pads might be 50 thou, but they might all be at multiples of 2 and 52 thou.

Although *Real Time Snap* will facilitate editing such a board to some extent, it will be better if you re-align it. This can be done by tagging the entire board, and then invoking the Align command on the Edit menu. This offsets all the tagged objects such that the majority of the pads lie on the current snap grid.

Systems Which Output a Single Pad File

We have come across one CAD system that outputs pads in a single 'pad master' file, and then outputs separate tracks only files for the top and bottom of the board. From an ARES viewpoint, the entities from the pad file need assigning to all layers, whilst the track files need assigning to the top and bottom copper layers, respectively. Unfortunately, GERBIT will only let you assign one Gerber file to a layer per session.

If you are faced with this, or a similar situation, you need run two sessions using the following strategy:

- On the first session, assign the pad file to *both* the top and bottom layers and the silk screen file to the *Top Silk* layer. Also, on the *Aperture Selection Screen*, use the *Write Style Set* command to preserve the style set for the second run.
- On the second session, assign the top and bottom tracking files to the *Top Copper* and *Bottom Copper* respectively

With ARES import each region file in turn, making a note of the co-ordinate readout the first time and then duplicating it the second time. Provided that the source design has a border on all the layers, this will guarantee registration between the two imports.

An alternative approach would be to assign the tracking files to a pair of inner layers in the first session. You can then either settle for this format for your imports, or if you are confident at using a text editor, you could edit the region file and change the layer assignments within it. Track blocks are designated with the line:

```
*LAYER <name>
```

where <name> is the short layer name.

2			
2D Graphics Styles		53	
A			
Align command			
in Gerber import		127	
Anchor point		17, 59	
Annotation		10	
Arcs		34	
ATTRIB		57	
Auto Trace Selection		39	
Auto Track Necking		13, 40	
in autorouter		72	
Auto Via Placement		10, 22, 39	
Automatic Insertion Machines		107	
Auto-Router command		82	
Auto-Routing		22, 81	
single sided boards		84	
using pads as through holes		84	
with off grid pads		85	
Auto-Save		48	
B			
Back-annotation		73	
Backup files		48	
Bitmap		102	
Blind vias		13	
Block editing		12, 44	
BMP file		102, 103	
Boxes		34	
Buried Vias		13	
C			
CADCAM output		105	
Circle step		101	
Circles		34	
Clipboard		102	
Component Re-Annotator		73	
Components			
affect of loading new netlist		63	
and back annotation		73	
distinction from packages		31	
editing		20, 37	
highlight by name		38	
not in netlist		31, 33	
on underside of board		32	
placing		19, 31	
unplaced		19	
Connections			
specifying manually		67	
Connectivity highlight		15, 66	
Connectivity rule check		24, 97	
CONNECTORS.LIB		57	
Coordinate Display		28	
Coordinates			
measuring		29	
snapping		30	
Copy icon		44	
Copying			
area		44	
route		14, 43	
CRC report		97	
Creating library parts		59	
Cursor types		8, 28	
Curved tracks		10, 40	
Cut-outs		107	
D			
D-Codes		105	
Decoupling capacitors		77	
DEFAULT.STY file		52	
Delete icon		45	
Deleting			
area		12, 45	
library parts from selectors		58	
nets		15, 66	
Power Planes		94	
route		14	
single object		37	
vias		12	
Design rules			
checking		24, 98	
violation message		40	
DEVICE.XLT file		65	

LABCENTER ELECTRONICS

Dimension fields	29
Dirty tracks	38, 39
Dragging	
objects	10
pads	20
part IDs	11, 20
single object	37
DRC function	98
Drivers	5
DXF	
export	103
import	111
DXFCVT	111

E

Edge layer	11
Editing library parts	61
Editing objects	36
Editing Window	26
Encapsulated Postscript	103
EPS file	103
Excellon format	106
Export	
Bitmap	102
DXF	103
EPS file	103
Overlay	103
Windows Metafile	102
Export Region command	47
Extra connections in CRC report	97

F

False origin	30
Features	1
File types	46
Flashing tracks	38
Force vectors	21, 67
Foward annotation	63

G

Gate-swap	67
Gate-Swap	68
Gerber	
file viewing	109
import	117
output	105

GERBIT	117
converting batches of files	126
memory usage	125
pin numbering	126
requirements for file conversion	117
tutorial	118
Graphics	
editing text	37
line width	101
placing	31, 34
types available	34
Graphics driver	5
Graphics files	102
Grid dots	30
Guard Gap	51

H

Hard copy	16
-----------	----

I

Imperial mode	28
Import Region command	47
Importing	
DXF	111
Gerber data	117
ISIS	63
ISIS symbols	62

K

Keyboard	54
Keyboard Configuration	54

L

Labels	
default position	59
dragging	20
size of	61
Last Loaded files	48
Layer Selector	8, 28
Layers	
selecting	8, 10, 28
toggling display of	8
Layout files	46
Library files	46, 57
Lines	34

Load Layout command	47	PACKAGE.LIB	57
M		Packages	
Make Package command	59	annotating	10
Make Symbol command	62	creating	17, 59
Manual routing	38	distinction from components	31
Map Cacheing	83	editing	37, 61
Markers	34, 59	global replacement of	58
MECH layers	107	placing	9, 31, 33
Mechanical data import	111	replacing	20
Mechanical Routing	107	specifying in netlist	65
Menu bar	25	Pad Master files in Gerber Import	127
Metafile	102	Pad stacks	50
Metric mode	28	Pad styles	34, 49
Missing connections in CRC report	97	Pads	
Move icon	45	off grid	85
N		placing	17, 31, 33
NC Drill output	106	placing on a given grid	30
Netlist Management	21	shapes available	33, 49
and block copy	44	smoothness when plotting	101
Netlists		style names	34
and existing tracking	64	Panelization	109
and strategies	70	Panning	8, 26
format of	63	Paste Mask	52
generating	74	Paths	34
loading	63	Pen colours	101
loading into existing layout	63	Pen width	101
loading into new layout	63	Pick & Place file	107
Nets		Pick command	58
deleting	15, 66	Pin number not found	64
highlighting	15, 66	Pin-swap	67
New Layout command	46	Pin-Swap	68
Nodes		Place Preview	27
redundant secondary	44	Placeable header	103
O		Placing	
Origin for library package	59	board outline	11, 21
Output		components	19, 31
origin	29	curved tracks	10, 40
Overlay	103	graphics	34
Overview Window	8, 27	packages	9, 33
P		pads	17, 33
Package library	58	power planes	92
		traces	39
		tracks	21
		vias	12, 39
		zones	35
		Plotter output	100
		Plotters	

LABCENTER ELECTRONICS

calibration	16	SMTCHIP.LIB	57
pen colours	101	SMTDISC.LIB	57
pen width	101	Snapping	30
Postscript	102, 103	Solder Paste Mask	52
Power Planes	89	Solder Resist	51
deleting	94	Statistics	6
editing	92	Strategies	23, 70
with no netlist	91	and router layers	72
Printer & Plotter drivers	5	editing	71
Printing	99	special names of	71
R		Styles	49
Ratnest		managing	52
manual entry	67	Symbol library	62
Ratsnest	66	Symbols	17
compiling	20	creating	62
editing	67	from ISIS	62
Read only libraries	57	placing	34
Real Time Snap	30	SYSTEM.LIB	57
Reference point	17	T	
Reflecting		Tag Filter	37
area	45	Tagging	
Region files	46, 47	group of objects	36
Reverse Engineering	74	objects	10
Rotate-while-drag	20, 32	route	13
Rotating		route	41
area	45	single object	36
Rotation icon	45	Technical support	6
Route editing	38	Thermal Reliefs	
Route Editing	12	setting default type for zone	93
Routing (mechanical)	107	Tidy command	44, 58
RS274X	105	Tidy Pass	82
S		Trace angle fixup	41
Save As command	47	Trace angle lock	41
Save Layout command	47	Trace styles	52
SDF file	63	Traces	
SDFGEN.EXE utility	74	copying	14, 43
Set Plotter command	101	deleting	14
Shift-Pan	8, 26	editing	13
Shift-Zoom	27	highlighting	15
Shortcut	54	placing with netlist loaded	21, 39
Single sided boards	71	placing without netlist loaded	10, 39
setting strategies for	84	re-routing	14, 43
Slots	50, 107	rubber banding	45
SMTBGA.LIB	57	tidying	44
		Tutorial	

basic techniques	9
block editing	12
introduction	7
making new packages	17
making new symbols	17
overview	7
printing	16
report generation	24
route editing	12
using a netlist	18

U

Untagging	36
USERPKG.LIB	57
USERSYM.LIB	57
Using the manual	3

V

Version information	6
Via styles	52
Vias	
deleting	12
placing	10, 12, 22, 39

W

WMF file	102
----------	-----

Z

Zones	35
deleting	94
editing	92
placing	31
Zoom	27