AutoTRAX™

Schematic Capture, Circuit Simulation and Layout

User Guide

www.kov.com

The philosophy behind Kovac Software's vision is the desire to provide a suite of software tools to enable today's engineers to develop world-class electronic solutions to today's problems; and have fun doing it!

- Ilija Kovacevic

Architect and developer of AutoTRAX

Notice

This program is a propriety product of Kovac Software and is protected by copyright laws and international treaty. Use of this software is subject to acceptance of the Kovac Software End User License agreement included in this software package.

Information in this manual is subject to change without notice and does not represent a commitment on the part of Kovac Software. The software described in this document is furnished under a license agreement, which states the terms for use of this product. The software may be used or copied only in accordance with the terms of that agreement. No part of this publication may be reproduced, transmitted, stored in a retrieval system or translated into another language without the written permission of Kovac Software. This manual utilizes fictitious names for the purpose of demonstration, references to actual persons, companies or organizations is strictly coincidental.

Trademarks and Copyrights

Patent Information

This software contains designs that are patent pending.

Kovac Software, Fair Oak, Skeath Lane, Sandon Bank, Stafford ST18 9TD, England.

www.kov.com

Document Number and Revision

Document Number 100-0001 Revision A

Effective Date: Wednesday, March 20, 2002

CONTENTS

Preface

Congratulations on choosing AutoTRAX from Kovac Software. We are confident that it will deliver many years of increased productivity, superior designs and above all, enjoyment.

Documentation Conventions

When an AutoTRAX manual refers to a toolbar button, you will see an image appearing in after the command.

This manual uses the convention **Menu/Item** to indicate a menu item. For example File/New means to chose the **New** command from the **File** menu.

Microsoft Windows 2000 Professional was used for illustrations in the manual.

Introduction

Welcome and congratulations on choosing AutoTRAX, a leading Electronic Design Automation (EDA) tool designed to help you produce world-class electronic designs that you will be proud of and also enjoy designing! We are confident that it will deliver many years of increased productivity, superior designs and above all, enjoyment.

Great emphasis was place on ease of use and your productivity during the design of AutoTRAX. This ensures that AutoTRAX utilizes the full power of your computer and puts it to work for you, not you working for the computer. You want to easily and quickly express your designs and test their working. You do not want to fight with the computer.

AutoTRAX is a Microsoft Windows based program suite designed to enable both the professional electronic engineer and the hobbyist electronic engineer to rapidly develop:

- Schematic designs based on both parts provided with the software and custom parts created by the user.
- **Built libraries of designs and parts.**
- Cooperate with others using the built-in **SQL distributed database** algorithms supporting both Internet and extranet collaboration.
- Simulate the electrical behavior of the schematic using the inbuilt **SPICE** simulation engine.
- Produce parts lists-Bill of Materials **(BOM)**
- Generate optimized and accurate PCBs using automatic layout and routing.
- View the PCB in 3D and export to 3rd party 3D CAD programs.
- Export the design to other programs including AutoCAD and PCB layout programs.
- During the design of AutoTRAX, the following prime targets was uppermost in the design objectives:
- **Must Have!** Above all, AutoTRAX is a must have product. The benefits and productivity improvements soon out-weight the purchase cost.
- **Minimize Cost Of Ownership.** Many programs carry with them a considerable cost of ownership. This costs includes maintenance and in particular training. AutoTRAX has been designed to minimize this cost and training costs by its ease of use.
- **Ease of Use**. A much overused phrase but one nevertheless that is very important and has a profound impact on the cost of the design and the users satisfaction.
- **Consistent User Interface.** Current Electronic Design Automation (EDA) programs often present a confusing set of different interfaces to the user. It appears that they are a collection of programs from different companies, purchased by the supplier, and packaged as a program 'suite'. This often is the case and the programs have been purchased from smaller companies that are often failing, are tight for money or are successful and the original team wishes to sell and move on… The design of AutoTRAX has been deliberately targeted to deliver a consistent interface with no gimmicks.
- **Highly Interactive.** Here again we have another overworked phrase but one that describes an area which has a major impact on productivity.
- **Graphics Quality**. Here, it is felt that current EDA developers have severely let down the user. If an engineer has spent many hours on a design it is only right that the final printed schematics reflect their professionalism and pride in their work. The grubby

blueprint and wiry line pen plots of the last century just will not do… A great deal of work was taken to during the design of AutoTRAX to ensure that printed output would be of the best and ultimate quality. In particular, the features of the new breed of large format photographic quality printers such as those from Hewlett-Packard are used to the best of the ability.

- **Reliability.** What it is the point of using a program that is unreliable? Sometimes users will tolerate an unreliable program (CorelDraw 4?) due to their unique capabilities but not for long. Users want a quite life and so do support personal. However, no program is 100% bug free. AutoTRAX's design has concentrated on reliability and reliability physics with the aim of maximizing the Mean Time Between Failure (MTBF) and the cost of maintenance.
- **FUN to Use!** Last, but certainly not least, AutoTRAX is be fun to use. Why should work be boring?

The Schematic Document

The main AutoTRAX document type is the Schematic, which consists of one or more named sheets. These sheets are presented to the user as a vertical pile of sheets with the selected sheet on top and all the others are hidden beneath the top sheet. A group of page tabs at the bottom of the main AutoTRAX window show the names of the pages.

Each page can be selected by clicking on its tab. This will bring it to the top of the pile of sheets.

Dragging a sheet tab to another location will change the order of the sheets.

Individual sheets can be deleted.

New sheets can be added.

The Schematic Sheet

The schematic sheet represents a single sheet or page of the design. Onto these sheets are placed the parts, electrical connections and non-electrical graphical elements that make up the circuit schematic or circuit diagram.

Each sheet can be assigned a standard sheet size such as A0 or they can be given a custom sheet size.

A Sheet consists of:

- An optional title block
- An optional reference border.
- A drawing area with a configurable grid.

The sheet background color can be changed.

The Title Block

Each sheet can contain an optional title block. This contains the following:

- The sheet title.
- A description of the sheet.
- The designer’s name.
- The design checkers name.

- The design approver's name.
- The date.
- The design revision number.
- The document number.

The Grid Reference Border

Each sheet can contain an optional grid reference border. This border consists of grid reference bars on the top, bottom, left and right edges of the sheet. These can be configured to be:

- Alphanumeric or numeric.
- Ascending or descending.
- The number of divisions.

The Grid

An optional grid can be placed on the sheet and is drawn underneath parts, the title block and the reference border. The grid can be one of three types:

- Graph paper. This has the appearance of graph paper.
- Grid. This is a rectangular grid of horizontal and vertical lines.
- Dots. Here the grid is drawn as a rectangular array of small points.

Viewports

The user views a sheet via a viewport. A viewport is a window onto the sheet. Each sheet can have one, two or four viewports.

The viewport displays a view of the sheet that can be panned or zoomed in and out to view the required details. There is effectively no limit on the precision of the pan or level of zoom. This contrasts sharply with other EDA software. AutoTRAX's data is represented using 8 byte double precision floating-point numbers. The IEEE (Institute of Electrical and Electronics Engineers) format has a range from 2.225e-308 to 1.798e+308; more than our universe can use!

Rulers

Vertical and horizontal rulers can be attached to the top and bottom of a sheet viewport.

As the cursor moves, lines are drawn on the rulers to indicate the mouse cursor position, object borders etc.

The Status Bar

A common status bar at the base of the application displays:

- The mouse cursor position.
- Object specific size and/or location
- Command prompts.

Printing

Each sheet can be printed to any Windows printer that supports raster graphics. Great care has been taken in the design of AutoTRAX to give the user the ultimate professional quality printing. The aim has been to produce artwork that you will truly proud of. Gone are the days of the 'blueprint' and the poor quality line plots produced by pen plotters.

Plots can be same size as the logical sheet size or the logical sheet size can be scaled up or down to fit the printers paper size.

AutoTRAX contains internal algorithms designed to produce the ultimate print quality, for instance in fountain filling.

Printer Setup

The printer to use can be selected using the 'File/Print Setup' menu command.

Print Preview

Prior to printing, it is possible to preview a print using the 'File/Print preview' menu command.

Email Documents

AutoTRAX has built in electronic email support via the 'File/Send' menu command. This allows the user to easily sends documents to more or more co-workers.

Editing Previous Work

AutoTRAX remembers up to the last 15 schematic documents that the user worked on. These are displayed in the File menu to allow rapid access to previous work.

In addition AutoTRAX can optionally automatically reload the last schematic or PCB document you worked on at startup. This eliminates the tedious repetitive chore of having to manually reload work every time the user starts work.

AutoTRAX warns the user if they try to exit without saving any edited work.

Multi-monitor Support

Not all application work well with multiple monitors; for instance they can get confused when the their main window is moved off the main monitor with popup dialogs appearing on the wrong monitor. AutoTRAX has been designed to work on system with multiple monitors. Windows 98 and Windows 2000 support multiple monitors. Multiple monitors allow you to organize their screen space more efficiently and to easily interact with more than one program at a time.

Undo/Redo

AutoTRAX has an effectively unlimited design history memory. All operations can be undo and redone. The limit is only controlled by physical memory but in general provides well over 1000 undo/redoes. Each sheet has its own separate undo/redo list.

As a design progresses, the design moves from one state to another with the operations being the link between the states. AutoTRAX names and remembers each of these states. The states are named after the operation that preceded it. So in addition to single undo/redo actions that move between adjacent states, AutoTRAX can display a list of named states so the user can rapidly move from one state to another.

The Recovery File

Before all operations, the current state of the design is saved on the undo stack. In addition, before any significant operation that modifies the design database, the current design is saved as a recovery file on the hard drive. When AutoTRAX terminates normally, the recovery file is deleted. However, if the operation fails and AutoTRAX terminates due to a serious error then the recovery file is not deleted. The presence of the recovery file will be automatically detected when AutoTRAX is restarted and the user will be prompted to load the recovery file. This will restore the design to the point just before the serious error occurred. The user will not have lost any work!

If the operating system fails due to say a power outage, AutoTRAX will detect the recovery file on restart allowing the user to recover their work. In this case at most only the last operation is lost.

One of the main advantages of the recovery file is the improved effective reliability of using AutoTRAX and the effect that recovery has after a program or system failure. Imagine, if you will, the user having worked on a design for 2 hours without saving the design and then the program crashes or the power goes off. The user will be none too pleased! Now imagine their delight when then start up AutoTRAX again to find that their work is still there. NOTE: Unfortunately a hard drive crash will not save enable recovery of the users work if a RAID system was not used.

By chance during the creation of this paragraph (using Word 2000 on Windows 2000 Professional) the computer decided to reboot. All the work was lost so it had to be retyped….

Object Selection

Objects can be selected individually by clicking the mouse over the object.

One or more objects can be selected by holding down the mouse button and dragging the mouse to reveal a selection rectangle (drag selection). All objects inside the selection rectangle will be selected and the display will dynamically change to show the selected items highlighted.

Selection can be inverted by menu command or by holding down the shift key during drag selection.

Grouping objects

One or more objects can be arranged into a group. Once formed, the collection of object can be manipulated as a single object (a group object).

Groups can contain groups. There is no limit to the level of nesting of grouped objects.

However, even though a selection of objects are contained in a group, it is possible to select and manipulate individual objects by holding down the **Alt** key during selection.

Mirroring Objects

One or more objects can be mirrored about their common central vertical or horizontal axis.

Rotating Objects

One or more objects can be rotated about their common center point by ±90° or 180°.

Delete/Cut/Copy

All objects or collections of objects can be:

- Deleted
- Cut (copied to the clipboard and then deleted)
- Copied to the clipboard

In addition to the objects being copied to the clipboard, if it is a single text object, then the text is copied to the clipboard. A bitmap representing the current viewport is also copied to the clipboard.

Paste

The following data can be pasted onto a design sheet from the clipboard:

- Objects cut or copied from ***any*** AutoTRAX sheet.
- Bitmaps
- Windows Metafiles.
- Text.

Alignment aids

The following aids are available.

Snap to Grid

Objects can be snapped to a regular rectangle grid. This is configurable.

Snap to Objects

Objects can snap to key positions on other objects, such as the endpoints of lines, and the corners of rectangles.

Guides

You can add point, horizontal and vertical guides to help you precisely align objects.

Ortho

Line can be restricted to horizontal, vertical or diagonal directions.

Object Alignment

If two or more objects are selected, they can be aligned together in several ways. Alignment is based on the position of one of the selected objects. This object is called the dominant object and is user specified.

Dominant Object

Alignment is based on the position of one of the selected objects. This object is called the dominant object and is user specified.

Align at Top

Objects are aligned so that the top of each object is level with the top of the dominant object.

Align at Bottom

Objects are aligned so that the bottom of each object is level with the bottom of the dominant object.

Align at Left

Objects are aligned so that the left edge of each object is level with the left edge of the dominant object.

Align at Right

Objects are aligned so that the right edge of each object is level with the right edge of the dominant object.

Align Horizontal Middle

Objects are aligned so that the center of each object is horizontally aligned with the center of the dominant object.

Align Vertical Middle

Objects are aligned so that the center of each object is vertically aligned with the center of the dominant object.

Object Vertical Order

Objects are drawn in a set order. The order is normally the order of creation. However, sometimes it is desirable for the order to change to force say one of the objects to appear above another.

To Front

This brings the selected objects to the front. They will be drawn last and cover objects before them.

To Back

This places the selected objects at the start of drawing. They will be drawn first and may be covered by objects drawn after them.

Forward One Level

This will move the selected objects forward one level. They will be drawn after the object that previously preceded them.

Backwards One Level

This will move the selected objects backward one level. They will be drawn before the object that previously preceded them and may be drawn behind them.

Distributing Objects

Object can be distributed evenly spaced either horizontally or vertically or both.

The Viewport

AutoTRAX displays a view of the current sheet. This view is can be split into 2 or 4 independent views.

Panning

You can pan around a sheet in real-time.

Zooming

You can dynamically zoom the view in and out in real-time.

Persistence

AutoTRAX automatically saves the position, size and visibility of the application and toolbars when the program exits.

Custom Toolbars

AutoTRAX comes with several predefined toolbars. However, the user can create their own toolbars and modify existing toolbars. These are saved on a per user basis.

Custom Menus

AutoTRAX comes with several predefined menus. However, the user can create their own menus and modify existing menus. These are saved on a per user basis.

Object Properties

All objects in a sheet have a user definable text name. This can be changed using the object properties dialog.

Each object has its own set of unique properties. Common properties include:

- Line properties such as color, width and corner/end styles.
- Fill properties such as color and style.
- The Layer on which the object resides.
- Object name.

In addition, objects can have properties unique to their type. For example, lines have a start and end point.

The properties can be inspected and/or changed using the dockable objects property dialog box.

If the user does not wish to have the object properties dialog permanently docked to either the left or right side of the application, these can have it either floating or non visible.

If it is floating the user can optionally have the object properties dialog box disappear during object movement and manipulation. This leads the user see the ‘wood for the trees’.

If it is invisible, then the user can double click on the object to reveal the properties dialog box.

Status Bar

At the base of the application is an optional status bar. This displays cursor position and/or selected object dimensions.

Design Library

You can save your designs to a hierarchical design library.

Component Library

AutoTRAX comes with a component library containing 1000s of parts.

Splitting the View

The main viewport can be split up into 2 or 4 viewports with each viewports having its own independent view of the sheet and its own optional rulers.

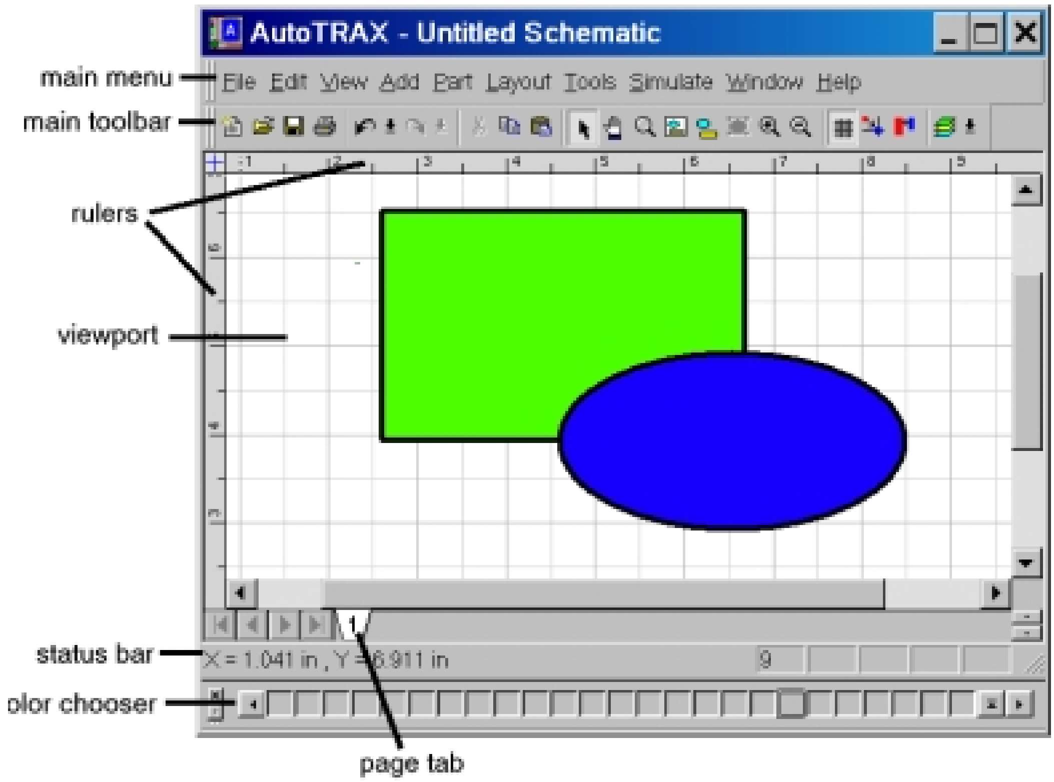
Each viewport can have its own viewport layout.

Full Screen

In order to provide a greater portion of the screen to be used for viewing, a full screen mode is available. The user can toggle between the normal view and a full screen mode that has reduced toolbars and no menu.

The User Interface

The main AutoTRAX application is shown below.



It consists of:

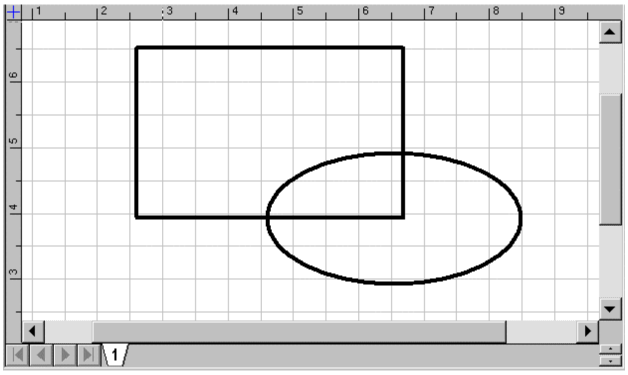
- The Main menu.
- The Viewport.
- The Rulers.
- The Status Bar.
- The Color Chooser
- The main Toolbar and other toolbars.
- The Part Library toolbar (Not shown above)
- The Design Library toolbar (Not shown above)
- The Object Properties Toolbar (Not shown above)
- The Help System. (Not shown above)

The Viewport

The Viewport displays the current view of the sheet. Normally it only contains one view, but you can have up to four views; See Splitting the viewport.

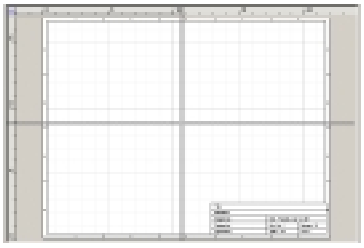
A typical viewport is shown below. It can have:

- Horizontal and vertical scrollbars. These will only be displayed if only part of the sheet is visible.
- Rulers.



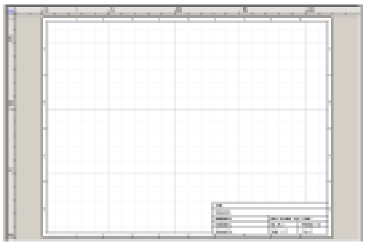
Splitting the viewport

You can split a viewport into 2 or 4 separate viewports. To create multiple viewports select View/Ortho/Split. A pair of large dotted lines will appear as shown below.

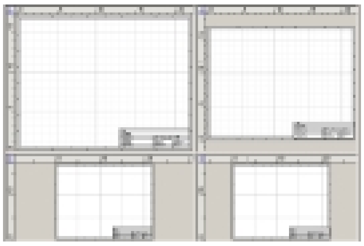


To split the viewport into 2 or 4 separate viewports:

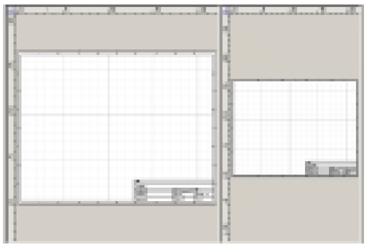
1. Select **<View/Split>** from the main menu.
2. Move the mouse to the center of the split viewports.

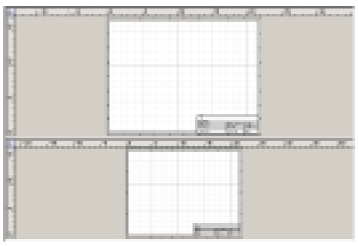


1 viewport



4 viewports





2 Viewports

To resize the viewports, position the mouse over a viewport edge and hold down the left mouse button and drag the mouse.

To **remove one or more viewports**, drag the viewport edge so that the ones you wish to remove are no longer visible.

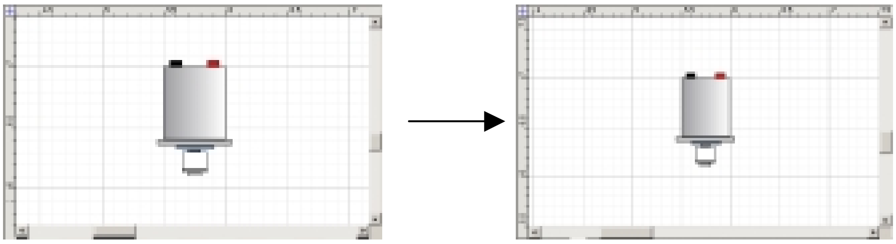
To revert to just a single viewport, Select **<View/Split>** again from the main menu.

Zooming in and out

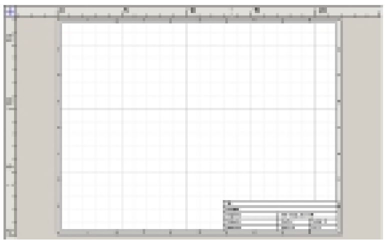
There are several ways you can zoom (magnify) the view of the current sheet.

- You can zoom in on a window.
- Zoom in by selecting **<View/Zoom/In>** from the main menu or pressing the **PgUp key.**

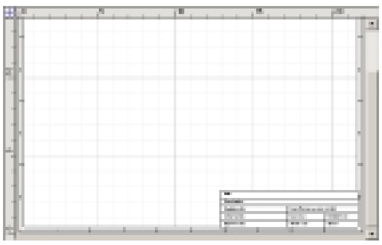
- Zoom out by selecting **<View/Zoom/Out>** from the main menu or pressing the **PgDown key.**



- **View the entire sheet** by selecting **<View/Zoom/Full Page>** from the main menu or pressing the **Home key.**

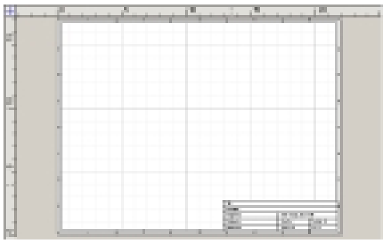


- Adjust the view to maximize the **page width** by selecting **<View/Zoom/Page Width>** from the main menu**.**

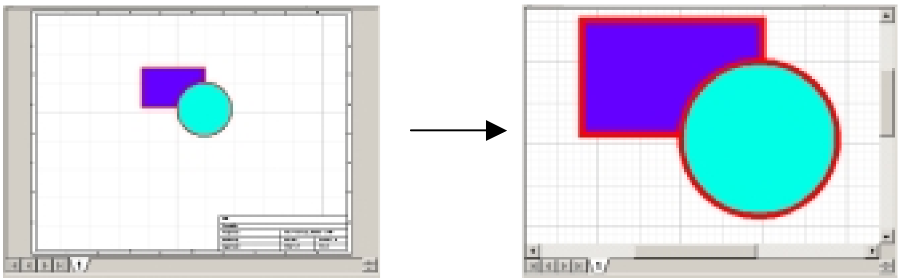


- Adjust the view to maximize the **page height** by selecting **<View/Zoom/Page**

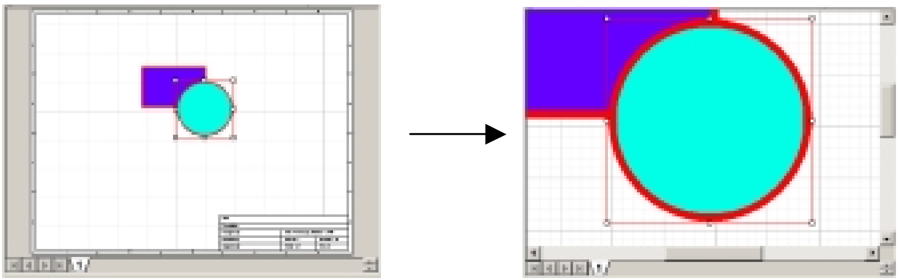
Height> from the main menu**.**



- View all the objects by selecting **<View/Zoom/All>** from the main menu.
-



- View all the selected objects by selecting **<View/Zoom/Selected>** from the main menu.



Zoom on a window

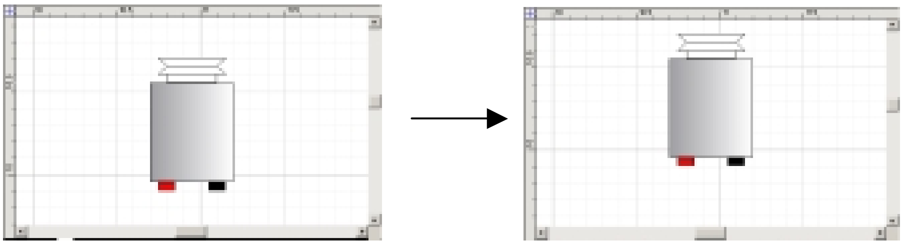
To zoom on a window:

1. Select **<View/Zoom/Widow>** from the main menu.
2. Hold down the left mouse button over one corner of the area to zoom in to.
3. Drag the mouse to the other corner and release the left mouse button.

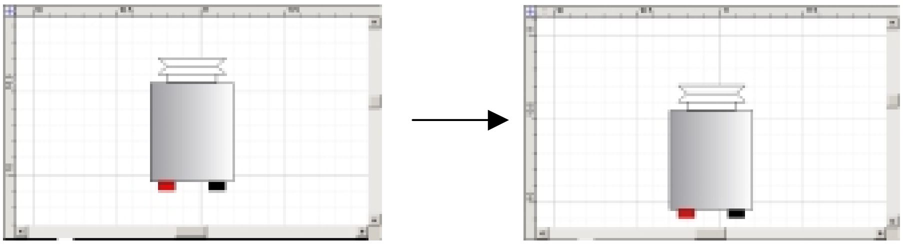
Panning

You can pan (change the view) by either:

- Holding down the middle mouse button (if you have one!) and **dragging the mouse**. The view will change in real-time to display the new view or
- Click the **Pan button** ( ) and press down the left mouse button and drag the mouse.
- Select **<View/Pan/Up>** from the main menu to pan the view upwards or



- Select **<View/Pan/Down>** from the main menu to pan the view downwards or



- Select **<View/Pan/Left>** from the main menu to pan the view left or

- Select **<View/Pan/Right>** from the main menu to pan the view right or

- Press the **left, right, up or down arrow keys** to pan left/right/up or down.

The Rulers

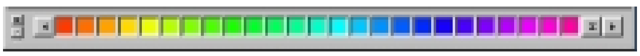
You can show rulers around the viewport to help to align, size and position object.
You can also set the ruler origins.

The Status Bar

The Status Bar is on the bottom on the application and contains information that continually changes to assist you in what you are doing.
You can **Hide/Show** the status bar by selecting **<View/Status Bar>** from the main menu.

The Color Chooser

The color chooser is shown below.

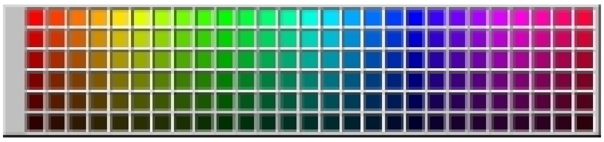


The color bar lets you set the line color and fill color of the selected entities.

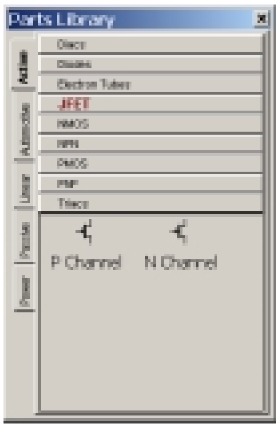
Click the **left mouse button** over a color patch to set the **fill color** to the color of the color patch.

Click the **right mouse button** over a color patch to set the **line color** to the color of the color patch.

Click the up arrow on the right to display the extended color patch matrix shown below. Now you can select from a wider choice of colors.



The Parts Library toolbar



The Parts Library toolbar is shown below. If you cannot see it, then select **<View/Part Library>** from the main menu.

It consists of:

1. A series of **vertical tabs** on the left representing part family groups such as Active part families.
2. A series **horizontal tabs** representing part families such as JFETs.
3. A **part display area** containing icons representing the parts in the database. Above we see 2 icons representing P and N channel JFETs.

The part display area has a popup menu activated by clicking the right mouse button.

Parts

To place a part on the sheet, click and drag the icon onto the sheet.

To edit a part, double click on the parts' icon or select **<Edit Part>** from the popup menu.

To rename a part, select the part and then click on the parts name or select rename from popup menu.

To delete one or more parts from the database, select them and select **<Delete>** from the popup menu.

You can edit the parts' icon by selecting **<Edit Icon...>** or **<Load Icon...>** from the popup menu.

Select **<New Part...>** from the popup menu to create a new part.

Part Families

A Part Family is a collection of related parts.

Select **<Family/Rename>** to rename a family collection.

To create a new family select **<Family/New>** from the popup menu.

To delete a collection select **<Family/Delete>** from the popup menu.

Part Family Collections

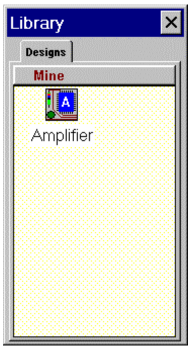
Part Family Collections are collection of families.

Select **<Collection/Rename>** to rename a family collection.

To create a new family selection select **<Collection/New>** from the popup menu.

To delete a collection select **<Collection/Delete>** from the popup menu.

The Design Library toolbar



The Design Library Toolbar is shown below.

If it is not visible select **<View/Design Library>** from the main menu.

- You can drag the icons, which represent design, onto the current sheet.
- You can double click the left mouse button over an icon to edit the design.

Right Mouse button menu.

You can right click the mouse button over the icon area to display popup menu with a list of commands. These commands are:

Add/to schematic. Adds the design to the current sheet.

Add/Selected to gallery. Adds the selected objects on the sheet to the gallery.

Add/Current schematic to gallery. Adds the current design to the gallery.

Edit. Edit the selected design.

Select/All. Select all icons.

Select/Invert. Invert the selection of icons.

Delete. Deletes the selected designs.

Cut. Copies the selected designs to the clipboard and then deletes them from the toolbar.

Copy. Copies the selected designs to the clipboard.

Paste. Pastes designs previously cut or copied to the clipboard.

View/Small Icons. Displays small icons.

View/Large Icons. Displays large icons.

View/List. Displays the designs as a list.

View/Report. Displays the designs as a report list.

Sort/Ascending. Sorts the designs by ascending names.

Sort/Descending. Sorts the designs by descending names.

Sort/None. Does not sort the designs.

Rename. Renames a design.

Properties. Display the designs' properties.

Library Path... Set the location for the library.

Collections

A collection is a collection of design folders.

Collection/Rename. Renames a collection.

Collection/New. Creates a new collection.

Collection/Rename. Deletes the current collection.

Folders

A folder is a group of designs.

Folder/Rename. Renames a folder.

Folder/New. Creates a new folder.

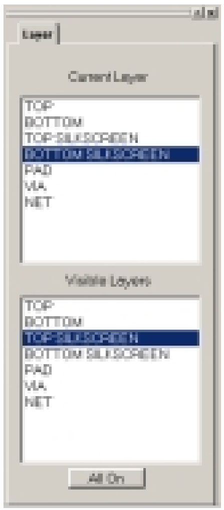
Folder/Rename. Deletes the current folder.

The Object Properties toolbar

The Object Properties toolbar is shown below. In this case it is for a line object.

See The Object Properties Toolbar

The Layers Toolbar



The Layers Toolbar is shown below.

If it is not visible select **<View/Layers Toolbar>** from the main menu.

The layers toolbar provides a quick means to:

- Set the current layer.
- See which layers are visible and which are not.
- Turn layers on or off.

At the top of the toolbar is a list box displaying all layers on the design. Click on the layer name to make it the current layer.

At the bottom of the toolbar is another list box that again displays all the layers in the current design. You can:

- Click on a layer name to display only that layer.
- Hold down the Ctrl button and click on a layer name to toggle its visibility.

Hold down the Shift button and click on a layer name to turn on only those layers between the selected name and the next selected name.

Toolbars

AutoTRAX comes with several standard toolbars. Some are visible when you first run AutoTRAX, while others are not.

To Show/Hide toolbars select **<View/Toolbars>** from the main menu and then select the toolbar to show or hide.

You can also use select **<Tools/Toolbars>** from the main menu.

To create your own custom toolbars see Custom Toolbars.

Docking Toolbars

Toolbars can either be “docked” or “floating.” You can move a docked toolbar to a floating location and vice-versa. Toolbars can be docked along all the outside edges of the Work Area, not just above it.

Moving a Docked Toolbar

A docked toolbar is attached to the top, left, or right sides of the Designer window. By default, the toolbars are docked to the top of the window when you launch Designer for the first time. To move a docked toolbar:

Click the toolbar anywhere but on a button.

Hold down the left mouse button.

Drag the toolbar to a new location.

Moving a Floating Toolbar

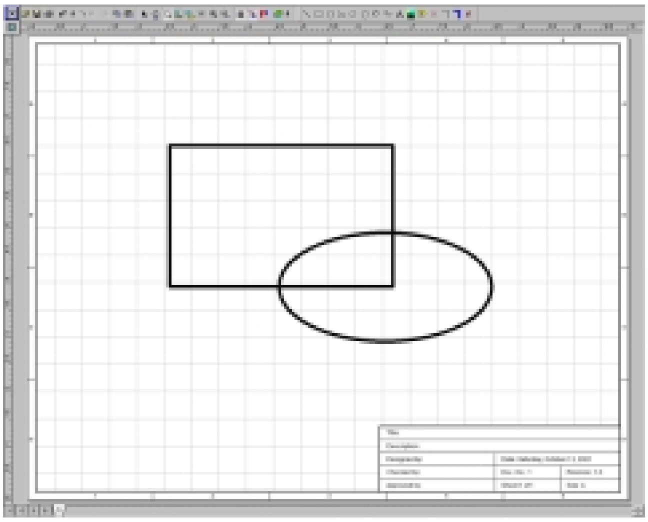
A floating toolbar can be placed anywhere within the Designer work space. To move one, click the blue toolbar title bar and drag it to a new location.

Using Full Screen mode

Select **<View/Full Screen>** from the main menu to utilize the full screen area.

Click the button to return to normal.

A Full screen view is shown below.



The Part Toolbar

The part toolbar is shown below:

If you cannot see the part toolbar, select **<View/Toolbars/Parts>** from the main menu.

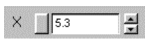
It consists of a collection of basic circuit elements.

Click on the element you wish to add to the current sheet and then move the mouse over the viewport. You will then see the new part follow the mouse. Click to place the part. Right click to cancel the part placement.

The toolbar can be docked to any of the 4 sides of the viewport.

Numeric Entry Controls

AutoTRAX incorporates a unique method of number entry. It consists of a combination of a numeric entry box, an up and down arrow and a popdown calculator as shown below for the variable x.



You can change the value by either:

Clicking on the numeric entry box (showing 4.75 in this case) and **typing** the new numeric value or

Clicking on the numeric entry box and then rotating the middle **thumbwheel** on your mouse. (If you have one!) or

Clicking either the **up or down arrows** on the right of the numeric entry box or

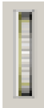
Press down the left mouse button over either the up or down arrow and **dragging the mouse** up or down the screen to increase or decrease the numeric value or

Click the small button on the left of the numeric entry field. The **calculator** show below will appear. You can then use it by clicking on the numbers and %/*- or + keys. Click Enter when you have done.

Thumbwheels

There are 2 types of thumbwheels. These are used to continuously vary a numeric value.

Vertical **Horizontal**



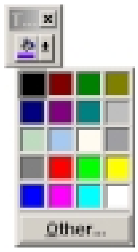
Scrollbars

AutoTRAX has it's own unique horizontal scrollbars such as the one shown below. Press and hold down the left mouse button over the slider button and drag the slider to the left or right to change its value. The new value is displayed in the center of the slider.

Unlike the conventional Windows scrollbars, if you move the cursor off the slider, the slider will not pop back to its original position.

Color Buttons

AutoTRAX incorporates unique dropdown color chooser. An example is the fill color chooser shown below.



Click on the paint bucket to fill the selected objects with the fill color.

Click on the down arrow to display the dropdown color chooser. Click the **other** button in the dropdown color chooser to display a more detailed color chooser.

Setting up your interface options

Select **<Tools/options>** from the main menu.

The Main menu

The main menu contains all the popdown menu commands.

It is initially at the top of the application but you can have it floating or docked to the bottom of the application.

Customizing AutoTRAX

Adding and removing keyboard shortcuts

Select <**Tools/shortcuts**> from the main menu. See the Shortcut Keys dialog below.

Adding new buttons to toolbars

Select <**Tools/Options**> from the main menu and select the Custom Toolbars tab. For more information, see the Custom Toolbars tab.

Creating new toolbars

Select <**Tools/Options**> from the main menu and select the Custom Toolbars tab. For more information, see the Custom Toolbars tab.

Showing and hiding toolbars

Select <**Tools/Options**> from the main menu and select the Toolbars tab. For more information, see the Toolbars tab.

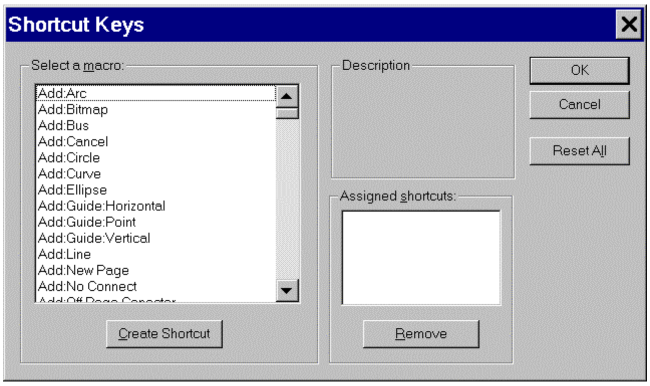
Setting Options

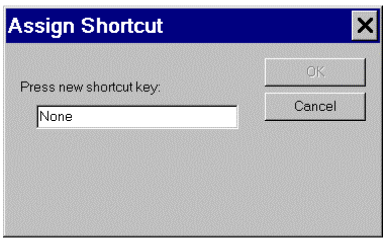
All AutoTRAX program settings are changeable using either

- The Options Dialog or
- The Shortcut Keys Dialog.

The Shortcut Keys dialog

The Shortcut Keys dialog is shown below.





Create Shortcut. First select one of the command/macros from the macro list. You will then be prompted for the keyboard key to act as the shortcut key.

Type key. You can press the **Alt**, **Shift** and **Ctrl** key at the same time.

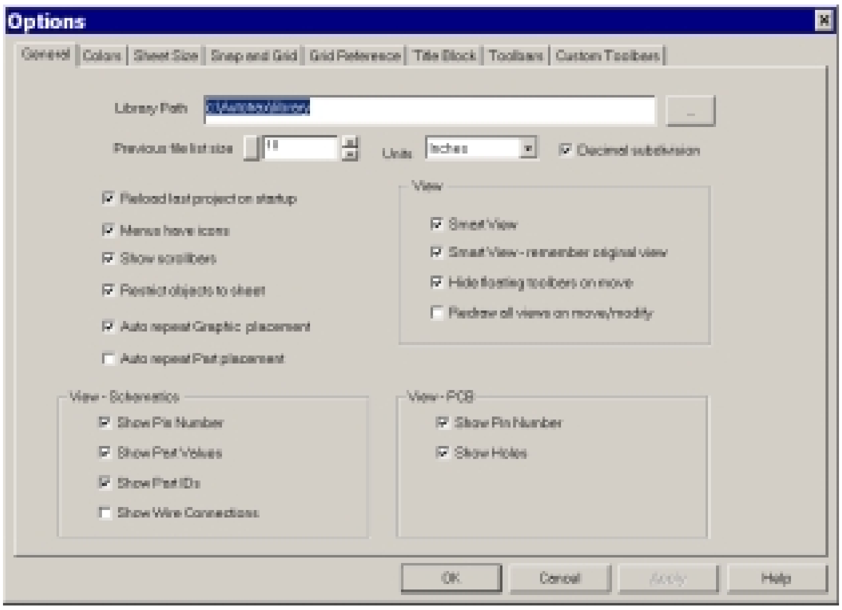
The Options dialog

Select <**Tools/Options**> to view the Options dialog. The options dialog contains several tabbed dialogs that group together configuration settings. These are:

- General.
- Colors.
- Sheet Size
- Snap and Grid
- Grid Reference
- Title Block
- Toolbars
- Custom Toolbars

General

The General options tab is shown below.



Library Path. This is the path for the components database. Click the to browse your file directories.

Previous file list size. Enter the maximum number of previous files to display in the File menu. The maximum number is 15.

Units. Enter the units that you wish to use. You can use millimeters, centimeters, meters, thou (thousandths of an inch) or inches.

Reload last project on startup. Check this to reload the last project that you saved when you start AutoTRAX.

Decimal subdivisions. If checked then rulers and the grid is divided into 10 units otherwise it is marked at ½ and ¼.

Show scrollbars. Check to show scrollbars on the viewports.

Restrict objects to sheet. Check to make it impossible to create and/or move objects off the sheet.

Highlight objects not on the sheet. Check this to display all objects that are partly or fully off sheet in red.

Auto repeat graphics placement. If checked, then if you start adding say a rectangle, after you have added one, you can start adding the next, until you cancel the command. Otherwise, the command mode cancels immediately after you have added a rectangle.

Auto repeat part placement. If checked, then if you start adding a part, after you have added one, you can start adding the next, until you cancel the command. Otherwise, the command mode cancels immediately after you have added a part.

View/Smart View. If checked, then the view will automatically pan when you move an object outside the view.

View/Smart View. Remember original view. In smart view, the viewport can remember your original position and will try to return the view to it if at all possible.

Hide floating toolbars on move. If checked, it will hide floating toolbars when you are moving objects. This lets you *'see the wood for the trees'*.

Redraw all views on move/modify. If checked, and you have split the viewport, each viewport will immediately reflect any change in the viewport in which you are working, otherwise they will be updated after you completed your action.

Show Pin Numbers. Show pin numbers on part terminals if checked.

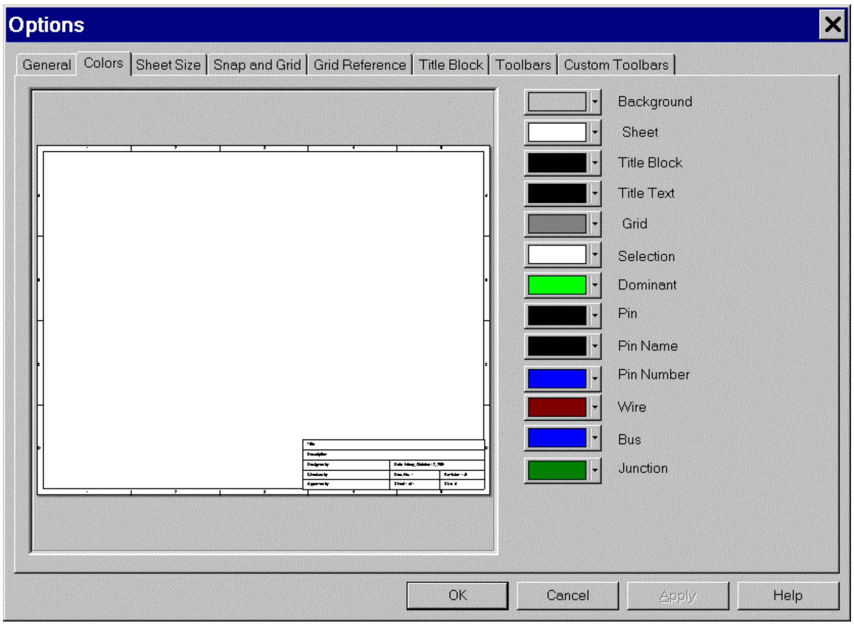
Show Part Values. Show part values if checked.

Show Part Ids. Show part ids if checked.

Show Wire Connections. Show wires junctions with other wises as a small solid circle if checked.

Colors

The Colors tab is shown below.



On the left is a view of the current sheet with the colors defined by the color buttons on the right.

Left click the mouse on a color button to change the color.

Background. The color of the background area surrounding the sheet.

Sheet. The color of the sheet.

Title Block. The color of the Title Block.

Title Text. The color of the text in the Title Block.

Grid. The grid color.

Selection. The color of the manipulator points for selected objects that are not the dominant selected objects.

Dominant. The color of the manipulator points for dominant selected objects.

Pin. Pin text color.

Pin Name. Pin name color.

Pin Number. The color of pin numbers.

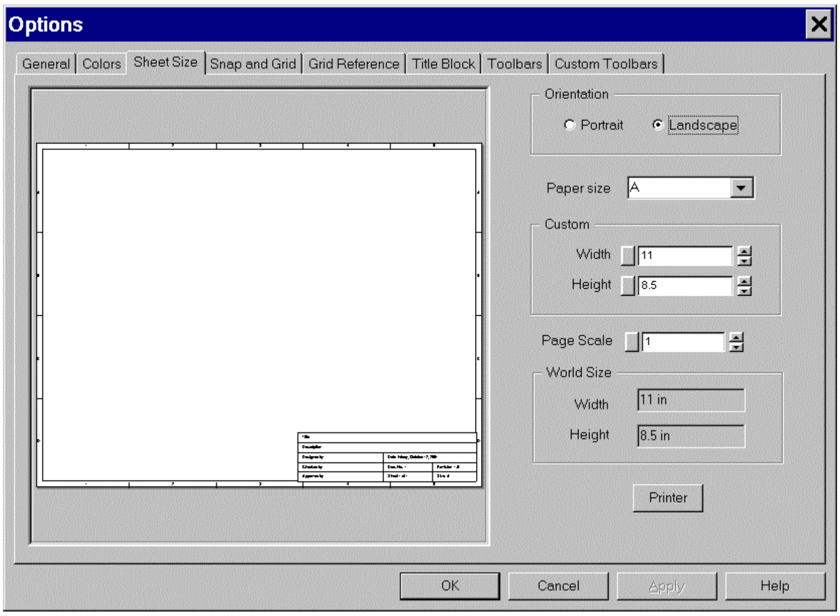
Wire. The color of wires (electrical connections).

Bus. The color of electrical buses.

Junction. The color of electrical junctions.

Sheet Size

The Sheet Size tab is shown below.



Orientation. Check the Portrait radio button to have the shortest edge of the sheet aligned horizontally or check the Landscape radio button to align the longest edge horizontally.

Paper Size. This dropdown list contains a list of standard paper sizes. A-E, A0-A4, and Custom.

Custom. This displays the Width and Height of the sheet. You can enter a custom size for either the width or the height. Alternatively you can set the sheet to a standard size using the Paper Size dropdown list.

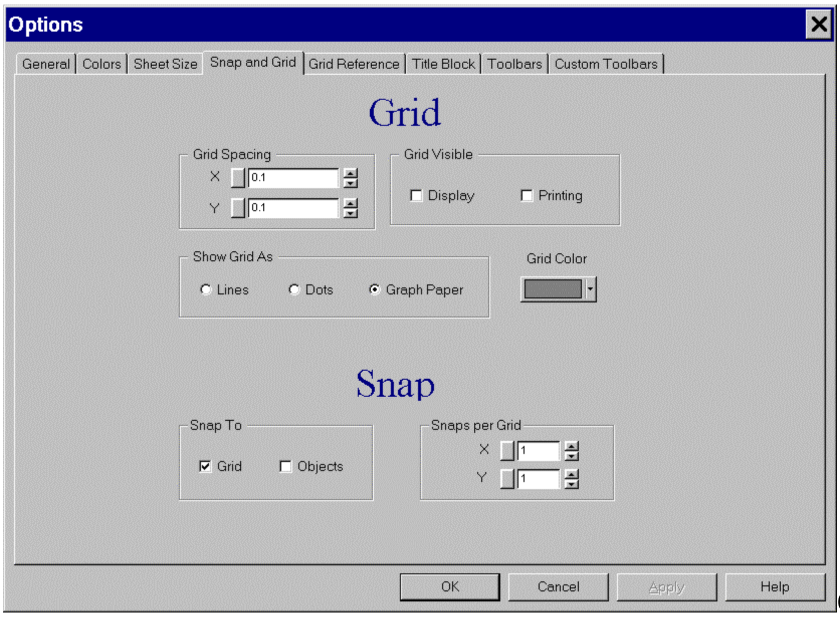
Page Scale. Here you can set the units scale for the sheet.

World Size. This is the scale of the world to paper size. If it is set to 1 then the page will print at a scale of 1:1. If it is set at 2 it will print half size and is set to 0.5 it will print twice the page size.

Printer. Click to set the printer.

Snap and Grid Settings

The Snap and Grid tab is shown below.



Grid Spacing

Here are displayed the current X and Y grid spacing. You can change each spacing separately.

Grid Visible

Display. If checked then the grid will be visible on the screen.

Printing. If checked then the grid will be drawn on all printed output.

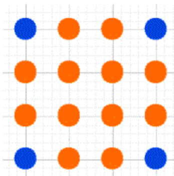
Snap To

Grid. If checked, then points will snap to the grid. See View/Grid/Snap to Grid.

Object. If checked, then points will snap to objects. See View/Grid/Snap to Objects.

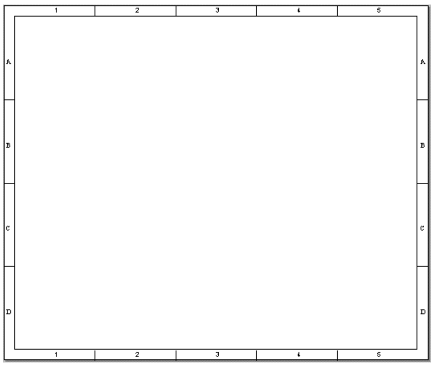
Snaps per Grid

This shows the number of snaps point per grid point. If set to 1 then there is a snap point only at a grid point. If set to another number then there are snap points between grid points. For instance, if set to 3 then there are 2 additional snap points between each grid point. The diagram below shows an array of dots representing snap points. The blue represent the grid spacing and the orange represent the additional snap points if Snaps Per Grid is set to 3 for both X and Y.

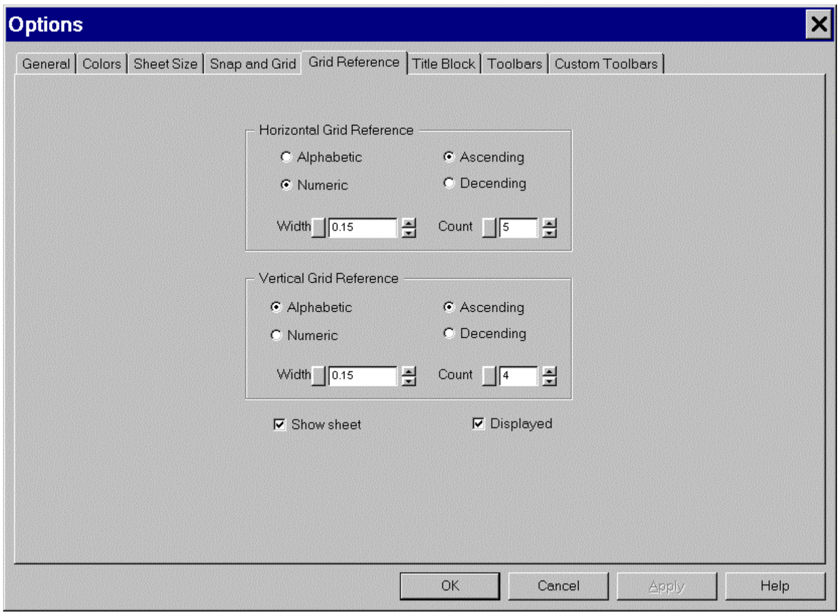


Grid Reference

The grid reference shown below consists of horizontal and vertical markers.



The Grid Reference tab is shown below.



An optional Grid Reference can place around the 4 edges of the sheet. An example is shown below. (The Title Box has been removed). It consists of 2 horizontal bars, one at the bottom and one at the top of the sheet, and 2 vertical bars on the left and right edges.

The bars have text reference ids. The type of text, the number of ids and their order can be set independently.

You can set the style for the horizontal and vertical rid References separately.

Alphabetic. Check this radio button to display the IDs as alphabetic text. The first starts with the letter A and then B etc.

Numeric. Check to display the IDs as numbers starting at 1.

Ascending. Check to have the IDs in ascending order from left to right/ top to bottom.

Descending. Check to have the IDs in descending order from left to right/ top to bottom.

Width. Enter the width of the reference bar.

Count. Enter the number of reference IDs on the bar.

Show sheet. Check to show the sheet border. If not checked, the sheet border is not drawn and no background will be drawn.

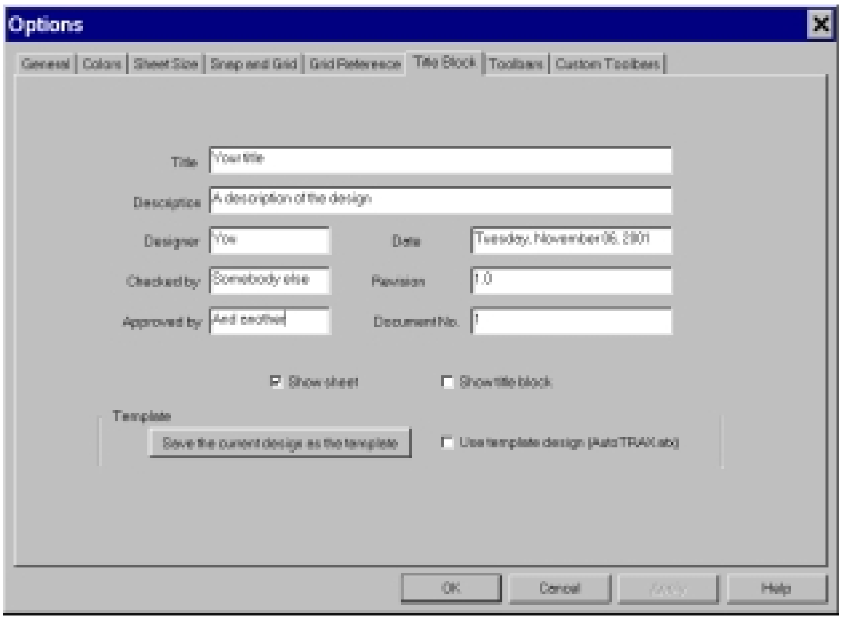
Displayed. Check to show the Grid Reference. Uncheck to hide it, the Grid Reference will not be drawn.

Title Block

The Title Block is shown below. It appears at the bottom right of the sheet.

Title:		
Description:		
Designed by:	Date: Saturday, October 13, 2001	
Checked by:	Doc. No.: 1	Revision: 1.0
Approved by:	Sheet 1 of 1	Size: Custom

The **<Tools/Options/Title Block>** tab is shown below. This allows you to define the Title Block that optionally appears at the bottom right of the sheet.



Title. The title of the sheet.

Description. A description of the design.

Designed by. The designers' name.

Checked by. The checkers' name.

Approved by. The name of the person who approved the design.

Doc No. The document number.

Revision. The design revision.

Sheet x of y. x = the sheet number and y = the number of sheets.

Size. The page size.

Show Sheet. Check to show the sheet and border, uncheck to hide it.

Show Title Block. Check to show the title block, uncheck to hide it.

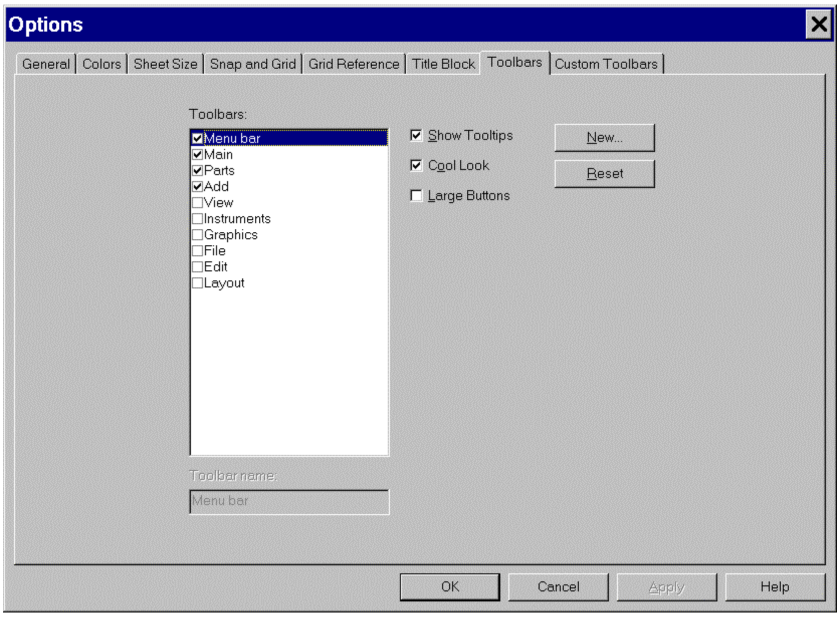
Click **OK** to accept your changes.

Toolbars

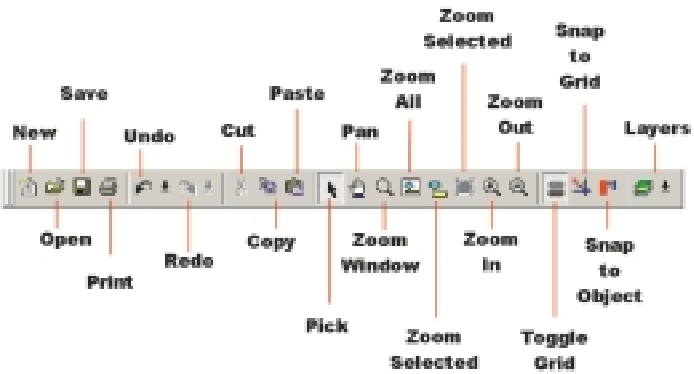
When the designers of AutoTRAX were developing AutoTRAX, they conducted usability tests to determine which commands and procedures users ran most often. From the results of these tests, they created a collection of toolbars that provide access to the commands and procedures that users found most helpful for a particular task. The most popular buttons were placed on the Main toolbar; commands for adding instruments were placed on the Instruments toolbar, and so on.

When you first start AutoTRAX, you will see one or two toolbars at the top of the application.

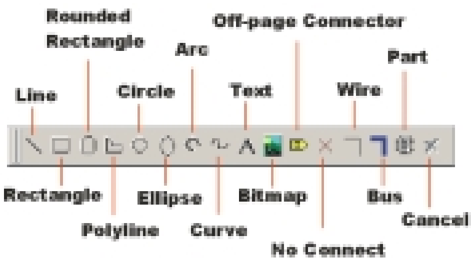
The **Tools/Options/Toolbars** tab is shown below. This displays a list of toolbars and allows you to show or hide a toolbar. If the toolbar is visible then the checkbox on the left will be checked. Check the checkbox to show/hide a toolbar.



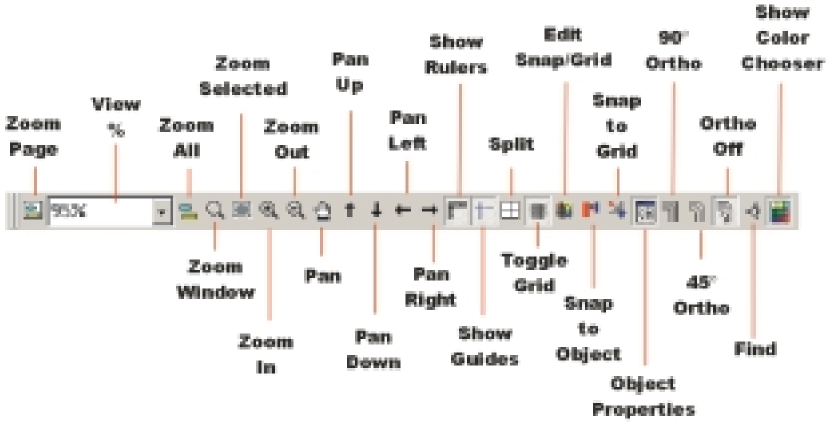
The Main Toolbar



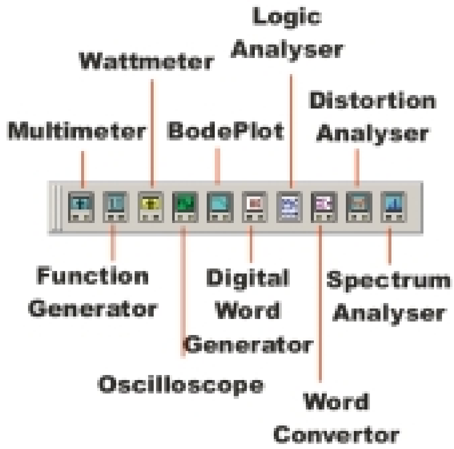
The Drawing Toolbar



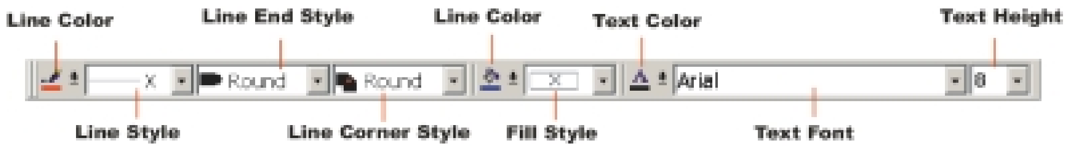
The View Toolbar



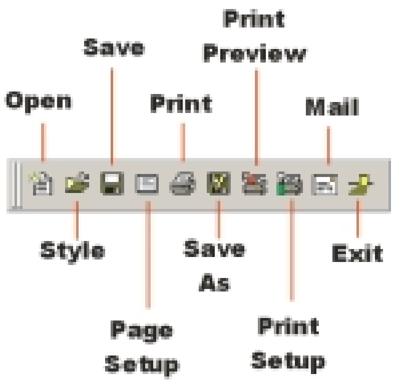
The Instruments Toolbar



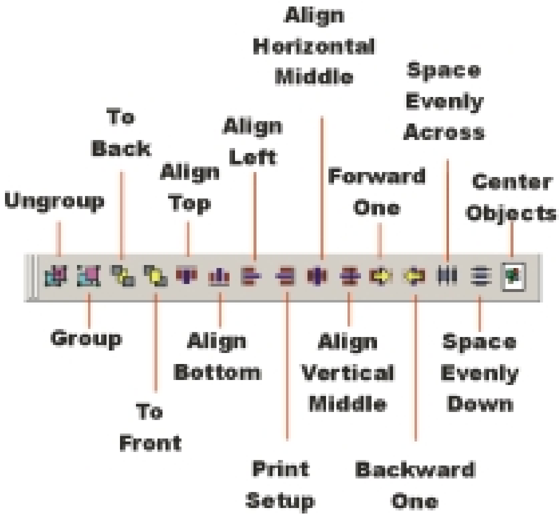
The Graphics Toolbar



The File Toolbar



The Layout Toolbar



Show Tooltips. Check to include tooltips on the toolbars. A tooltip is a text message with a yellow background that appears when you place the mouse cursor over a toolbar button. The message will disappear after about 4 seconds. An example of a tooltip is shown below for the View Full Page button .

Cool Look. Check to have Cool Look toolbars. Uncheck to revert to standard toolbars.

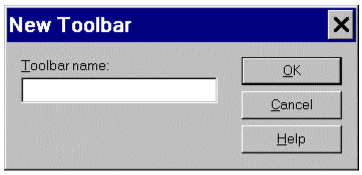
Standard Toolbar

Cool Look Toolbar

Large Buttons. Check to display large buttons.

Small Button Large Button

New… Click this button to create a new toolbar. You will be prompted for the name of the new toolbar.

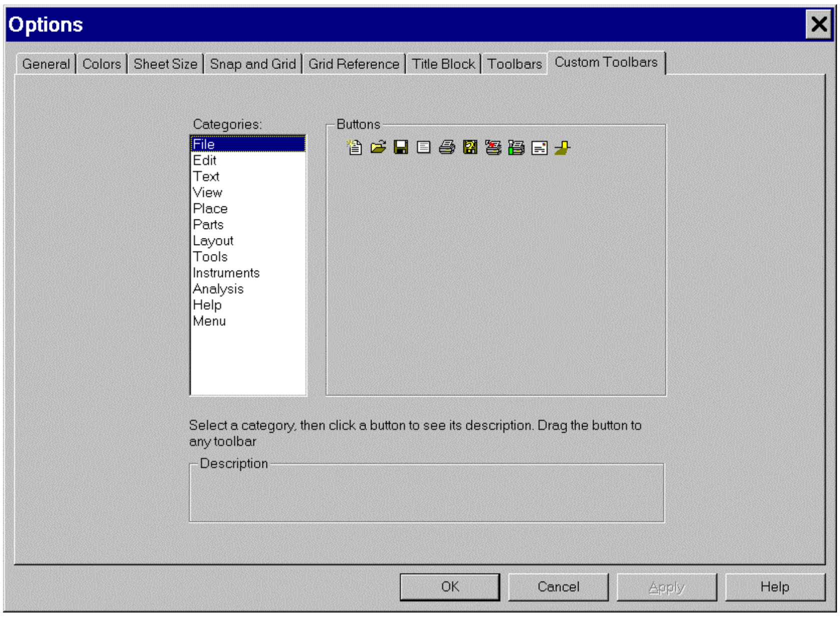


Enter the name for the new toolbar and click the OK button.

Reset. This will reset the toolbars to the *'factory'* setting. These are the settings when AutoTRAX was first installed.

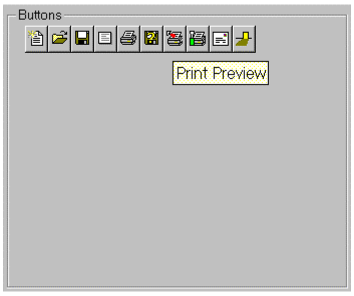
Custom Toolbars

The Tools/Options/Custom Toolbars tab is shown below. AutoTRAX gives you complete control over the toolbars displayed in your workspace. You can create toolbars, add buttons to a toolbar, remove buttons and rearrange buttons.



Adding Buttons to a Toolbar

You can add buttons to a toolbar. The Custom Toolbars tab shown above contains a Categories list box, which displays the type of commands you can use. When you select a category in the list box, the associated commands appear in the Buttons box to the right. Manly of the commands featured in the functional categories are not included in the default toolbars, so take a few minutes to review your options and pick you favorites to help improve your productivity.



Many of the command names also have buttons associated with them, and these tend to make the most efficient toolbar buttons. If you want to learn more about what a button either:

- Click the button. The Description area will display information on what the button.
- Hover' the mouse cursor over the button. A tooltip will appear as shown below.

To add a button to a toolbar, follow these steps:

1. Display the toolbar you wish to modify.
2. Choose Tools/Options from the main menu and select the Custom Toolbars tab.
3. Click the group in the Categories list box containing the command you wish to select. For example to select the Mail button (a button than does the same as the <File/Send> menu item) click the File category.
4. Drag the button you want from the dialog box to the target toolbar. (As you drag, the mouse pointer changes to a toolbar pointer). Place the button exactly where you want it on the toolbar, and then release the mouse button. AutoTRAX will shift existing buttons to the right to make room for the new button.

Repeat steps 3 and 4 to add additional buttons, if necessary. When you have finished, click the OK button.

Removing Unwanted Buttons

If you have wish to remove one or more buttons from an existing toolbar, follow these steps:

1. Display the toolbar you wish to modify.
2. Choose Tools/Options from the main menu and select the Custom Toolbars tab.

3. Pick the toolbar and then the button you want to remove. Click and drag the toolbar button (the one on the toolbar, not the one in the dialog box) away from the toolbar and release the mouse button and the toolbar will be removed from your toolbar.
4. Continue to remove as many buttons as you like. Click OK when you are finished.

Rearranging Toolbar Buttons.

You can also change the order of toolbar buttons, or move buttons from one toolbar to another, while the Options dialog box is open and the Custom Toolbars tab is selected. Like adding and removing toolbar buttons, rearranging toolbar buttons is a matter of dragging the buttons from one location to another.

To rearrange toolbar buttons, follow these steps:

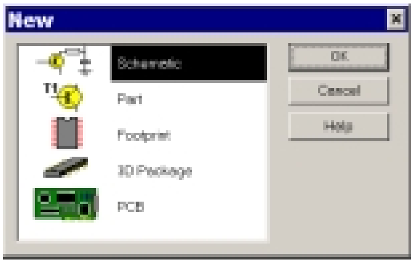
1. Display the toolbar you want to rearrange. If you want to move buttons from one toolbar to another, display both toolbars.
2. Choose Tools/Options from the main menu and select the Custom Toolbars tab.
3. Drag toolbar buttons from one location to the next. When you release the mouse button, the button you have just dragged will be relocated.
4. Continue to rearrange as many buttons as you like. Click OK when you are finished.

Creating, saving and opening designs

To create a new design select **<File/New>** from the main menu.

The **New dialog box** shown below will appear. Select either:

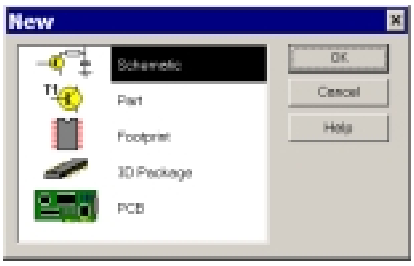
1. Schematic. This will create a schematic.
2. Part. This will create a component part for use in a schematic.
3. Footprint. This will create a footprint for use in a PCB design.
4. 3D Package. This will create a 3D package for use in a PCB design.
5. PCB. This will create a PCB.



Creating a new Schematic

To create a new design select **<File/New>** from the main menu.

The **New dialog box** shown below will appear. Select **Schematic** to create a new schematic design.



Creating a schematic template

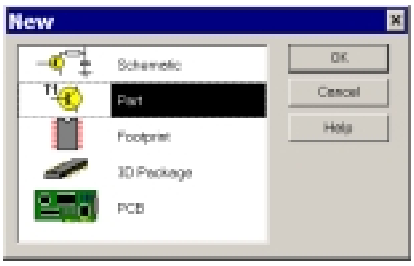
You can create a schematic template that is optionally used as the base template for all new schematic. To create a template:

1. Create a new schematic.
2. Select **<File/Save as Schematic Template...>** from the main menu.

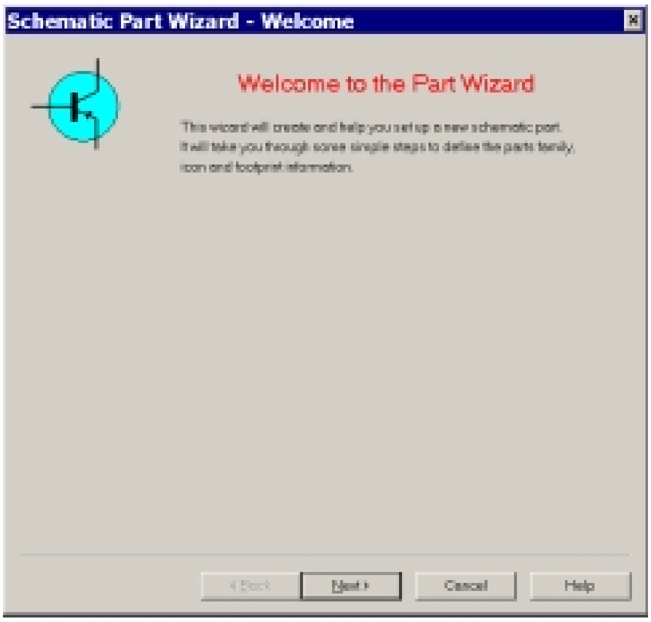
Creating a part

To create a new part, select **<File/New>** from the main menu.

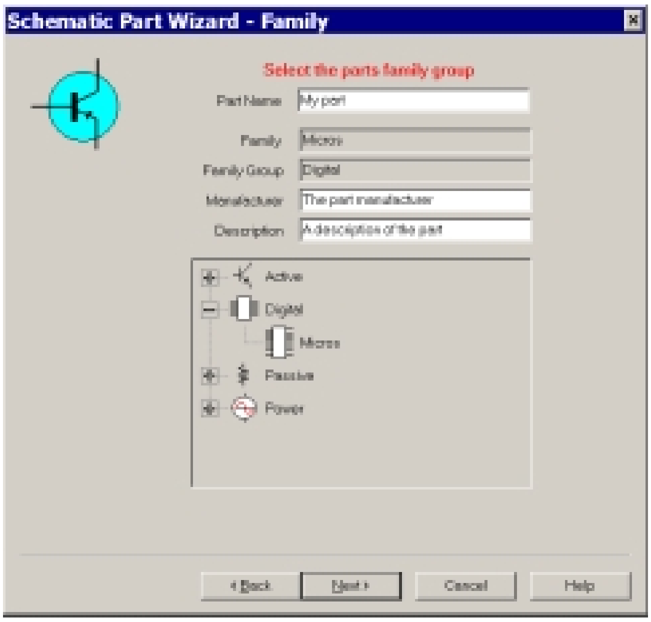
The **New dialog box** shown below will appear. Select **Part** to create a new part.

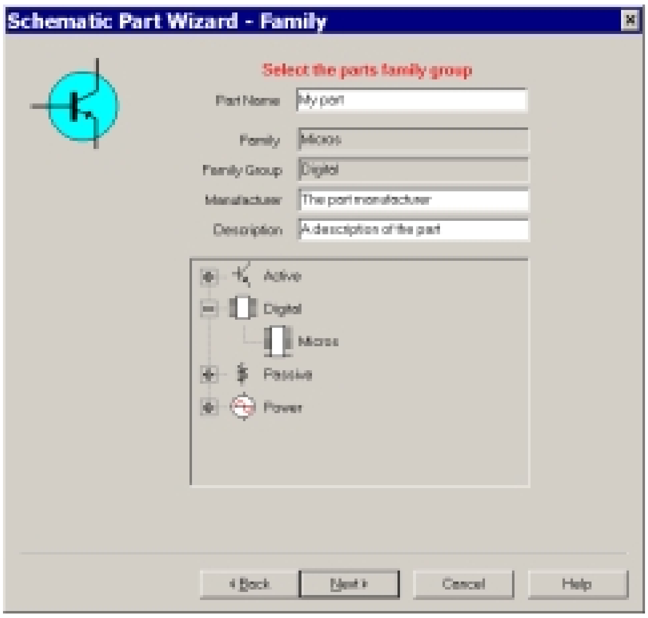


This will start the part wizard. The first page is shown below.



Click on the **Next** button to start.





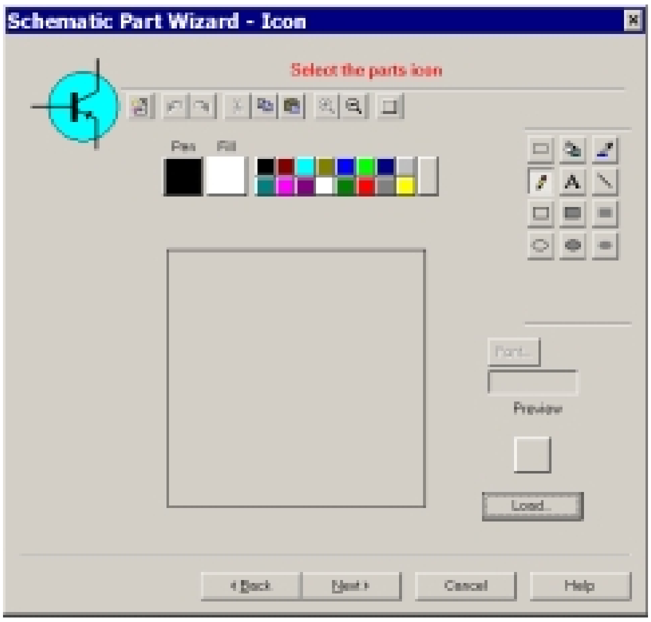
Select the family that you want the part to be in and enter the **Part Name.** This will save a basic part in the database.

Manufacturer. Enter the manufacturer's name

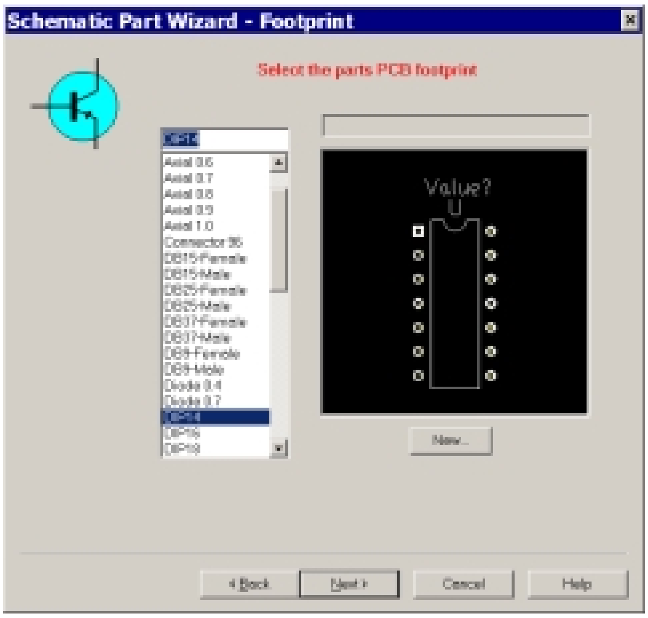
Description. Enter a description for the part.

Click **OK** to continue or **Cancel** to cancel creating a new part.

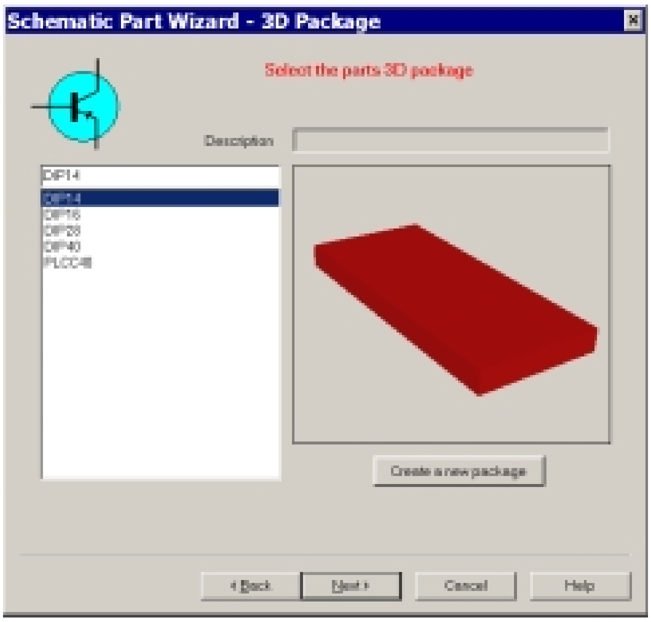
Next the Icon Editor will appear. You need to associate an icon with the part.



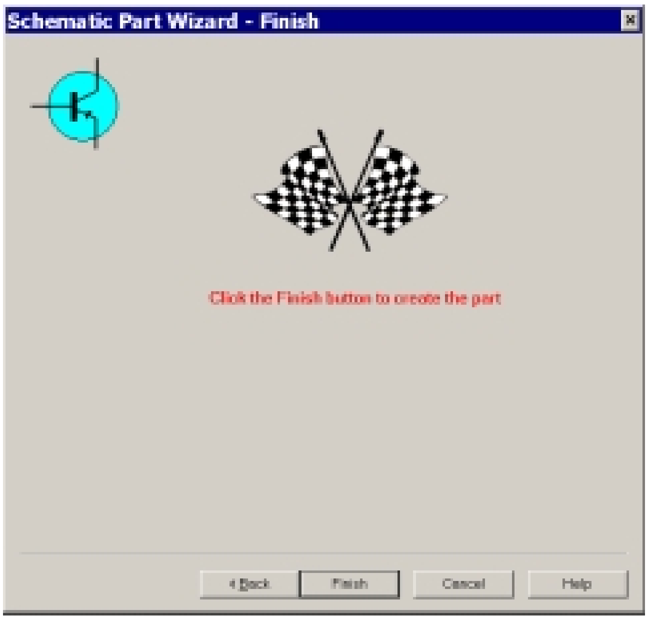
Next you will be asked to associate a PCB footprint with the part. You do not need to assign one as you can assign one later.



When you have selected a footprint. Click the Next button. You will then be able to assign a 3D package.



Select the 3D package from the list of packages and then click the **Next** button. Again you do not need to specify a 3D package, you can assign it later.



Finally click the **Finish** button.

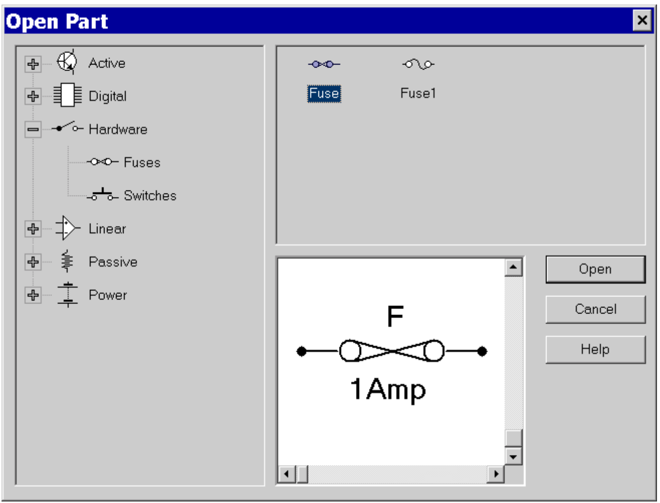
You will now see a basic part with:

1. A part border.
2. **A part reference.**
3. **A part value.**

Opening schematics

To open an existing part select **<File/Open/Schematic>** from the main menu.

Opening parts



To open an existing part select **<File/Open/Part>** from the main menu. The dialog box shown below will appear.

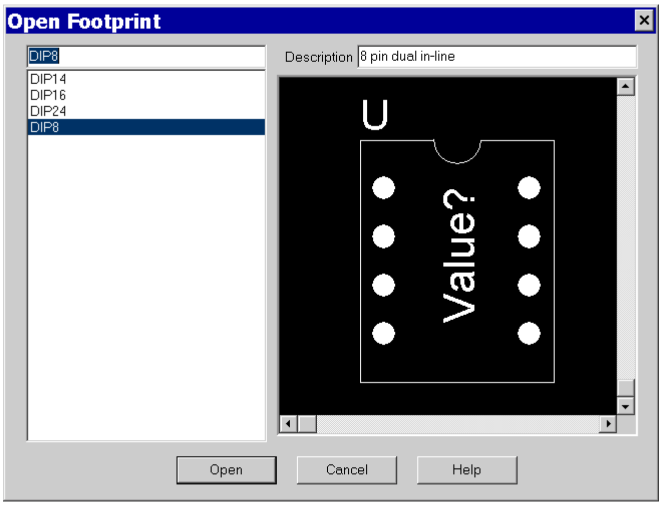
1. By clicking on the icons that in the part family tree control on the left of the dialog box, you can select a part family.
2. The parts in the selected family are shown in the parts area at the top left of the dialog box. Click on one of them to select a part. The selected part will appear in the part view area just below the parts area. You can pan around the view of the part by holding down the middle mouse button and then dragging the mouse. Rotating the thumbwheel will zoom in/out on the part.
3. Click on the **Open** button to open the part for editing or double click the left mouse button on a part icon.

Click the **Cancel** button to cancel opening a part.

Clicking the **Help** button will display this page.

Opening footprints

To open an existing part select **<File/Open/Footprint>** from the main menu. The dialog box shown below will appear.



A list of available footprints is displayed in the list box on the left of the dialog boxes. Click on one of them to view the footprint in the viewer on the right. You can pan around the view of the footprint by holding down the middle mouse button and then dragging the mouse. Rotating the thumbwheel will zoom in/out on the footprint.

A description of the footprint is displayed above the view of the footprint.

Click the **Open** button to open the footprint for editing or double click the left mouse button on a footprint name.

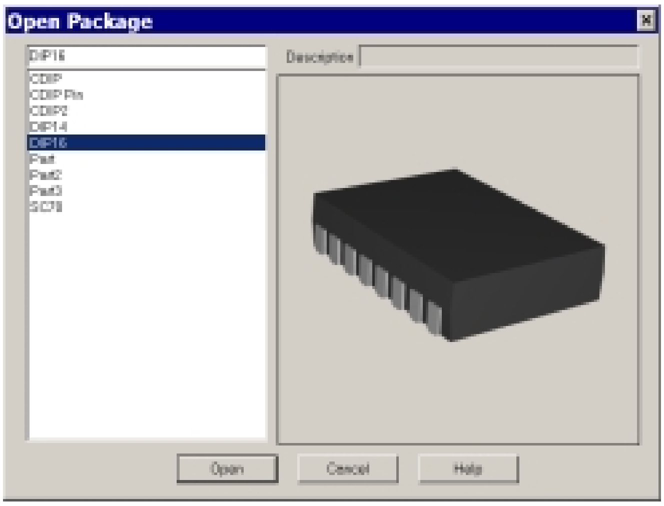
Click the **Cancel** button to cancel opening the footprint.

Clicking the **Help** button will display this help page.

Opening PCB designs

To open an existing part select **<File/Open/PCB>** from the main menu.

Opening 3D packages



To open an existing part select **<File/Open/3D Package>** from the main menu. The dialog box shown below will appear.

A list of available packages is displayed in the list box on the left of the dialog boxes. Click on one of them to view the package in the viewer on the right. You can pan around the view of the package by holding down the middle mouse button and then dragging the mouse. Rotating the thumbwheel will zoom in/out on the package.

A description of the package is displayed above the view of the package.

Click the **Open** button to open the package for editing or double click the left mouse button on a 3D package name.

Click the **Cancel** button to cancel opening the package.

Clicking the **Help** button will display this help page.

Saving a design

To save an existing design select **<File/Save>** from the main menu. If it has not already been saved, you will be prompted for a design name to save it as.

To save an existing design to a different name select **<File/Save As>** from the main menu. You will be prompted for a design name to save it as.

NOTE: *If you exit AutoTRAX and have failed to save a design that has been modified, you will be prompted to save it.*

Saving a design to a different name

To save an existing design to a different name select **<File/Save As>** from the main menu. You will be prompted for a design name to save to.

Sheets and Pages

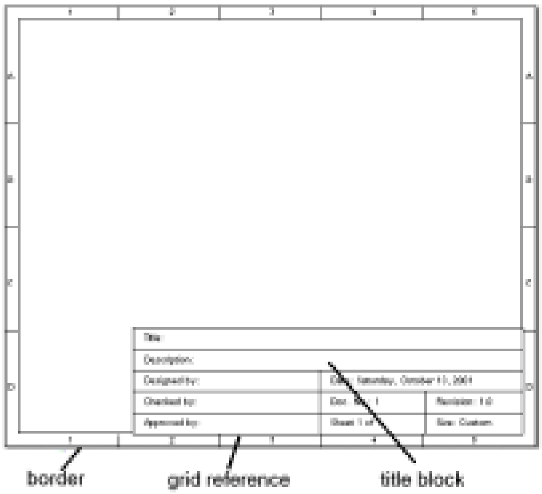
What is a sheet?

A design is consists of one or more sheets (pages).

A sheet can have a:

- Border. A rectangular area defined by the page size.
- Grid reference. This consists of reference markers around the sheet.
- A title box. This contains information about the design.

All sheets are named. The name is displayed in the tab at the bottom of the viewport.



You can

- Resize a sheet
- Add a sheet
- Delete a sheet
- Rename a sheet
- Hide/Show the border
- Hide/Show and configure the grid reference
- Hide/Show and configure the title box

Adding a sheet

To add a new sheet either:

- Select **<Add/New Sheet>** from the main menu or
- **Click the right mouse button** over any of the sheet tabs. A popup menu will appear. Select **<New Sheet>** from the menu.

Deleting sheets

To delete a sheet either:

1. Select the sheet by clicking the left mouse button over its name tab at the bottom of the viewport.
2. Select **<Edit/Delete Current Sheet>** from the main menu

or

1. Select the sheet by clicking the left mouse button over its name tab at the bottom of the viewport.
2. Click **the right mouse button** over any of the sheet tabs. A popup menu will appear. Select **<Delete>** from the menu.

Changing the size of the sheet

See the Sheet Size tab in the Options dialog.

Changing the name of a sheet

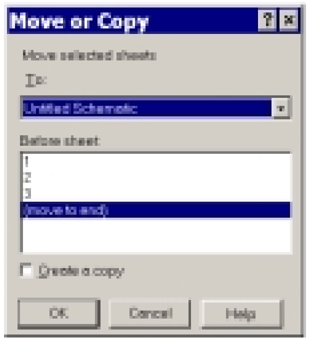
To rename a sheet, double click the left mouse button on the name tab at the bottom of the viewport and type in the new name.

Switching between sheets

To switch to a different sheet, click the left mouse button on the sheets' name tab at the bottom of the viewport.

Alternatively you can right click the mouse over the sheets' tab and select **<Move or copy...>** from the popup menu. This will display the popup menu shown below.

Changing the sheet order



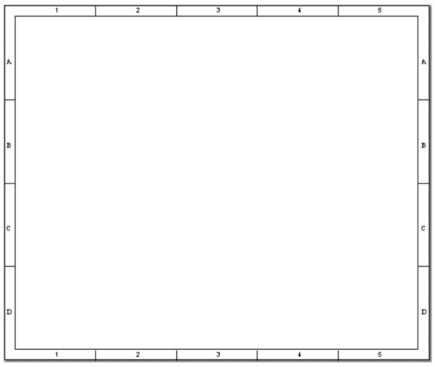
To change the sheet order, press down the left mouse button over the sheets' name tab and drag the tab to its new position.

Hiding and showing the border

You can hide the sheet and its border. See the Title Block tab in the Options dialog.

The Grid Reference

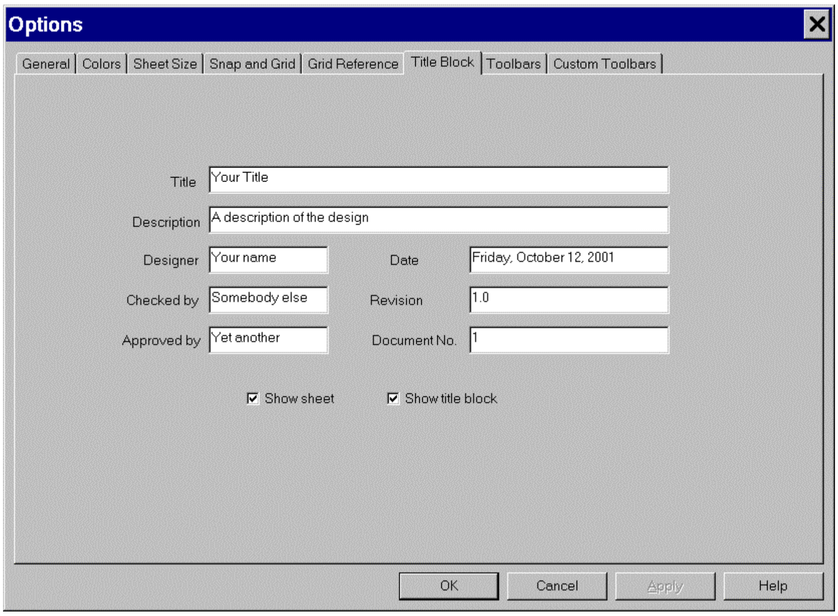
The grid reference is shown below and consists of horizontal and vertical markers.



To setup the grid reference see the Grid Reference tab in the Options dialog.

The Title Box

The title box is shown below.



Title. The title of the sheet.

Description. A description of the design.

Designed by. The designers' name.

Checked by. The checkers' name.

Approved by. The name of the person who approved the design.

Doc No. The document number.

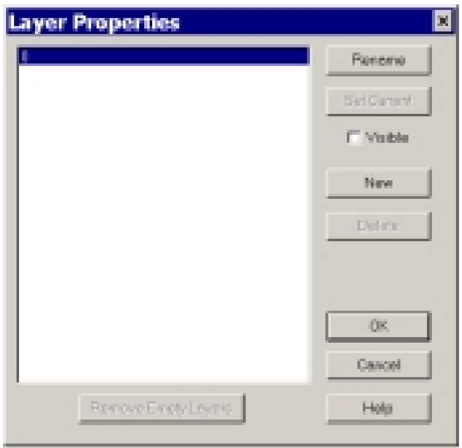
Revision. The design revision.

Sheet x of y. x = the sheet number and y = the number of sheets.

Size. The page size.

You can set the fields using the Title Box tab in the Options dialog.

Adding and deleting layers



To add or delete layers select **<Edit/Layers...>** from the main menu.

Rename. Rename the selected layer.

Set Current. Sets the current layer to the selected layer.

New. Adds a new layer.

Delete. Delete the selected layer.

Remove Empty Layers. Remove layers that do not have objects on them.

OK. Accept your changes.

Cancel. Cancel the changes.

Help. Display this help topic.

Printing

Setting the printed page size

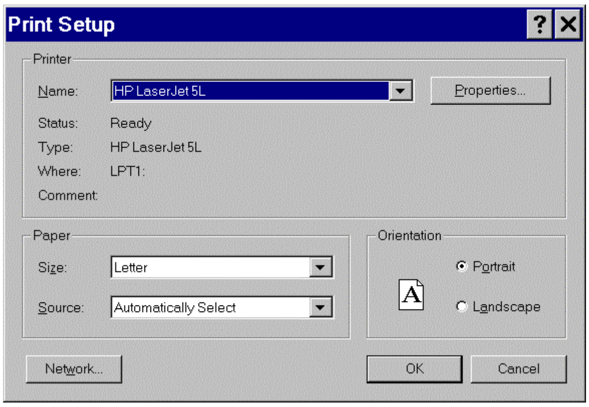
See the Sheet Size tab in the Options dialog.

Selecting and configuring the printer

To setup and configure the printer:

1. Select <**File/Print Setup…**> from the main menu.

The Print Setup dialog will be shown. The dialog below shows the print setup dialog for a HP LaserJet 5L printer.



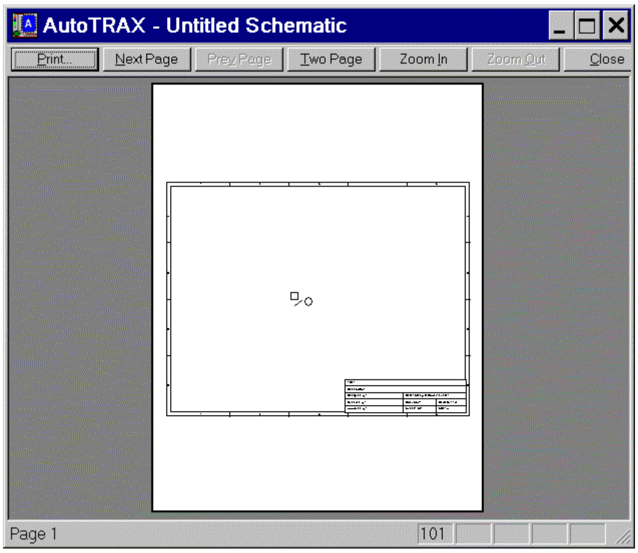
Click the **Properties...** button to configure the printer.

Previewing printed output

To preview the printed output:

1. Select <**File/Print Preview**> from the main menu.

The Print Preview dialog will be shown.

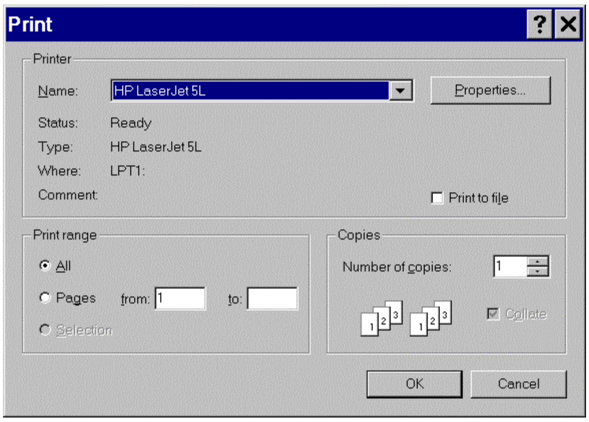


Printing the current design

To print the current design:

1. Select **<File/Print...>** from the main menu.

The Print Dialog shown below will be displayed. The dialog below shows the print dialog for a HP LaserJet 5L printer. If may be different from your printer as some printers have their own dialogs.

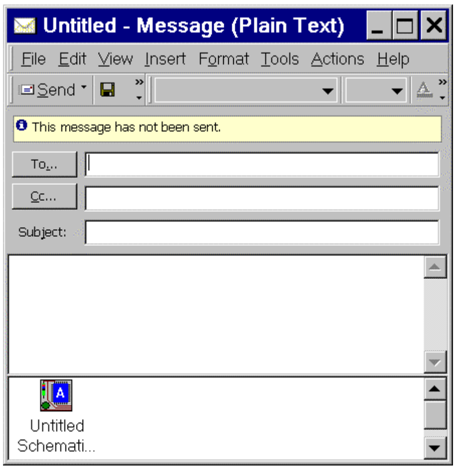


E-Mailing designs to others

Mailing your design to others

To mail the current design to another person

1. Select <**File/Send...**> from the main menu. The mail dialog box shown below will be displayed.



The current design has been attached to the email message.

Fill in the **To... CC...** and **Subject** fields.

Type in any notes in the text message area.

Click the **Send** button at the top left to send the design.

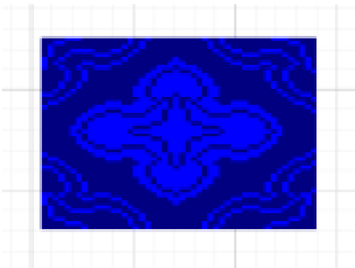
Drawing Shapes

With AutoTRAX you can draw the following non-electrical shapes:

- Lines
- Polylines
- Rectangles
- Rectangles with rounded corners
- Circles
- Ellipses
- Arcs
- Curves
- Images (bitmaps)

Adding an image (bitmap)

To draw one or more bitmaps:



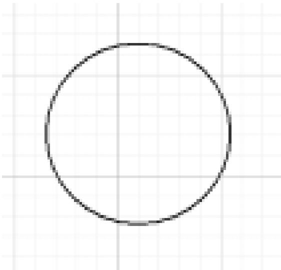
2. Select **<Add/Bitmap>** from the main menu.
3. Move the mouse to the first corner of the bitmap. Then either
4. Press the left mouse button down and drag the mouse to the other corner of the bitmap rectangle and release the left mouse button when the bitmap is the correct size to complete the bitmap or
5. Click the left mouse button at the first corner of the bitmap, and move the mouse until the bitmap is the required size and then click the left mouse button again to complete the bitmap.
6. You will then see a File Open Dialog. Select the bitmap and click on the **Open** button.
7. Repeat 2,3 and 4 to create more bitmaps or right click to return to the normal mouse mode.

If you press and hold down the **SHIFT** button while dragging the bitmap, then the bitmap will be centered at the position of the left mouse press.

You can press the **ESC** key or the right mouse button to cancel the image creation command.

Drawing a circle

To draw one or more circles:



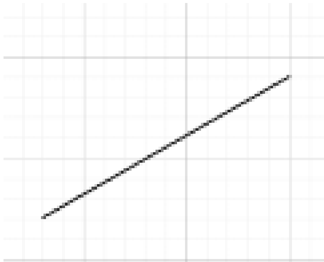
1. Select **<Add/Circle>** from the main menu.
2. Move the mouse to a corner of the circles' bounding box. Then either
3. Press down the left mouse button and drag the mouse to the other corner of the circle's bounding box and release the left mouse button when the circle is the correct size or
4. Click the left mouse button to start the circle, drag the mouse until the circle is the required size and then click the left mouse button again to complete the circle.

5. Repeat 2,3 and 4 to create more circles or right click to return to normal mouse mode.

If you press and hold down the **SHIFT** button while dragging the circle, then the circle will be centered at the position at which you pressed left mouse button.

You can press the **ESC** key or the right mouse button to cancel the circle creation command.

Drawing a line



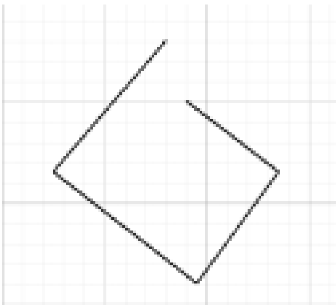
To draw one or more lines:

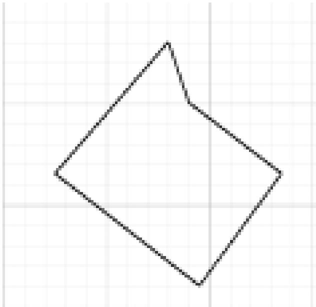
1. Select **<Add/Line>** from the main menu.
2. Move the mouse to the start of the line. Then either
3. Press down the left mouse button and drag the mouse. The line will then appear. Release the left mouse button when the line is the correct length and direction or
4. Click the left mouse button to start the line, and then drag the mouse until the line is the required length and direction. Finally click the left mouse button again to complete the line.
5. Repeat 2,3 and 4 to create more lines or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the line creation command.

You can restrict lines to vertical, horizontal of 45º by turning ortho on.

Drawing a polyline or polygon





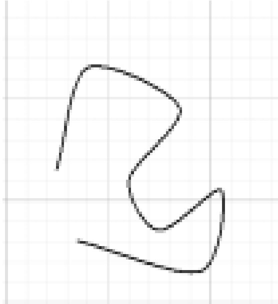
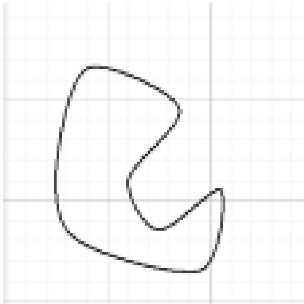
To draw one or more polylines:

1. Select **<Add/Polyline>** from the main menu.
2. Move the mouse to the start of the polyline.
3. Click the left mouse button to start the polyline, and then drag the mouse until the polyline segment is the required length and direction. Click the left mouse button again to start another segment. Again drag the mouse until the new polyline segment is the required length and direction. Keep on repeating this until the polyline is complete. *(If you click on the start of the first polyline segment, then the polyline will be **closed**.)* Finally click the right mouse button to end.
4. Repeat 2 and 3 to create more polylines or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the polyline creation command.

You can restrict lines to vertical, horizontal of 45º by turning ortho on.

Drawing a curve

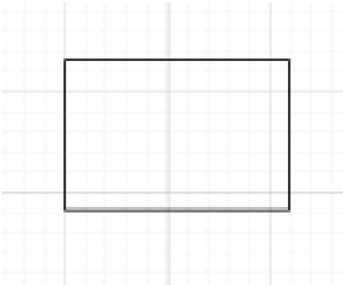
To draw one or more curves:

1. Select **<Add/Curve>** from the main menu.
2. Move the mouse to the start of the curve. Then either
3. Click the left mouse button to start the curve, and then drag the mouse until the curve segment is the required length and direction. Click the left mouse button again to start another segment. Again drag the mouse until the new curve segment is the required length and direction. Keep on repeating this until the curve is complete. *(If you click on the start of the first curve segment, then the curve will be* ***closed****.)* Finally click the right mouse button to end.
4. Repeat 2 and 3 to create more curves or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the line creation command.

You can restrict lines to vertical, horizontal of 45º by turning ortho on.

Drawing a rectangle



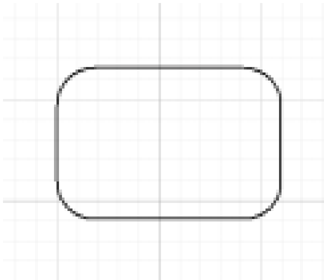
To draw one or more rectangles:

1. Select **<Add/Rectangle>** from the main menu.
2. Move the mouse to the first corner of the rectangle. Then either
3. Press the left mouse button down and drag the mouse to the other corner of the rectangle and release the left mouse button when the rectangle is the correct size to complete the rectangle or
4. Click the left mouse button at the first corner of the rectangle, and move the mouse until the rectangle is the required size and then click the left mouse button again to complete the rectangle.
5. Repeat 2,3 and 4 to create more rectangles or right click to return to the normal mouse mode.

If you press and hold down the **SHIFT** button while dragging the rectangle, then the rectangle will be centered at the position of the left mouse press.

You can press the **ESC** key or the right mouse button to cancel the rectangle creation command.

Drawing a rectangle with rounded corners



To draw one or more rectangles:

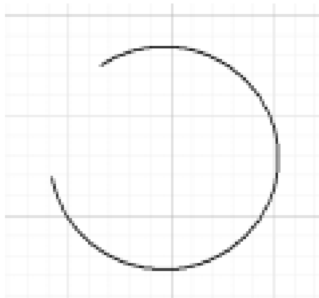
Select **<Add/Rounded Rectangle>** from the main menu.

1. Move the mouse to the first corner of the rectangle. Then either
2. Press the left mouse button down and drag the mouse to the other corner of the rectangle and release the left mouse button when the rectangle is the correct size to complete the rectangle or
3. Click the left mouse button at the first corner of the rectangle, and move the mouse until the rectangle is the required size and then click the left mouse button again to complete the rectangle.
4. Repeat 2,3 and 4 to create more rectangles or right click to return to the normal mouse mode.

If you press and hold down the **SHIFT** button while dragging the rectangle, then the rectangle will be centered at the position of the left mouse press.

You can press the **ESC** key or the right mouse button to cancel the rectangle creation command.

Drawing an arc



To draw one or more arcs:

Select **<Add/Arc>** from the main menu.

Move the mouse to the center of the arc and click the left mouse button.

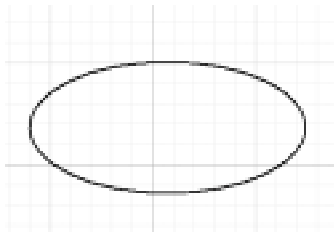
Move the mouse to the start of the arc and click the left mouse button.

Finally move the mouse to the end of the arc and click the left mouse button to complete the arc.

Repeat 2,3 and 4 to create more arcs or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the line creation command.

Drawing an ellipse



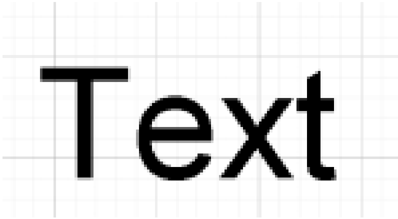
To draw one or more ellipses:

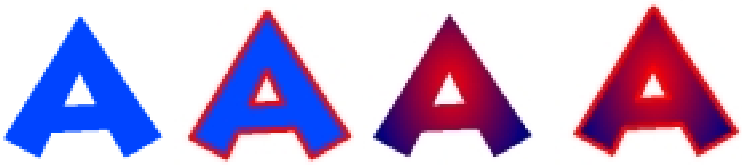
1. Select **<Add/Ellipse>** from the main menu.
2. Move the mouse to a corner of the ellipse's bounding box. Then either
3. Press down the left mouse button and drag the mouse to the other corner of the ellipse's bounding box and release the left mouse button when the ellipse is the correct size or
4. Click the left mouse button to start the ellipse, drag the mouse until the ellipse is the required size and then click the left mouse button again to complete the ellipse.
5. Repeat 2,3 and 4 to create more ellipses or right click to return to normal mouse mode.

If you press and hold down the **SHIFT** button while dragging the ellipse, then the ellipse will be centered at the position at which you pressed left mouse button.

You can press the **ESC** key or the right mouse button to cancel the ellipse creation command.

Drawing text





To draw text:

1. Select **<Add/Text>** from the main menu.
2. Move the mouse to the center of the text and click the left mouse button.

You can press the **ESC** key or the right mouse button to cancel the text creation command.

Drawing Aids

Changing grid settings

To change grid settings select <Tools/Options> from the main menu.

The Options dialog will appear. Select the **Snap and Grid Settings** tab.

See Setting Snap and Grid Settings.

Changing the drawing units (inches or millimeters?)

See the General Tab in the options dialog.

Restricting lines and wires to vertical, horizontal or diagonal directions

To restrict lines to vertical, horizontal or diagonal directions select one of the following from the main menu:

- **<Edit/Ortho/90º>**. Restricts line to 90º.
- **<Edit/Ortho/45º>.** Restricts line to 90º and 45º.
- **<Edit/Ortho/None>.** Turns off restrictions.

Showing and hiding the grid

To hide/show the Grid select **<View/Grid/Show the Grid>** from the main menu. If the grid is visible it will be hidden, conversely, if it is visible then it will be hidden.

To set the grid type and spacing, see Changing grid settings.

Snapping to a grid

To toggle snapping to the Grid select **<View/Grid/Snap Grid>** from the main menu. If the snapping to grid is enabled then it will be disabled, conversely, if it is disabled then it will be enabled.

Snapping to other objects

To toggle snapping to the other Objects select <View/Grid/Snap To Objects> from the main menu. If the snapping to objects is enabled then it will be disabled, conversely, if it is disabled then it will be enabled.

Undoing and redoing commands

Select **<Edit/Undo>** from the main menu to undo the last command.

Select **<Edit/Redo>** from the main menu to redo the last undo.

Using horizontal, vertical and point guides

You can add horizontal, vertical and point guides to help you align objects.

Guidelines can help you align several objects along an arbitrary axis. You might create guidelines to represent the printable area of a sheet, so you will not accidentally place objects in the unprintable region of the sheet.

Adding Horizontal Guides

Make sure the Rulers are displayed.

1. Place your cursor inside the horizontal (x-axis) ruler.
2. Hold down the left mouse button and drag the cursor downward.
3. Release the left mouse button when the line is properly placed.

Adding Vertical Guides

Make sure the Rulers are displayed.

1. Place your cursor inside the vertical (y-axis) ruler.
2. Hold down the left mouse button and drag the cursor downward.
3. Release the left mouse button when the line is properly placed.

Adding Point Guides

Point Guides serve as "anchors" or reference points for objects. You can place as many Point Guides as needed anywhere on the form.

Your rulers must be visible.

1. Click the Point Guide icon and hold down the left mouse button. The Point Guide icon is found at the intersection of the horizontal and vertical rulers.
2. Drag the Point Guide onto your form and release the left mouse button.

You can move and delete Point Guides like any other form object

Deleting guides

To delete a guide:

1. Select the guide.
2. Select **<Edit/Delete>** from the main menu or drag the guide onto the ruler.

Using rulers

You can show rulers around the viewport to help to align, size and position object.

You can set the ruler origins.

Showing and hiding rulers

Select <View/Rulers> from the main menu to hide/show the rulers around the viewport.

Setting the ruler origins

Press down the left mouse button over the ruler origin and drag the move in the viewport to set the new ruler origin.

Resetting the ruler origin

- Double click the left mouse button over the ruler origin to reset both the X and Y origin.
- Double click the left mouse button over the horizontal ruler origin to **reset the X origin.**
- Double click the left mouse button over the vertical ruler origin to **reset the Y origin.**

Setting the ruler units

See Changing the drawing units.

Selecting Objects

AutoTRAX has several methods of selecting objects on the sheet. You an:

Select the Pointer toolbar button . Click the left mouse button over on an object. To **select multiple objects**, hold down the **CONTROL** key while you click each object. To **deselect an object,** hold down the **SHIFT** key and click on the object that you wish to deselect.

Select all the objects.

Invert the objects selection.

Click and drag a "marquee" around all the objects.

Dragging a marquee

You can select multiple objects by:

1. Select the Pointer toolbar button .
2. holding down the left mouse button and dragging a "marquee" around the objects that you wish to select.

Selecting all the objects

Select **<Edit/Select All>** from the main menu to select all the objects on a sheet.

Inverting the selection

Select **<Edit/Invert Selection>** from the main menu to invert the selection.

Changing objects

Editing Objects

You can edit objects by selecting the objects in the viewport and either

- Manipulating it with the mouse or
- Set the numeric parameters using the objects properties dialog box.

Editing arcs

To edit an arc select the arc. You can edit an arc either by moving the start, middle or end points or the arc itself. In addition you can use the arc's properties box.

The arc's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the arc.

X and **Y** are the coordinates of the center of the arc.

X radius is the horizontal radius.

Y radius is the vertical radius.

Enter the new coordinates and radii using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the arc's color, width and style.

Editing circles and ellipses

AutoTRAX treats circles and ellipses as the same. A circle is an ellipse with the X ad Y radii being the same.

To edit an ellipse select the ellipse. You can edit an ellipse either by moving the corners, edges or the ellipse itself. In addition you can use the ellipse's properties box.

The ellipse's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the ellipse.

X and **Y** are the coordinates of the center of the ellipse.

X radius is the horizontal radius.

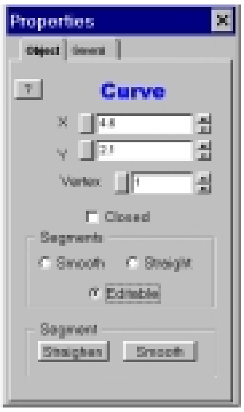
Y radius is the vertical radius.

Enter the new coordinates and radii using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the ellipse's color, width and style and the fill color and pattern.

Editing curves



To edit a curve select the curve. You can edit a curve either by moving the vertices or the curve itself. In addition you can use the curve's properties box.

The curve's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the curve.

X and **Y** are the coordinates of the selected vertex. To select a vertex, click on it. A red manipulator point will then mark it.

Vertex is the vertex number.

Check **Closed** to close the curve.

Check **Smooth** to have smooth curve or check **Straight** to have straight segments.

Check Editable to allow more control over the curvature.

Enter the new values using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the curve's color, width and style and the fill color and pattern.

Editing images (bitmaps)

To edit a bitmap select the bitmap. You can edit a bitmap either by moving the corners, edges or the bitmap itself. In addition you can use the bitmap's properties box.

The bitmap's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the bitmap.

Center X and **Y** are the coordinates of the center of the bitmap.

Width is the width of the bitmap.

Height is the height of the height of the bitmap.

Load Bitmap. Click to load a different bitmap.

Click the button to display online help.

Enter the new coordinates using the numeric entry boxes.

Editing lines

To edit a line select the line. You can edit a line either by moving the end points of the line or by moving the line itself. In addition you can use the line's properties box.

The line's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the line.

Start

X and **Y** are the coordinates of the start of the line.

End

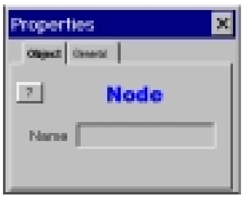
X and **Y** are the coordinates of the end of the line.

Enter the new coordinates using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the lines' color, width and style.

Editing nodes



To edit a node select the node. You can edit a node by moving the vertices. In addition you can use the node's properties box.

The node's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, double click on the node.

Name. This is the name of the node.

Click the button to display online help.

Enter the new values using the numeric entry boxes.

Editing buses

To edit a bus select the bus. You can edit a bus by moving the vertices. In addition you can use the bus's properties box.

The bus's properties are shown in the Properties dialog box below.

NOTE: If you cannot see the properties dialog box, double click on the bus.

Name. This is the name of the bus.

Click the button to display online help.

Enter the new values using the numeric entry boxes.

Editing groups

To edit a group select the group.

The group's properties are shown in the Properties dialog box below.

NOTE: If you cannot see the properties dialog box, double click on the group.

X and **Y** are the coordinates of the center of the group.

Click the button to display online help.

Enter the new coordinates using the numeric entry boxes.

Editing off-page connectors

To edit an off-page connector, first select the off-page connector. You can edit a connector either by moving the connector or you can use the connector's properties box.

NOTE: If you cannot see the properties dialog box, double click on the group.

Enter the name in the text area.

X and **Y** are the coordinates of the off-page connector.

Click the button to display online help.

Enter the new coordinates using the numeric entry boxes.

Editing instruments

To edit an instrument select the instrument. You can edit an instrument either by moving the instrument or you can use the instrument's properties box.

The instrument's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, double click on the instrument.

X and **Y** are the coordinates of the center of the instrument.

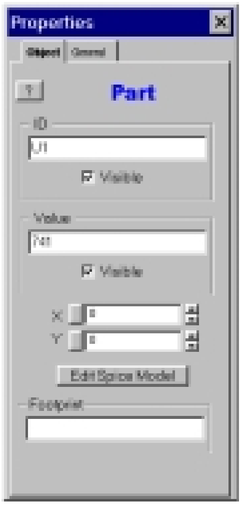
Enter the new coordinates and sizes using the numeric entry boxes.

Click the button to display online help.

Editing parts

To edit a part select the part. You can edit a part either by moving the part or you can use the part's properties box.

The part's properties are shown in the Properties dialog box.



NOTE: If you cannot see the properties dialog box; double click on the part.

ID. Enter the parts ID text. Uncheck Visible to hide the ID text.

Value. Enter the parts Value text. Uncheck Visible to hide the value text.

X and **Y** are the coordinates of the center of the part.

Width is the width of the border and

Height is the height of the border.

Edit Spice Model. Click to edit the spice model.

Footprint. Enter the parts PCB footprint.

Click the button to display online help.

Enter the new coordinates and sizes using the numeric entry boxes.

Editing a part border

To edit a parts border select the border. You can edit a border either by moving the corners, edges or the border itself. In addition you can use the border's properties box.

The border's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, double click on the border.

Center X and **Center Y** are the coordinates of the center of the border.

Width is the width of the border and

Height is the height of the border.

Uncheck the **Visible** checkbox to hide the border.

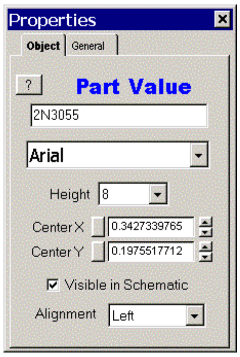
Enter the new coordinates and sizes using the numeric entry boxes.

Alignment. Set the text alignment to center, left or right.

Click the button to display online help.

The General Tab allows you to set the arc's color, width and style.

Editing a parts value



To edit a parts value select the value. You can edit a part value either by moving the corners, edges or the id itself. In addition you can use the part value's properties box.

The part value's properties are shown in the Properties dialog box below.

NOTE: If you cannot see the properties dialog box, double click on the part value.

Enter the text in the text edit box.

Use the **font combo box** to select the font.

Height is the height of the part value.

Center X and **Y** are the coordinates of the center of the part value.

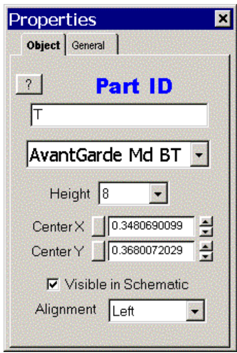
Alignment. Set the text alignment to center, left or right.

Click the button to display online help.

Enter the new coordinates using the numeric entry boxes.

The General Tab allows you to set the value's color, width and line style and the fill color and pattern.

Editing a parts id



To edit a parts id select the id. You can edit a border either by moving the corners, edges or the id itself. In addition you can use the id's properties box.

The id's properties are shown in the Properties dialog box below.

NOTE: If you cannot see the properties dialog box, double click on the id.

Enter the id in the **text edit box**. This text is used as the prefix for a parts id e.g. if you enter U then all parts will be labels U1, U2, U3....

Use the **font combo box** to select the font.

Height is the height of the id.

Center X and **Y** are the coordinates of the center of the id.

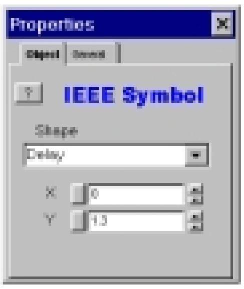
Click the button to display online help.

Enter the new coordinates using the numeric entry boxes.

The General Tab allows you to set the id's color, width and line style and the fill color and pattern.

Editing IEEE symbols

To edit an IEEE symbol select the symbol. You can edit a symbol by moving the symbol. In addition you can use the symbol's properties box.



The symbol's properties are shown in the Properties dialog box below.

NOTE: If you cannot see the properties dialog box, double click on the symbol.

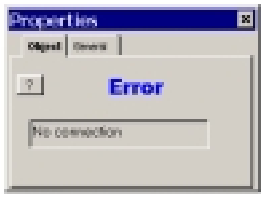
Shape. This dropdown list sets the shape type.

X and **Y** are the coordinates of the center of the symbol.

Click the button to display online help.

Enter the new values using the numeric entry boxes.

Editing error markers



Error markers are not editable. However, if you select an error marker. You will see more information about the error in the objects property toolbar.

Select **<Tools/Show Design Errors>** from the main menu to show/hide design error symbols.

Click the button to display online help.

Editing guides

You can change the position of guides and delete them.

- Point guide
- Vertical guide
- Horizontal guide

Editing horizontal guides

To edit a horizontal guide select the guide.

The horizontal's properties are shown in the Properties dialog box.

Position. This is the Y coordinate of the guide.

Click the button to display online help.

NOTE: If you cannot see the properties dialog box, double click on the horizontal.

Enter the new coordinate using the numeric entry boxes.

Editing vertical guides

To edit a vertical guide select the guide.

The vertical's properties are shown in the Properties dialog box.

Position. This is the X coordinate of the guide.

Click the button to display online help.

NOTE: If you cannot see the properties dialog box, double click on the vertical.

Enter the new coordinate using the numeric entry boxes.

Editing point guides

To edit a point guide select the guide.

The point's properties are shown in the Properties dialog box.

X. This is the X coordinate of the guide.

Y. This is the X coordinate of the guide.

Click the  button to display online help.

NOTE: If you cannot see the properties dialog box, double click on the point.

Enter the new coordinates using the numeric entry boxes.

Editing terminals

To edit a terminal select the terminal. You can edit the terminal either by moving it and/or using the terminal's properties box.

The terminal's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the terminal.

Name. Enter the pin name. Enter the pin name. (If you select Bus then this name is used as the base for all pin names).

Pin Number. Enter the physical pin number.

Bus. Check this box is it is a bus terminal.

Shape. Select the shape.

Label Visible. Uncheck to hide the label.

Pin Number Visible. Uncheck to hide the pin number.

Ground Terminal. Check to connect this terminal to the ground node. (Node 0).

Node. Enter a name for all nodes that are connected to this terminal. e.g. VCC for all nodes to be common to VCC. This does not apply to bus nodes.

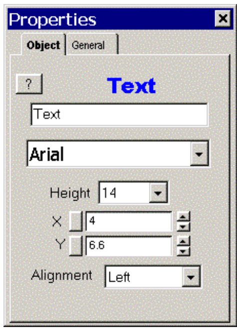
Click on the **pin list** to select it. You can then enter its new layout pin number.

Start. Enter the first bus pin number.

End. Enter the last bus pin number.

Count. Enter the number of bus pins. Click the  button to display online help.

Editing text



To edit a text object select the text. You can edit a text object either by moving the corners or the text itself. In addition you can use the text's properties box.

The text's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the text.

Enter the text in the **text edit box.**

Use the **font combo box** to select the font.

Height is the height of the text.

Center X and **Y** are the coordinates of the center of the text.

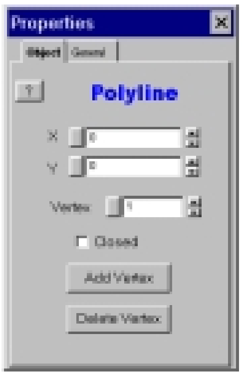
Enter the new coordinates using the numeric entry boxes.

Alignment. Set the text alignment to center, left or right.

Click the button to display online help.

The General Tab allows you to set the text object's color, width and line style and the fill color and pattern.

Editing polylines and polygons



A polygon is a closed polyline.

To edit a polyline select the polyline. You can edit a polyline either by moving the vertices or the polyline itself. In addition you can use the polyline's properties box.

The polyline's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the polyline.

X and **Y** are the coordinates of the selected vertex. To select a vertex, click on it. A red manipulator point will then mark it.

Vertex is the vertex number.

Check **Closed** to close the polyline. It will become a polygon.

Add Vertex. Click to add a vertex.

Delete Vertex. Click to delete the current vertex.

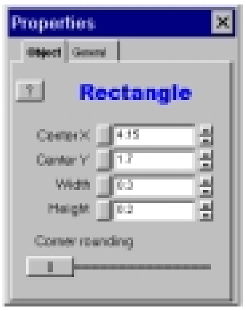
Enter the new values using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the polyline's color, width and style and the fill color and pattern.

Editing rectangles and rounded rectangles

To edit a rectangle select the rectangle. You can edit a rectangle either by moving the corners, edges or the rectangle itself. In addition you can use the rectangle's properties box.



The rectangle's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, **double click** on the rectangle.

Center X and **Y** are the coordinates of the center of the rectangle.

Width is the width of the rectangle.

Height is the height of the height of the rectangle.

Corner rounding is the amount of rounding of the corner. Try it! (See scrollbars)

Enter the new coordinates using the numeric entry boxes.

Click the button to display online help.

The General Tab allows you to set the rectangle's color, width and style and the fill color and pattern.

Editing the PCB border

The PCB border is a closed polygon defining the extent of a PCB.

To edit the border select the border. You can edit the border either by moving the vertices or the border itself. In addition you can use the border's properties box.

The border's properties are shown in the Properties dialog box.

NOTE: I*f you cannot see the properties dialog box,* ***double click*** *on the* border.

X and **Y** are the coordinates of the selected vertex. To select a vertex, click on it. A red manipulator point will then mark it.

Vertex is the currently selected vertex number.

Add Vertex. Click to add a vertex.

Delete Vertex. Click to delete the current vertex.

Click this to display this help topic.

Enter the new values using the numeric entry boxes.

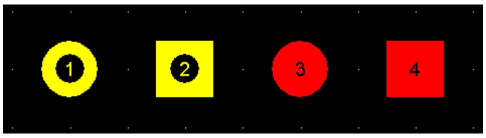
Editing nets

A net is an unrouted electrical connection. They can only be deleted or routed. To delete a net:

- Select the net.
- Select <**Edit\Delete**> from the main menu.

Editing pads

To edit a pad select the pad. You can edit a pad object either by moving the corners or the pad itself. In addition you can use the pad's properties box.



The pad's properties are shown in the Properties dialog box.

NOTE: I*f you cannot see the properties dialog box,* ***double click*** *on the pad.*

X and **Y** are the location of the center of the pad.

X size is the horizontal size of the pad.

Y size is the horizontal size of the pad.

Hole size is the diameter of the hole (through-hole pads only)

Use the **Shape** dropdown list to either a round or rectangular pad.

Use the **Layer** dropdown list to select SMT (single sided) or Thru-hole (through-hole pad with drill hole)

Pin Number. Sets the electrical pin number of the pad.

Click this to display this help topic.

Enter the pin number, new coordinates and sizes using the numeric entry boxes.

Editing traces

Traces cannot be edited. They can only be unrouted. See unrouting.

Editing coordinates

To edit a coordinate select the line. You can edit a line either by moving the coordinate. In addition you can use the coordinate's properties box.

The coordinate's properties are shown in the Properties dialog box.

NOTE: If you cannot see the properties dialog box, double click on the coordinate.

X and **Y** are the coordinates of the coordinate.

Enter the new coordinates using the numeric entry boxes.

 Click this to display this help topic.

The **General Tab** allows you to set the coordinate's color, width and style.

Editing keepouts

A keepout is a closed "no go" polygonal region within which components and/or tracks are to be excluded.

To edit a keepout select the keepout. You can edit a keepout either by moving the vertices or the keepout itself. In addition you can use the keep out’s properties box.

The keepout's properties are shown in the Properties dialog box.

NOTE: I*f you cannot see the properties dialog box,* ***double click*** *on the* keepout.

X and **Y** are the coordinates of the selected vertex. To select a vertex, click on it. It will then be marked by a red manipulator point.

Vertex is the currently selected vertex number.

Add Vertex. Click to add a vertex.

Delete Vertex. Click to delete the current vertex.

Click this to display this help topic.

Enter the new values using the numeric entry boxes.

Editing layout rooms

A room is a closed Polygon area defining a layout area for PCB footprints.

To edit a room select the room. You can edit a room either by moving the vertices or the room itself. In addition you can use the room's properties box.

The room's properties are shown in the Properties dialog box.

NOTE: I*f you cannot see the properties dialog box,* ***double click*** *on the* room.

X and **Y** are the coordinates of the selected vertex. To select a vertex, click on it. A red manipulator point will then mark it.

Vertex is the currently selected vertex number.

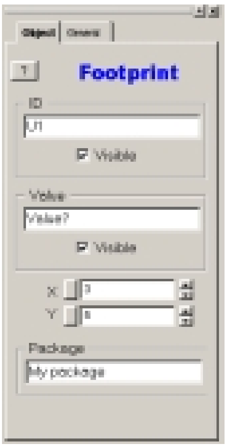
Add Vertex. Click to add a vertex.

Delete Vertex. Click to delete the current vertex.

?Click this to display this help topic.

Enter the new values using the numeric entry boxes.

Editing footprints



To edit a footprint select the footprint. You can edit a footprint using the footprint's properties box.

The footprint's properties are shown in the Properties dialog box.

NOTE: I*f you cannot see the properties dialog box,* ***double click*** *on the* footprint.

ID. Enter the footprints ID text. Uncheck Visible to hide the ID text.

Value. Enter the footprints Value text. Uncheck Visible to hide the value text.

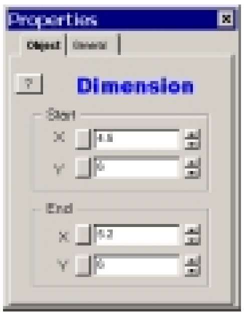
X and **Y** are the coordinates of the center of the footprint.

Package is the 3D package.

?Click this to display this help topic.

Enter the new coordinates and sizes using the numeric entry boxes.

Editing dimensions



To edit a dimension first select the dimension. You can then edit the dimension either by moving the ends of dimension In addition you can use the dimenions's properties box.

NOTE: If you cannot see the properties dialog box, double click on the line.

Start

X and Y are the coordinates of the start of the dimension.

End

X and Y are the coordinates of the end of the dimension.

Enter the new coordinates using the numeric entry boxes.

Click this to display this help topic.

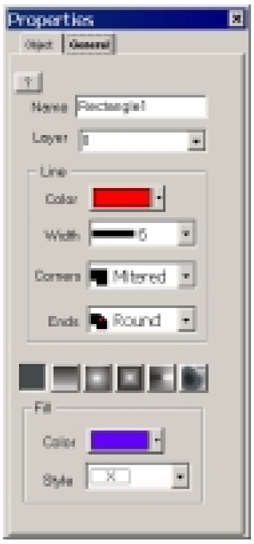
The General Tab allows you to set the dimensions' color, width and style.

Setting line colors, widths and styles

To set an objects line color, width or style first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

The General Tab



The General Tab is shown below.

It allows you to set

- The name of an object
- Set the objects layer
- Lock an object. This prevents accidental movement and resizing.
- Set the objects line color, line width and line style
- Set the objects fill color and fill style.

Name Enter the objects name.

Layer. See the layer using this dropdown combo box.

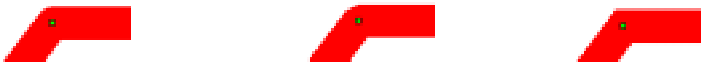
NOTE: If you cannot see the properties dialog box, **double click** on the object.

Color. Click to set the line color. This is a color button.

Width. Sets the width of the line or style.

Corners. Set the corner style to Round, Beveled or Mitered

Round Beveled Mitered

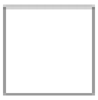
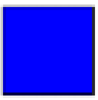
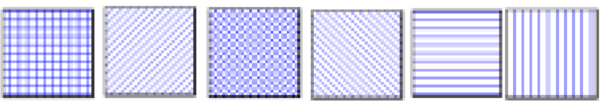
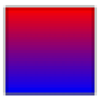


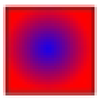
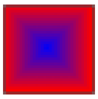
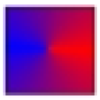
Ends. Set end style to: flat, square or round.

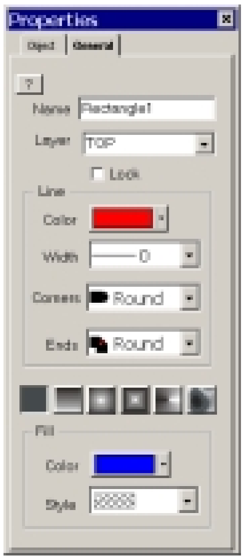
Flat Square Round

Setting fill colors and styles

To set an objects fill colors, and style, first select the object. The object's properties are shown in the Properties dialog box. Here the general tab has been selected. Fill style can be:

-  Hollow
-  Solid
-  1 of 6 different hatch patterns.
-  Linear fill from one color to another. This can be horizontal or vertical.

-  Radial fill from one color to another. The center of the fill and rate of change can be changed.
-  Square fill from one color to another. The center of the fill and rate of change can be changed.
-  Conical fill from one color to another. The center of the fill and angle can be changed.



Bitmap fill. A color bitmap/picture can be used to fill the object

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Solid Fill

To set an object's fill style to solid, first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Color. Click this color button to set the fill color.

Style. Use this combo box to set the fill pattern.

Linear fill

To set an object's fill style to solid, first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Select the Linear Fill button 

Top. Click to set the top color.

Bottom. Click to set the bottom color.

Swap. Click to swap the top and bottom colors.

Square fill

To set an object's fill style to square, first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Select the Square Fill button

Edge. Click to set the edge color.

Center. Click to set the center color.

Swap. Click to swap the edge and center colors.

To alter the center, press down the left mouse button and drag the mouse in the color area at the bottom of the dialog.

Radial fill

To set an object's fill style to radial, first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Select the Radial Fill button

Edge. Click to set the edge color.

Center. Click to set the center color.

Swap. Click to swap the edge and center colors.

To alter the center, press down the left mouse button and drag the mouse in the color area at the bottom of the dialog.

Conical fill

To set an object's fill style to conical, first select the object.

The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Select the Conical Fill button

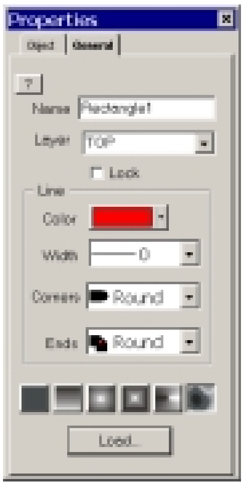
From. Click to set the 'from' color.

Top. Click to set the 'to' color.

Swap. Click to swap the 'from' and to colors.

To alter the center, press down the left mouse button and drag the mouse in the color area at the bottom of the dialog.

Bitmap fill



To set an object's fill style to bitmap, first select the object.

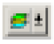
The object's properties are shown in the Properties dialog box. Here the general tab has been selected.

NOTE: If you cannot see the properties dialog box, **double click** on the object.

Select the Bitmap Fill button 

Load. Click to load the bitmap.

Setting an objects layer

Select the layer button  from the main toolbar.

Copying objects to the clipboard

To copy objects to the clipboard.

1. Select the objects.
2. Select <**Edit/Copy**> from the main menu.

Deleting all objects

To delete all objects on a sheet you either:

1. Select <**Edit/Clear**> from the main menu.

Or

2. Select all the objects on the sheet.
3. Select <**Edit/Delete**> from the main menu.

Or:

1. Select all the objects on the sheet.
2. Select <**Edit/Cut**> from the main menu. This will place the deleted objects on the clipboard.

Deleting objects

To delete objects on a sheet:

1. Select the objects to delete.
2. Select **<Edit/Delete>** from the main menu.

Or:

1. Select the objects to delete.
2. Select **<Edit/Cut>** from the main menu. This will place the deleted objects on the clipboard.

Mirroring objects

You can mirror objects about either their:

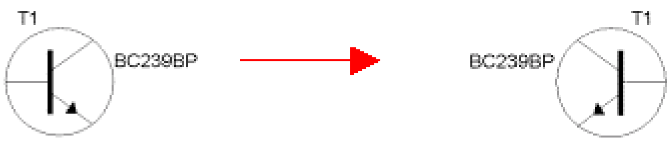
- Vertical axis or
- Horizontal axis.

Mirroring about vertical axis

To mirror objects about their vertical axis

1. Select the objects to mirror.

Select <Edit/Mirror/Vertical> from the main menu.

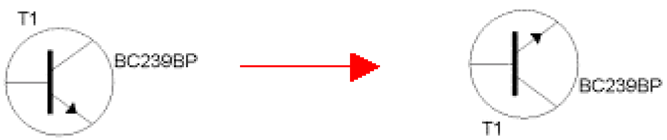


NOTE: When parts are mirrored, the text is still drawn from left to right.

Mirroring about horizontal axis

To mirror objects about their horizontal axis

1. Select the objects to mirror.
2. Select **<Edit/Mirror/Horizontal** from the main menu.



NOTE: When parts are mirrored, the text is still drawn from left to right.

Moving objects

To move an object, move the mouse over an object to move, press down left mouse button and drag the object to its new location.

Pasting objects from the clipboard

To paste objects from the clipboard.

1. Select **<Edit/Paste>** from the main menu.

Resizing objects

To resize objects:

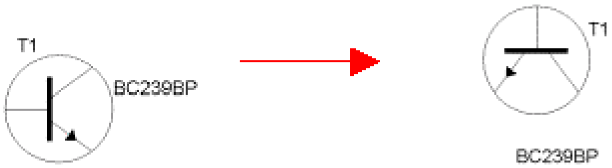
2. Select the object.
3. Move its object manipulator points.

For more details on each object see Editing Objects.

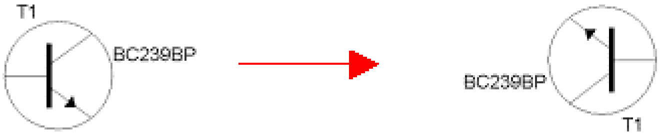
Rotating objects

You can rotate objects:

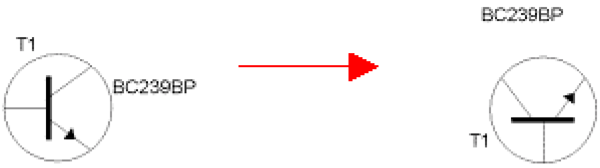
90º clockwise. Select **<Edit/Rotate/90ºclockwise>** from the main menu.



180º. Select **<Edit/Rotate/180º>** from the main menu.



90º anticlockwise. Select **<Edit/Rotate/90º anticlockwise>** from the main menu.



NOTE: When parts are rotated, the text is still drawn from left to right.

Ungrouping objects that are in a group

To ungroup a group:

1. Select the group.
2. Select **<Edit/Ungroup>** from the main menu.

Using the object properties dialog box

All objects have an associated Properties dialog box.

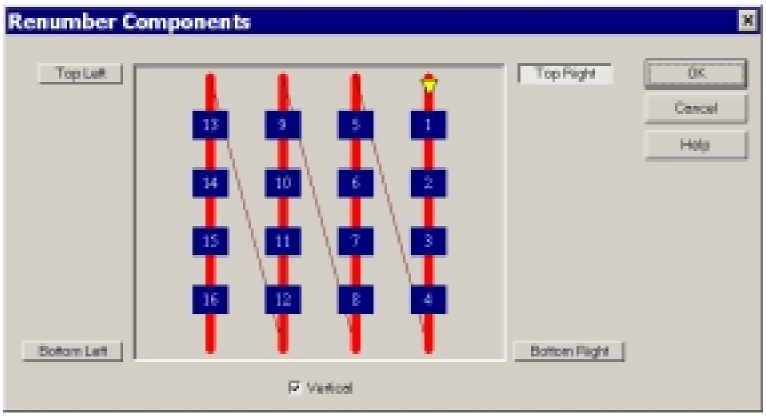
NOTE: If you cannot see the properties dialog box, **double click** on the object.

Renumbering all components

To renumber all components based on their position:

1. Select **<Tools/Renumber components...>** from the main menu.

The dialog box shown below will appear.

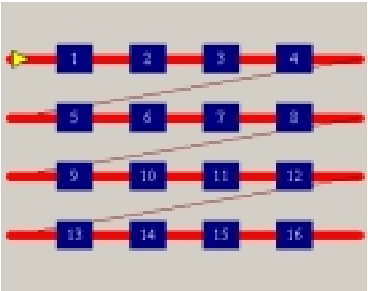
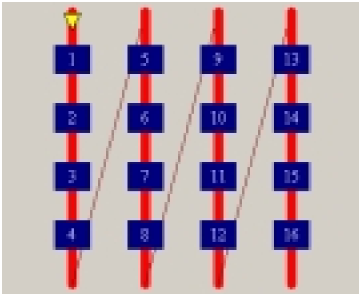


Vertical

Click to order the components vertically. Unclick to order them horizontally.

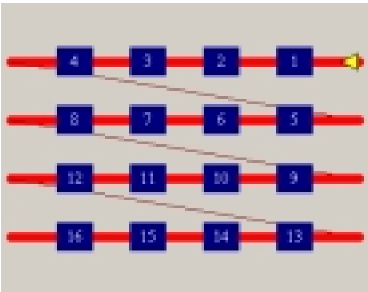
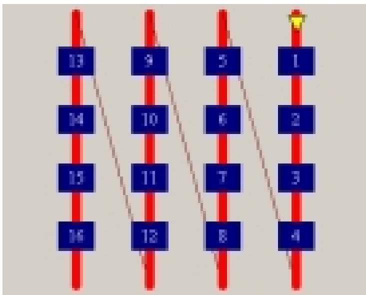
Top Left.

Starts the order from the top left as shown below. The first is ordered horizontally while the second is ordered vertically (Check the vertical tick box).

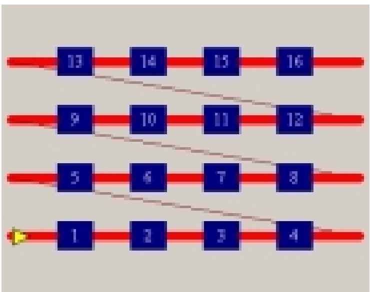
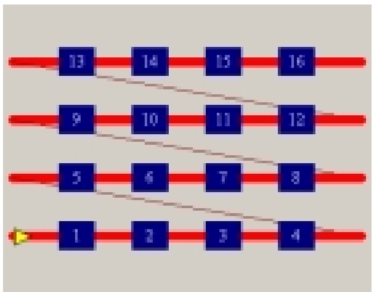
Top Right

Starts the order from the top right as shown below. The first is ordered horizontally while the second is ordered vertically (Check the vertical tick box).

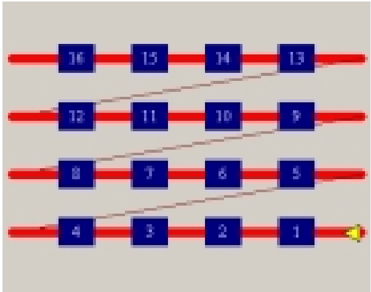
Bottom Left

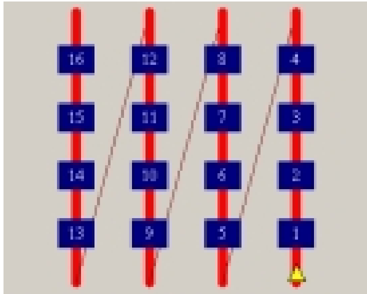
Starts the order from the bottom left as shown below. The first is ordered horizontally while the second is ordered vertically (Check the vertical tick box).

Bottom Right

Starts the order from the bottom right as shown below. The first is ordered horizontally while the second is ordered vertically (Check the vertical tick box).





Arranging Objects

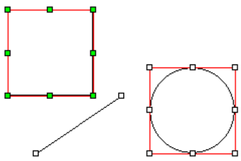
You can align objects to the dominant object by their:

- Left edges
- Right edges
- Top edges
- Bottom edges
- You can also align objects
- Horizontally by their centers
- Vertically by their centers

The dominant Object

The dominant object is the selected object that is used as the reference object to arrange other selected objects against.

When several items are selected, the object with the green selection handles is the dominant object. In the following picture, the square is the dominant object.



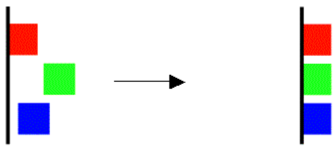
If you hold the **CONTROL** or **SHIFT** key to select multiple objects, the dominant object will be the first object you click. If you drag a marquee around multiple objects, the dominant object will be the first object you enveloped with the marquee.

Changing the Dominant Object

This procedure will work only if more than one objects is selected.

Press the **CONTROL** key and click on an object. The object you click will become the dominant object.

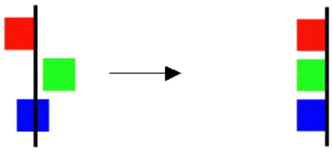
Aligning objects by their left edge



1. Select the objects you wish to align.
2. Select <Layout/Align/Align At Left> from the main menu.

This will align the left edge of all selected objects with the left edge of the dominant object (red square).

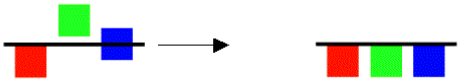
Aligning objects by their right edge



1. Select the objects you wish to align.
2. Select **<Layout/Align/Align At Right>** from the main menu.

This will align the right edge of all selected objects with the right edge of the dominant object (red square).

Aligning objects by their top edge



1. Select the objects you wish to align.
2. Select **<Layout/Align/Align At Top>** from the main menu.

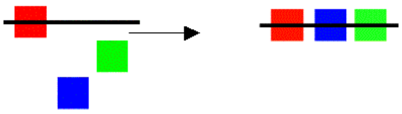
This will align the top edge of all selected objects with top selected edge of the dominant object (red square).

Aligning objects by their bottom edge

1. Select the objects you wish to align.
2. Select <Layout/Align/Align At Bottom> from the main menu.

This will align the bottom edge of all selected objects with the bottom edge of the dominant object (red square).

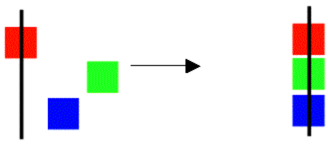
Aligning objects horizontally by their centers



1. Select the objects you wish to align.
2. Select <Layout/Align/Align At Vertical Middle> from the main menu.

This will align the vertical middle of all selected objects with the vertical middle of the dominant object (red square).

Aligning objects vertically by their centers

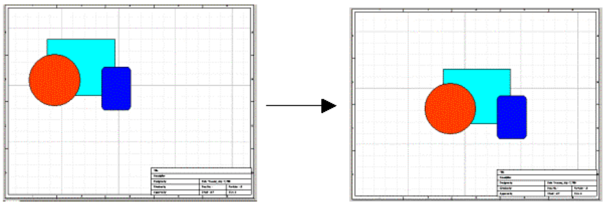


1. Select the objects you wish to align.

2. Select <Layout/Align/Align At Horizontal Middle> from the main menu.

This will align the horizontal middle of all selected objects with the horizontal middle of the dominant object (red square).

Centering everything on the sheet



To center all the objects at the center of the sheet

1. Select **<Layout/Center Drawing>** from the main menu.

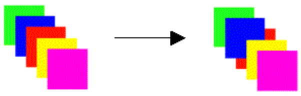
Changing overlapping objects

All graphical objects are drawn in order. As a consequence if an object is drawn after another, then it is possible for it to either partially or completely over the one drawn earlier. However, if is possible to alter the order in which objects are drawn.

You can:

- Move the selected objects back one level.
- Move the selected objects forward one level.
- Move the selected objects to the back.
- Move the selected objects to the front.

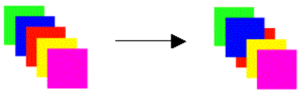
Sending objects backward one level



2. Select the objects you wish to send backwards one level.
3. Select <Layout/Order/Backward One Level> from the main menu.

In the example above, the red object is moved backwards one level.

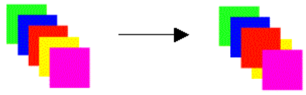
Sending objects to the back



1. Select the objects you wish to send to the back.
2. Select **<Layout/Order/To Back>** from the main menu.

In the example above, the red object is moved to the back.

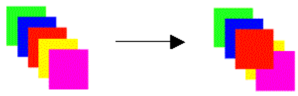
Bringing objects forward one level



1. Select the objects you wish to bring forward one level.
2. Select <Layout/Order/Forward One Level> from the main menu.

In the example above, the red object is moved forward one level.

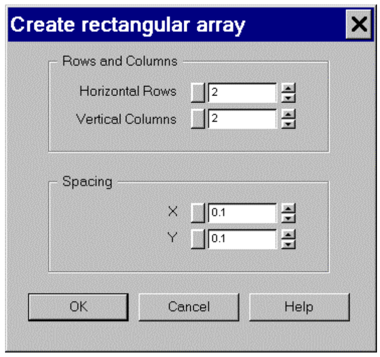
Bringing objects to the front



1. Select the objects you wish to bring to the front.
2. Select **<Layout/Order/To Front>** from the main menu.

In the example above, the red object is moved to the front.

Creating a rectangular array of objects



To create a rectangular array of objects:

1. Select the objects to array.
2. Select **<Tools/Array Rectangular>** from the main menu. This will display the Array Rectangular dialog shown below.
3. Fill in the entries and click the **OK** button.

Horizontal Rows. Enter the number of rows. There must be at least 1 row.

Vertical Columns. Enter the number of columns. There must be at least 1 column.

Spacing. Enter the spacing between rows an column. Here the spacing between columns will be 3 and the spacing between rows will be 2.

OK. Click this to create the array of selected objects.

Cancel. Cancel the tool. Nothing will be arrayed.

Help. Get help on the rectangular array tool.

The above entries will form an array consisting of 3 columns and 2 rows with spacing between columns of 3 and spacing between rows of 2.

Below an object selection consisting of a rectangle and a circle has been arrayed using the above parameters.

Distributing objects

You can distribute objects evenly:

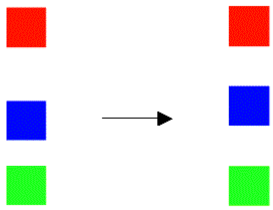
- In the horizontal direction or
- In the vertical direction.

Distributing objects evenly across

To distribute the selected objects evenly in the horizontal direction.

1. Select the objects you wish to distribute.
2. Select <Layout/Distribute/Evenly Across> from the main menu.

Distributing objects evenly down



To distribute the selected objects evenly in the vertical direction:

1. Select the objects you wish to distribute.
2. Select <Layout/Distribute/Evenly Down> from the main menu.

Grouping objects

Grouping objects allows you to manipulate several distinct objects as one. Grouping saves you the effort of selecting multiple objects over and over again.

To group two or more objects:

1. Select the objects you wish to group.
2. Select **<Edit/Group>** from the main menu.

The selected objects will now appear and behave as one object. If you resize the group, all the objects within that group will be resized. If you delete the group, all the objects within it will be deleted.

Ungrouping objects

To ungroup one or more objects:

1. Select the objects you wish to align.
2. Select **<Edit/Ungroup>** from the main menu.

Adding Parts

Using the parts library toolbar

Drag a part from the Parts Library toolbar onto the sheet.

Using the parts toolbar

Click on a part in the parts toolbar and move the mouse over the viewport. Click to place the part.

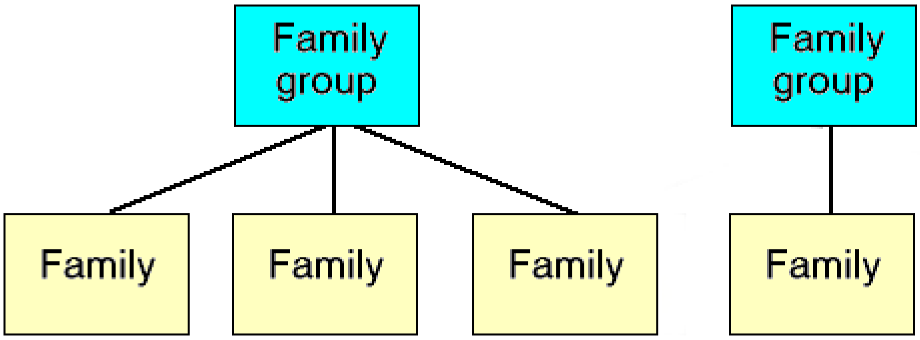
Creating New parts

Overview

AutoTRAX hold all parts in a relational database called AutoTRAX.mdb.

You add parts to a design using either the parts toolbar or the parts library toolbar.

AutoTRAX comes with thousands of parts. These are grouped into Family Groups and Families, see below. Here we have one family group which has 3 families and another family group which has only 1 family.



A Family Group is a collection of related part families such as a group named transistors with a JFET transistors family and another family named NPN transistors.

A family is a collection of related parts, such as NPN transistors.

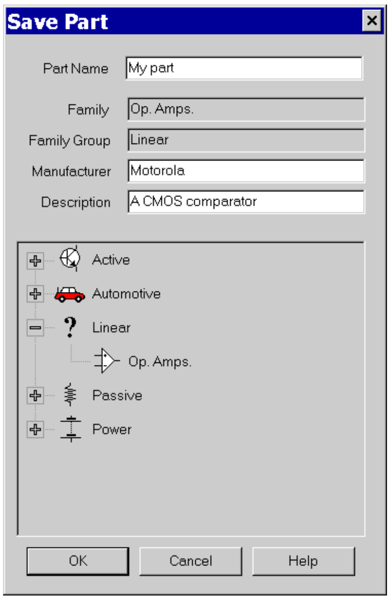
All parts consist of a physical image and associated electrical terminals. In addition, a part can have a SPICE model.

Creating a part

To create a new part, select **<File/New>** from the main menu.

The New dialog box shown below will appear. Select Part to create a new part.

The Save Part Dialog shown below will appear.



Select the family that you want the part to be in and enter the Part Name. This will save a basic part in the database.

Manufacturer. Enter the manufacturer's name

Description. Enter a description for the part.

Click **OK** to create the new part.

Next the Icon Editor will appear. You need to associate an icon with the part.

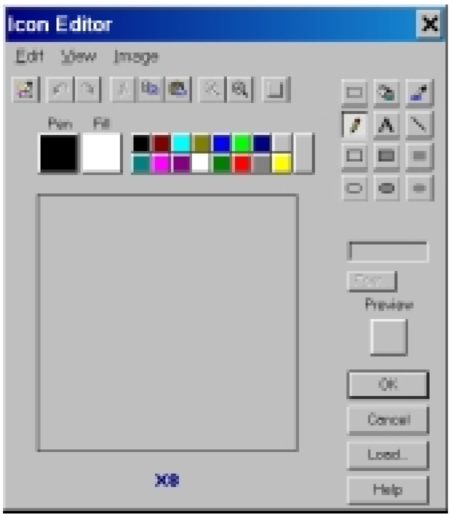
Finally you will be asked to associated a PCB footprint with the part. You do not need to assign one or you can assign one later.

You will now see a basic part with:

1. A part border.

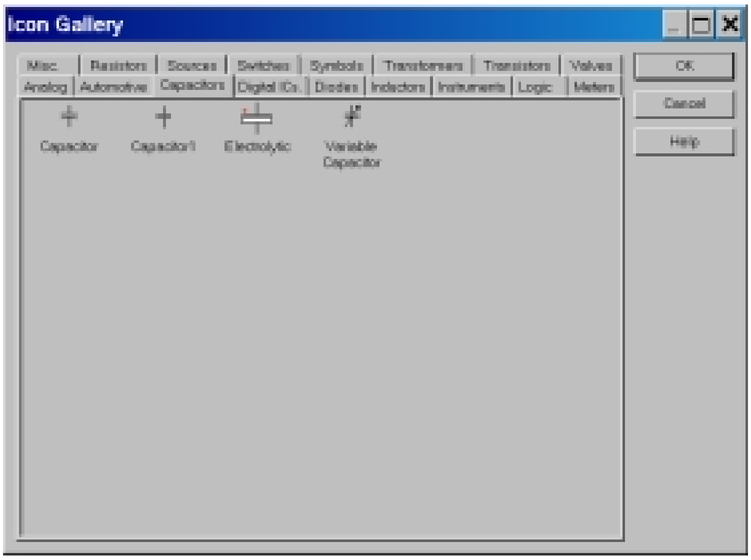
2. A part reference.
3. A part value.

The Icon Editor



The Icon Gallery

The Icon Gallery is shown below.



Select the tab for the group of icons that you are interested in viewing.

Click the right mouse button over the icon display area and select the appropriate menu item to:

Select one or more icons.

Create a new icon.

Edit a selected icon.

Rename a selected icon.

Cut selected icons.

Delete selected icons.

Paste icons from the clipboard.

Add a new group of icons (add a tab)

Delete the current group of icons (delete tab).

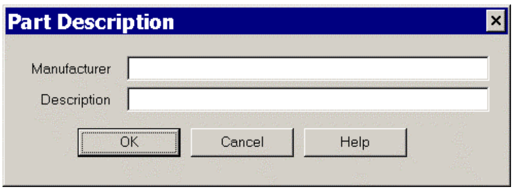
Rename the current icon group (rename tab)

Setting the PCB footprint

To set the PCB footprint for a part, select **<Edit/Footprint...>** from the main menu. and enter the name of the footprint.

The footprint name is used when exporting the netlist to PCB layout programs.

Editing a parts description



You can set the part manufacturers name and provide a description of the part. This information is used when generating part lists and in the parts library toolbar.

Select **<Edit/Description>** from the main menu to change the description of a part. The Part Description dialog box shown below will appear.

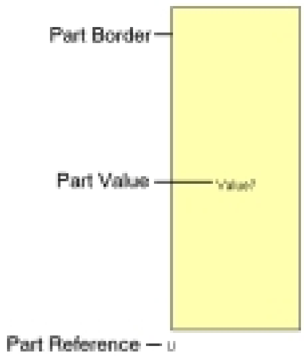
Manufacturer. Enter the name of the part manufacturer.

Description. Enter a description of the part.

Then click **OK** to accept your changes or click **Cancel** to ignore them.

Help. Displays this help topic.

The Part Border



This part border is a rectangular area that terminals are connected to. It is used for generic ICs. If you are creating say a transistor, then you can delete the part border by selecting it and then deleting it.

Moving the Part Border

You can move the part border by pressing down the left mouse button over the border's center and dragging it.

Resizing the Part Border

You can resize the part border by pressing down the left mouse button over the border's edge and dragging it.

The part Reference ID

The part reference is the reference for the part e.g. U1, T1 etc. This can be hidden.

The Part Value

The part value is the type of the device, e.g. 2N3055. This can be hidden.

Adding terminals

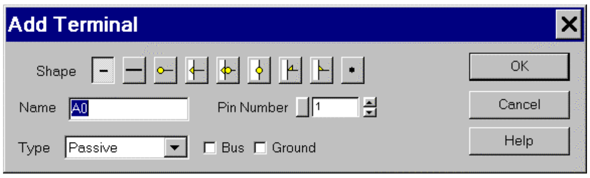
Select <Add/Terminal> from the main menu. Select from the list of type.

Enter the name of the terminal in the popup dialog. (Shown below)

Enter the pin number.

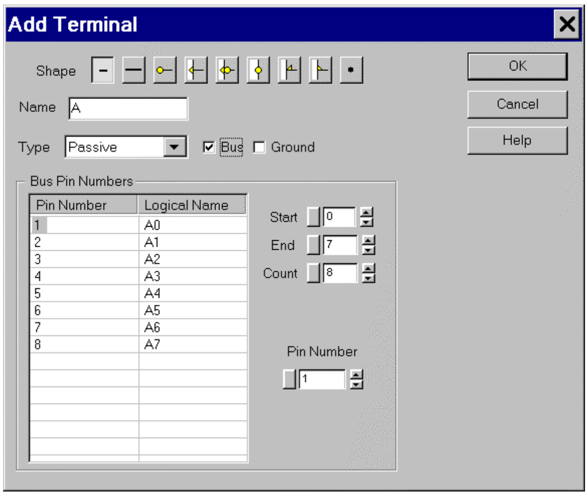
Click Ground if it is a ground terminal. This will automatically be connected to the ground node (Node 0).

Move the move and click the left mouse button when the terminal is at the correct location.



If you enter a name ending with a number such as A5, you can place successive terminals, with each one having the trailing number incremented. e.g. A5,A6,A7...

If you wish to enter a bus terminal check the Bus box. The dialog box will expand to one similar to the one below. Enter the bus pin numbers and logical names.



You can press the **ESC** key or the right mouse button to cancel the Add Terminal command.

Adding IEEE symbols

1. Select **<Add/IEEE Symbol>** from the main menu and then select the symbol to add.
2. Move the mouse in the viewport; the symbol will follow the mouse. You can press the **ESC** key to cancel adding the symbol.
3. Click the left mouse button to place the symbol.

Creating a Bipolar Junction Transistor

To create a bipolar transistor:

1. Create a basic part.
2. Delete the part border.
3. Draw the JFET transistor using non-electrical shapes.
4. Add the E, C and B terminals.
5. Select **<Edit/Spice Model...>** from the main menu.

6. Select the Bipolar transistor model from the modeler and complete the dialog.

Creating a capacitor

To create a capacitor:

1. Create a basic part.
2. Delete the part border.
3. Draw the capacitor using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the capacitor model from the modeler and complete the dialog.

Creating a diode

To create a diode:

1. Create a basic part.
2. Delete the part border.
3. Draw the diode using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model…>** from the main menu.
6. Select the diode model from the modeler and complete the dialog.

Creating a JFET transistor

To create a JFET transistor:

1. Create a basic part.
2. Delete the part border.
3. Draw the JFET transistor using non-electrical shapes.
4. Add the A, K and G terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the JFET transistor model from the modeler and complete the dialog.

Creating a lossless transmission line

To create a lossless transmission line:

1. Create a basic part.
2. Delete the part border.
3. Draw the lossless transmission line using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the lossless transmission line model from the modeler and complete the dialog.

Creating a lossy transmission line

To create a lossy transmission line:

1. Create a basic part.
2. Delete the part border.
3. Draw the lossy transmission line using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the lossy transmission line model from the modeler and complete the dialog.

Creating a MESFET transistor

To create a MESFET transistor:

Create a basic part.

1. Delete the part border.
2. Draw the MESFET transistor using non-electrical shapes.
3. Add the A, K and G terminals.
4. Select **<Edit/Spice Model...>** from the main menu.
5. Select the MESFET transistor model from the modeler and complete the dialog.

Creating a MOSFET transistor

To create a MOSFET transistor:

1. Create a basic part.
2. Delete the part border.
3. Draw the MOSFET transistor using non-electrical shapes.
4. Add the A, K and G terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the MOSFET transistor model from the modeler and complete the dialog.

Creating a resistor

To create a resistor:

1. Create a basic part.
2. Delete the part border.
3. Draw the resistor using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the resistor model from the modeler and complete the dialog.

Creating a switch

To create a switch:

1. Create a basic part.

2. Delete the part border.
3. Draw the switch using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the switch model from the modeler and complete the dialog.

Creating a transformer

To create a transformer:

1. Create a basic part.
2. Delete the part border.
3. Draw the transformer using non-electrical shapes.
4. Add the terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the transformer from the modeler and complete the dialog.

Creating an inductor

To create an inductor:

1. Create a basic part.
2. Delete the part border.
3. Draw the inductor using non-electrical shapes.
4. Add the N+ and N- terminals.
5. Select **<Edit/Spice Model...>** from the main menu.
6. Select the inductor model from the modeler and complete the dialog.

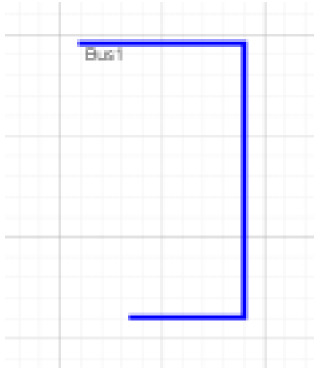
Deleting terminals

To delete a terminal:

1. Select the terminal.
2. Select **<Edit/Delete>** from the main menu.

Connecting Components

Adding a bus

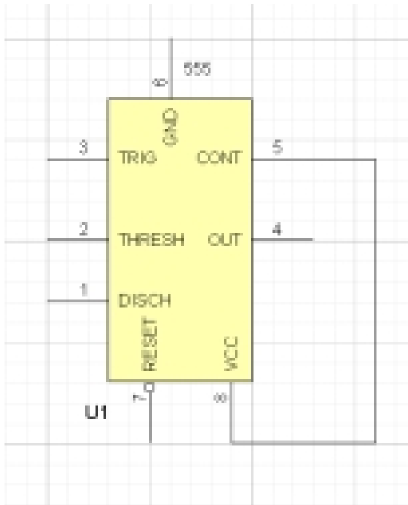


To draw one or more buses:

1. Select **<Add/Bus>** from the main menu.
2. Move the mouse to the start of the bus.
3. Click the left mouse button to start the bus, and then drag the mouse until the bus segment is the required length and direction. Click the left mouse button again to start another segment. Again drag the mouse until the new bus segment is the required length and direction. Keep on repeating this until the bus is complete. Finally click the right mouse button to end.
4. Repeat 2 and 3 to create more buses or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the bus creation command.

Adding a wire

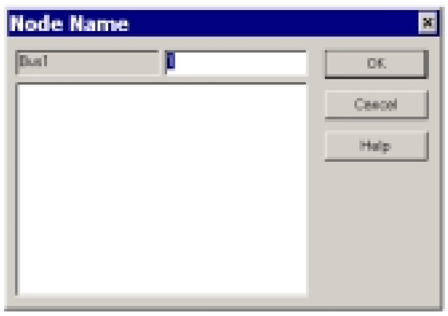


To draw one or more wires:

1. Select **<Add/Wire>** from the main menu.
2. Move the mouse over a part terminal or bus and click to start the wire.
3. Drag the mouse until the wire segment is the required length and direction. Click the left mouse button again to start another segment. Again drag the mouse until the new wire segment is the required length and direction.
4. Move the wire segment until it is over another part terminal or a bus. Left click to end the wire.
5. Repeat 2 and 4 to create more wires or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the bus creation command.

Connecting a wire to a bus



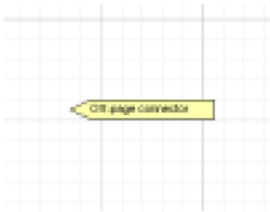
A bus is a series of connected line segments and represents 0 or more nodes.

If you connect a wire to a bus, you will be prompted for the bus's virtual node to connect to.

Adding off-page connectors

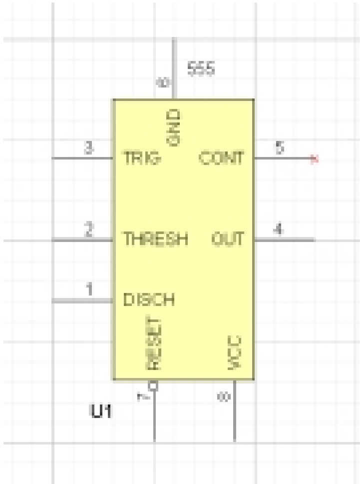
To draw one or more wires:

1. Select **<Add/Off Page Connector>** from the main menu.
2. Move the move in the viewport to position the Off Page Connector and click the left mouse button.



You can press the **ESC** key or the right mouse button to cancel the off page connector command.

Marking a terminal as no connection.



To mark a terminal as no connection required:

Select **<Add/No Connect>** from the main menu.

Move the move in the viewport to position the Cross and click the left mouse button.

You can press the **ESC** key or the right mouse button to cancel the command.

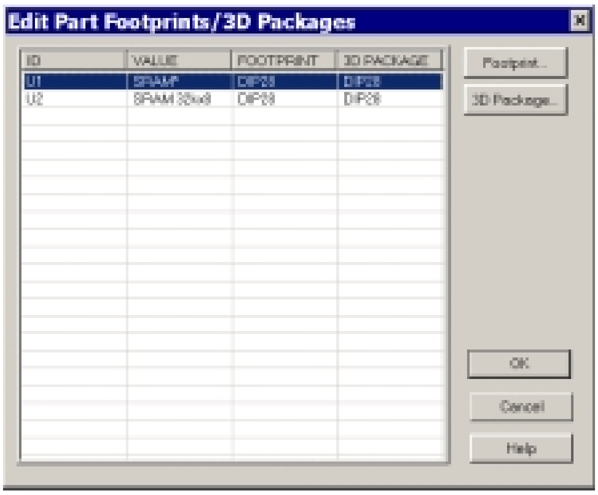
What is a node?

A node is an electrical connection between 2 or more part terminals.

PCB Design

Setting the part footprints

To view and set the part footprints on a schematic select **<Edit/Edit Footprints...>** from the main menu.



Double click the left mouse button on the footprint name to change it.

Click **OK** to accept your changes or click the **Cancel** button to ignore them.

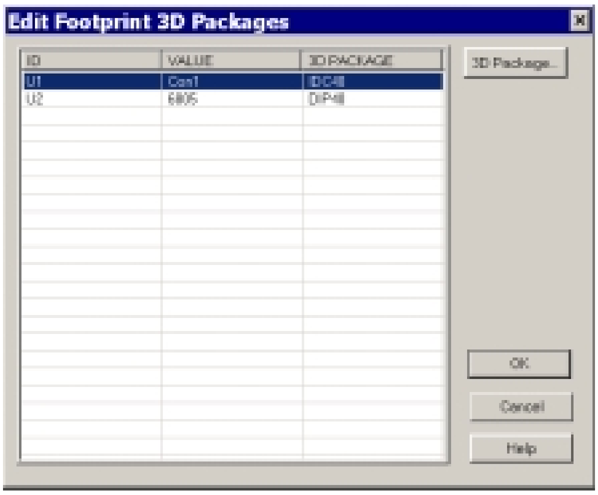
Click the **Help** button to get this help topic.

Setting the packages in the PCB Designer

Before you can visualize a PCB in 3D, each footprint needs to have a 3D package assigned to it. This is normally done when the part is created and the information is saved in the part database. However, you can assign packages in the PCB Designer.

To change the packages select **<Edit\Edit 3D Packages...>** from the main menu.

The dialog box shown below will appear.



Double click the left mouse button on the footprint to change it.

Click **OK** to accept your changes or click the **Cancel** button to ignore them.

Click the **Help** button to get this help topic.

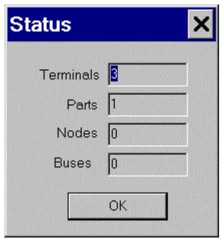
Design Checking

Displaying part design status

Select <**Tools/Status**> from the main menu to display the Design Status dialog.

Terminals. The number of terminals in the part.

Displaying schematic design status



Select <**Tools/Status**> from the main menu to display the Design Status dialog.

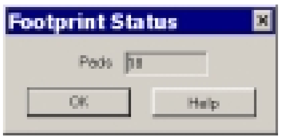
Terminals. The number of terminals on the sheet.

Parts. The number of parts on the sheet.

Nodes. The number of electrical nodes on the sheet.

Buses. The number of electrical buses on the sheet.

Displaying footprint design status



Select <**Tools/Status**> from the main menu to display the Design Status dialog.

Pads. The number of pads in the footprint.

Displaying PCB design status

Select <**Tools/Status**> from the main menu to display the Design Status dialog.

Pads. The number of pads on the pcb.

Parts. The number of parts on the pcb.

Nets. The number of electrical nets on the pcb.

Vias. The number of electrical vias on the pcb.

Showing and hiding design error markers

Select <Tools/Show Design Errors> from the main menu show/hide design error symbols.

Clearing design errors

Select <Tools/Clear Design Errors> from the main menu to remove all design error symbols.

Electrical Test Instruments

Adding electrical instruments

AutoTRAX has several electrical instruments that you can add to a sheet and help you test the design.

You can add:

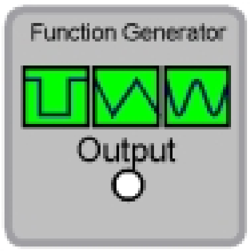
Multimeters.

Function Generators.

Oscilloscopes.

Function Generators

Adding a function generator



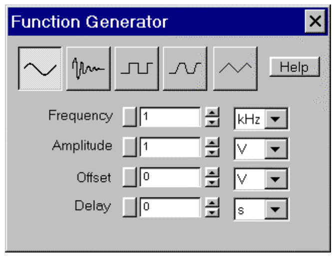
To add a function generator select <Simulate/Instruments/Function Generator> from the main menu.

Drag the function generator to where you want to place it, and click the left mouse button.

The function generator is a single output signal generator. You can connect a wire from the output to any electrical node in the design.

Double click on the function generator to view the output.

Viewing a function generator



Double click the left mouse button on the function generator. The Function Generator dialog shown below will appear.

You have a choice of:

- Sin wave
- Damped sin wave
- Perfect square wave
- Square wave with rise and fall time
- Saw tooth

You can set the:

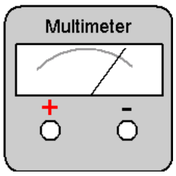
- Frequency
- Amplitude
- Offset
- Delay
- Duty cycle (square wave only)
- Rise time/fall time (square wave only)

Multimeters

Adding a multimeter

To add an oscilloscope select <Simulate/Instruments/Multimeter> from the main menu.

Drag the multimeter to where you want to place it, and click the left mouse button.



The multimeter is a digital voltmeter. You can connect a wire from the + or - input to any electrical node in the design.

Double click on the multimeter to view the output.

Viewing a multimeter

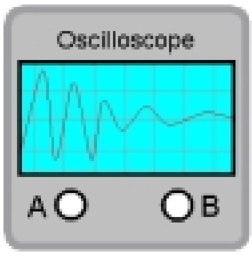
Double click the left mouse button over a multimeter to view the multimeter output. The multimeter output dialog is shown below.

Oscilloscopes

Adding an oscilloscope

To add an oscilloscope select <Simulate/Instruments/Oscilloscope> from the main menu.

Drag the oscilloscope to where you want to place it, and click the left mouse button.

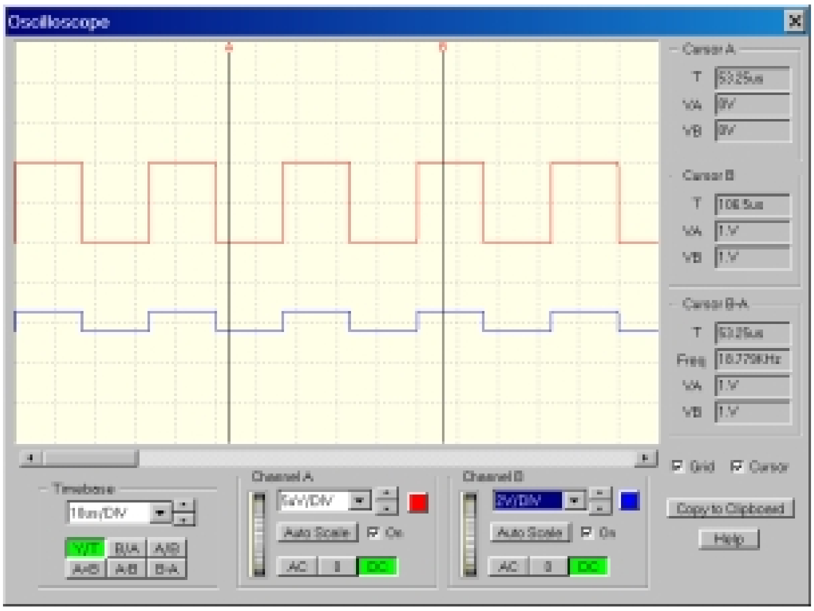


The oscilloscope is a 2-channel scope with channels A and B. You can connect wires from either of the terminals to any electrical node in the design.

Double click on the oscilloscope to view the output.

Viewing an oscilloscope

Double click the left mouse button over an oscilloscope to view the oscilloscope output.



The oscilloscope has 2 channels A and B.

Time base. Sets the timebase scale.

Y/T. Display channel voltage vs. time.

B/A. Display Channel B vs. channel A.

A/B. Display Channel A vs. channel B.

A+B. Display the sum of channel A and B voltages vs. time.

A-B. Display the channel A voltage minus channel B voltage vs. time.

B-A. Display the channel B voltage minus channel A voltage vs. time.

Channel A and Channel B. Set channel scale.

Autoscale. Autoscale the voltage trace to fit in the display.

AC. Display signal with DC component removed.

0. Display a ground signal.

DC. Display the signal.

On. Check to show the trace. Uncheck to hide the trace.

Copy to Clipboard. Copy the image of the display to the clipboard.

Help. Display this help topic.

Cursor A / Cursor B / Cursor A-B displays information about the 2 vertical cursors.

T. The cursor time.

VA. The voltage of trace A.

VB. The voltage of trace B.

Freq. The effective frequency between cursor A and B = 1/(time difference).

Grid. Check to show the grid. Uncheck to hide the grid.

Cursor. Check to show the cursors. Uncheck to hide the cursors.

Spice Circuit Simulation

What is SPICE

SPICE* is an engineering design tool for analog circuit simulation. Computer simulation is an important supplement to traditional design techniques. In most design methodologies, simulation can be an aid to the initial design development, during the breadboarding phase, and during debugging and diagnostics.

SPICE was developed at the University of California, Berkeley and has become the defacto analog circuit simulator.

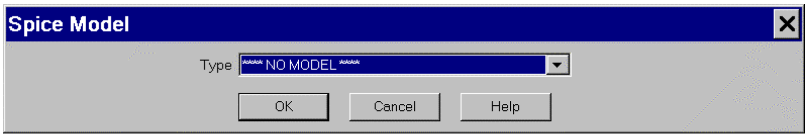
AutoTRAX incorporates version SPICE 3f5, the latest version available.

For books discussing SPICE see the books topic in the Resources Book.

* SPICE stands for **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

The Spice modeler

The Spice modeler dialog box is shown below.



The following device types are available.

- NO MODEL
- Bipolar Junction Transistor (BJT)
- Capacitor
- Diode
- Independent Current Source - DC
- Independent Current Source - Sinusoidal
- Independent Current Source - Pulse
- Independent Current Source - Single Frequency FM
- Independent Current Source - Piece-Wise Linear
- Independent Current Source - Exponential
- Independent Voltage Source - DC
- Independent Voltage Source - Sinusoidal
- Independent Voltage Source - Pulse
- Independent Voltage Source - Single Frequency FM
- Independent Voltage Source - Piece-Wise Linear
- Independent voltage source - Exponential
- Inductor
- Junction Field Effect Transistor (JFET)
- Linear Current Controlled Current Source (CCCS)
- Linear Current Controlled Voltage Source (CCVS)
- Linear Voltage Controlled Current Source (VCCS)
- Linear Voltage Controlled Voltage Source (VCVS)
- MESFET
- MOSFET
- Resistor
- Switch
- Transformer
- Transmission Line - lossless
- Transmission Line - lossy (LTRA)
- Transmission Line - lossy (URC)
- USER MODEL
- Database TABLE

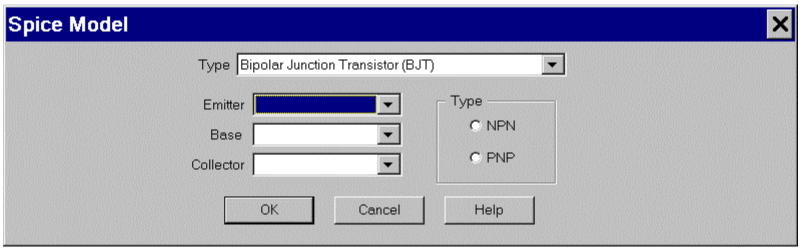
Modeling a Bipolar Junction Transistor

The Bipolar Junction Transistor model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

Emitter is the terminal to use as the emitter.

Base is the terminal to use as the base.

Collector is the terminal to use as the collector

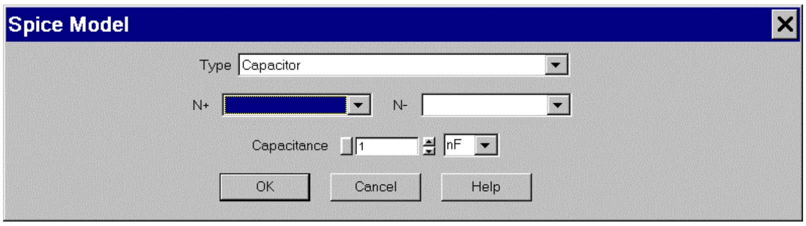


Modeling a capacitor

The capacitor model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

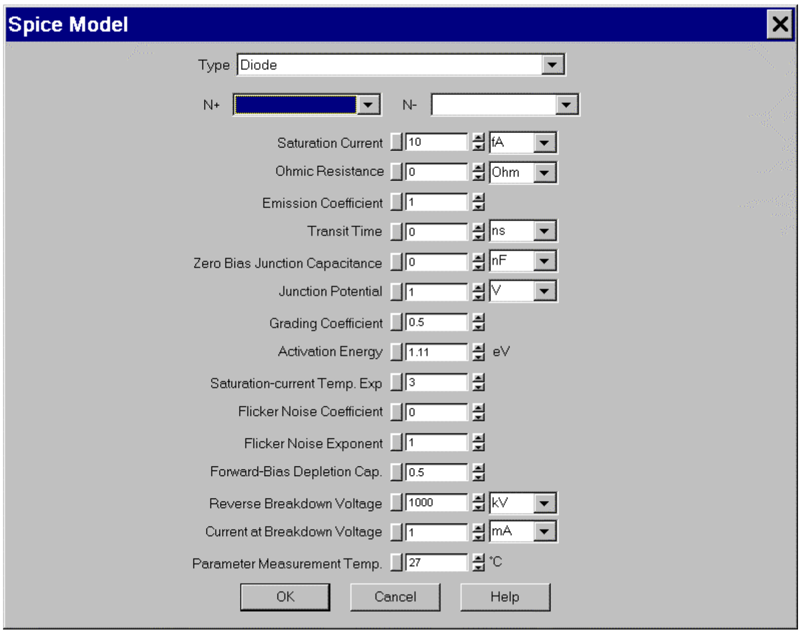


Modeling a diode

The diode model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the terminal to use as the anode.

N+ is the terminal to use as the cathode.

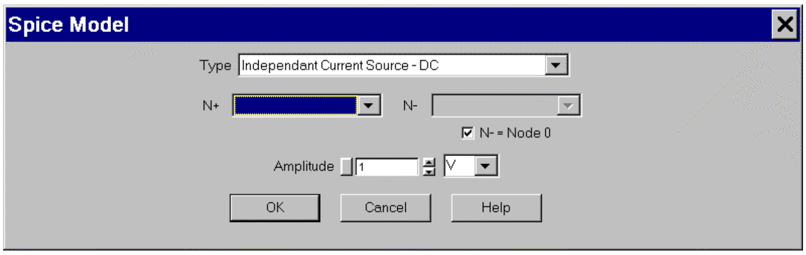


Independent current source - DC

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

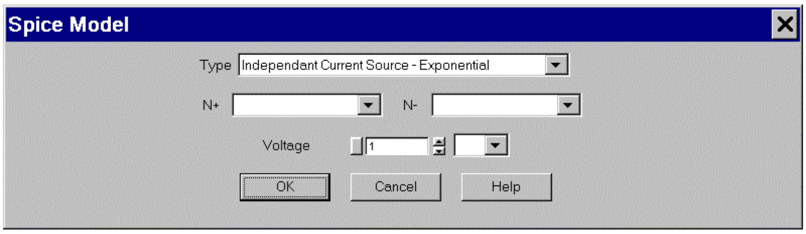


Independent current source - Exponential

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.



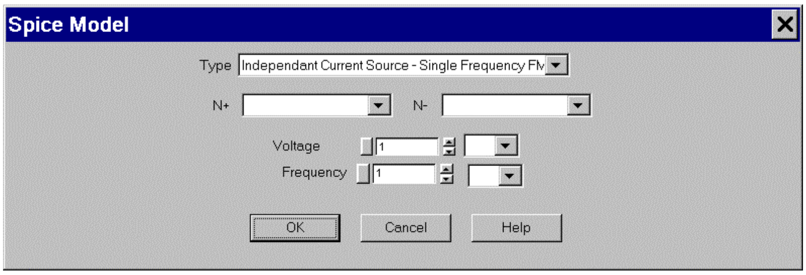
Independent current source - FM

The model dialog is shown below. For details of the model parameters see the Spice

Reference Manual.

N+ is the positive terminal.

N+ is the negative.

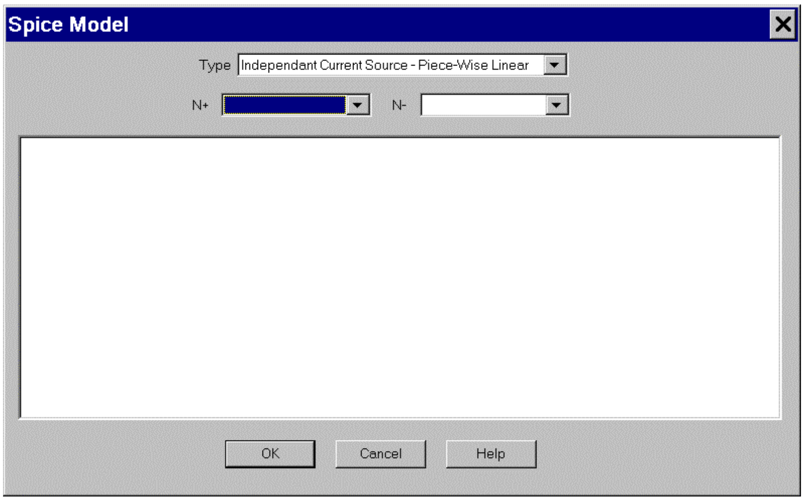


Independent current source - piecewise linear

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

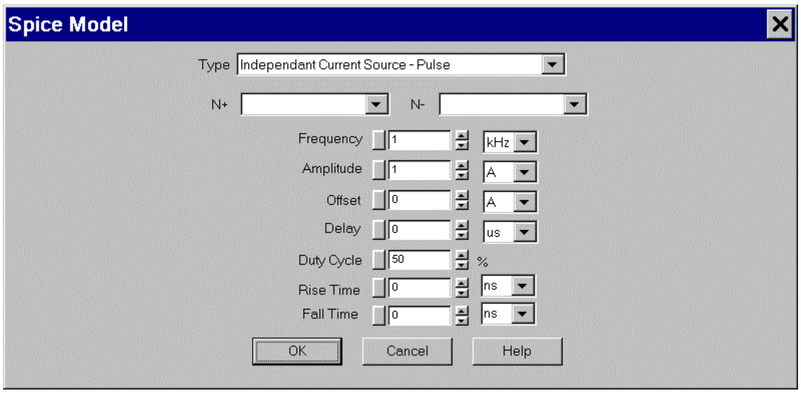


Independent current source - Pulse

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

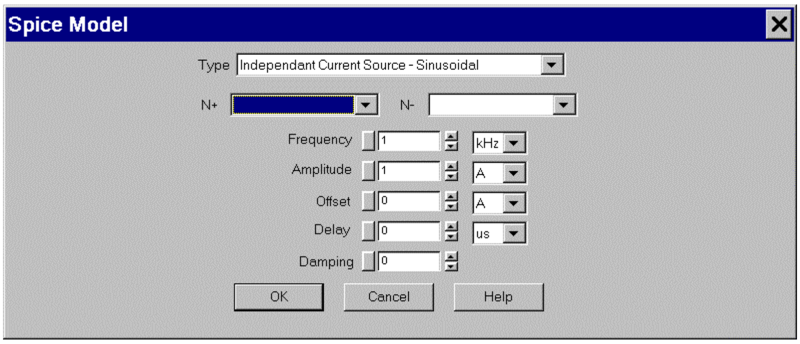


Independent current source - Sinusoidal

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

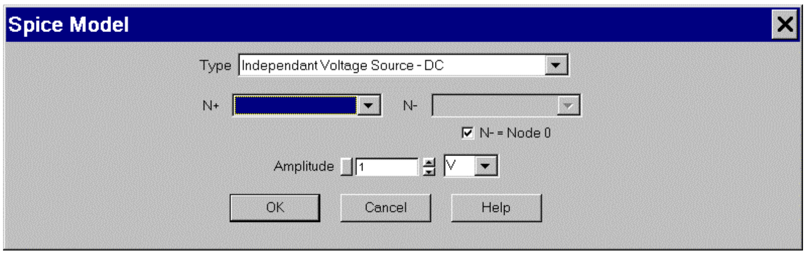


Independent voltage source - DC

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

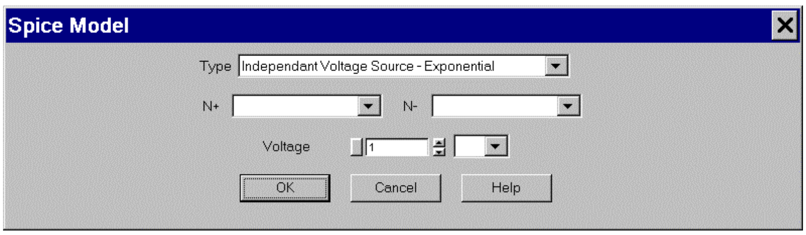


Independent voltage source - Exponential

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.



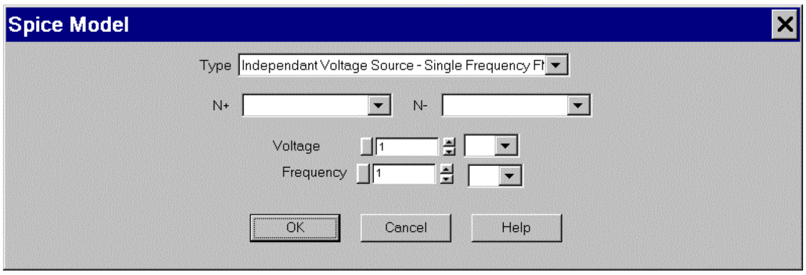
Independent voltage source - FM

The model dialog is shown below. For details of the model parameters see the Spice

Reference Manual.

N+ is the positive terminal.

N+ is the negative.

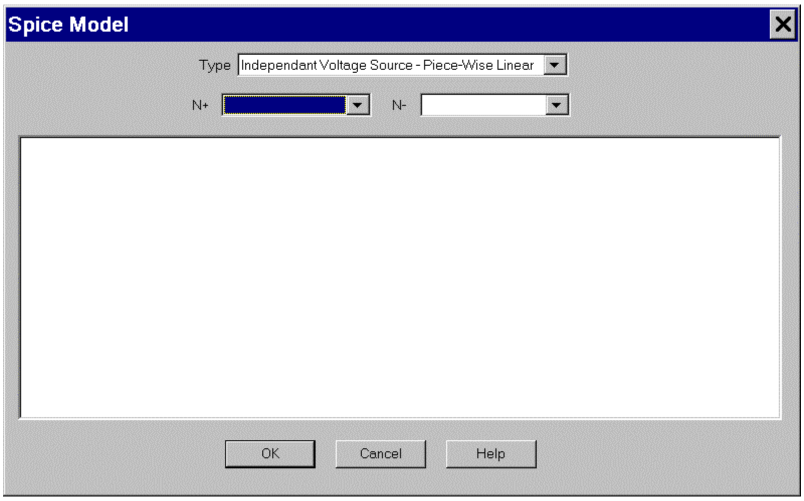


Independent voltage source - Piecewise linear

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

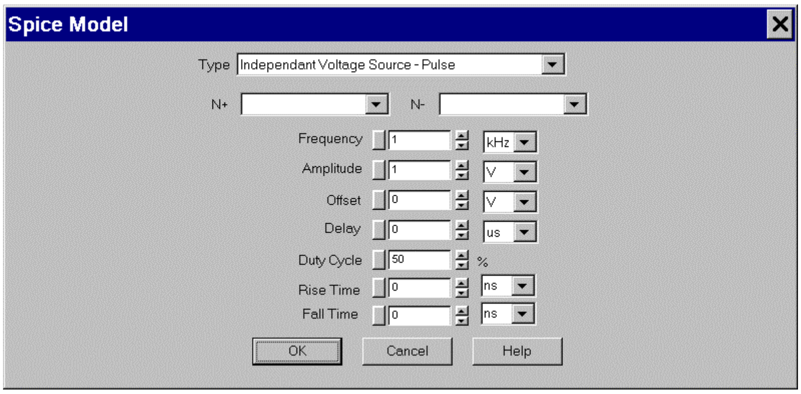


Independent voltage source - Pulse

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

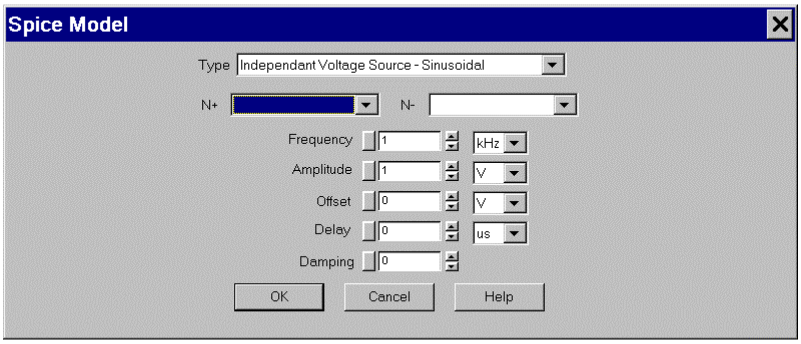


Independent voltage source - Sinusoidal

The model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.

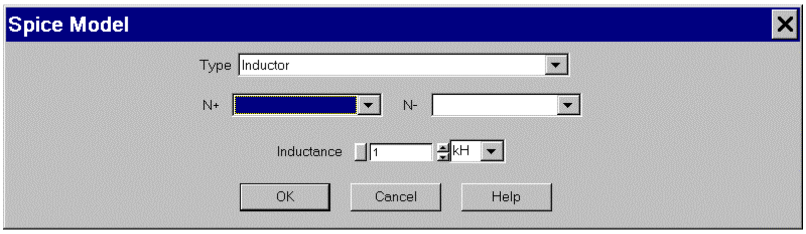


Modeling an inductor

The inductor model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

N+ is the positive terminal.

N+ is the negative.



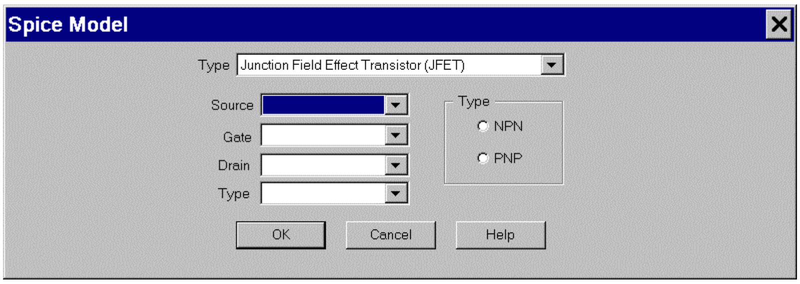
Modeling a JFET transistor

The JFET model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

Source is the terminal to use as the source.

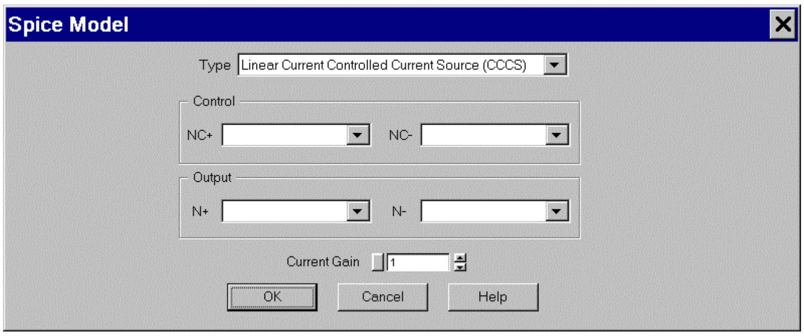
Gate is the terminal to use as the gate.

Drain is the terminal to use as the drain.



Modeling a linear current controlled current source

The linear current controlled current source model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



NC+ The control positive terminal.

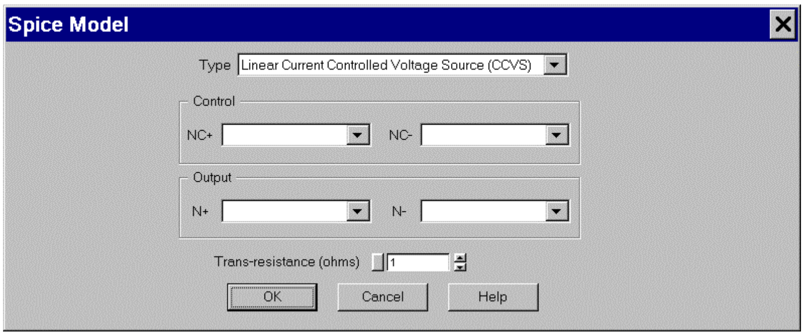
NC+ The control negative terminal.

N+ The positive terminal.

N+ The negative terminal.

Modeling a linear current controlled voltage source

The linear current controlled current voltage model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



NC+ The control positive terminal.

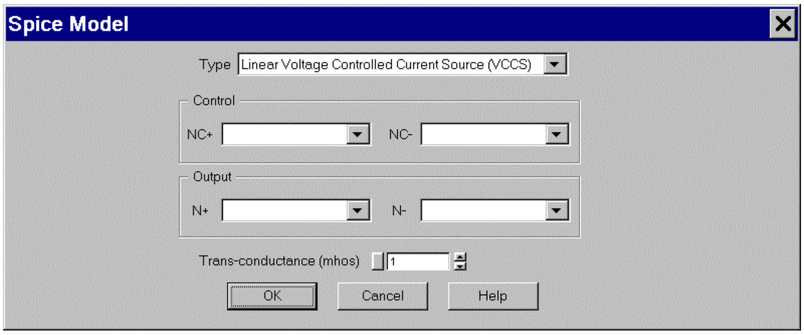
NC+ The control negative terminal.

N+ The positive terminal.

N+ The negative terminal.

Modeling a linear voltage controlled current source

The linear voltage controlled current source model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



NC+ The control positive terminal.

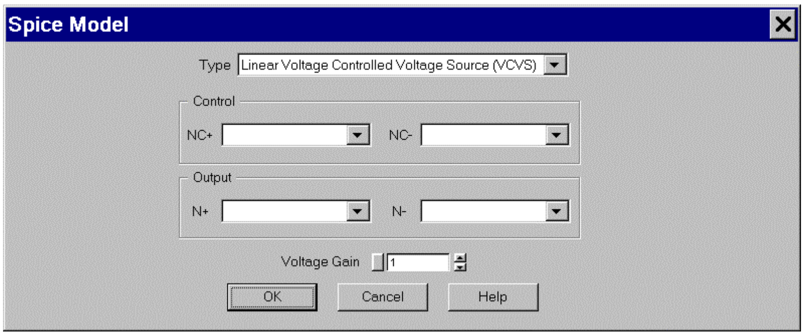
NC+ The control negative terminal.

N+ The positive terminal.

N+ The negative terminal.

Modeling a linear voltage controlled voltage source

The linear voltage controlled current source model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



NC+ The control positive terminal.

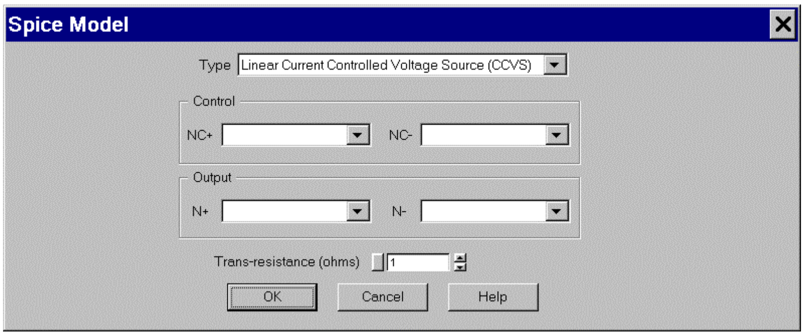
NC+ The control negative terminal.

N+ The positive terminal.

N+ The negative terminal.

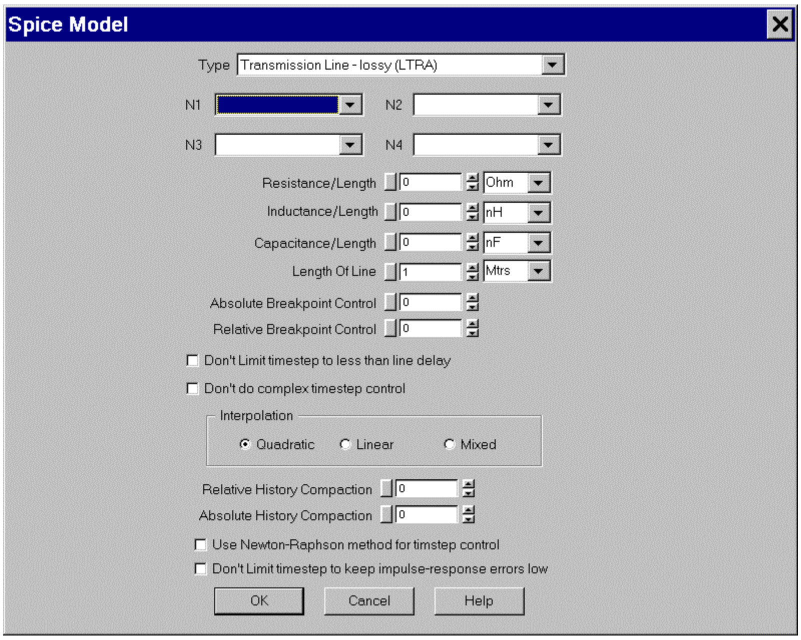
Modeling a lossless transmission line

The lossless transmission model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



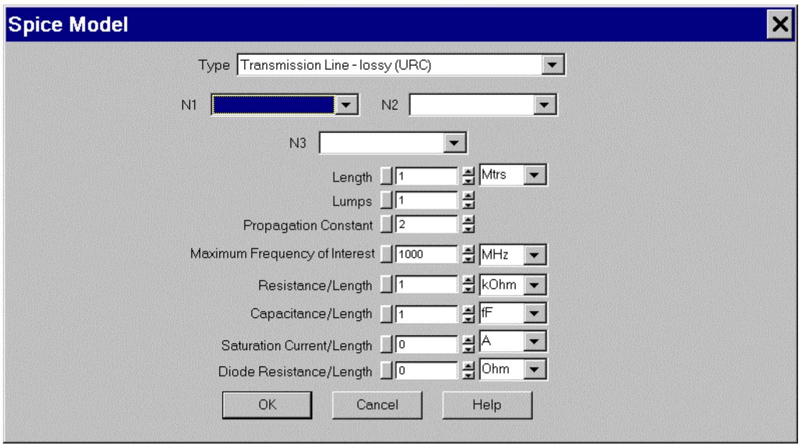
Modeling a lossy transmission line (LTRA)

The lossy transmission (LTRA) model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



Modeling a lossy transmission line (URC)

The lossy transmission (URC) model dialog is shown below. For details of the model parameters see the Spice Reference Manual.



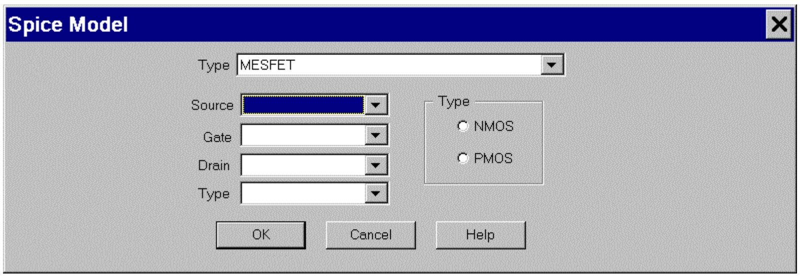
Modeling a MESFET transistor

The MESFET model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

Source is the terminal to use as the source.

Gate is the terminal to use as the gate.

Drain is the terminal to use as the drain.



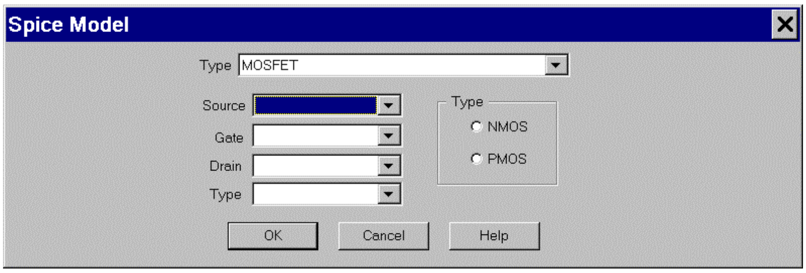
Modeling a MOSFET transistor

The MOSFET model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

Source is the terminal to use as the source.

Gate is the terminal to use as the gate.

Drain is the terminal to use as the drain.

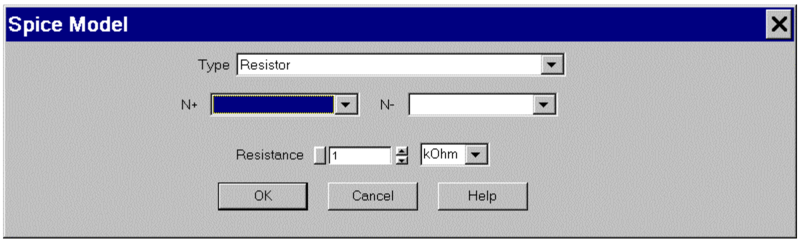


Modeling a resistor

The resistor model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

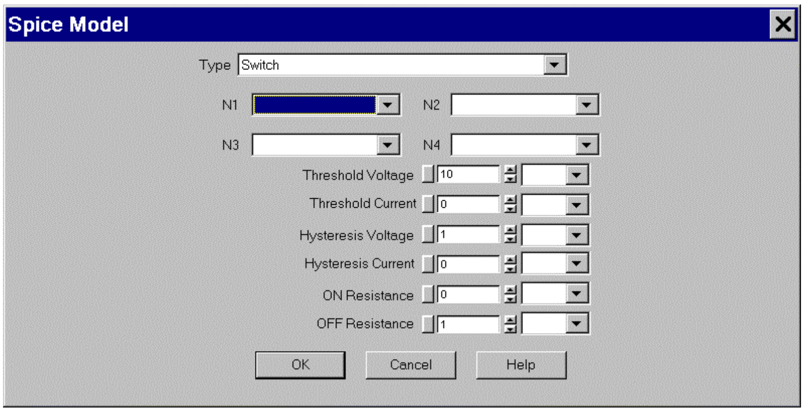
N+ is the positive terminal.

N+ is the negative.



Modeling a switch

The switch model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

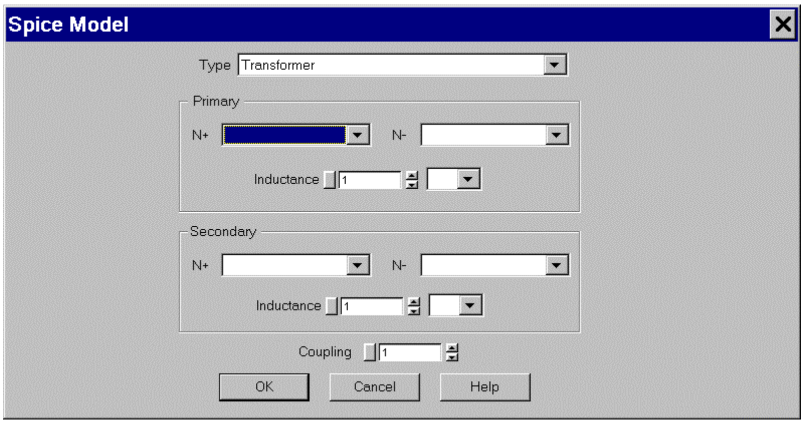


Modeling a transformer

The resistor model dialog is shown below. For details of the model parameters see the Spice Reference Manual.

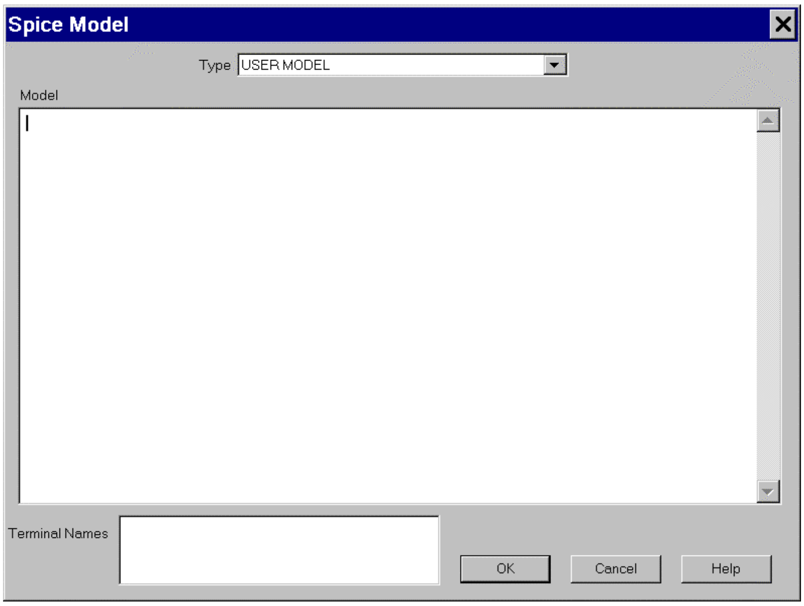
N+ is the positive terminal.

N+ is the negative.



Adding your own model

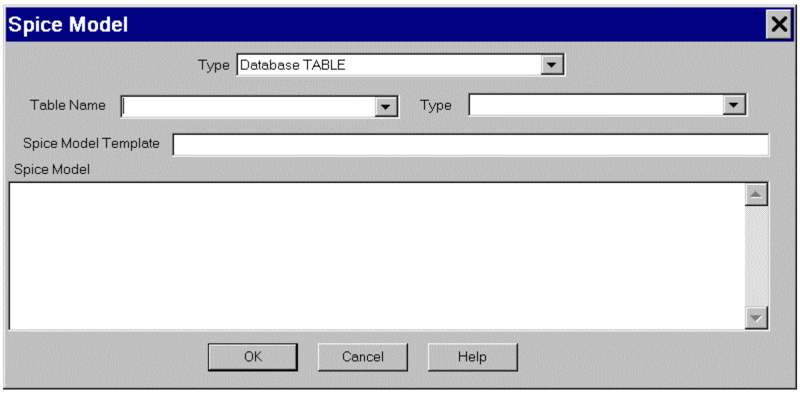
The user model dialog is shown below. Type in the SPICE model for the part.



Terminal Names. This is a list of available terminals.

Modeling a group of similar devices

The similar devices model dialog is shown below.



SPICE 3f5 MANUAL

INTRODUCTION

SPICE is a general-purpose circuit simulation program for nonlinear dc, nonlinear transient, and linear ac analyses. Circuits may contain resistors, capacitors, inductors, mutual inductors, independent voltage and current sources, four types of dependent sources, lossless and lossy transmission lines (two separate implementations), switches, uniform distributed RC lines, and the five most common semiconductor devices: diodes, BJTs, JFETs, MESFETs, and MOSFETs.

The SPICE3 version is based directly on SPICE 2G.6. While SPICE3 is being developed to include new features, it continues to support those capabilities and models that remain in extensive use in the SPICE2 program.

SPICE has built-in models for the semiconductor devices, and the user need specify only the pertinent model parameter values. The model for the BJT is based on the integral-charge model of Gummel and Poon; however, if the Gummel- Poon parameters are not specified, the model reduces to the simpler Ebers-Moll model. In either case, charge-storage effects, ohmic resistances, and a current-dependent output conductance may be included. The diode model can be used for either junction diodes or Schottky barrier diodes. The JFET model is based on the FET model of Shichman and Hodges. Six MOSFET models are implemented: MOS1 is described by a square-law I-V characteristic, MOS2 [1] is an analytical model, while MOS3 [1] is a semi-empirical model; MOS6 [2] is a simple analytic model accurate in the short-channel region; MOS4 [3, 4] and MOS5 [5] are the BSIM (Berkeley Short-channel IGFET Model) and BSIM2. MOS2, MOS3, and MOS4 include second-order effects such as channel-length modulation, sub threshold conduction, scattering-limited velocity saturation, small-size effects, and charge-controlled capacitances.

TYPES OF ANALYSIS

DC Analysis

The dc analysis portion of SPICE determines the dc operating point of the circuit with inductors shorted and capacitors opened. The dc analysis options are specified on the .DC, .TF, and .OP control lines. A dc analysis is automatically performed prior to a transient analysis to determine the transient initial conditions, and prior to an ac small-signal analysis to determine the linearized, small-signal models for nonlinear devices. If requested, the dc small-signal value of a transfer function (ratio of output variable to input source), input resistance, and output resistance is also computed as a part of the dc solution. The dc analysis can also be used to generate dc transfer curves: a specified independent voltage or current source is stepped over a user-specified range and the dc output variables are stored for each sequential source value.

AC Small-Signal Analysis

The ac small-signal portion of SPICE computes the ac output variables as a function of frequency. The program first computes the dc operating point of the circuit and determines linearized, small-signal models for all of the nonlinear devices in the circuit. The resultant linear circuit is then analyzed over a user-specified range of frequencies. The desired output of an ac small- signal analysis is usually a transfer function (voltage gain, transimpedance, etc). If the circuit has only one ac input, it is convenient to set that input to unity and zero phase, so that output variables have the same value as the transfer function of the output variable with respect to the input.

Transient Analysis

The transient analysis portion of SPICE computes the transient output variables as a function of time over a user-specified time interval. The initial conditions are automatically determined by a dc analysis. All sources that are not time dependent (for example, power supplies) are set to their dc value. The transient time interval is specified on a .TRAN control line.

Pole-Zero Analysis

The pole-zero analysis portion of SPICE computes the poles and/or zeros in the small-signal ac transfer function. The program first computes the dc operating point and then determines the linearized, small-signal models for all the nonlinear devices in the circuit. This circuit is then used to find the poles and zeros of the transfer function.

Two types of transfer functions are allowed: one of the form (output voltage)/(input voltage) and the other of the form (output voltage)/(input current). These two types of transfer functions cover all the cases and one can find the poles/zeros of functions like input/output impedance and voltage gain. The input and output ports are specified as two pairs of nodes.

The pole-zero analysis works with resistors, capacitors, inductors, linear-controlled sources, independent sources, BJTs, MOSFETs, JFETs and diodes. Transmission lines are not supported.

The method used in the analysis is a sub-optimal numerical search. For large circuits it may take a considerable time or fail to find all poles and zeros. For some circuits, the method becomes "lost" and finds an excessive number of poles or zeros.

Small-Signal Distortion Analysis

The distortion analysis portion of SPICE computes steady-state harmonic and intermodulation products for small input signal magnitudes. If signals of a single frequency are specified as the input to the circuit, the complex values of the second and third harmonics are determined at every point in the circuit. If there are signals of two frequencies input to the circuit, the analysis finds out the complex values of the circuit variables at the sum and difference of the input frequencies, and at the difference of the smaller frequency from the second harmonic of the larger frequency.

Distortion analysis is supported for the following nonlinear devices: diodes (DIO), BJT, JFET, MOSFETs (levels 1, 2, 3, 4/BSIM1, 5/BSIM2, and 6) and MESFETS. All linear devices are automatically supported by distortion analysis. If there are switches present in the circuit, the analysis continues to be accurate provided the switches do not change state under the small excitations used for distortion calculations.

Sensitivity Analysis

Spice3 will calculate either the DC operating-point sensitivity or the AC small-signal sensitivity of an output variable with respect to all circuit variables, including model parameters. Spice calculates the difference in an output variable (either a node voltage or a branch current) by perturbing each parameter of each device independently. Since the method is a numerical approximation, the results may demonstrate second order affects in highly sensitive parameters, or may fail to show very low but non-zero sensitivity. Further, since each variable is perturbed by a small fraction of its value, zero-valued parameters are not analyized (this has the benefit of reducing what is usually a very large amount of data).

Noise Analysis

The noise analysis portion of SPICE does analysis device-generated noise for the given circuit. When provided with an input source and an output port, the analysis calculates the noise contributions of each device (and each noise generator within the device) to the output port voltage. It also calculates the input noise to the circuit, equivalent to the output noise referred to the specified input source. This is done for every frequency point in a specified range - the calculated value of the noise corresponds to the spectral density of the circuit variable viewed as a stationary gaussian stochastic process.

After calculating the spectral densities, noise analysis integrates these values over the specified frequency range to arrive at the total noise voltage/current (over this frequency range). This calculated value corresponds to the variance of the circuit variable viewed as a stationary gaussian process.

ANALYSIS AT DIFFERENT TEMPERATURES

All input data for SPICE is assumed to have been measured at a nominal temperature of 27 C, which can be changed by use of the TNOM parameter on the .OPTION control line. This value can further be overridden for any device which models temperature effects by specifying the TNOM parameter on the model itself. The circuit simulation is performed at a temperature of 27 C, unless overridden by a TEMP parameter on the .OPTION control line. Individual instances may further override the circuit temperature through the specification of a TEMP parameter on the instance.

Temperature dependent support is provided for resistors, diodes, JFETs, BJTs, and level 1, 2, and 3 MOSFETs. BSIM (levels 4 and 5) MOSFETs have an alternate temperature dependency scheme which adjusts all of the model parameters before input to SPICE. For details of the BSIM temperature adjustment, see [6] and [7].

Temperature appears explicitly in the exponential terms of the BJT and diode model equations. In addition, saturation currents have a built-in temperature dependence. The temperature dependence of the saturation current in the BJT models is determined by:

$$I_S(T_1) = I_S(T_0)\left[\frac{T_1}{T_0}\right]^{XTI} \exp\left[\frac{E_g q(T_1 T_0)}{k(T_1 - T_0)}\right]$$

Where k is Boltzmann's constant, q is the electronic charge, E_g is the energy gap which is a model parameter, and XTI is the saturation current temperature exponent (also a model parameter, and usually equal to 3).

The temperature dependence of forward and reverse beta is according to the formula:

$$\beta(T_1) = \beta(T_0)\left[\frac{T_1}{T_0}\right]^{XTB}$$

Where T_1 and T_0 are in degrees Kelvin, and XTB is a user-supplied model parameter. Temperature effects on beta are carried out by appropriate adjustment to the values of $_F$, I_{SE}, $_R$, and I_{SC} (spice model parameters BF, ISE, BR, and ISC, respectively).

Temperature dependence of the saturation current in the junction diode model is determined by:

$$I_S(T_1) = I_S(T_0)\left[\frac{T_1}{T_0}\right]^{\frac{XTI}{N}} \exp\left[\frac{E_g q(T_1 T_0)}{Nk(T_1 - T_0)}\right]$$

Where N is the emission coefficient, which is a model parameter, and the other symbols have the same meaning as above. Note that for Schottky barrier diodes, the value of the saturation current temperature exponent, XTI, is usually 2.

Temperature appears explicitly in the value of junction potential,   (in spice PHI), for all the device models. The temperature dependence is determined by:

$$\phi(T) = \frac{kT}{q}\ln\left[\frac{N_a N_d}{N_i(T)^2}\right]$$

where k is Boltzmann's constant, q is the electronic charge, N_a is the acceptor impurity density, N_d is the donor impurity density, N_i is the intrinsic carrier concentration, and E_g is the energy gap.

Temperature appears explicitly in the value of surface mobility, $_0$ (or UO), for the MOSFET model. The temperature dependence is determined by:

$$\mu_0(T) = \frac{\mu_0(T_0)}{\left(\frac{T}{T_0}\right)^{1.5}}$$

The effects of temperature on resistors is modeled by the formula:

$$R(T) = R(T_0)[1 + TC_1(T - T_0) + TC_2(T - T_0)^2]$$

where T is the circuit temperature, T_0 is the nominal temperature, and TC_1 and TC_2 are the first- and second-order temperature coefficients.

CIRCUIT DESCRIPTION

GENERAL STRUCTURE AND CONVENTIONS

The circuit to be analyzed is described to SPICE by a set of element lines, which define the circuit topology and element values, and a set of control lines, which define the model parameters and the run controls. The first line in the input file must be the title, and the last line must be ".END". The order of the remaining lines is arbitrary (except, of course, that continuation lines must immediately follow the line being continued).

Each element in the circuit is specified by an element line that contains the element name, the circuit nodes to which the element is connected, and the values of the parameters that determine the electrical characteristics of the element. The first letter of the element name specifies the element type. The format for the SPICE element types is given in what follows. The strings XXXXXXX, YYYYYYY, and ZZZZZZZ denote arbitrary alphanumeric strings. For example, a resistor name must begin with the letter R and can contain one or more characters. Hence, R, R1, RSE, ROUT, and R3AC2ZY are valid resistor names. Details of each type of device are supplied in a following section.

Fields on a line are separated by one or more blanks, a comma, an equal ('=') sign, or a left or right parenthesis; extra spaces are ignored. A line may be continued by entering a '+' (plus) in column 1 of the following line; SPICE continues reading beginning with column 2.

A name field must begin with a letter (A through Z) and cannot contain any delimiters.

A number field may be an integer field (12, -44), a floating point field (3.14159), either an integer or floating point number followed by an integer exponent (1e-14, 2.65e3), or either an integer or a floating point number followed by one of the following scale factors:

T = 10^{12} G = 10^{9} Meg = 10^{6} K = 10^{3} mil = 25.4 10^{-6}
m = 10^{-3} u = 10^{-6} n = 10^{-9} p = 10^{-12} f = 10^{-15}

Letters immediately following a number that are not scale factors are ignored, and letters immediately following a scale factor are ignored. Hence, 10, 10V, 10Volts, and 10Hz all represent the same number, and M, MA, MSec, and MMhos all represent the same scale factor. Note that 1000, 1000.0, 1000Hz, 1e3, 1.0e3, 1KHz, and 1K all represent the same number.

Nodes names may be arbitrary character strings. The datum (ground) node must be named '0'. Note the difference in SPICE3 where the nodes are treated as character strings and not evaluated as numbers, thus '0' and '00' are distinct nodes in SPICE3 but not in SPICE2. The circuit cannot contain a loop of voltage sources and/or inductors and cannot contain a cut-set of current sources and/or capacitors. Each node in the circuit must have a dc path to ground. Every node must have at least two connections except for transmission line nodes (to permit unterminated transmission lines) and MOSFET substrate nodes (which have two internal connections anyway).

TITLE LINE, COMMENT LINES AND .END LINE

Title Line

Examples:

POWER AMPLIFIER CIRCUIT

TEST OF CAM CELL

The title line must be the first in the input file. Its contents are printed verbatim as the heading for each section of output.

.End line

Examples:

.END

The "End" line must always be the last in the input file. Note that the period is an integral part of the name.

Comments

General Form:

* <any comment>

Examples:

* RF=1K Gain should be 100

* Check open-loop gain and phase margin

The asterisk in the first column indicates that this line is a comment line. Comment lines may be placed anywhere in the circuit description. Note that SPICE3 also considers any line with leading white space to be a comment.

DEVICE MODELS

General form:

.MODEL MNAME TYPE(PNAME1=PVAL1 PNAME2=PVAL2 ...)

Examples:

.MODEL MOD1 NPN (BF=50 IS=1E-13 VBF=50)

Most simple circuit elements typically require only a few parameter values. However, some devices (semiconductor devices in particular) that are included in SPICE require many parameter values. Often, many devices in a circuit are defined by the same set of device model parameters. For these reasons, a set of device model parameters is defined on a separate .MODEL line and assigned a unique model name. The device element lines in SPICE then refer to the model name.

For these more complex device types, each device element line contains the device name, the nodes to which the device is connected, and the device model name. In addition, other optional parameters may be specified for some devices: geometric factors and an initial condition (see the following section on Transistors and Diodes for more details).

MNAME in the above is the model name, and type is one of the following fifteen types:

R	Semiconductor resistor model
C	Semiconductor capacitor model
SW	Voltage controlled switch
CSW	Current controlled switch
URC	Uniform distributed RC model
LTRA	Lossy transmission line model
D	Diode model
NPN	NPN BJT model

PNP	PNP BJT model
NJF	N-channel JFET model
PJF	P-channel JFET model
NMOS	N-channel MOSFET model
PMOS	P-channel MOSFET model
NMF	N-channel MESFET model
PMF	P-channel MESFET model

Parameter values are defined by appending the parameter name followed by an equal sign and the parameter value. Model parameters that are not given a value are assigned the default values given below for each model type. Models, model parameters, and default values are listed in the next section along with the description of device element lines.

SUBCIRCUITS

A subcircuit that consists of SPICE elements can be defined and referenced in a fashion similar to device models. The subcircuit is defined in the input file by a grouping of element lines; the program then automatically inserts the group of elements wherever the subcircuit is referenced. There is no limit on the size or complexity of subcircuits, and subcircuits may contain other subcircuits. An example of subcircuit usage is given in *(AA.

.SUBCKT

General form:

.SUBCKT subnam N1 <N2; N3 ...>

Examples:

.SUBCKT OPAMP 1 2 3 4

A circuit definition is begun with a .SUBCKT line. SUBNAM is the subcircuit name, and N1, N2, ... are the external nodes, which cannot be zero. The group of element lines which immediately follow the .SUBCKT line define the subcircuit. The last line in a subcircuit definition is the .ENDS line (see below). Control lines may not appear within a subcircuit definition; however, subcircuit definitions may contain anything else, including other subcircuit definitions, device models, and subcircuit calls (see below). Note that any device models or subcircuit definitions included as part of a subcircuit definition are strictly local (i.e., such models and definitions are not known outside the subcircuit definition). Also, any element nodes not included on the .SUBCKT line are strictly local, with the exception of 0 (ground) which is always global.

.ENDS

General form:

.ENDS <SUBNAM>

Examples:

.ENDS OPAMP

The "Ends" line must be the last one for any subcircuit definition. The subcircuit name, if included, indicates which subcircuit definition is being terminated; if omitted, all subcircuits being defined are terminated. The name is needed only when nested subcircuit definitions are being made.

Subcircuit Calls

General form:

XYYYYYYY N1 <N2 N3 ...> SUBNA

Examples:

X1 2 4 17 3 1 MULTI

Subcircuits are used in SPICE by specifying pseudo-elements beginning with the letter X, followed by the circuit nodes to be used in expanding the subcircuit.

COMBINING FILES: .INCLUDE LINES

General form:

.INCLUDE *filename*

Examples:

.INCLUDE /users/spice/common/wattmeter.cir

Frequently, portions of circuit descriptions will be reused in several input files, particularly with common models and subcircuits. In any spice input file, the ".include" line may be used to copy some other file as if that second file appeared in place of the ".include" line in the original file. There is no restriction on the file name imposed by spice beyond those imposed by the local operating system.

CIRCUIT ELEMENTS AND MODELS

Data fields that are enclosed in less-than and greater-than signs ('< >') are optional. All indicated punctuation (parentheses, equal signs, etc.) is optional but indicate the presence of any delimiter. Further, future implementations may require the punctuation as stated. A consistent style adhering to the punctuation shown here makes the input easier to understand. With respect to branch voltages and currents, SPICE uniformly uses the associated reference convention (current flows in the direction of voltage drop).

Resistors

General form:

RXXXXXXX N1 N2 VALUE

Examples:

R1 1 2 100

RC1 12 17 1K

N1 and N2 are the two element nodes. VALUE is the resistance (in ohms) and may be positive or negative but not zero.

Semiconductor Resistors

General form:

RXXXXXXX N1 N2 <VALUE> <MNAME> <L=LENGTH> <W=WIDTH> <TEMP=T>

Examples:

RLOAD 2 10 10K

RMOD 3 7 RMODEL L=10u W=1u

This is the more general form of the resistor presented in section 6.1, and allows the modeling of temperature effects and for the calculation of the actual resistance value from strictly geometric information and the specifications of the process. If VALUE is specified, it overrides the geometric information and defines the resistance. If MNAME is specified, then the resistance may be calculated from the process information in the model MNAME and the given LENGTH and WIDTH. If VALUE is not specified, then MNAME and LENGTH **must** be specified. If WIDTH is not specified, then it is taken from the default width given in the model. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

Semiconductor Resistor Model (R)

The resistor model consists of process-related device data that allow the resistance to be calculated from geometric information and to be corrected for temperature. The parameters available are:

name	parameter	units	default	example
TC1	first order temperature coeff.	Ω/°C	0	-
TC2	second order temperature coeff.	Ω/oC^2	0	-
RSH	sheet resistance	Ω/q	-	50
DEFW	default width	meters	1e-6	2e-6
NARROW	narrowing due to side etching	meters	0	1e-7
TNOM	parameter measurement temperature	°C	27	50

The sheet resistance is used with the narrowing parameter and L and W from the resistor device to determine the nominal resistance by the formula

$$R = RSH\frac{L - NARROW}{W - NARROW}$$

DEFW is used to supply a default value for W if one is not specified for the device. If either RSH or L is not specified, then the standard default resistance value of 1kΩ is used. TNOM is used to override the circuit-wide value given on the .OPTIONS control line where the parameters of this model have been measured at a different temperature. After the nominal resistance is calculated, it is adjusted for temperature by the formula:

$$R(T) = R(T_0)[1 + TC_1(T - T_0) + TC_2(T - T_0)^2]$$

Capacitors

General form:

CXXXXXXX N+ N- VALUE <IC=INCOND>

Examples:

CBYP 13 0 1UF

COSC 17 23 10U IC=3V

N+ and N- are the positive and negative element nodes, respectively. VALUE is the capacitance in Farads.

The (optional) initial condition is the initial (time-zero) value of capacitor voltage (in Volts). Note that the initial conditions (if any) apply 'only' if the UIC option is specified on the .TRAN control line.

Semiconductor Capacitors

General form:

CXXXXXXX N1 N2 <VALUE> <MNAME> <L=LENGTH> <W=WIDTH> <IC=VAL>

Examples:

CLOAD 2 10 10P

CMOD 3 7 CMODEL L=10u W=1u

This is the more general form of the Capacitor presented in section 6.2, and allows for the calculation of the actual capacitance value from strictly geometric information and the specifications of the process. If VALUE is specified, it defines the capacitance. If MNAME is specified, then the capacitance is calculated from the process information in the model MNAME and the given LENGTH and WIDTH. If VALUE is not specified, then MNAME and LENGTH **must** be specified. If WIDTH is not specified, then it is taken from the default width given in the model. Either VALUE or MNAME, LENGTH, and WIDTH may be specified, but not both sets.

Semiconductor Capacitor Model (C)

The capacitor model contains process information that may be used to compute the capacitance from strictly geometric information.

name	parameter	units	default	example
CJ	junction bottom capacitance	F/meters^2	-	5.e-5
CJSW	junction sidewall capacitance	F/meters	-	2.e-11
DEFW	default device width	meters	1.e-6	2.e-6
NARROW	narrowing due to side etching	meters	0.0	1.e-7

The capacitor has a capacitance computed as

$$CAP = CJ(LENGTH - NARROW)(WIDTH - NARROW) + 2CJSW(LENGTH + WIDTH - 2NARRO$$

Inductors

General form:

LYYYYYYY N+ N- VALUE <IC=INCOND>

Examples:

LLINK 42 69 1UH

LSHUNT 23 51 10U IC=15.7MA

N+ and N- are the positive and negative element nodes, respectively. VALUE is the inductance in Henries.

The (optional) initial condition is the initial (time-zero) value of inductor current (in Amps) that flows from N+, through the inductor, to N-. Note that the initial conditions (if any) apply only if the UIC option is specified on the .TRAN analysis line.

Coupled (Mutual) Inductors

General form:

KXXXXXXX LYYYYYYY LZZZZZZZ VALUE

Examples:

K43 LAA LBB 0.999

KXFRMR L1 L2 0.87

LYYYYYYY and LZZZZZZZ are the names of the two coupled inductors, and VALUE is the coefficient of coupling, K, which must be greater than 0 and less than or equal to 1. Using the 'dot' convention, place a 'dot' on the first node of each inductor.

Switches

General form:

SXXXXXXX N+ N- NC+ NC- MODEL <ON><OFF>

WYYYYYYY N+ N- VNAM MODEL <ON><OFF>

Examples:

s1 1 2 3 4 switch1 ONs2 5 6 3 0 sm2 off

Switch1 1 2 10 0 smodel1

w1 1 2 vclock switchmod1

W2 3 0 vramp sm1 ON

wreset 5 6 vclck lossyswitch OFF

Nodes 1 and 2 are the nodes between which the switch terminals are connected. The model name is mandatory while the initial conditions are optional. For the voltage controlled switch, nodes 3 and 4 are the positive and negative controlling nodes respectively. For the current controlled switch, the controlling current is that through the specified voltage source. The direction of positive controlling current flow is from the positive node, through the source, to the negative node.

Switch Model (SW/CSW)

The switch model allows an almost ideal switch to be described in SPICE. The switch is not quite ideal, in that the resistance can not change from 0 to infinity, but must always have a finite positive value. By proper selection of the on and off resistances, they can be effectively zero and infiniy in comparison to other circuit elements. The parameters available are:

name	parameter	units	default	switch
VT	threshold voltage	Volts	0.0	S
IT	threshold current	Amps	0.0	W
VH	hysteresis voltage	Volts	0.0	S
IH	hysteresis current	Amps	0.0	W
RON	on resistance	Ω	1.0	both
ROFF	off resistance	Ω	1/GMIN*	both

*(See the .OPTIONS control line for a description of GMIN, its default value results in an off-resistance of 1.0e+12 ohms.)

The use of an ideal element that is highly nonlinear such as a switch can cause large discontinuities to occur in the circuit node voltages. A rapid change such as that associated with a switch changing state can cause numerical roundoff or tolerance problems leading to erroneous results or timestep difficulties. The user of switches can improve the situation by taking the following steps:

First, it is wise to set ideal switch impedances just high or low enough to be negligible with respect to other circuit elements. Using switch impedances that are close to "ideal" in all cases aggravates the problem of discontinuities mentioned above. Of course, when modeling real devices such as MOSFETS, the on resistance should be adjusted to a realistic level depending on the size of the device being modeled.

If a wide range of ON to OFF resistance must be used in the switches (ROFF/RON >1e;+12), then the tolerance on errors allowed during transient analysis should be decreased by using the .OPTIONS control line and specifying TRTOL to be less than the default value of 7.0. When switches are placed around capacitors, then the option CHGTOL should also be reduced. Suggested values for these two options are 1.0 and 1e-16 respectively. These changes inform SPICE3 to be more careful around the switch points so that no errors are made due to the rapid change in the circuit.

VOLTAGE AND CURRENT SOURCES

Independent Sources

General form:

VXXXXXXX N+ N- <DC<> DC/TRAN VALUE> <AC <ACMAG <ACPHASE>>>

+ <DISTOF1 <F1MAG <F1PHASE>>> <DISTOF2 <F2MAG <F2PHASE>>>

IYYYYYYY N+ N- <<DC> DC/TRAN VALUE> <AC <ACMAG <ACPHASE>>>

+ <DISTOF1 <F1MAG <F1PHASE>>> <DISTOF2 <F2MAG <F2PHASE>>>

Examples:

VCC 10 0 DC 6

VIN 13 2 0.001 AC 1 SIN(0 1 1MEG)

ISRC 23 21 AC 0.333 45.0 SFFM(0 1 10K 5 1K)

VMEAS 12 9

VCARRIER 1 0 DISTOF1 0.1 -90.0

VMODULATOR 2 0 DISTOF2 0.01

IIN1 1 5 AC 1 DISTOF1 DISTOF2 0.001

N+ and N- are the positive and negative nodes, respectively. Note that voltage sources need not be grounded. Positive current is assumed to flow from the positive node, through the source, to the negative node. A current source of positive value forces current to flow out of the N+ node, through the source, and into the N- node. Voltage sources, in addition to being used for circuit excitation, are the 'ammeters' for SPICE, that is, zero valued voltage sources may be inserted into the circuit for the purpose of measuring current. They of course have no effect on circuit operation since they represent short-circuits.

DC/TRAN is the dc and transient analysis value of the source. If the source value is zero both for dc and transient analyses, this value may be omitted. If the source value is time-invariant (e.g., a power supply), then the value may optionally be preceded by the letters DC.

ACMAG is the ac magnitude and ACPHASE is the ac phase. The source is set to this value in the ac analysis. If ACMAG is omitted following the keyword AC, a value of unity is assumed. If ACPHASE is omitted, a value of zero is assumed. If the source is not an ac small-signal input, the keyword AC and the ac values are omitted.

DISTOF1 and DISTOF2 are the keywords that specify that the independent source has distortion inputs at the frequencies F1 and F2 respectively (see the description of the .DISTO control line). The keywords may be followed by an optional magnitude and phase. The default values of the magnitude and phase are 1.0 and 0.0 respectively.

Any independent source can be assigned a time-dependent value for transient analysis. If a source is assigned a time-dependent value, the time-zero value is used for dc analysis. There are five independent source functions: pulse, exponential, sinusoidal, piece-wise linear, and single-frequency FM. If parameters other than source values are omitted or set to zero, the default values shown are assumed. (TSTEP is the printing increment and TSTOP is the final time (see the .TRAN control line for explanation)).

Pulse

General form:

PULSE(V1 V2 TD TR TF PW PER)

Examples:

VIN 3 0 PULSE(-1 1 2NS 2NS 2NS 50NS 100NS)

parameter	default value	units
V1 (initial value)		Volts or Amps
V2 (pulsed value)		Volts or Amps
TD (delay time)	0.0	seconds
TR (rise time)	TSTEP	seconds
TF (fall time)	TSTEP	seconds
PW (pulse width)	TSTOP	seconds
PER(period)	TSTOP	seconds

A single pulse so specified is described by the following table:

time	value
0	V1
TD	V1
TD+TR	V2
TD+TR+PW	V2
TD+TR+PW V2	V1
TSTOP	V1

Intermediate points are determined by linear interpolation.

Sinusoidal

General form:

SIN(VO VA FREQ TD THETA)

Examples:

VIN 3 0 SIN(0 1 100MEG 1NS 1E10)

parameters	default value	units
VO (offset)		Volts or Amps

VA (amplitude)		Volts or Amps
FREQ (frequency)	1/TSTOP	Hz
TD (delay)	0.0	seconds
THETA (damping factor)	0.0	1/seconds

The shape of the waveform is described by the following table:

time	value
0 to TD	VO
TD to TSTOP	$VO + VA\, e^{-\frac{(t-TD)}{THETA}} \sin(2\pi FREQ(t+TD))$

Exponential

General Form:
EXP(V1 V2 TD1 TAU1 TD2 TAU2)
Examples:
VIN 3 0 EXP(-4 -1 2NS 30NS 60NS 40NS)

parameter	default value	units
V1 (initial value)		Volts or Amps
V2 (pulsed value)		Volts or Amps
TD1 (rise delay time)	0.0	seconds
TAU1 (rise time constant)	TSTEP	seconds
TD2 (fall delay time)	TD1+TSTEP	seconds
TAU2 (fall time	TSTEP	seconds

The shape of the waveform is described by the following table:

time	value
0 to TD1	V1
TD1 to TD2	$V1 + \left(V2 - V1\right)\left[1 - e^{\frac{-(t-TD1)}{TAU1}}\right]$

TD2 to TSTOP	$V1+\left(V2-V1\right)\left[1-e^{\frac{-(t-TD1)}{TAU1}}\right]+\left(V1-V2\right)\left[1-e^{\frac{-(t-TD2)}{TAU2}}\right]$

Piece-Wise Linear

General Form:

PWL(T1 V1 <T2; V2 T3 V3 T4 V4 ...>)

Examples:

VCLOCK 7 5 PWL(0 -7 10NS -7 11NS -3 17NS -3 18NS -7 50NS -7)

Each pair of values (Ti, Vi) specifies that the value of the source is Vi (in Volts or Amps) at time=Ti. The value of the source at intermediate values of time is determined by using linear interpolation on the input values.

Single-Frequency FM

General Form:

SFFM(VO VA FC MDI FS)

Examples:

V1 12 0 SFFM(0 1M 20K 5 1K)

parameter	default value	units
VO (offset)		Volts or Amps
VA (amplitude)		Volts or Amps
FC (carrier frequency)	1/TSTOP	Hz
MDI (modulation index)		
FS (signal frequency)	1/TSTOP	Hz

The shape of the waveform is described by the following equation:

$$V(t) = V_0 + V_A \sin(2\pi FCt + MDI \sin(2\pi FSt))$$

Linear Dependent Sources

SPICE allows circuits to contain linear dependent sources characterized by any of the four equations

i = g v	v = e v	i = f i	v = h i

where g, e, f, and h are constants representing transconductance, voltage gain, current gain, and transresistance, respectively.

Linear Voltage-Controlled Current Sources

General form:

GXXXXXXX N+ N- NC+ NC- VALUE

Examples:

G1 2 0 5 0 0.1MMHO

N+ and N- are the positive and negative nodes, respectively. Current flow is from the positive node, through the source, to the negative node. NC+ and NC- are the positive and negative controlling nodes, respectively. VALUE is the transconductance (in mhos).

Linear Voltage-Controlled Voltage Sources

General form:

EXXXXXXX N+ N- NC+ NC- VALUE

Examples:

E1 2 3 14 1 2.0

N+ is the positive node, and N- is the negative node. NC+ and NC- are the positive and negative controlling nodes, respectively. VALUE is the voltage gain.

Linear Current-Controlled Current Sources

General form:

FXXXXXXX N+ N- VNAM VALUE

Examples:

F1 13 5 VSENS 5

N+ and N- are the positive and negative nodes, respectively. Current flow is from the positive node, through the source, to the negative node. VNAM is the name of a voltage source through which the controlling current flows. The direction of positive controlling current flow is from the positive node, through the source, to the negative node of VNAM. VALUE is the current gain.

Linear Current-Controlled Voltage Sources

General form:

HXXXXXXX N+ N- VNAM VALUE

Examples:

HX 5 17 VZ 0.5K

N+ and N- are the positive and negative nodes, respectively. VNAM is the name of a voltage source through which the controlling current flows. The direction of positive controlling current flow is from the positive node, through the source, to the negative node of VNAM. VALUE is the transresistance (in ohms).

Non-linear Dependent Sources

General form:

BXXXXXXX N+ N- <I=EXPR> <V=EXPR>

Examples:

B1 0 1 I=cos(v(1))+sin(v(2))

B1 0 1 V=ln(cos(log(v(1,2)^2)))-v(3)^4+v(2)^v(1)

B1 3 4 I=17

B1 3 4 V=exp(pi^i(vdd))

N+ is the positive node, and *N-* is the negative node. The values of the **V** and **I** parameters determine the voltages and currents across and through the device, respectively. If **I** is given then the device is a current source, and if **V** is given the device is a voltage source. One and only one of these parameters must be given.

The small-signal AC behavior of the nonlinear source is a linear dependent source (or sources) with a proportionality constant equal to the derivative (or derivatives) of the source at the DC operating point.

The expressions given for **V** and **I** may be any function of voltages and currents through voltage sources in the system. The following functions of real variables are defined:

abs	asinh	cosh	sin
acos	atan	exp	sinh
acosh	atanh	ln	sqrt
asin	cos	log	tan

The function "u" is the unit step function, with a value of one for arguments greater than one and a value of zero for arguments less than zero. The function "uramp" is the integral of the unit step: for an input *x*, the value is zero if *x* is less than zero, or if *x* is greater than zero the value is *x*. These two functions are useful in sythesizing piece-wise non-linear functions, though convergence may be adversely affected.

The following standard operators are defined:

+ - * / ^ unary -

If the argument of log, ln, or sqrt becomes less than zero, the absolute value of the argument is used. If a divisor becomes zero or the argument of log or ln becomes zero, an error will result. Other problems may occur when the argument for a function in a partial derivative enters a region where that function is undefined.

To get time into the expression you can integrate the current from a constant current source with a capacitor and use the resulting voltage (don't forget to set the initial voltage across the capacitor). Non-linear resistors, capacitors, and inductors may be synthesized with the nonlinear dependent source. Non-linear resistors are obvious. Non-linear capacitors and inductors are implemented with their linear counterparts by a change of variables implemented with the nonlinear dependent source. The following subcircuit will implement a nonlinear capacitor:

```
.Subckt nlcap pos neg
* Bx: calculate f(input voltage)
Bx 1 0 v = f(v(pos,neg))
* Cx: linear capacitance
Cx 2 0 1
* Vx: Ammeter to measure current into the capacitor
Vx 2 1 DC 0Volts
* Drive the current through Cx back into the circuit
Fx pos neg Vx 1
.ends
```

Non-linear inductors are similar.

TRANSMISSION LINES

Lossless Transmission Lines

General form:

TXXXXXXX N1 N2 N3 N4 Z0=VALUE <TD=VALUE> <F=FREQ

<NL=NRMLEN>>

+ <IC;=V1, I1, V2, I2>

Examples:

T1 1 0 2 0 Z0=50 TD=10NS

N1 and N2 are the nodes at port 1; N3 and N4 are the nodes at port 2. Z0 is the characteristic impedance. The length of the line may be expressed in either of two forms. The transmission delay, TD, may be specified directly (as TD=10ns, for example). Alternatively, a frequency F may be given, together with NL, the normalized electrical length of the transmission line with respect to the wavelength in the line at the frequency F. If a frequency is specified but NL is omitted, 0.25 is assumed (that is, the frequency is assumed to be the quarter-wave frequency). Note that although both forms for expressing the line length are indicated as optional, one of the two must be specified.

Note that this element models only one propagating mode. If all four nodes are distinct in the actual circuit, then two modes may be excited. To simulate such a situation, two transmission-line elements are required. (see the example in *(AA for further clarification.)

The (optional) initial condition specification consists of the voltage and current at each of the transmission line ports. Note that the initial conditions (if any) apply 'only' if the UIC option is specified on the .TRAN control line.

Note that a lossy transmission line (see below) with zero loss may be more accurate than than the lossless transmission line due to implementation details.

Lossy Transmission Lines

General form:

OXXXXXXX N1 N2 N3 N4 MNAME

Examples:

O23 1 0 2 0 LOSSYMOD

OCONNECT 10 5 20 5 INTERCONNECT

This is a two-port convolution model for single-conductor lossy transmission lines. N1 and N2 are the nodes at port 1; N3 and N4 are the nodes at port 2. Note that a lossy transmission line with zero loss may be more accurate than than the lossless transmission line due to implementation details.

Lossy Transmission Line Model (lTRA)

The uniform RLC/RC/LC/RG transmission line model (referred to as the LTRA model henceforth) models a uniform constant-parameter distributed transmission line. The RC and LC cases may also be modeled using the URC and TRA models; however, the newer LTRA model is usually faster and more accurate than the others. The operation of the LTRA model is based on the convolution of the transmission line's impulse responses with its inputs (see [8]).

The LTRA model takes a number of parameters, some of which must be given and some of which are optional.

name	parameter	units/type	default	example
R	resistance/length		0.0	0.2

L	inductance/length	henrys/unit	0.0	9.13e-9
G	conductance/length	mhos/unit	0.0	0.0
C	capacitance/length	farads/unit	0.0	3.65e-12
LEN	length of line		No default	1.0
REL	breakpoint control	arbitrary unit	1	0.5
ABS	breakpoint control		1	5
NOSTEPLIMIT	don't limit timestep to less than line delay	flag	not set	set
NOCONTROL	don't do complex timestep control	flag	not set	set
LININTERP	use linear interpolation	flag	not set	set
MIXEDINTERP	use linear when quadratic seems bad		not set	set
COMPACTREL	special reltol for history compaction	flag	RELTOL	1.0e-3
COMPACTABS	special abstol for history compaction		ABSTOL	1.0e-9
TRUNCNR	use Newton-Raphson method for timestep control	flag	not set	set
TRUNCDONTCUT	don't limit timestep to keep impulse-response errors low	flag	not set	set

The following types of lines have been implemented so far: RLC (uniform transmission line with series loss only), RC (uniform RC line), LC (lossless transmission line), and RG (distributed series resistance and parallel conductance only). Any other combination will yield erroneous results and should not be tried. The length LEN of the line must be specified.

NOSTEPLIMIT is a flag that will remove the default restriction of limiting time-steps to less than the line delay in the RLC case. NOCONTROL is a flag that prevents the default limiting of the time-step based on convolution error criteria in the RLC and RC cases. This speeds up simulation but may in some cases reduce the accuracy of results. LININTERP is a flag that, when specified, will use linear interpolation instead of the default quadratic interpolation for calculating delayed signals. MIXEDINTERP is a flag that, when specified, uses a metric for judging whether quadratic interpolation is not applicable and if so uses linear interpolation; otherwise it uses the default quadratic interpolation. TRUNCDONTCUT is a flag that removes the default cutting of the time-step to limit errors in the actual calculation of impulse-response related quantities. COMPACTREL and COMPACTABS are quantities that control the compaction of the past history of values stored for convolution. Larger values of these lower accuracy but usually increase simulation speed. These are to be used with the TRYTOCOMPACT option, described in the .OPTIONS section. TRUNCNR is a flag that turns on the use of Newton-Raphson iterations to determine an appropriate timestep in the timestep control routines. The default is a trial and error procedure by cutting the previous timestep in half. REL and ABS are quantities that control the setting of breakpoints.

The option most worth experimenting with for increasing the speed of simulation is REL.

The default value of 1 is usually safe from the point of view of accuracy but occasionally increases computation time. A value greater than 2 eliminates all breakpoints and may be worth trying depending on the nature of the rest of the circuit, keeping in mind that it might not be safe from the viewpoint of accuracy. Breakpoints may usually be entirely eliminated if it is expected the circuit will not display sharp discontinuities. Values between 0 and 1 are usually not required but may be used for setting many breakpoints.

COMPACTREL may also be experimented with when the option TRYTOCOMPACT is specified in a .OPTIONS card. The legal range is between 0 and 1. Larger values usually decrease the accuracy of the simulation but in some cases improve speed. If TRYTOCOMPACT is not specified on a .OPTIONS card, history compaction is not attempted and accuracy is high. NOCONTROL, TRUNCDONTCUT and NOSTEPLIMIT also tend to increase speed at the expense of accuracy.

Uniform Distributed RC Lines (lossy)

General form:

UXXXXXXX N1 N2 N3 MNAME L=LEN <N=LUMPS>

Examples:

U1 1 2 0 URCMOD L=50U

URC2 1 12 2 UMODL l=1MIL N=6

N1 and N2 are the two element nodes the RC line connects, while N3 is the node to which the capacitances are connected. MNAME is the model name, LEN is the length of the RC line in meters. LUMPS, if specified, is the number of lumped segments to use in modeling the RC line (see the model description for the action taken if this parameter is omitted).

Uniform Distributed RC Model (URC)

The URC model is derived from a model proposed by L. Gertzberrg in 1974. The model is accomplished by a subcircuit type expansion of the URC line into a network of lumped RC segments with internally generated nodes. The RC segments are in a geometric progression, increasing toward the middle of the URC line, with K as a proportionality constant. The number of lumped segments used, if not specified for the URC line device, is determined by the following formula:

$$N = \frac{\log\left[F_{max}\frac{R}{L}\frac{C}{L}2\pi L^2\left(\frac{K-1}{K}\right)^2\right]}{\log K}$$

The URC line is made up strictly of resistor and capacitor segments unless the ISPERL parameter is given a non-zero value, in which case the capacitors are replaced with reverse biased diodes with a zero-bias junction capacitance equivalent to the capacitance replaced, and with a saturation current of ISPERL amps per meter of transmission line and an optional series resistance equivalent to RSPERL ohms per meter.

	name	parameter	units	default	example	area
1	K	Propagation Constant	-	2.0	1.2	-
2	FMAX	Maximum Frequency of interest	Hz	1.0G	6.5Meg	-
3	RPERL	Resistance per unit length	Ω	1000	10	-
4	CPERL	Capacitance per unit length	F/m	1.0e-15	1pF	-

5	ISPERL	Saturation Current per unit length	A/m	0	-	-
6	RSPERL	Diode Resistance per unit length	Ω	0	-	-

TRANSISTORS AND DIODES

The area factor used on the diode, BJT, JFET, and MESFET devices determines the number of equivalent parallel devices of a specified model. The affected parameters are marked with an asterisk under the heading 'area' in the model descriptions below. Several geometric factors associated with the channel and the drain and source diffusions can be specified on the MOSFET device line.

Two different forms of initial conditions may be specified for some devices. The first form is included to improve the dc convergence for circuits that contain more than one stable state. If a device is specified OFF, the dc operating point is determined with the terminal voltages for that device set to zero. After convergence is obtained, the program continues to iterate to obtain the exact value for the terminal voltages. If a circuit has more than one dc stable state, the OFF option can be used to force the solution to correspond to a desired state. If a device is specified OFF when in reality the device is conducting, the program still obtains the correct solution (assuming the solutions converge) but more iterations are required since the program must independently converge to two separate solutions. The .NODESET control line serves a similar purpose as the OFF option. The .NODESET option is easier to apply and is the preferred means to aid convergence.

The second form of initial conditions are specified for use with the transient analysis. These are true 'initial conditions' as opposed to the convergence aids above. See the description of the .IC control line and the .TRAN control line for a detailed explanation of initial conditions.

Junction Diodes

General form:

DXXXXXXX N+ N- MNAME <AREA>> <OFF> <IC=VD> <TEMP>

Examples:

DBRIDGE 2 10 DIODE1

DCLMP 3 7 DMOD 3.0 IC=0.2

N+ and N- are the positive and negative nodes, respectively. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) starting condition on the device for dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification using IC=VD is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

Diode Model (D)

The dc characteristics of the diode are determined by the parameters IS and N. An ohmic resistance, RS, is included. Charge storage effects are modeled by a transit time, TT, and a nonlinear depletion layer capacitance which is determined by the parameters CJO, VJ, and M. The temperature dependence of the saturation current is defined by the parameters EG, the energy and XTI, the saturation current temperature exponent. The nominal temperature at which these parameters were measured is TNOM, which defaults to the circuit-wide value specified on the .OPTIONS control line. Reverse breakdown is modeled by an exponential increase in the reverse diode current and is determined by the parameters BV and IBV (both of which are positive numbers).

	name	parameter	units	default	example	area
1	IS	saturation current	A	1.0e-14	1.0e-14	*
2	RS	ohmic resistance	Ω	0	10	*
3	N	emission coefficient	-	1	1.0	
4	TT	transit-time	sec	0	0.1ns	
5	CJO	zero-bias junction capacitance	F	0	2pF	*
6	VJ	junction potential	V	1	0.6	
7	M	grading coefficient	-	0.5	0.5	
8	EG	activation energy	eV	1.11	1.11 Si 0.69 Sbd 0.67 Ge	
9	XTI	saturation-current temp. exp	-	3.0	3.0jn 2.0Sbd	
10	KF	flicker noise coefficient	-	0		
11	AF	flicker noise exponent	-	1		
12	FC	coefficient for forward-bais depletion capacitance formula	-	0.5		
13	BV	reverse breakdown voltage	V	infinite	40.0	
14	IBV	current at breakdown voltage	A	1.0e-3		
15	TNOM	parameter measurement temperature	oC	27	50	

Bipolar Junction Transistors (BJTs)

General form:

QXXXXXXX NC NB NE <NS> MNAME <AREA> <OFF> <IC=VBE, VCE> <TEMP=T>

Examples:

Q23 10 24 13 QMOD IC=0.6, 5.0

Q50A 11 26 4 20 MOD1

NC, NB, and NE are the collector, base, and emitter nodes, respectively. NS is the (optional) substrate node. If unspecified, ground is used. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) initial condition on the device for the dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification using IC=VBE, VCE is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. See the .IC control line description for a better way to set transient initial conditions. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

BJT Models (NPN/PNP)

The bipolar junction transistor model in SPICE is an adaptation of the integral charge control model of Gummel and Poon. This modified Gummel-Poon model extends the original model to include several effects at high bias levels. The model automatically simplifies to the simpler Ebers-Moll model when certain parameters are not specified. The parameter names used in the modified Gummel-Poon model have been chosen to be more easily understood by the program user, and to reflect better both physical and circuit design thinking.

The dc model is defined by the parameters IS, BF, NF, ISE, IKF, and NE which determine the forward current gain characteristics, IS, BR, NR, ISC, IKR, and NC which determine the reverse current gain characteristics, and VAF and VAR which determine the output conductance for forward and reverse regions. Three ohmic resistances RB, RC, and RE are included, where RB can be high current dependent. Base charge storage is modeled by forward and reverse transit times, TF and TR, the forward transit time TF being bias dependent if desired, and nonlinear depletion layer capacitances which are determined by CJE, VJE, and MJE for the B-E junction , CJC, VJC, and MJC for the B-C junction and CJS, VJS, and MJS for the C-S (Collector-Substrate) junction. The temperature dependence of the saturation current, IS, is determined by the energy-gap, EG, and the saturation current temperature exponent, XTI. Additionally base current temperature dependence is modeled by the beta temperature exponent XTB in the new model. The values specified are assumed to have been measured at the temperature TNOM, which can be specified on the .OPTIONS control line or overridden by a specification on the .MODEL line.

The BJT parameters used in the modified Gummel-Poon model are listed below. The parameter names used in earlier versions of SPICE2 are still accepted.

Modified Gummel-Poon BJT Parameters

	name	**parameter**	**units**	**default**	**example**	**area**
1	IS	transport saturation current	A	1.0e-16	1.0e-15	*
2	BF	ideal maximum forward beta	-	100	100	
3	NF	forward current emission coefficient	-	1.0	1	
4	VAF	forward Early voltage	V	Infinite	200	
5	IKF	corner for forward beta high current roll-off	A	infinite	0.01	*
6	ISE	B-E leakage saturation current	A	0	1.0e-13	*
7	NE	B-E leakage emission coefficient	-	1.5	2	

8	BR	ideal maximum reverse beta	-	1	0.1	
9	NR	reverse current emission coefficient	-	1	1	
10	VAR	reverse Early voltage	V	infinite	200	
11	IKR	corner for reverse beta high current roll-off	A	infinite	0.01	*
12	ISC	leakage saturation current	A	0		8
13	NC	leakage emission coefficient	-	2	1.5	
14	RB	zero bias base resistance	Ω	0	100	*
15	IRB	current where base resistance falls halfway to its min value	A	infinte	0.1	*
16	RBM	minimum base resistance at high currents	Ω	RB	10	*
17	RE	emitter resistance	Ω	0	1	*
18	RC	collector resistance	Ω	0	10	*
19	CJE	B-E zero-bias depletion capacitance	F	0	2pF	*
20	VJE	B-E built-in potential	V	0.75	0.6	
21	MJE	B-E junction exponential factor	-	0.33	0.33	
22	TF	ideal forward transit time	sec	0	0.1ns	
23	XTF	coefficient for bias dependence of TF	-	0		
24	VTF	voltage describing VBC dependence of TF	V	infinite		
25	ITF	high-current parameter for effect on TF	A	0		*
26	PTF	excess phase at freq=1.0/(TF*2PI) Hz	deg	0		
27	CJC	B-C zero-bias depletion capacitance	F	0	2pF	*

28	VJC	B-C built-in potential	V	0.75	0.5	
29	MJC	B-C junction exponential factor	-	0.33	0.5	
30	XCJC	fraction of B-C depletion capacitance connected to internal base node	-	1		
31	TR	ideal reverse transit time	sec	0	10ns	
32	CJS	zero-bias collector-substrate capacitance	F	0	2pF	*
33	VJS	substrate junction built-in potential	V	0.75		
34	MJS	substrate junction exponential factor	-	0	0.5	
35	XTB	forward and reverse beta temperature exponent	-	0		
36	EG	energy gap for temperature effect on IS	eV	1.11		
37	XTI	temperature exponent for effect on IS	-	3		
38	KF	flicker-noise coefficient	-	0		
39	AF	flicker-noise exponent	-	1		
40	FC	coefficient for forward-bias depletion capacitance formula	-	0.5		
41	TNOM	Parameter measurement temperature	oC	27	50	

Junction Field-Effect Transistors (JFETs)

General form:

JXXXXXXX ND NG NS MNAME <AREA> <OFF> <IC=VDS, VGS> <TEMP>

Examples:

J1 7 2 3 JM1 OFF

ND, NG, and NS are the drain, gate, and source nodes, respectively. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) initial condition on the device for dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification, using IC=VDS, VGS is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. See the .IC control line for a better way to set initial conditions. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line.

JFET Models (NJF/PJF)

The JFET model is derived from the FET model of Shichman and Hodges. The dc characteristics are defined by the parameters VTO and BETA, which determine the variation of drain current with gate voltage, LAMBDA, which determines the output conductance, and IS, the saturation current of the two gate junctions. Two ohmic resistances, RD and RS, are included. Charge storage is modeled by nonlinear depletion layer capacitances for both gate junctions which vary as the -1/2 power of junction voltage and are defined by the parameters CGS, CGD, and PB.

Note that in Spice3f and later, a fitting parameter B has been added. For details, see [9].

	name	parameter	units	default	example	area
1	VTO	threshold voltage (V_{T0})	V	-2.0	-2.0	
2	BETA	transconductance parameter (β) transconductance parameter	A/V^2	1.0e-4	1.0e-3	*
3	LAMBDA	channel-length modulation parameter (λ)	1/V	0	1.0e-4	
4	RD	drain ohmic resistance	Ω	0	100	*
5	RS	source ohmic resistance	Ω	0	100	*
6	CGS	zero-bias G-S junction capacitance (C_{gs})	F	0	5pF	*
7	CGD	zero-bias G-D junction capacitance (C_{gs})	F	0	1pF	*
8	PB	gate junction potential	V	1	0.6	
9	IS	gate junction saturation current (I_S)	A	1.0e-14	1.0e-14	*
10	B	doping tail parameter	-	1	1.1	
11	KF	flicker noise coefficient	-	0		
12	AF	flicker noise exponent	-	1		
13	FC	coefficient for forward-bias	-	0.5		

14	TNOM	parameter measurement temperature	oC	27	50	

MOSFETs

General form:

MXXXXXXX ND NG NS NB MNAME <L=VAL> <W=VAL> <AD=VAL> <AS=VAL>

+ <PD=VAL> <PS=VAL> <NRD=VAL> <NRS=VAL> <OFF>

+ <IC=VDS, VGS, VBS> <TEMP=T>

Examples:

M1 24 2 0 20 TYPE1

M31 2 17 6 10 MODM L=5U W=2U

M1 2 9 3 0 MOD1 L=10U W=5U AD=100P AS=100P PD=40U PS=40U

ND, NG, NS, and NB are the drain, gate, source, and bulk (substrate) nodes, respectively. MNAME is the model name. L and W are the channel length and width, in meters. AD and AS are the areas of the drain and source diffusions, in m^2. Note that the suffix U specifies microns (1e-6 m) and P sq-microns (1e-12 m^2). If any of L, W, AD, or AS are not specified, default values are used. The use of defaults simplifies input file preparation, as well as the editing required if device geometries are to be changed. PD and PS are the perimeters of the drain and source junctions, in meters. NRD and NRS designate the equivalent number of squares of the drain and source diffusions; these values multiply the sheet resistance RSH specified on the .MODEL control line for an accurate representation of the parasitic series drain and source resistance of each transistor. PD and PS default to 0.0 while NRD and NRS to 1.0. OFF indicates an (optional) initial condition on the device for dc analysis. The (optional) initial condition specification using IC=VDS, VGS, VBS is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. See the .IC control line for a better and more convenient way to specify transient initial conditions. The (optional) TEMP value is the temperature at which this device is to operate, and overrides the temperature specification on the .OPTION control line. The temperature specification is ONLY valid for level 1, 2, 3, and 6 MOSFETs, not for level 4 or 5 (BSIM) devices.

MOSFET Models (NMOS/PMOS)

SPICE provides four MOSFET device models, which differ in the formulation of the I-V characteristic. The variable LEVEL specifies the model to be used:

LEVEL=1 -> Shichman-Hodges

LEVEL=2 -> MOS2 (as described in [1])

LEVEL=3 -> MOS3, a semi-empirical model(see [1])

LEVEL=4 -> BSIM (as described in [3])

LEVEL=5 -> new BSIM (BSIM2; as described in [5])

LEVEL=6 -> MOS6 (as described in [2])The dc characteristics of the level 1 through level 3 MOSFETs are defined by the device parameters VTO, KP, LAMBDA, PHI and GAMMA. These parameters are computed by SPICE if process parameters (NSUB, TOX, ...) are given, but user-specified values always override. VTO is positive (negative) for enhancement mode and negative (positive) for depletion mode N-channel (P-channel) devices. Charge storage is modeled by three constant capacitors, CGSO, CGDO, and CGBO which represent overlap capacitances, by the nonlinear thin-oxide capacitance which is distributed among the gate, source, drain, and bulk regions, and by the nonlinear depletion-layer capacitances for both substrate junctions divided into bottom and periphery, which vary as the MJ and MJSW power of junction voltage respectively, and are determined by the parameters CBD, CBS, CJ, CJSW, MJ, MJSW and PB. Charge storage effects are modeled by the piecewise linear voltages-dependent capacitance model proposed by Meyer. The thin-oxide charge-storage effects are treated slightly different for the LEVEL=1 model. These voltage-dependent capacitances are included only if TOX is specified in the input description and they are represented using Meyer's formulation.

There is some overlap among the parameters describing the junctions, e.g. the reverse current can be input either as IS (in A) or as JS (in A/m^2). Whereas the first is an absolute value the second is multiplied by AD and AS to give the reverse current of the drain and source junctions respectively. This methodology has been chosen since there is no sense in relating always junction characteristics with AD and AS entered on the device line; the areas can be defaulted. The same idea applies also to the zero-bias junction capacitances CBD and CBS (in F) on one hand, and CJ (in F/m^2) on the other. The parasitic drain and source series resistance can be expressed as either RD and RS (in ohms) or RSH (in ohms/sq.), the latter being multiplied by the number of squares NRD and NRS input on the device line.

A discontinuity in the MOS level 3 model with respect to the KAPPA parameter has been detected (see [10]). The supplied fix has been implemented in Spice3f2 and later. Since this fix may affect parameter fitting, the option "BADMOS3" may be set to use the old implementation (see the section on simulation variables and the ".OPTIONS" line).

SPICE level 1, 2, 3 and 6 parameters:

	name	paraeter	units	default	example
1	LEVEL	model	index	-	1
2	VTO	zero-bias threshold voltage (V_{T0})	V	0.0	1.0
3	KP	transconductance parameter	A/V^2	2.0e-5	3.1e-5
4	GAMMA	bulk threshold parameter (γ)	$V^{1/2}$	0.0	0.37
5	PHI	surface potential ( )	V	0.6	0.65
6	LAMBDA	channel-length modulation (MOS1 and MOS2 only) (λ)	1/V	0.0	0.02
7	RD	drain ohmic resistance	Ω	0.0	1.0
8	RS	source ohmic resistance	Ω	0.0	1.0
9	CBD	zero-bias B-D junction capacitance	F	0.0	20fF
10	CBS	zero-bias B-S junction capacitance	F	0.0	20fF

11	IS	bulk junction saturation current (I_S)	A	1.0e-14	1.0e-15
12	PB	bulk junction potential	V	0.8	0.87
13	CGSO	gate-source overlap capacitance per meter channel width	F/m	0.0	4.0e11
14	CGDO	gate-drain overlap capacitance per meter channel width	F/m	0.0	4.0e-11
15	CGBO	gate-bulk overlap capacitance per meter channel length	F/m	0.0	2.0e-10
16	RSH	drain and source diffusion sheet resistance	Ω/q	0.0	10.0
17	CJ	zero-bias bulk junction bottom cap. per sq-meter of junction area	F/m^2	0.0	2.0e-4
18	MJ	bulk junction bottom grading coeff.	-	0.5	0.5
19	CJSW	zero-bias bulk junction sidewall cap. per meter of junction perimeter	F/m	0.0	1.0e-9
20	MJSW	bulk junction sidewall grading coeff.	-	0.50(level1) 0.33(level2,3)	
21	JS	bulk junction saturation current per sq-meter of junction area	A/m^2		1.0e-8
22	TOX	oxide thickness	meter	1.0e-7	1.0e-7
23	NSUB	substrate doping	$1/cm^3$	0.0	4.0e15
24	NSS	surface state density	$1/cm^2$	0.0	1.0e10
	NFS	fast surface state density	$1/cm^2$	0.0	1.0e10
	TPG	type of gate material: +1 opp. to substrate -1 same as substrate 0 Al gate	-	1.0	
	XJ	metallurgical junction depth	meter	0.0	1μ
	LD	lateral diffusion	meter	0.0	0.8μ

	UO	surface mobility	cm^2/Vs	600	700
	UCRIT	critical field for mobility degradation (MOS2 only)	V/cm	1.0e4	1.0e4
	UEXP	critical field exponent in mobility degradation (MOS2 only)	-	0.0	0.1
	UTRA	transverse field coeff. (mobility) (deleted for MOS2)	-	0.0	0.3
	VMAX	maximum drift velocity of carriers	m/s	0.0	5.0e4
	NEFF	total channel-charge (fixed and mobile) coefficient (MOS2 only)	-	1.0	5.0
	KF	flicker noise coefficient	-	0.0	1.0e-26
36	AF	flicker noise exponent	-	1.0	1.2
37	FC	coefficient for forward-bias depletion capacitance formula	-	0.5	
38	DELTA	width effect on threshold voltage (MOS2 and MOS3)	-	0.0	1.0
39	THETA	mobility modulation (MOS3 only)	1/V	0.0	0.1
40	ETA	static feedback (MOS3 only)	-	0.0	1.0
41	KAPPA	saturation field factor (MOS3 only)	-	0.2	0.5
42	TNOM	parameter measurement temperature	oC	27	50

The level 4 and level 5 (BSIM1 and BSIM2) parameters are all values obtained from process characterization, and can be generated automatically. J. Pierret [4] describes a means of generating a 'process' file, and the program Proc2Mod provided with SPICE3 converts this file into a sequence of BSIM1 ".MODEL" lines suitable for inclusion in a SPICE input file. Parameters marked below with an * in the l/w column also have corresponding parameters with a length and width dependency. For example, VFB is the basic parameter with units of Volts, and LVFB and WVFB also exist and have units of Volt-μmeter The formula

$$P = P_0 + \frac{P_L}{L_{effective}} + \frac{P_W}{W_{effective}}$$

is used to evaluate the parameter for the actual device specified with

$$L_{effective} = L_{input} - DL$$

and

$$W_{effective} = W_{input} - DW$$

Note that unlike the other models in SPICE, the BSIM model is designed for use with a process characterization system that provides all the parameters, thus there are no defaults for the parameters, and leaving one out is considered an error. For an example set of parameters and the format of a process file, see the SPICE2 implementation notes[3].

For more information on BSIM2, see reference [5].

SPICE BSIM (level 4) parameters:

name	parameter	units	l/w
VFB	flat-band voltage	V	*
PHI	surface inversion potential	V	*
K1	body effect coefficient	$V^{1/2}$	*
K2	drain/source depletion charge-sharing coefficient	-	*
ETA	zero-bias drain-induced barrier-lowering coefficient	-	*
MUZ	zero-bias mobility	$cm^2/V\text{-}s$	
DL	shortening of channel	μm	
DW	narrowing of channel	μm	
U0	zero-bias transverse-field mobility degradation coefficient	V^{-1}	*
U1	zero-bias velocity saturation coefficient	μm/V	*
X2MZ	sens. of mobility to substrate bias at $v_{ds}=0$	$cm^2/V^2\text{-}s$	*
X2E	sens. of drain-induced barrier lowering effect to substrate bias	V^{-1}	*
X3E	sens. of drain-induced barrier lowering effect to drain bias at $V_{ds}=V_{dd}$	V^{-1}	*
X2U0	sens. of transverse field mobility degradation effect to substrate bias	V^{-2}	*
X2U1	sens. of velocity saturation effect to substrate bias	μmV^{-2}	*
MUS	mobility at zero substrate bias and at $V_{ds}=V_{dd}$	$cm^2/V^2\text{-}s$	

X2MS	sens. of mobility to substrate bias at $V_{ds}=V_{dd}$	cm^2/V^2-s	*
X3MS	sens. of mobility to drain bias at $V_{ds}=V_{dd}$	cm^2/V^2-s	*
X3U1	sens. of velocity saturation effect on drain bias at $V_{ds}=V_{dd}$	μmV^{-2}	*
TOX	gate oxide thickness	μm	
TEMP	temperature at which parameters were measured	oC	
VDD	measurement bias range	V	
CGDO	gate-drain overlap capacitance per meter channel width	F/m	
CGSO	gate-source overlap capacitance per meter channel width	F/m	
CGBO	gate-bulk overlap capacitance per meter channel length	F/m	
XPART	gate-oxide capacitance-charge model flag	-	
N0	zero-bias subthreshold slope coefficient	-	*
NB	sens. of subthreshold slope to substrate bias	-	*
ND	sens. of subthreshold slope to drain bias	-	*
RSH	drain and source diffusion sheet resistance	Ω/q	
JS	source drain junction current density	A/m^2	
PB	built in potential of source drain junction	V	
MJ	Grading coefficient of source drain junction	-	
PBSW	built in potential of source, drain junction sidewall	V	
MJSW	grading coefficient of source drain junction sidewall	-	
CJ	Source drain junction capacitance per unit area	F/m^2	
CJSW	source drain junction sidewall capacitance per unit length	F/m	

WDF	source drain junction default width	m	
DELL	Source drain junction length reduction	m	

XPART = 0 selects a 40/60 drain/source charge partition in saturation, while XPART=1 selects a 0/100 drain/source charge partition.

ND, NG, and NS are the drain, gate, and source nodes, respectively. MNAME is the model name, AREA is the area factor, and OFF indicates an (optional) initial condition on the device for dc analysis. If the area factor is omitted, a value of 1.0 is assumed. The (optional) initial condition specification, using IC=VDS, VGS is intended for use with the UIC option on the .TRAN control line, when a transient analysis is desired starting from other than the quiescent operating point. See the .IC control line for a better way to set initial conditions.

MESFETs

General form:

ZXXXXXXX ND NG NS MNAME <AREA> <OFF> <IC=VDS, VGS>

Examples:

Z1 7 2 3 ZM1 OFF

MESFET Models (NMF/PMF)

The MESFET model is derived from the GaAs FET model of Statz et al. as described in [11]. The dc characteristics are defined by the parameters VTO, B, and BETA, which determine the variation of drain current with gate voltage, ALPHA, which determines saturation voltage, and LAMBDA, which determines the output conductance. The formula are given by:

$$I_d = \frac{\beta(V_{gs}-V_T)^2}{1+b(V_{gs}-V_T)}\left[1-\left[1-\alpha\frac{V_{ds}}{3}\right]^3\right](1+\lambda V_{ds}) \quad \text{for } 0 < V_{ds} < \frac{3}{\alpha}$$

$$I_d = \frac{\beta(V_{gs}-V_T)^2}{1+b(V_{gs}-V_T)}(1+\lambda V_{ds}) \qquad \text{for } V_{ds} > \frac{3}{\alpha}$$

Two ohmic resistances, RD and RS, are included. Charge storage is modeled by total gate charge as a function of gate-drain and gate-source voltages and is defined by the parameters CGS, CGD, and PB.

	name	parameter	units	default	example	area
1	VTO	pinch-off voltage	V	-2.0	-2.0	
2	BETA	transconductance parameter	A/V^2	1.0e-4	1.0e-3	*
3	B	doping tail extending parameter	1/V	0.3	0.3	*
4	ALPHA	saturation voltage parameter	1/V	2	2	*
5	LAMBDA	channel-length modulation parameter	1/V	0	1.0e-4	
6	RD	drain ohmic resistance	Ω	0	100	*
7	RS	source ohmic resistance	Ω	0	100	*

8	CGS	zero-bias G-S junction capacitance	F	0	5pF	*
9	CGD	zero-bias G-D junction capacitance	F	0	1pF	*
10	PB	gate junction potential	V	1	0.6	
11	KF	flicker noise coefficient	-	0		
12	AF	flicker noise exponent	-	1		
13	FC	coefficient for forward-bias depletion capacitance formula	-	0.5		

ANALYSES AND OUTPUT CONTROL

The following command lines are for specifying analyses or plots within the circuit description file. Parallel commands exist in the interactive command interpreter (detailed in the following section). Specifying analyses and plots (or tables) in the input file is useful for batch runs. Batch mode is entered when either the **-b** option is given or when the default input source is redirected from a file. In batch mode, the analyses specified by the control lines in the input file (e.g. ".ac", ".tran", etc.) are immediately executed (unless ".control" lines exists; see the section on the interactive command interpretor). If the **-r** *rawfile* option is given then all data generated is written to a Spice3 **rawfile**. The rawfile may be read by either the interactive mode of Spice3 or by **nutmeg**; see the previous section for details. In this case, the .SAVE line (see below) may be used to record the value of internal device variables (see Appendix B).

If a rawfile is not specified, then output plots (in "line-printer" form) and tables can be printed according to the **.PRINT**, **.PLOT**, and **.FOUR** control lines, described next. **.PLOT**, **.PRINT**, and **.FOUR** lines are meant for compatibility with Spice2.

SIMULATOR VARIABLES (.OPTIONS)

Various parameters of the simulations available in Spice3 can be altered to control the accuracy, speed, or default values for some devices. These parameters may be changed via the "set" command (described later in the section on the interactive front-end) or via the ".OPTIONS" line:

General form:

.OPTIONS OPT1 OPT2 ... (or OPT=OPTVAL ...)

Examples:

.OPTIONS RELTOL=.005 TRTOL=8

The options line allows the user to reset program control and user options for specific simulation purposes. Additional options for Nutmeg may be specified as well and take effect when Nutmeg reads the input file. Options specified to Nutmeg via the 'set' command are also passed on to SPICE3 as if specified on a .OPTIONS line. See the following section on the interactive command interpreter for the parameters which may be set with a .OPTIONS line and the format of the 'set' 'command. Any combination of the following options may be included, in any order. 'x' (below) represents some positive number.

option	**effect**
ABSTOL=x	resets the absolute current error tolerance of the program. The default value is 1 picoamp.
BADMOS3	Use the older version of the MOS3 model with the "kappa" discontinuity.
CHGTOL=x	resets the charge tolerance of the program. The default value is 1.0e-14.
DEFAD=x	resets the charge tolerance of the program. The default value is 1.0e-14.
DEFAS=x	resets the value for MOS source diffusion area; the default is 0.0.
DEFL=x	resets the value for MOS source diffusion area; the default is 0.0.
DEFW=x	resets the value for MOS channel width; the default is 100.0 micrometer

GMIN=x	resets the value of GMIN, the minimum conductance allowed by the program. The default value is 1.0e-12.
ITL1=x	resets the dc iteration limit. The default is 100.
ITL2=x	resets the dc transfer curve iteration limit. The default is 50.
ITL3=x	resets the lower transient analysis iteration limit. the default value is 4. (Note: not implemented in Spice3).
ITL4=x	resets the transient analysis timepoint iteration limit. the default is 10.
ITL5=x	resets the transient analysis total iteration limit. the default is 5000. Set ITL5=0 to omit this test. (Note: not implemented in Spice3).
KEEPOPINFO	Retain the operating point information when either an AC, Distortion, or Pole-Zero analysis is run. This is particularly useful if the circuit is large and you do not want to run a (redundant) ".OP" analysis.
METHOD=name	Reets the numerical integration method used by SPICE. Possible names are "Gear" or "trapezoidal" (or just "trap"). The default is trapezoidal.
PIVREL=x	resets the relative ratio between the largest column entry and an acceptable pivot value. The default value is 1.0e-3. In the numerical pivoting algorithm the allowed minimum pivot value is determined by EPSREL=AMAX1(PIVREL*MAXVAL, PIVTOL) where MAXVAL is the maximum element in the column where a pivot is sought (partial pivoting).
PIVTOL=x	resets the absolute minimum value for a matrix entry to be accepted as a pivot. The default value is 1.0e-13.
RELTOL=x	resets the relative error tolerance of the program. The default value is 0.001 (0.1%).
TEMP=x	Resets the operating temperature of the circuit. The default value is 27 deg C (300 deg K). TEMP can be overridden by a temperature specification on any temperature dependent instance.
TNOM=x	resets the nominal temperature at which device parameters are measured. The default value is 27 deg C (300 deg K). TNOM can be overridden by a specification on any temperature dependent device model.
TRTOL=x	resets the transient error tolerance. The default value is 7.0. This parameter is an estimate of the factor by which SPICE overestimates the actual truncation error.
TRYTOCOMPACT	Applicable only to the LTRA model. When specified, the simulator tries to condense LTRA transmission lines' past history of input voltages and currents.
VNTOL=x	resets the absolute voltage error tolerance of the program. The default value is 1 microvolt.

In addition, the following options have the listed effect when operating in spice2 emulation mode:

option	effect
ACCT	causes accounting and run time statistics to be printed
LIST	causes the summary listing of the input data to be printed
NOMOD	suppresses the printout of the model parameters
NOPAGE	suppresses page ejects
NODE	causes the printing of the node table.
OPTS	causes the option values to be printed.

INITIAL CONDITIONS

.NODESET: Specify Initial Node Voltage Guesses

General form:

.NODESET V(NODNUM)=VAL V(NODNUM)=VAL ...

Examples:

.NODESET V(12)=4.5 V(4)=2.23

The Nodeset line helps the program find the dc or initial transient solution by making a preliminary pass with the specified nodes held to the given voltages. The restriction is then released and the iteration continues to the true solution. The .NODESET line may be necessary for convergence on bistable or a-stable circuits. In general, this line should not be necessary.

.IC: Set Initial Conditions

General form:

.IC V(NODNUM)=VAL V(NODNUM)=VAL ...

Examples:

.IC V(11)=5 V(4)=-5 V(2)=2.2

The IC line is for setting transient initial conditions. It has two different interpretations, depending on whether the UIC parameter is specified on the .TRAN control line. Also, one should not confuse this line with the .NODESET line. The .NODESET line is only to help dc convergence, and does not affect final bias solution (except for multi-stable circuits). The two interpretations of this line are as follows:

1. When the UIC parameter is specified on the .TRAN line, then the node voltages specified on the .IC control line are used to compute the capacitor, diode, BJT, JFET, and MOSFET initial conditions. This is equivalent to specifying the IC=... parameter on each device line, but is much more convenient. The IC=... parameter can still be specified and takes precedence over the .IC values. Since no dc bias (initial transient) solution is computed before the transient analysis, one should take care to specify all dc source voltages on the .IC control line if they are to be used to compute device initial conditions.

2. When the UIC parameter is not specified on the .TRAN control line, the dc bias (initial transient) solution is computed before the transient analysis. In this case, the node voltages specified on the .IC control line is forced to the desired initial values during the bias solution. During transient analysis, the constraint on these node voltages is removed. This is the

preferred method since it allows SPICE to compute a consistent dc solution.

ANALYSES

.AC: Small-Signal AC Analysis

General form:

.AC DEC ND FSTART FSTOP

.AC OCT NO FSTART FSTOP

.AC LIN NP FSTART FSTOP

Examples:

.AC DEC 10 1 10K

.AC DEC 10 1K 100MEG

.AC LIN 100 1 100HZ

DEC stands for decade variation, and ND is the number of points per decade. OCT stands for octave variation, and NO is the number of points per octave. LIN stands for linear variation, and NP is the number of points. FSTART is the starting frequency, and FSTOP is the final frequency. If this line is included in the input file, SPICE performs an AC analysis of the circuit over the specified frequency range. Note that in order for this analysis to be meaningful, at least one independent source must have been specified with an ac value.

.DC: DC Transfer Function

General form:

.DC SRCNAM VSTART VSTOP VINCR [SRC2 START2 STOP2 INCR2]

Examples:

.DC VIN 0.25 5.0 0.25

.DC VDS 0 10 .5 VGS 0 5 1

.DC VCE 0 10 .25 IB 0 10U 1U

The DC line defines the dc transfer curve source and sweep limits (again with capacitors open and inductors shorted). SRCNAM is the name of an independent voltage or current source. VSTART, VSTOP, and VINCR are the starting, final, and incrementing values respectively. The first example causes the value of the voltage source VIN to be swept from 0.25 Volts to 5.0 Volts in increments of 0.25 Volts. A second source (SRC2) may optionally be specified with associated sweep parameters. In this case, the first source is swept over its range for each value of the second source. This option can be useful for obtaining semiconductor device output characteristics. See the second example circuit description in Appendix A.

.DISTO: Distortion Analysis

General form:

.DISTO DEC ND FSTART FSTOP <F2OVERF1>

.DISTO OCT NO FSTART FSTOP <F2OVERF1>

.DISTO LIN NP FSTART FSTOP <F2OVERF1>

Examples:

```
.DISTO DEC 10 1kHz 100Mhz
.DISTO DEC 10 1kHz 100Mhz 0.9
```

The Disto line does a small-signal distortion analysis of the circuit. A multi-dimensional Volterra series analysis is done using multi-dimensional Taylor series to represent the nonlinearities at the operating point. Terms of up to third order are used in the series expansions.

If the optional parameter F2OVERF1 is not specified, .DISTO does a harmonic analysis - i.e., it analyses distortion in the circuit using only a single input frequency F1, which is swept as specified by arguments of the .DISTO command exactly as in the .AC command. Inputs at this frequency may be present at more than one input source, and their magnitudes and phases are specified by the arguments of the DISTOF1 keyword in the input file lines for the input sources (see the description for independent sources). (The arguments of the DISTOF2 keyword are not relevant in this case). The analysis produces information about the A.C. values of all node voltages and branch currents at the harmonic frequencies 2 F1 and 3 F1, vs. the input frequency F1 as it is swept. (A value of 1 (as a complex distortion output) signifies cos(2 (2 F1) t) at 2 F1 and cos (2 (3 F1) t) at 3 F1, using the convention that 1 at the input fundamental frequency is equivalent to cos(2 F1 t).) The distortion component desired (2 F1 or 3 F1) can be selected using commands in nutmeg, and then printed or plotted. (Normally, one is interested primarily in the magnitude of the harmonic components, so the magnitude of the AC distortion value is looked at). It should be noted that these are the A.C. values of the actual harmonic components, and are not equal to HD2 and HD3. To obtain HD2 and HD3, one must divide by the corresponding A.C. values at F1, obtained from an .AC line. This division can be done using nutmeg commands.

If the optional F2OVERF1 parameter is specified, it should be a real number between (and not equal to) 0.0 and 1.0; in this case, .DISTO does a spectral analysis. It considers the circuit with sinusoidal inputs at two different frequencies F1 and F2. F1 is swept according to the .DISTO control line options exactly as in the .AC control line. F2 is kept fixed at a single frequency as F1 sweeps - the value at which it is kept fixed is equal to F2OVERF1 times FSTART. Each independent source in the circuit may potentially have two (superimposed) sinusoidal inputs for distortion, at the frequencies F1 and F2. The magnitude and phase of the F1 component are specified by the arguments of the DISTOF1 keyword in the source's input line (see the description of independent sources); the magnitude and phase of the F2 component are specified by the arguments of the DISTOF2 keyword. The analysis produces plots of all node voltages/branch currents at the intermodulation product frequencies F1 + F2, F1 - F2, and (2 F1) - F2, vs the swept frequency F1. The IM product of interest may be selected using the setplot command, and displayed with the print and plot commands. It is to be noted as in the harmonic analysis case, the results are the actual AC voltages and currents at the intermodulation frequencies, and need to be normalized with respect to .AC values to obtain the IM parameters.

If the DISTOF1 or DISTOF2 keywords are missing from the description of an independent source, then that source is assumed to have no input at the corresponding frequency. The default values of the magnitude and phase are 1.0 and 0.0 respectively. The phase should be specified in degrees.

It should be carefully noted that the number F2OVERF1 should ideally be an irrational number, and that since this is not possible in practice, efforts should be made to keep the denominator in its fractional representation as large as possible, certainly above 3, for accurate results (i.e., if F2OVERF1 is represented as a fraction A/B, where A and B are integers with no common factors, B should be as large as possible; note that A < B because F2OVERF1 is constrained to be < 1). To illustrate why, consider the cases where F2OVERF1 is 49/100 and 1/2. In a spectral analysis, the outputs produced are at F1 + F2, F1 - F2 and 2 F1 - F2. In the latter case, F1 - F2 = F2, so the result at the F1-F2 component is erroneous because there is the strong fundamental F2 component at the same frequency. Also, F1 + F2 = 2 F1 - F2 in the latter case, and each result is erroneous individually. This problem is not there in the case where F2OVERF1 = 49/100, because F1-F2 = 51/100 F1 < > 49/100 F1 = F2. In this case, there are two very closely spaced frequency components at F2 and F1 - F2. One of the advantages of the Volterra series technique is that it computes distortions at mix frequencies expressed symbolically (i.e. n F1   m F2), therefore one is able to obtain the strengths of distortion components accurately even if the separation between them is very small, as opposed to transient analysis for example. The disadvantage is of course that if two of the mix frequencies coincide, the results are not merged together and presented (though this could presumably be done as a postprocessing step). Currently, the interested user should keep track of the mix frequencies himself or herself and add the distortions at coinciding mix frequencies together should it be necessary.

.NOISE: Noise Analysis

General form:

.NOISE V(OUTPUT <,REF>) SRC (DEC | LIN | OCT) PTS FSTART FSTOP

+ <PTS_PER_SUMMARY>

Examples:

.NOISE V(5) VIN DEC 10 1kHZ 100Mhz

.NOISE V(5,3) V1 OCT 8 1.0 1.0e6 1

The Noise line does a noise analysis of the circuit. OUTPUT is the node at which the total output noise is desired; if REF is specified, then the noise voltage V(OUTPUT) - V(REF) is calculated. By default, REF is assumed to be ground. SRC is the name of an independent source to which input noise is referred. PTS, FSTART and FSTOP are .AC type parameters that specify the frequency range over which plots are desired. PTS_PER_SUMMARY is an optional integer; if specified, the noise contributions of each noise generator is produced every PTS_PER_SUMMARY frequency points.

The .NOISE control line produces two plots - one for the Noise Spectral Density curves and one for the total Integrated Noise over the specified frequency range. All noise voltages/currents are in squared units V^2/Hz and A^2/Hz for spectral density, V^2 and A^2 for integrated noise).

.OP: Operating Point Analysis

General form:

.OP

The inclusion of this line in an input file directs SPICE to determine the dc operating point of the circuit with inductors shorted and capacitors opened. Note: a DC analysis is automatically performed prior to a transient analysis to determine the transient initial conditions, and prior to an AC small-signal, Noise, and Pole-Zero analysis to determine the linearized, small-signal models for nonlinear devices (see the KEEPOPINFO variable above).

.PZ: Pole-Zero Analysis

General form:

```
.PZ NODE1 NODE2 NODE3 NODE4 CUR POL
.PZ NODE1 NODE2 NODE3 NODE4 CUR ZER
.PZ NODE1 NODE2 NODE3 NODE4 CUR PZ
.PZ NODE1 NODE2 NODE3 NODE4 VOL POL
.PZ NODE1 NODE2 NODE3 NODE4 VOL ZER
.PZ NODE1 NODE2 NODE3 NODE4 VOL PZ
```

Examples:

```
.PZ 1 0 3 0 CUR POL
.PZ 2 3 5 0 VOL ZER
.PZ 4 1 4 1 CUR PZ
```

CUR stands for a transfer function of the type (output voltage)/(input current) while VOL stands for a transfer function of the type (output voltage)/(input voltage). POL stands for pole analysis only, ZER for zero analysis only and PZ for both. This feature is provided mainly because if there is a nonconvergence in finding poles or zeros, then, at least the other can be found. Finally, NODE1 and NODE2 are the two input nodes and NODE3 and NODE4 are the two output nodes. Thus, there is complete freedom regarding the output and input ports and the type of transfer function.

In interactive mode, the command syntax is the same except that the first field is PZ instead of .PZ. To print the results, one should use the command 'print all'.

.SENS: DC or Small-Signal AC Sensitivity Analysis

General form:

```
.SENS OUTVAR
.SENS OUTVAR AC DEC ND FSTART FSTOP
.SENS OUTVAR AC OCT NO FSTART FSTOP
.SENS OUTVAR AC LIN NP FSTART FSTOP
```

Examples:

```
.SENS V(1,OUT)
.SENS V(OUT) AC DEC 10 100 100k
.SENS I(VTEST)
```

The sensitivity of OUTVAR to all non-zero device parameters is calculated when the SENS analysis is specified. OUTVAR is a circuit variable (node voltage or voltage-source branch current). The first form calculates sensitivity of the DC operating-point value of OUTVAR. The second form calculates sensitivity of the AC values of OUTVAR. The parameters listed for AC sensitivity are the same as in an AC analysis (see ".AC" above). The output values are in dimensions of change in output per unit change of input (as opposed to percent change in output or per percent change of input).

.TF: Transfer Function Analysis

General form:

```
.TF OUTVAR INSRC
```

Examples:

.TF V(5, 3) VIN

.TF I(VLOAD) VIN

The TF line defines the small-signal output and input for the dc small-signal analysis. OUTVAR is the small-signal output variable and INSRC is the small-signal input source. If this line is included, SPICE computes the dc small-signal value of the transfer function (output/input), input resistance, and output resistance. For the first example, SPICE would compute the ratio of V(5, 3) to VIN, the small-signal input resistance at VIN, and the small-signal output resistance measured across nodes 5 and 3.

.TRAN: Transient Analysis

General form:

.TRAN TSTEP TSTOP <TSTART <TMAX>>

Examples:

.TRAN 1NS 100NS

.TRAN 1NS 1000NS 500NS

.TRAN 10NS 1US

TSTEP is the printing or plotting increment for line-printer output. For use with the post-processor, TSTEP is the suggested computing increment. TSTOP is the final time, and TSTART is the initial time. If TSTART is omitted, it is assumed to be zero. The transient analysis always begins at time zero. In the interval <zero, TSTART>, the circuit is analyzed (to reach a steady state), but no outputs are stored. In the interval <TSTART, TSTOP>, the circuit is analyzed and outputs are stored. TMAX is the maximum stepsize that SPICE uses; for default, the program chooses either TSTEP or (TSTOP-TSTART)/50.0, whichever is smaller. TMAX is useful when one wishes to guarantee a computing interval which is smaller than the printer increment, TSTEP.

UIC (use initial conditions) is an optional keyword which indicates that the user does not want SPICE to solve for the quiescent operating point before beginning the transient analysis. If this keyword is specified, SPICE uses the values specified using IC=... on the various elements as the initial transient condition and proceeds with the analysis. If the .IC control line has been specified, then the node voltages on the .IC line are used to compute the initial conditions for the devices. Look at the description on the .IC control line for its interpretation when UIC is not specified.

BATCH OUTPUT

.SAVE Lines

General form:

.SAVE vector vector vector ...

Examples:

.SAVE i(vin) input output

.SAVE @m1[id]

The vectors listed on the .SAVE line are recorded in the rawfile for use later with spice3 or nutmeg (nutmeg is just the data-analysis half of spice3, without the ability to simulate). The standard vector names are accepted. If no .SAVE line is given, then the default set of vectors are saved (node voltages and voltage source branch currents). If .SAVE lines are given, only those vectors specified are saved. For more discussion on internal device data, see Appendix B. See also the section on the interactive command interpretor for information on how to use the rawfile.

.PRINT Lines

General form:

.PRINT PRTYPE OV1 <OV2 ... OV8>

Examples:

.PRINT TRAN V(4) I(VIN)

.PRINT DC V(2) I(VSRC) V(23, 17)

.PRINT AC VM(4, 2) VR(7) VP(8, 3)

The Print line defines the contents of a tabular listing of one to eight output variables. PRTYPE is the type of the analysis (DC, AC, TRAN, NOISE, or DISTO) for which the specified outputs are desired. The form for voltage or current output variables is the same as given in the previous section for the **print** command; Spice2 restricts the output variable to the following forms (though this restriction is not enforced by Spice3):

V(N1<,N2>)

specifies the voltage difference between nodes N1 and N2. If N2 (and the preceding comma) is omitted, ground (0) is assumed. See the **print** command in the previous section for more details. For compatibility with spice2, the following five additional values can be accessed for the ac analysis by replacing the "V" in V(N1,N2) with:

VR - real part

VI - imaginary part

VM - magnitude

VP - phase

VDB - 20 log10(magnitude)

I(VXXXXXXX) specifies the current flowing in the independent voltage source named VXXXXXXX. Positive current flows from the positive node, through the source, to the negative node. For the ac analysis, the corresponding replacements for the letter I may be made in the same way as described for voltage outputs.Output variables for the noise and distortion analyses have a different general form from that of the other analyses.

There is no limit on the number of .PRINT lines for each type of analysis.

.PLOT Lines

General form:

.PLOT PLTYPE OV1 <(PLO1, PHI1)> <OV2 <(PLO2, PHI2)> ... OV8>

Examples:

.PLOT DC V(4) V(5) V(1)

.PLOT TRAN V(17, 5) (2, 5) I(VIN) V(17) (1, 9)

.PLOT AC VM(5) VM(31, 24) VDB(5) VP(5)

.PLOT DISTO HD2 HD3(R) SIM2

.PLOT TRAN V(5, 3) V(4) (0, 5) V(7) (0, 10)

The Plot line defines the contents of one plot of from one to eight output variables. PLTYPE is the type of analysis (DC, AC, TRAN, NOISE, or DISTO) for which the specified outputs are desired. The syntax for the OVI is identical to that for the .PRINT line and for the **plot** command in the interactive mode.

The overlap of two or more traces on any plot is indicated by the letter X.

When more than one output variable appears on the same plot, the first variable specified is printed as well as plotted. If a printout of all variables is desired, then a companion .PRINT line should be included.

There is no limit on the number of .PLOT lines specified for each type of analysis.

.FOUR: Fourier Analysis of Transient Analysis Output

General form:

.FOUR FREQ OV1 <OV2 OV3 ...>

Examples:

.FOUR 100K V(5)

The Four (or Fourier) line controls whether SPICE performs a Fourier analysis as a part of the transient analysis. FREQ is the fundamental frequency, and OV1, ..., are the output variables for which the analysis is desired. The Fourier analysis is performed over the interval <TSTOP-period, TSTOP>, where TSTOP is the final time specified for the transient analysis, and period is one period of the fundamental frequency. The dc component and the first nine harmonics are determined. For maximum accuracy, TMAX (see the .TRAN line) should be set

Circuit Simulation Analyses

What is a circuit analysis?

AutoTRAX can perform the following circuit analyses:

- AC small-signal analysis
- DC operating Point analysis
- Distortion analysis
- Fourier analysis
- Noise analysis
- Pole-Zero analysis
- Small-Signal distortion analysis
- Sensitivity analysis
- Sweep analysis
- Transient analysis

The results are displayed in the analysis result dialog.

Creating an AC analysis

Select **<Simulate/Analysis/AC...>** from the main menu.
The results are displayed in the analysis result dialog.

Creating a DC Operating point analysis

Select **<Simulate/Analysis/DC Operating Point>** from the main menu.
The results are displayed in the analysis result dialog.

Creating a distortion analysis

Select **<Simulate/Analysis/Distortion...>** from the main menu.
The results are displayed in the analysis result dialog.

Creating a Fourier analysis

Select **<Simulate/Analysis/Fourier...>** from the main menu.
The results are displayed in the analysis result dialog.

Creating a noise analysis

Select **<Simulate/Analysis/Noise...>** from the main menu.
The results are displayed in the analysis result dialog.

Creating a Pole-Zero analysis

Select **<Simulate/Analysis/Pole Zero...>** from the main menu.

The results are displayed in the analysis result dialog.

Creating a sensitivity analysis

Select **<Simulate/Analysis/Sensitivity...>** from the main menu.

The results are displayed in the analysis result dialog.

Creating a sweep analysis

Select **<Simulate/Analysis/Sweep...>** from the main menu.

The results are displayed in the analysis result dialog.

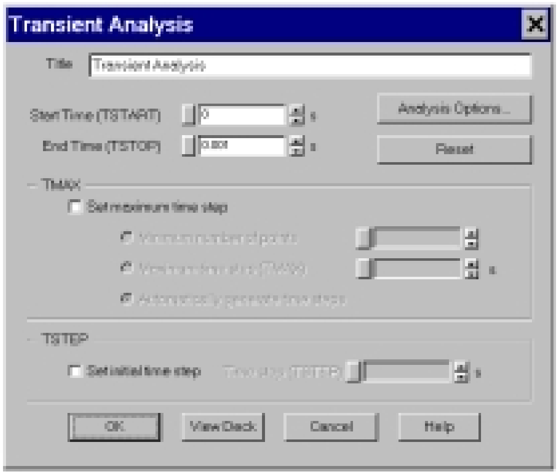
Creating a transient analysis

Select **<Simulate/Analysis/Transient...>** from the main menu.

Title. The title of the analysis.

Analysis Option. Click to set the simulator options.

Reset. Click to reset all parameters.



TSTEP is the printing or plotting increment for line-printer output. For use with the post-processor, TSTEP is the suggested computing increment. **TSTOP** is the final time, and TSTART is the initial time. If **TSTART** is omitted, it is assumed to be zero. The transient analysis always begins at time zero. In the interval <zero, TSTART>, the circuit is analyzed (to reach a steady state), but no outputs are stored. In the interval <TSTART, TSTOP>, the circuit is analyzed and outputs are stored. **TMAX** is the maximum step size that SPICE uses; for default, the program chooses either TSTEP or (TSTOP-TSTART)/50.0, whichever is smaller. TMAX is useful when one wishes to guarantee a computing interval which is smaller than the printer increment, TSTEP.

OK. Click to perform the analysis.

View Deck. Click to view the spice source deck for this analysis.

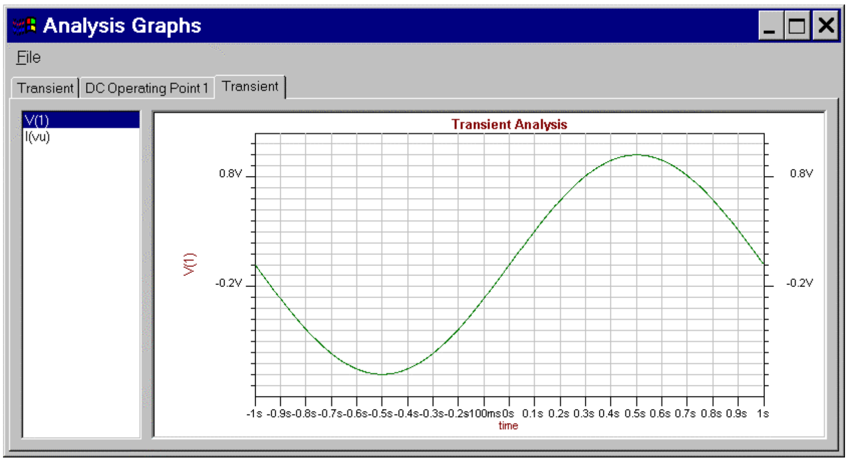
Cancel. Click to cancel the analysis.

Help. Displays this help topic.

The results are displayed in the analysis result dialog.

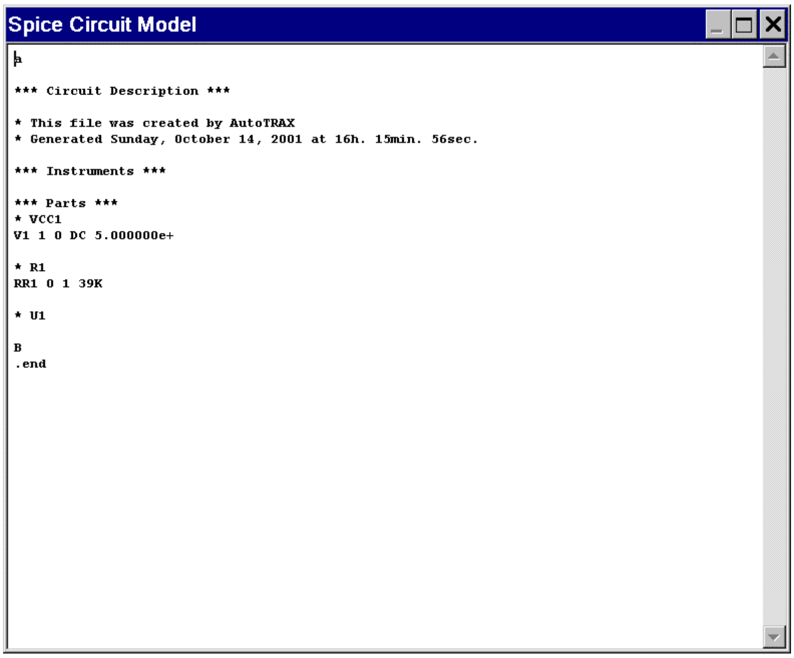
Viewing the analysis results

Analysis results are displayed in the tabbed dialog as shown below.

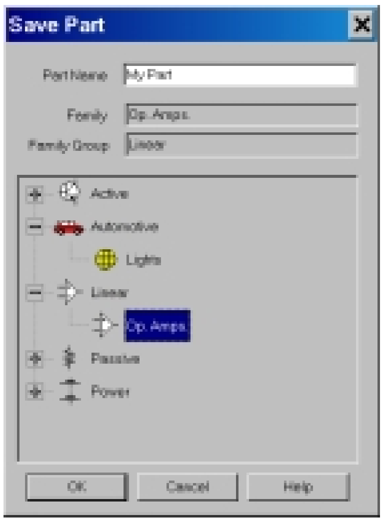


Viewing the design Spice model

Select **<Simulate/Spice Model...>** from the main menu. The **Spice Deck** dialog box will be displayed containing the SPICE source deck.



Creating a part



To create a new part, select **<File/New>** from the main menu.

The **New dialog box** shown below will appear. Select **Part** to create a new part.

The **Save Part Dialog** shown below will appear.

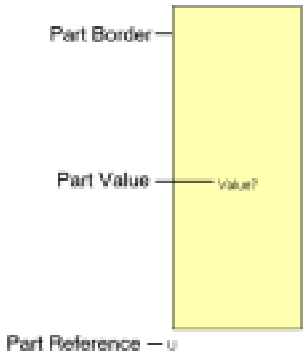
Select the family that you want the part to be in and enter the Part Name. This will save a basic part in the database.

Next the Icon Editor will appear. You need to associate an icon with the part.

Finally you will be asked to associate a PCB footprint with the part. You do not need to assign one or you can assign one later.

You will now see a basic part with:

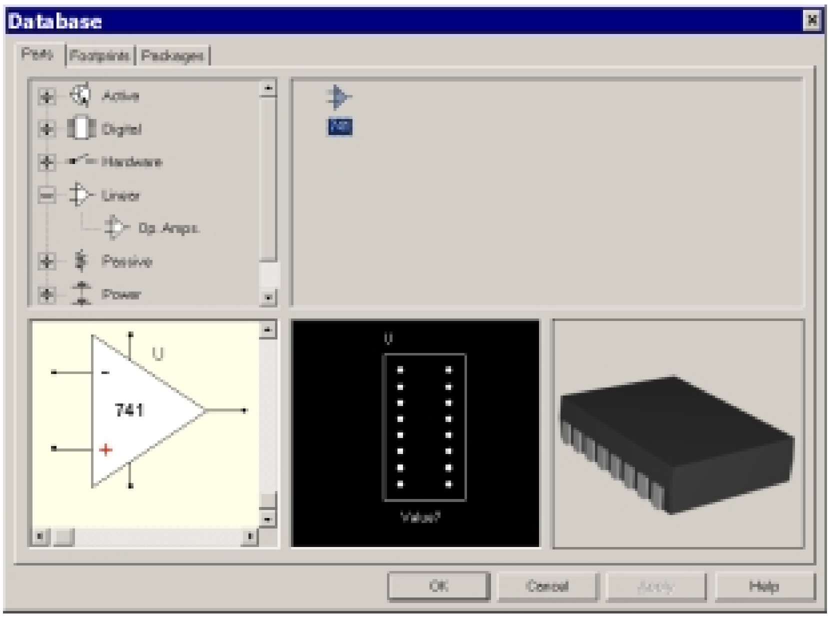
- A part border.
- A part reference.
- A part value.



The Part Database

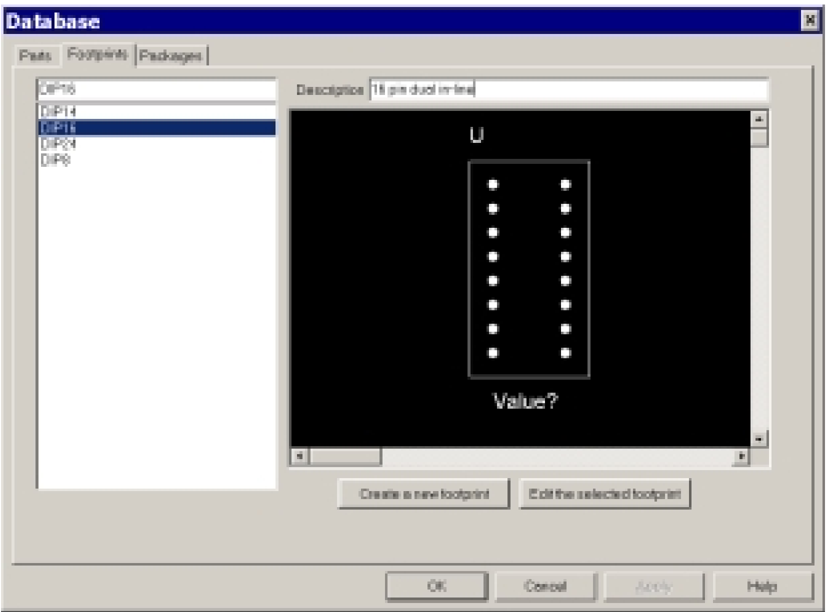
Editing the part Database

Select **<Tools/Database>** from the main menu. The Part Database dialog shown below will appear.

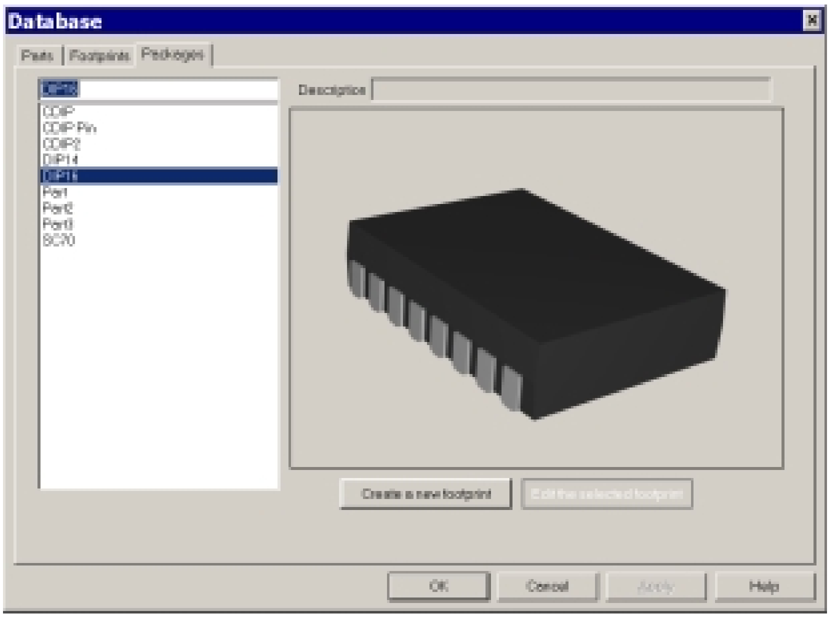


The database dialog consists of 3 tabbed dialogs. The first is the parts dialog shown above. The other two are shown.

The Footprint tab.



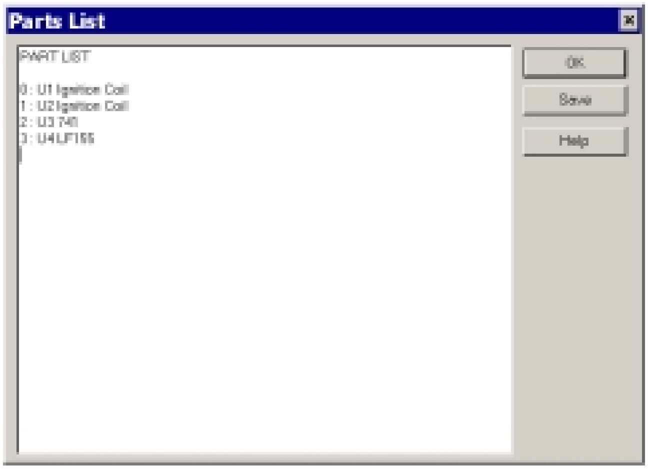
The 3D Package Tab.



Generating a parts list

To generate a parts list select **<Tools/Parts List...>** from the main menu.

A typical partlist list is shown below.



The Design Library

What is the design library

The design library is a hierarchical depository for your design.

You can add and manage designs and drag them onto new design.

For more information on using the design library, see the Design Library toolbar.

PCB Design

Overview

Completing the PCB layout is the first part of the process that culminates in the fabrication and assembly of your PCB.

When you start designing, you should have a clear idea of the output requirements of the PCB technology and production methods you will be using.

To begin the PCB design phase of a project, create a new PCB document in your design database (see the Adding a new document or folder to a design topic in the Links section below).

Before bringing design information from the schematic, you should first create the mechanical and electrical board outline for your board, and configure the layer stack.

The mechanical outline defines the physical shape and size of the board, and also includes items such as dimension detail, photo tool targets and other company and fabrication specific information.

The electrical board outline defines the routing and component placement limits of the board. This is done by adding a PCB border.

AutoTRAX includes a powerful Board Wizard that guides you through the complete process of creating a new PCB design. The Wizard includes a number of pre-defined board templates, and also allows you to create your own templates.

Creating the mechanical definition of a PCB

A PCB board will depend on company and manufacturer requirements. Generally, manufacturers require board corner markers, a reference hole location and external dimensions as a minimum. Contact your PCB manufacturer for details.

To create the mechanical definition of the PCB, you place tracks, dimensions and other PCB design objects on the four Mechanical layers available.

In general it is advisable to use one mechanical layer to draw the physical outline of the board, and then place dimensions, alignment markers, header information, etc. on the other mechanical layers.

Completing the PCB layout is the first part of the process that culminates in the fabrication and assembly of your PCB.

The link between your design and the finished board are the print, Gerber and NC drill fabrication files, as well as the Bill of Materials, and pick and place assembly files.

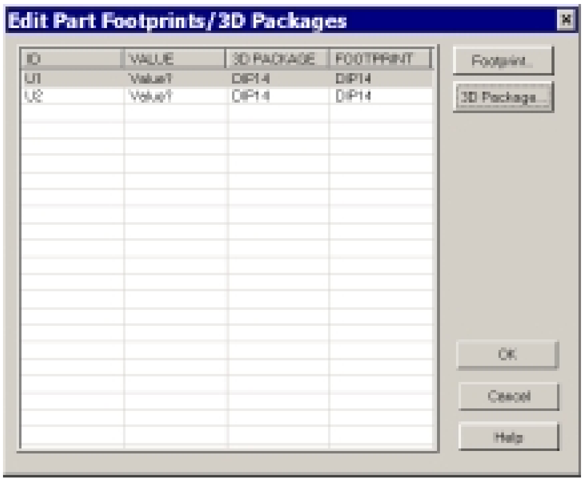
When you start designing, you should have a clear idea of the output requirements of the PCB technology and production methods you will be using.

If you intend to use the services of a plotting bureau or PCB manufacturer take the time to consult with them before you start generating artwork. Bureaus and manufacturers often have specific requirements that must be reflected in the files or artwork that you submit. For example, you may wish to either "step and repeat" or panelize your Gerber files for efficient quantity fabrication.

To accomplish this, you have to know the film size accepted by the photoplotter, clearance requirements, etc, as well as the manufacturing tolerances involved. Planning for Numeric Control (NC) drilling requires similar consideration.

In some instances, the bureau or fabrication facility will prefer to work directly with "raw" Gerber files (or even PCB files) rather than panelized Gerber files. Understanding these requirements will help you to plan the entire design process for efficient and trouble-free completion.

Check with your manufacturer to see if they can accept AutoTRAX PCB files directly. Many manufacturers will be able to produce a PCB directly from an AutoTRAX binary PCB file. The Gerber file output setup topic also includes a number of questions that you should discuss with the manufacturer.

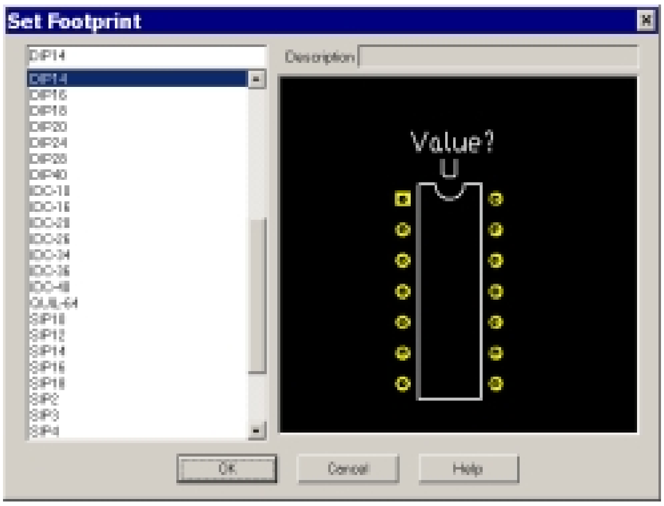


Setting the part footprints

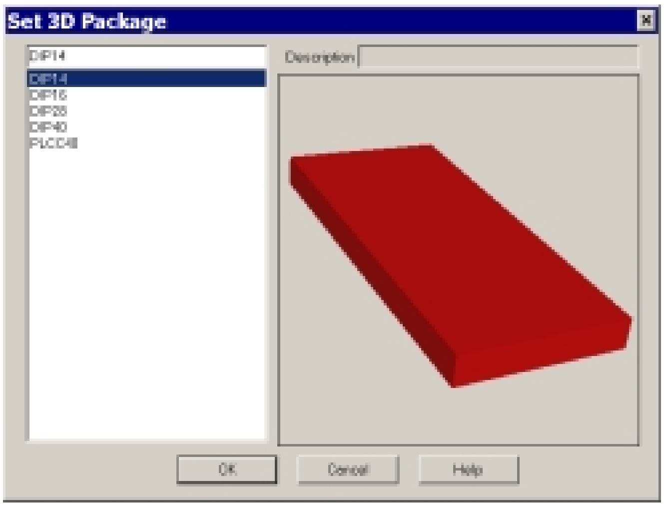
In the schematic, to view and set the part footprints on a schematic select **<Edit/Edit Footprints...>** from the main menu.

Click the left mouse button on the footprint name to select it.

Click the **Footprint...** button to change the footprint.



Select an appropriate footprint and click **OK** to accept your changes or click the **Cancel** button to ignore them.

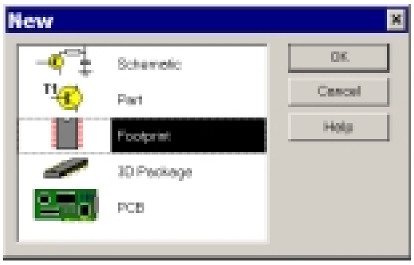


Click the **3D Package** to change the package

Select an appropriate package and click **OK** to accept your changes or click the **Cancel** button to ignore them.

Click the **Help** button to get this help topic.

Creating a Footprint

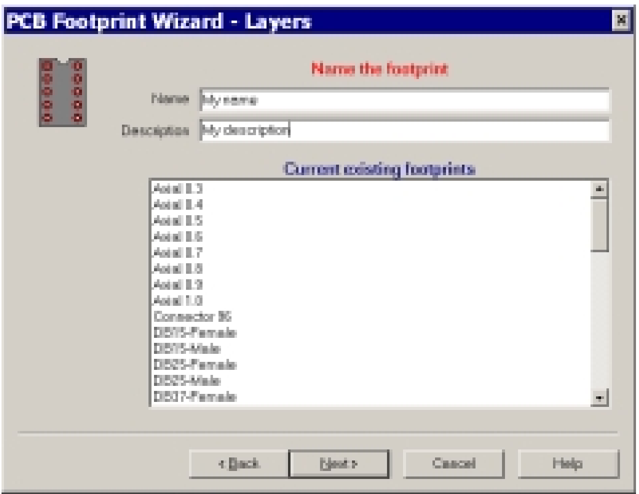


To create a new footprint, select **<File/New>** from the main menu.

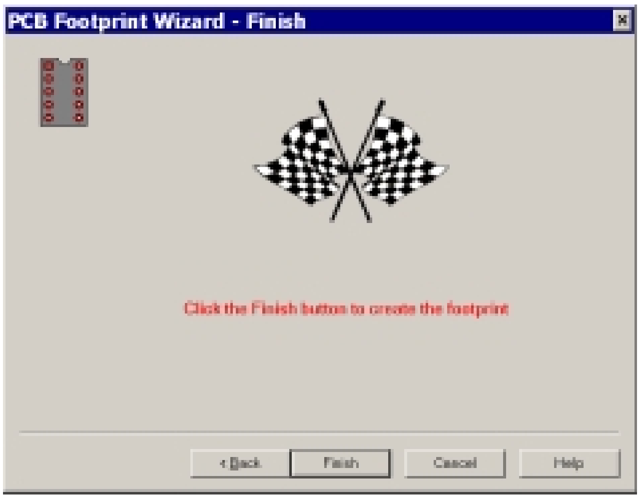
The **New dialog box** shown below will appear. Select **Footprint** to create a new part.

The PCB Footprint wizard will start. The first page of the wizard is shown below. Click on the **Next** button to start.

You will now be asked for the name of the new footprint and an optional description. The second wizard page is shown below.



Enter the name in the **Name** field and enter an optional description in the **Description** field for the footprint. A list of current footprints are displayed in the list box below the Description field. Click the **Next** button to continue. This will display the final wizard page.



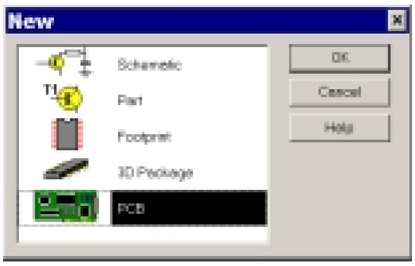
Click the Finish button to finally create the footprint.

NOTE: Always build SMT footprints on the top layer.

Creating a PCB

You can create either a blank PCB or one from a schematic design.

Creating a blank PCB



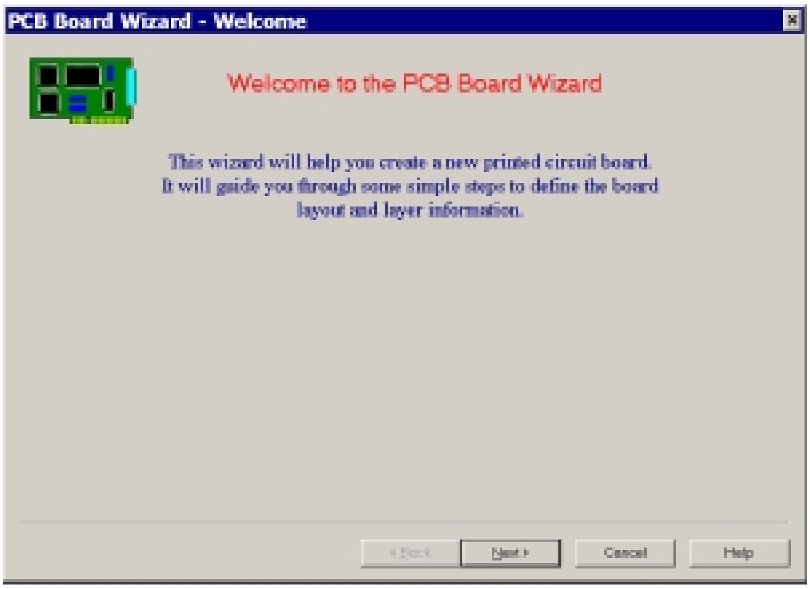
To create a new PCB, select **<File/New>** from the main menu.

The **New dialog box** shown below will appear. Select **PCB** to create a new part.

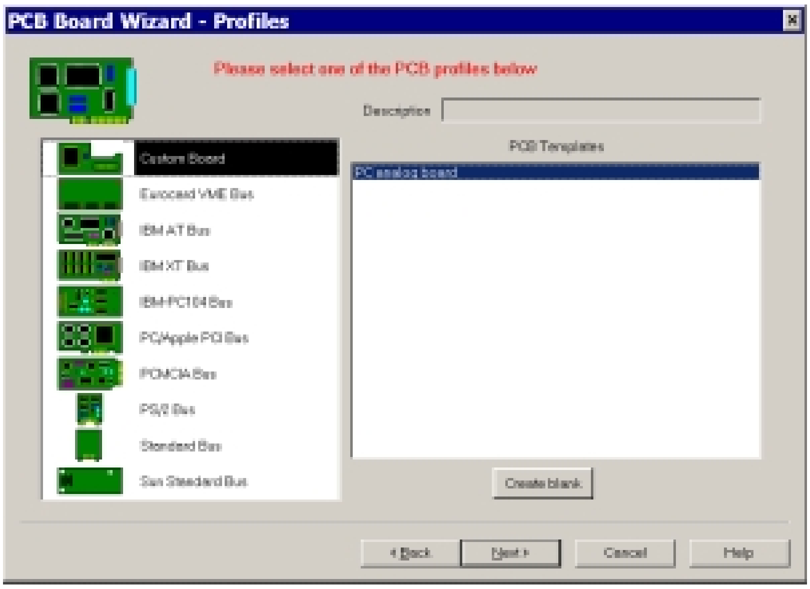
This will start the PCB Design Wizard.

The PCB Design Wizard

The PCB Design wizard will guide you through the process of creating a PCB. The first page of the wizard is shown below.



Click on the **Next** button to see the next page. (Shown below).



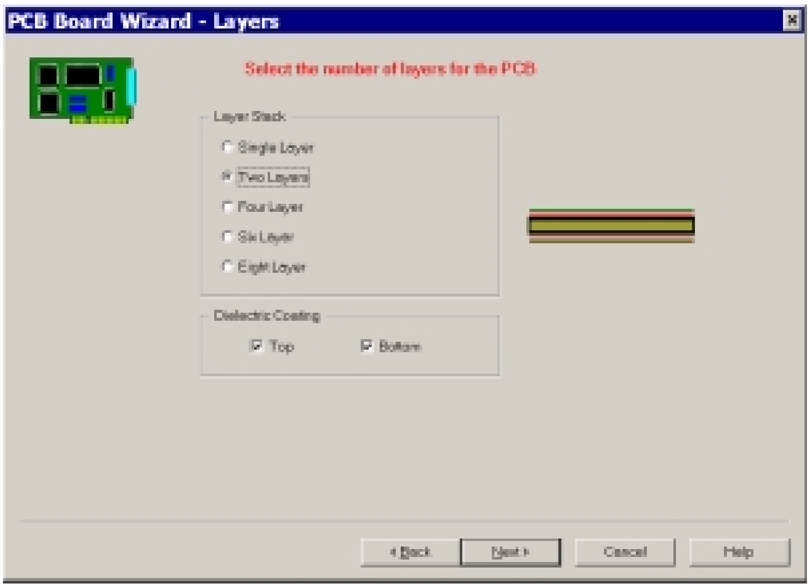
AutoTRAX maintains a database of predefined PCB board templates. These are grouped into 10 families.

- Custom Boards
- Eurocard VME Bus
- IBM AT Bus
- IBM XT Bus
- IBM-PC104 Bus
- PC/Apple PCI Bus
- PCMCIA Bus
- PS/2 Bus
- Standard Bus
- Sun Standard Bus

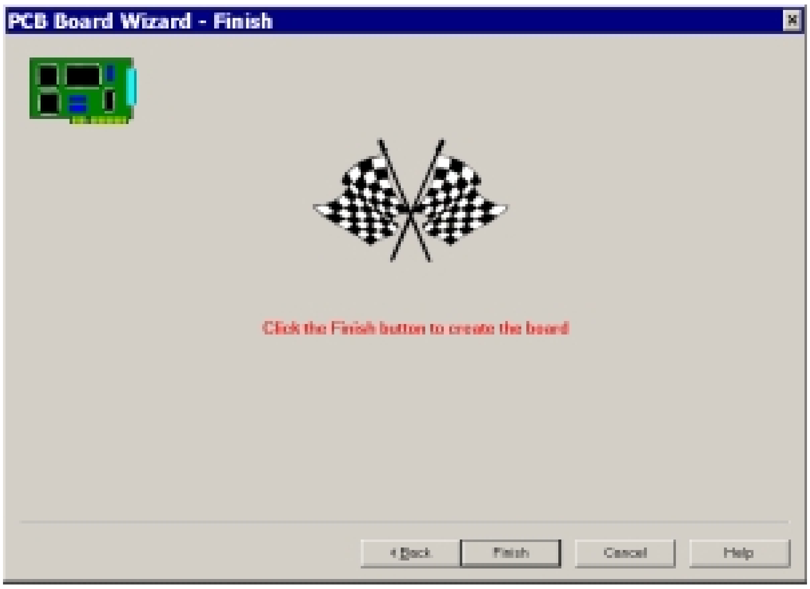
Select a board family. Once selected, a list of available boards are displayed in the list box on the right. Click on the name of the PCB template you wish to use and then click the Next button to continue to the next page.

Alternatively, if you do not wish to use one of the templates, click on the **Create Blank** button to create a blank PCB.

You will now be asked to define the layer structure. The Layers page shown below will be displayed.



Selected the number of layers and click the **Next** button. This will display the last page of the wizard.



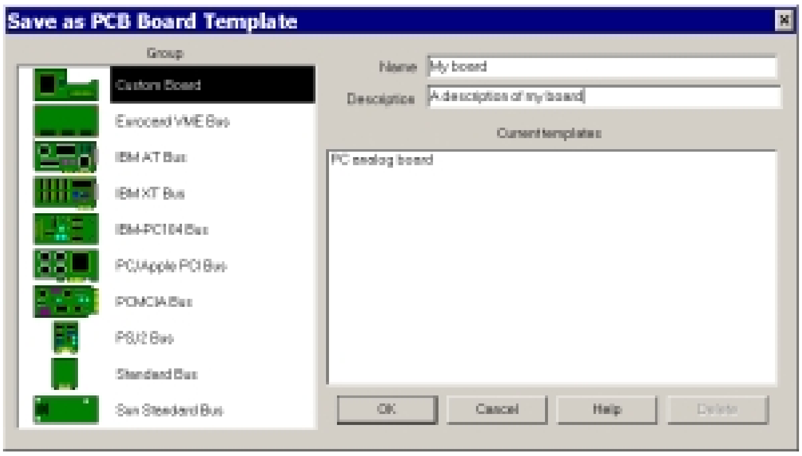
Click the **Finish** button to create the PCB.

Creating and deleting PCB board templates

AutoTRAX maintains a database of predefined PCB board templates. These are grouped into 10 families.

- Custom Boards
- Eurocard VME Bus
- IBM AT Bus
- IBM XT Bus
- IBM-PC104 Bus

- PC/Apple PCI Bus
- PCMCIA Bus
- PS/2 Bus
- Standard Bus
- Sun Standard Bus



To create a PCB template.

1. Create a new PCB board.
2. Add a PCB border etc.
3. Click on the group you wish to save the current PCB in.
4. Enter the name of the board template in the **Name** field.
5. Optionally enter a description for the board template in the **Description** field.
6. Click the **OK** button to add it.

To delete an existing PCB template.

1. Select the group containing the template.
2. Select the name in the current templates list.
3. Click on the **Delete** button.

Setting the PCB outline

A PCB outline is a closed polygon that defines the edge limits of the PCB board.

Only 1 PCB outline can exist, so if there is already an outline, it will be automatically deleted.

1. Select <**Add\Border**> from the main menu.
2. Move the mouse to a corner of the outline.
3. Click the left mouse button to start the outline, and then drag the mouse until the outline edge is the required length and direction. Click the left mouse button again to start another edge. Again drag the mouse until the new edge is the required length and direction. Keep on repeating this until the outline is complete.
4. Finally double click the left mouse button or click the right mouse button to end.

You can press the **ESC** key or the right mouse button to cancel the outline creation command.

You can restrict edges to vertical, horizontal of 45º by turning ortho on.

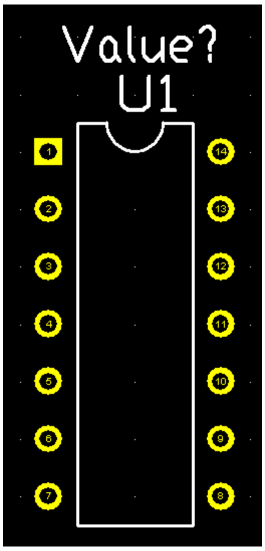
Adding a layout room

A layout room is a closed polygon that defines an area for the automatic layout of footprints.

- Select **<Add/\Border>** from the main menu.
- Move the mouse to a corner of the room.
- Click the left mouse button to start the room, and then drag the mouse until the room edge is the required length and direction. Click the left mouse button again to start another edge. Again drag the mouse until the new edge is the required length and direction. Keep on repeating this until the room is complete.
- Finally double click the left mouse button or click the right mouse button to end.

You can press the **ESC** key or the right mouse button to cancel the room creation command.

You can restrict edges to vertical, horizontal of 45º by turning ortho



Adding a footprint

A Footprint is collection of pads and silkscreen graphics representing a physical component. It defines the electrical points to route to.

Select **<Add\Component>** from the main menu.

Locking a footprint

To lock/unlock one or more footprint(s) and thus prevent its(their) movement.

Select the footprint(s).

Select **<Edit\Lock>** from the main menu.

To lock/unlock all the footprints.

Select **<Edit\Lock All>** from the main menu.

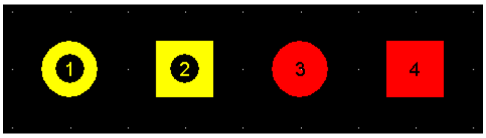
Adding pads

One of the most points in creating a new footprint is positioning the pads that will be used to solder the component to the PCB. These must be placed in exactly the right positions to correspond to the pins on the physical device.

To add a pad, Select **<Add\Round Pad>** or **<Add\Square Pad>** from the main menu.

Pads can be either round or rectangular and either position on only the top or bottom or the PCB and on both sides with a drilled hole.

Below are 4 pads. The first 2 are through-hole pads and the second 2 are SMT pads (only 1 side with no hole)



Always build SMT footprints on the top layer.

Adding a net

To add one or more nets:

1. Select **<Add\Net>** from the main menu.
2. Move the mouse to the start of the net.
3. Click the left mouse button on a pad to start the net
4. Drag the mouse to another pad and click the left mouse button again.
5. Repeat 2 and 4 to add more nets or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the net creation command.

Adding a keepout region

A keepout is a closed "no go" polygonal region within which components and/or tracks are to be excluded.

- Select **<Add\Keepout>** from the main menu.
- Move the mouse to a corner of the keepout.
- Click the left mouse button to define the first edge of the keepout, and then drag the mouse until the keepout edge is the required length and direction. Click the left mouse button again to start another edge. Again drag the mouse until the new edge is the required length and direction. Keep on repeating this until the keepout is complete.
- Finally double click the left mouse button or click the right mouse button to end.

You can press the **ESC** key or the right mouse button to cancel the keepout creation command.

You can restrict edges to vertical, horizontal of 45º by turning ortho on.

Adding a trace

To add a trace (electrical track)

- Select **<Add\Trace>** from the main menu.
- Move the mouse to the start of the trace.
- Click the left mouse button on a pad to start the trace
- Drag the mouse to another position and click the left mouse button again.
- If you move the mouse over another pad, then the trace will be finished.
- Repeat 2 and 4 to add more trace or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the net creation command.

Adding a coordinate marker

A coordinate marker is a point with associated text, defining the position if the point.

1. Select **<Add\Coordinate>** from the main menu.
2. Move the mouse to the position of the point and click the left mouse button.
3. Repeat 3 to add more coordinate markers or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the coordinate marker creation command.

Adding a dimension

Dimension objects place dimensioning information on the current PCB layer. The dimension value is the distance between the start and end markers.

1. Select **<Add\Dimension>** from the main menu.
2. Move the mouse to the first point and click the left mouse button.
3. Move the mouse to the second point and click the left mouse button.
4. Repeat 2 and3 to add more dimensions or right click to return to normal mouse mode.

You can press the **ESC** key or the right mouse button to cancel the dimension creation command.

Back annotation

Select **<Tools\Back Annotate>** from the main menu.

Layout

Laying out components

You can layout components automatically or intractively/

Automatic layout

Select **<Layout\Auto Layout>** from the main menu.

Interactive layout

Select **<Layout\Interactive Layout>** from the main menu.

Routing

Automatic routing

Routing all the board

To automatically route all the PCB select **<Route\All >** from the main menu.

Routing 1 or more components

To automatically route all nets connected to 1 or more components:

1. Select the component(s)
2. Select **<Route\Component>** from the main menu.

Routing 1 or more nets

To automatically route 1 or more nets:

1. Select the net(s)
2. Select **<Route\Net>** from the main menu.

Manual routing

See adding a net and adding a trace.

Unrouting

Unrouting all the board

To automatically route all the PCB select **<Route\Unroute\All>** from the main menu.

Unrouting 1 or more components

To automatically unroute all traces connected to 1 or more components:

1. Select the component(s)
2. Select **<Route\Unroute\Component>** from the main menu.

Unrouting 1 or more nets

To automatically unroute 1 or more traces:

1. Select the trace(s)
2. Select **<Route\Unroute\Net>** from the main menu.

CAM

Exporting an AutoCAD DXF file

The AutoCAD DXF (Drawing eXchange Format) format is the native vector file formats of Autodesk's AutoCAD CAD application.

DXF is probably one of the most widely supported vector formats in the world today. DXF is rich in features, including: support for 2D and 3D objects, curves, and text.

To export a schematic, part, footprint or a PCB to an AutoCAD DXF file select **<File\Export to DXF>** from the main menu.

AutoTRAX exports to AutoCAD version Release 11 and later.

NOTE: *Due to the limitations of AutoCAD:*

1. *Fonts are only available as AutoCAD stick fonts.*
2. *No pattern or bitmap fills are available.*
3. *No line style or line endstyles are available.*
4. *Curve and polygons are not filled.*

Photoplotting

If you intend to use the services of a plotting bureau or PCB manufacturer take the time to consult with them before you start generating artwork. Bureaus and manufacturers often have specific requirements that must be reflected in the files or artwork that you submit. For example, you may wish to either "step and repeat" or panelize your Gerber files for efficient quantity fabrication.

Once the PCB design process is complete and the design has passed all the design rule checks, the Gerber files are generated, one for each layer needed in the fabrication process. The Gerber language is the standard language format used to transfer PCB layout data from the PCB design software to the phototool creation process. These Gerber files are then sent to the manufacturer, who loads them into a photoplotting machine and creates the phototools.

Each phototool is created by exposing the film to build up the image required for that layer. The information needed to form the image includes the shape and size of the objects on that layer, and the coordinates of these objects. The shapes are specified in the Gerber file as apertures, and typically these apertures are created from the board and included in each Gerber file, when they are referred to as embedded apertures. If the apertures are not embedded then they must be stored in a separate aperture file and supplied with the Gerber files.

To accomplish this, you have to know the film size accepted by the photoplotter, clearance requirements, etc, as well as the manufacturing tolerances involved. Planning for Numeric Control (NC) drilling requires similar consideration.

In some instances, the bureau or fabrication facility will prefer to work directly with "raw" Gerber files (or even PCB files) rather than panelized Gerber files. Understanding these requirements will help you to plan the entire design process for efficient and trouble-free completion.

Check with your manufacturer to see if they can accept AutoTRAX files directly. Many manufacturers will be able to produce a PCB directly from an AutoTRAX file. The Gerber file output setup topic also includes a number of questions that you should discuss with the manufacturer.

Raster plotters do not use a system of fixed apertures. They read the Gerber file, storing an "image" of the whole plot, which is then scanned onto the film, line-by-line, not unlike a television image. Raster photoplotters can synthesize a virtually unlimited variety of different apertures, providing a great amount of flexibility to the designer.

Some photoplotters use the Postscript language. Photoplot files for these devices are prepared using an appropriate Postscript driver. For information about Postscript printing, see the Postscript Printing Tips topic later in this chapter.

Contact your photoplot bureau before generating any photoplots. Matching available plotting options at the edit level can save considerable time and expense when generating Gerber phototools.

Gerber files

A PCB is fabricated as a series of layers that the manufacturer assembles into a board through a variety of chemical and mechanical processes. To fabricate each physical layer in the PCB the manufacturer uses an image of that layer -- this image is referred to as a phototool. A phototool is a piece of clear film, with black lines, circles and other shapes forming exactly the same patterns as the content of that layer in AutoTRAX.

Gerber data is a simple, generic means of transferring printed circuit board information to a wide variety of devices that convert the electronic PCB data to artwork produced by a photoplotter. Virtually every PCB CAD system generates Gerber data because all photoplotters read it. It is a software structure consisting of X,Y coordinates supplemented by commands that define where the PCB image starts, what shape it will take, and where it ends. In addition to the coordinates, Gerber data contains aperture information, which defines the shapes and sizes of lines, holes, and other features.

Gerber Format, which is the format in which Gerber data is expressed, actually is a family of data formats that are subsets of EIA Standard RS-274D. Extended Gerber Format, which is also called RS-274X, provides enhancements that handle polygon fill codes, positive/negative image compositing, and custom apertures, and other features. RS-274X also encapsulates the aperture list in the header of the Gerber data file and therefore allows files to pass from one system to another without the need to re-input the aperture table. RS-274X produces a variety of Gerber data called X data.

RS-274X is a superset of the EIA Standard RS-274D format. RS-274X supports some of the parameter data codes (G codes) and aperture codes (D codes) contained in RS-274D, as well as codes referred to as mass parameters. Mass parameters are plot parameters that define characteristics that can affect an entire plot, or only specific parts of the plot, called layers. Mass parameters extend the capabilities of Gerber Format. Their presence makes the Gerber data X data. RS-274X is maintained by Gerber Systems Corporation (GS), a leading supplier of CAD/CAM systems, large-area plotting systems, and precision cutting systems since 1965.

An RS-274X plot file is a file consisting of RS-274X parameters and standard RS-274D codes which, when correctly interpreted, result in an image that may be displayed or plotted.

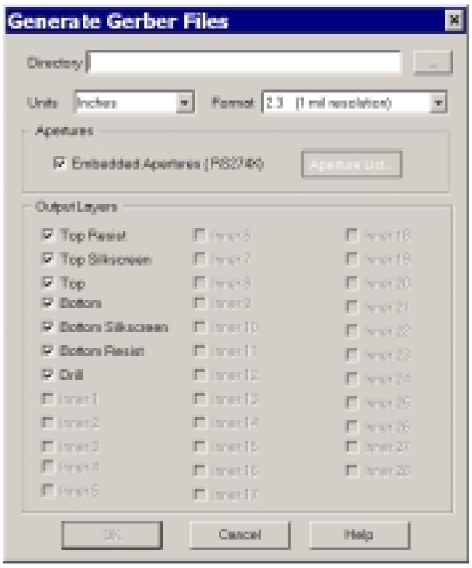
Generating Gerber files

To generate Gerber files, select <CAM\Generate Gerber files...> from the main menu. The dialog box shown below will appear.

Directory. Enter the output directory for the Gerber files.

Units. Select the output units.

Format. Select the output format.



Apertures. AutoTRAX can automatically create the apertures required, and automatically embed the information in the Gerber file.

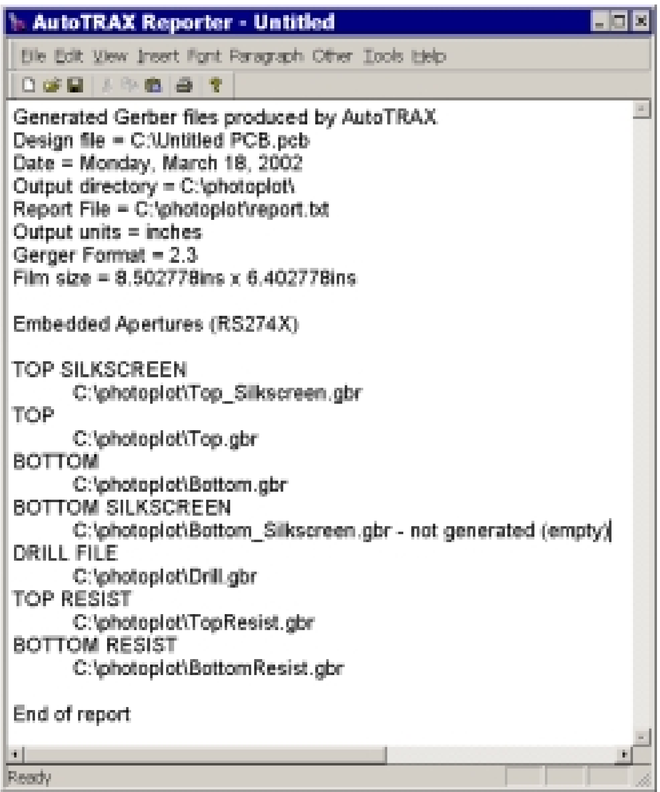
Embedded Apertures (RS274X). Check this box to automatically embed the apertures. If this box is not checked, the **Aperture List** button is enabled. Click on the **Aperture List** button to edit the apertures.

Note: when you enable the Embedded Apertures option in the Apertures Tab of the Gerber Setup dialog the aperture list is created automatically, each time you generate the Gerber files. The apertures are then embedded in the Gerber files, according to the RS274X standard. This feature means that you do not need to worry if the current aperture list includes all the required apertures -- unless your PCB manufacturer does not support embedded apertures it is highly recommended that you use this option.

To ensure that the finished PCB meets your design and manufacturing requirements it is important that you contact the fabrication house and discuss their requirements before generating the Gerber files.

Some of the requirements you should discuss include:

Any apertures restrictions -- most modern photoplotters are raster plotters which can accept any size aperture. Generally they also accept Gerber files with embedded apertures. In this situation enable the Embedded Apertures option in the Apertures Tab of the Gerber Setup dialog.



When you generate the Gerber output a series of files are created, each one corresponding to one of the layers enabled in the Gerber setup. These files are then loaded into a Gerber photoplotter, which produces the necessary phototools for PCB manufacture.

Output Layers. Check the box next to the layer name to generate a plot for that layer.

Click **OK** to generate the Gerber files.

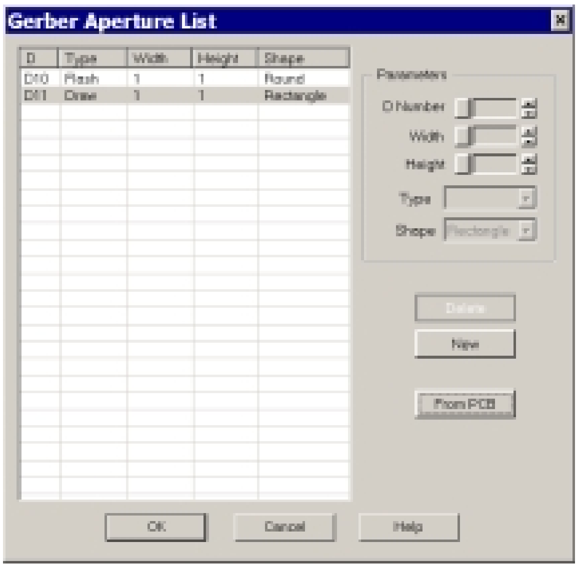
Once the files are generated, a report file similar to the one shown below will be displayed.

Creating an aperture list

Note that when you enable the Embedded Apertures option in the Apertures Tab of the Gerber Setup dialog the aperture list is created automatically, each time you generate the Gerber files. The apertures are then embedded in the Gerber files, according to the RS274X standard. This feature means that you do not need to worry if the current aperture list includes all the required apertures -- unless your PCB manufacturer does not support embedded apertures it is highly recommended that you use this option.

An RS-274X plot file is a file consisting of RS-274X parameters and standard RS-274D codes which, when correctly interpreted, result in an image that may be displayed or plotted.

Vector plotters control the width and shape of features by projecting light through a series of openings, or apertures, in a rotating wheel. Each position on the wheel is identified by a unique D code. When the D code appears in the data, the wheel rotates to the referenced position for exposure.



Unlike a vector device, a raster device has no apertures and therefore requires a description of the aperture geometry to create the required lines and other features. The aperture parameters provide the description.

Standard RS-274D codes (D codes, G codes, M codes, etc.) specify how the coordinate data should be manipulated. Each code applies to coordinate data located in the same data block as the code (that is, between EOB characters). It also applies to coordinate data following it until another code of the same type is encountered, or until a new layer is generated. This continuing action is referred to as modal. For example, G02 specifies clockwise, single-quadrant circular interpolation and is modal. All coordinate data following it will be considered clockwise arc data until another interpolation code is encountered, or until a new layer is generated. When a new layer is generated, interpolation will be reset to linear (G01). Like parameters, standard RS-274D codes may be grouped according to function. They generally appear in the file in the following sequence:

1. N codes (sequence numbers) are similar to line numbers and may be assigned to data blocks to simplify organization. Sequence numbers may be 0 to 99999. N codes are not necessary.

2. G codes (general functions) specify how to interpolate and move to the coordinate locations following the code until changed or until a new layer is generated (modal).

3. D codes (plot functions) select and control tools, specify line type, etc.

4. M codes (miscellaneous functions) perform a variety of functions such as program stop and origin specification.

Standard apertures

The AD parameter identifies standard apertures by D-code number and describes them in terms of shape (circular, rectangular, obround, or polygonal) and size (diameter if round, height and width if rectangular or obround, outside dimension and number of sides if polygonal). Apertures may be solid or open (that is, with a hole) and are always centered.

Special apertures

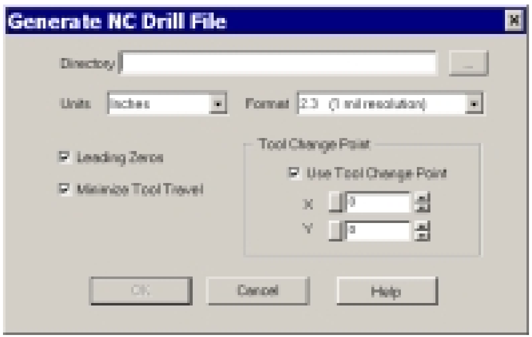
The AD parameter is also used to assign a D-code to a file containing an aperture description. The aperture description file may be a .mac file created by the AM (Aperture Macro) parameter or a .des file created by an Aperture Editor such as the Gerber GPC Aperture Editor. See the AM parameter description for further information on aperture macros. AD parameter syntax rules

Creating an NC drill file

NC drill files are used to program a drilling machine with the information required to drill the holes in the PCB during the PCB fabrication process. Drill files specify the drill sizes, drill tool assignments and hole locations.

To create a NC drill file, select **<CAM/Generate NC Drill file...>** from the main menu.

The **Generate NC Drill File** shown below will appear.



Directory. Enter the directory for the output drill file.

Format. Select the resolution. The NC Drill files should be created with the same format, or precision, as the Gerber files. For example, if the Gerber setup has been configured to use the 2.4 format, then the corresponding NC drill setup should use the same format.

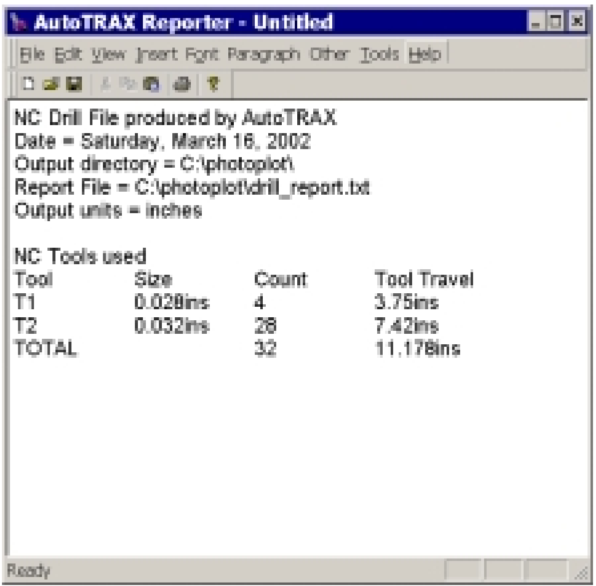
Leading Zeros. Check to include leading zeros.

Minimize Tool Travel. Check to optimize tool movement and reduce the time taken to drill a board.

Use Tool Change Point. When changing tools (drills) the drilling machine can either change the tool without moving or it may have to return to a tool collection point. If the drilling machine has to return to a pick uo point, check this box. Enter the **X** and **Y** coordinates of the tool change point.

Click **OK** to generate the NC drill file.

Once a drill file is created, a report file is created. An example report is shown below.



Exporting the netlist

To export the netlist for an open schematic, select <**Tools\Export Netlist...>** from the main menu.

You can export to the following formats.

1. AutoTRAX PCB (.xml)
2. Eagle PCB (.scr)
3. Layo1 PCB (.cmp,net)
4. MultiSIM PCB (.plc,.net)
5. OrCAD PCB (.plc,.net)

6. PADS PCB (.adc,.asc)
7. PCAD PCB (.net)
8. TANGO PCB (.net)
9. Ultimate PCB (.net)

Exporting to an IDF file

The Intermediate Data Format (IDF) is a specification designed to exchange printed circuit assembly (PCA) data between mechanical design and PCB layout. Initially developed in 1992, the IDF has since become a de facto industry standard, implemented by most CAD vendors and widely used by their customers.

IDF 4.0 includes all information that is commonly shared among mechanical design, circuit board layout, and physical analysis during the design and analysis of products containing PCAs including:

Each of the major interconnect technologies (traditional PCB, MCM, hybrid)

- Panel and board assemblies
- Board design variants
- Panel, board, and component parts
- 3D part shapes consisting of extrusions, cutouts, and cavities
- Mounting side and opposite side component part shapes
- Holes (mounting, tooling, pin, via, thermal via)
- Conductors (pads, traces, filled areas)
- Restriction regions (keepins and keepouts)
- Graphics (to represent miscellaneous board features such as fiducials and silkscreen)
- Annotations (to communicate miscellaneous design information between designers)
- Figures, footprints, and sublayouts (to group related features and component instances)
- Properties (thermal and structural)

The IDF 4.0 data model is based on a hierarchy of assemblies, parts, and features. Assemblies are constructed from instances of parts and other assemblies. Parts are constructed from features. Features define the geometric shape and other physical characteristics of parts, and convey functional information as well.

To create an IDF file, select **<CAM\Output IDF file)>** from the main menu.

Creating a Pick and Place file

Pick and place file requirements are configured in the Pick and Place Setup dialog. This dialog appears when you edit an existing pick and place setup in a CAM output configuration document, or when you select Edit " Insert Pick and Place from the CAM Manager menus to add a new pick and place setup to the current CAM output configuration document.

Pick and place files are used to program machines that automatically load components onto the PCB during assembly. They are called pick and place because they pick the required component from a feeder tube and place it in the correct location on the PCB. Once a PCB has the components loaded it is then passed through another machine that solders all the connections.

Pick and place component location files are generated in Spreadsheet, comma separated text formats in imperial or metric units.

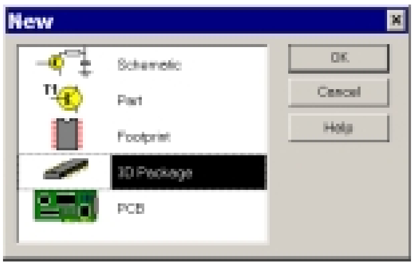
To generate a Pick and place file, select **<CAM\Pick and Place>** from the main menu.

Creating a Bill of Materials file

To create a Bill of Materials (BOM) file, select **<CAM\Bill of Materials (BOM)>** from the main menu.

3D Packages

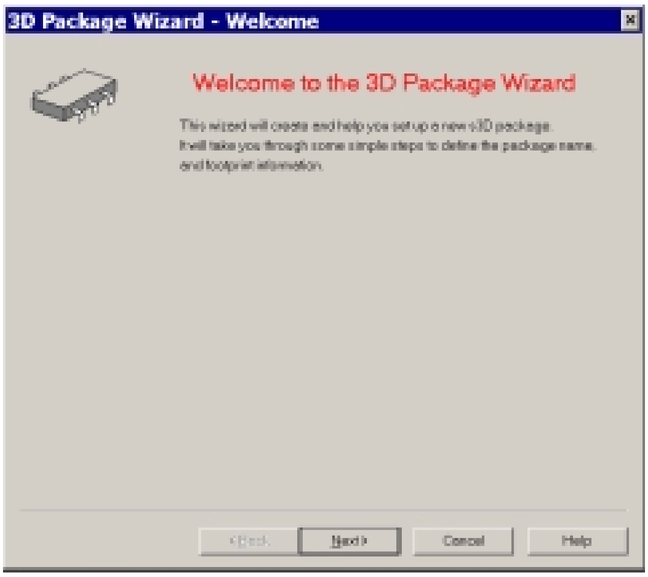
Creating a 3D Package



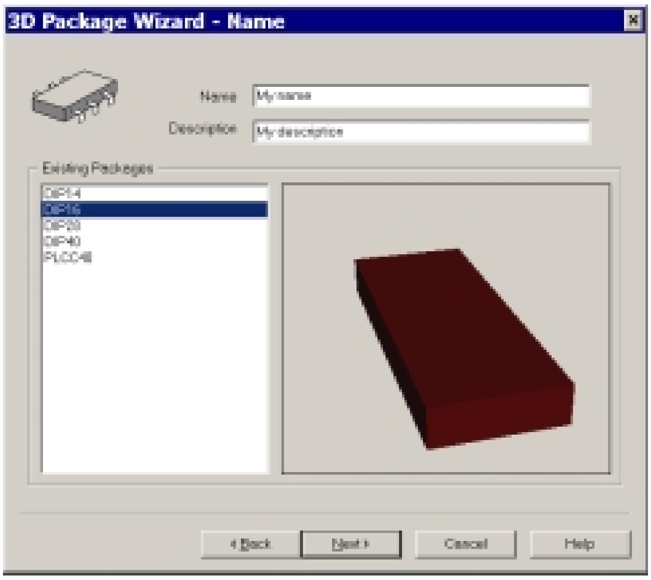
To create a new 3D package, select **<File/New>** from the main menu.

The **New dialog box** shown below will appear. Select **3D Package** to create a new package.

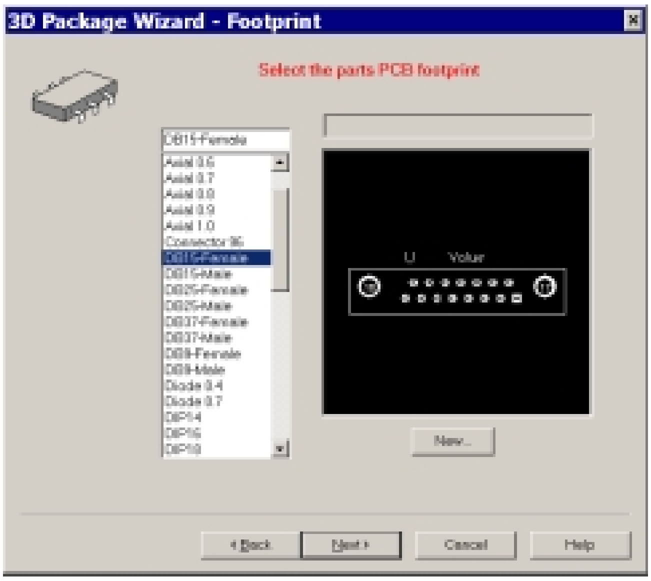
The **3D Package Wizard** will then appear. The first page is shown below.



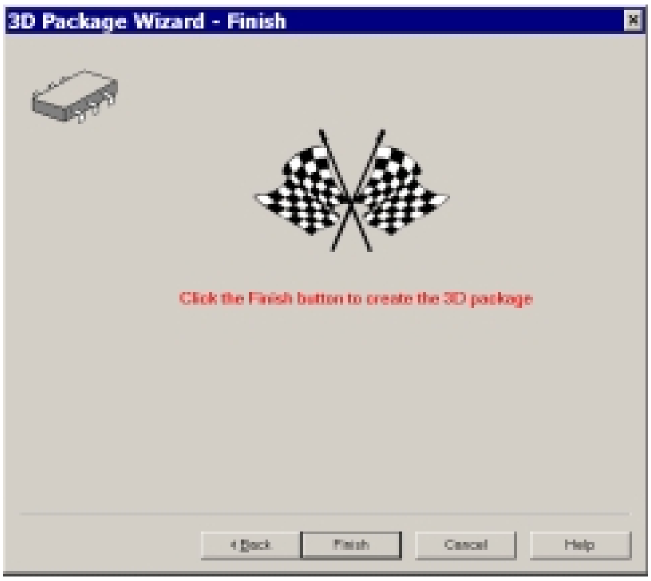
Click the Next button to display the next page.



Enter the name of the new 3D package and click the **Next** button.



Select the footprint and again press the **Next** button.



Finally click the **Finish** button. This will start the Active3D modeler.

Resources

Kovac Software on the Net

For the latest news on AutoTRAX, Internet resources and information go to www.kov.com

Books

Semiconductor Device Modeling with SPICE. Giuseppe Massobrio and Paolo Antogenetti, 2nd (30 November, 1998) McGraw Hill (Tx); ISBN: 0071349553

Inside SPICE. Ron Kielkowski 2nd Ed (March 1998) McGraw-Hill Publishing Company; ISBN: 0079137121

The Spice Book, Andre Vladimirescu, (26 April, 1994) John Wiley and Sons (WIE); ISBN: 0471609269

SPICE Practical Device Modeling. Ron Kielkowski (September 1995) McGraw-Hill Publishing Company; ISBN: 0079115241

Mosfet Modeling with Spice. Daniel Foty.(14 January, 1997) Prentice Hall; ISBN: 0132279355

The Designer's Guide to SPICE and Spectre, Kenneth S. Kundert. (30 May, 1995)

Kluwer Academic Publishers; ISBN: 0792395719

Switch-Mode Power Supply SPICE Cookbook. Christophe P. Basso, Christopher Basso. (19 March, 2001) McGraw-Hill Companies, Inc.; ISBN: 0071375090

SMPS Simulation with SPICE 3. Steven M, Sander. McGraw-Hill Companies, Inc.; ISBN: 0079132278

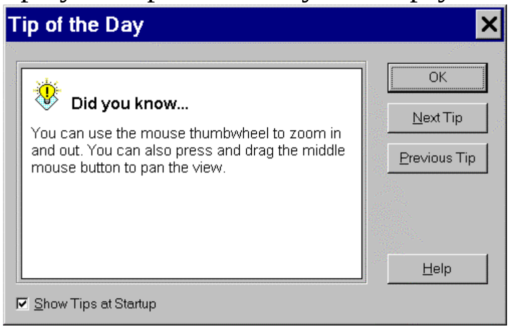
Spice Models

Many manufacturers have spice models for the parts that they sell that are available for download from their web sites.

Getting help and support

Getting a tip of the day when you start AutoTRAX

Every time you start AutoTRAX you can optionally display a 'Tip of the Day' to help you learn how the get the most out of AutoTRAX. The 'Tip of the Day' dialog is shown below:



Every time it is displayed, it displays another helpful tip.

To display the 'Tip of the Day' select **<Help/Tip of the Day>** from the main menu.

If you do not want to see the tips at startup, uncheck **Show Tip at Startup.**

To cycle through the tips, click the **Next Tip** or **Previous Tip** buttons.

I wish AutoTRAX would do this...

Do you have a feature that you would like to see in AutoTRAX?

If so, select <Help/Wish List> from the main menu.

This will guide you to Kovac Software's web site and will allow you to request new features. Your features will be considered and hopefully included in new versions. You will be notified by email.

Kovac Software on the Web

Select <Help/Kovac Software on the Net> from the main menu to go to the Kovac Software home web page. There you will find information on new versions, other useful products and design resources.

Obtaining information about your copy of AutoTRAX

Select <Help/About> from the main menu to display information about your copy of AutoTRAX.

A dialog box similar to the one shown below will be displayed.

Registering your copy of AutoTRAX

To register your copy of AutoTRAX Select <Help/Register...> from the main menu. You will then be taken to the registration page on Kovac Software's web site.

The recovery file

Before all operations, the current state of the design is saved on an undo stack. In addition, before any significant operation that modifies the design database, the current design is save to as recovery file on the hard drive. When AutoTRAX terminates normally, the recovery file is deleted. However, if the operation fails and AutoTRAX terminates due to a serious error then the recovery file is not deleted. The presence of the recovery file will be detected when AutoTRAX is restarted and the user will be prompted to load the recovery file. This will restore the design to the point just before the serious error occurred. The user will not have lost any work!

If the operating system fails due to say a power outage, AutoTRAX will detect the recovery file on restart allowing the user to recover their work. In this case at most only the last operation is lost.

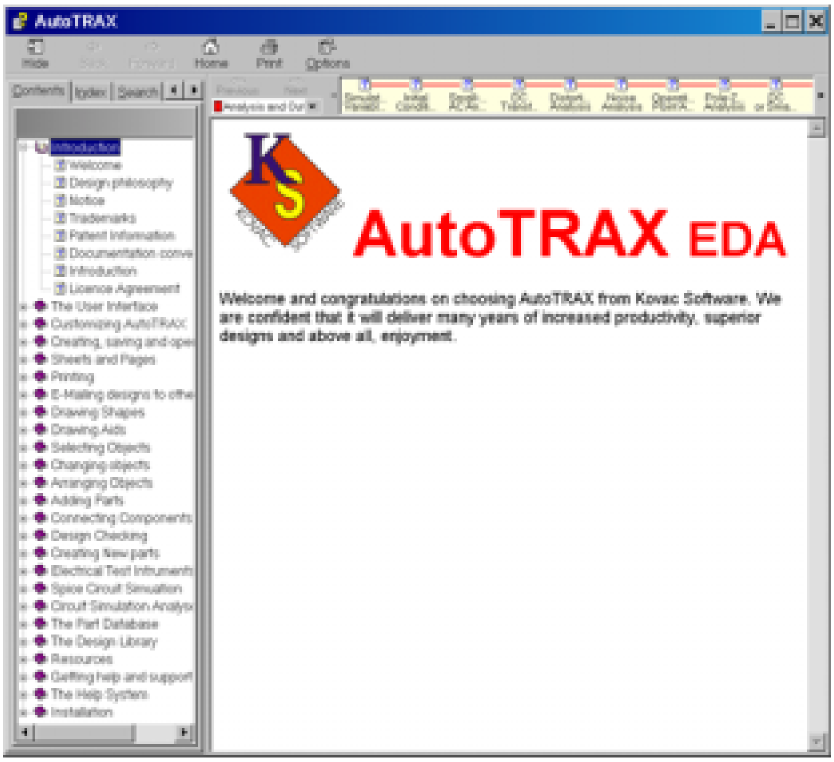
One of the main advantages of the recovery file is the improved effective reliability of using AutoTRAX and the effect that recovery has after a program or system failure. Imagine, if you will, the user having worked on a design for 2 hours without saving the design and then the program crashes or the power goes off. The user will be none too pleased! Now imagine their delight when then start up AutoTRAX again to find that their work is still there. NOTE: Unfortunately a hard drive crash will not save enable recovery of the users work if a RAID system was not used.

By chance during the creation of the actual paragraph (using Word 2000 on Windows 2000 Professional) the computer decided to reboot. All the work was lost so it had to be retyped….

The Help System

The AutoTRAX Help System is designed to rapidly assist you in learning how to use AutoTRAX. It is task orientated with each topic based on 'How do I do...' questions. This is in sharp contrast to conventional help systems, which are more like a reference manual.

The Help System is displayed in a window that looks and acts very much like a web browser, see below.



On the top is a navigation bar.

On the left is a set of tabbed windows; these are:

Contents. A list of **Books** with **'How to'** topic pages.

Index. A index of all words in the help topics.

Search. A powerful search engine to help you find relevant topics.

Favorites. A list of your favorite topics.

A Glossary.

To the right of the tabbed windows is the information pane that contains the selected topic.

Finally, above the information pane is a browse sequence to assist you in browsing through related topics.

Viewing the contents

The Table of Contents for the Help system contains a collection of Books on different subjects.

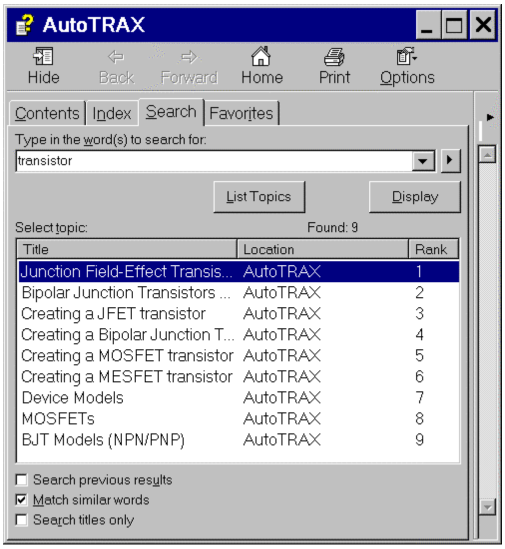
Click a book icon ( ) to open the book and view the topic pages in the book. Books can also contain other Books. A

Book can also be views as a Chapter.

The help topics have been designed to answer 'How To' questions.

Searching for help

If you can't find the Help you're looking for with the Contents or the Index tab, use the Search function. To search for a phrase or a combination of words, you can use Boolean terms (AND, OR, NOT).



Type a word or a phrase in the Type in the keyword to find field.

Click the List Topics button.

A list of topics will appear in the Select Topic to display window.

Select a topic and click the Display button.

The topic will appear in the Help window.

Many topics contain drop-down and expanding links. The word or phrase you are searching for may be in one of these links and not immediately visible. Click these links (which are not underlined, making them distinguishable from hyperlink jumps) to view the entire contents of the Help topic.

The information pane

The pane showing this text is the information pane.

The Information Pane is where you will find the actual topic you have requested. If the topic is larger than the available display space, you can use the scroll bars to view the rest of the topic. You can also expand the entire Help window so more of the topic is visible.

Navigating help topics

The navigation bar at the top of the Help System assists you in moving from topic to topic.

Hide. Hides the contents tab. If you select this, the contents tab will be hidden. A Show button will then replace the button. Select this to show the contents tab.

Back. This will be taken to the last topic that you viewed.

Forward. The Forward button will only have an effect after you use the Back button. If you click it, the Help window will display the topic you viewed BEFORE you clicked the Back button.

Home. This will take you to the Kovac Software web site. **www.kov.com.**

Print. This will print the current topic.

Options. This will be display a popdown list of options. See below.

Hide Tabs. Hides the contents tab. If you select this, the contents tab will be hidden. A Show button will then replace the button. Select this to show the contents tab.

Back. This will be taken to the last topic that you viewed.

Forward. The Forward button will only have an effect after you use the Back button. If you click it, the Help window will display the topic you viewed BEFORE you clicked the Back button.

Home. This will take you to the Kovac Software web site. **www.kov.com.**

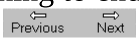
Stop. Stops search.

Refresh. Updated the information pane.

Internet Options... Sets Internet options. See your Internet Explorer Help System.

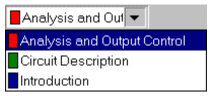
Search Highlight On/Off. Turns On/Off Highlighting text in displayed topics.

Browse sequences

Many topics in the AutoTRAX Help System are part of an overall "stream" of information. For example, there are several topics that cover how to start a form. These topics are part of the "Starting a Design" Browse Sequence, which you can read from beginning to end without having to click a hyperlink. Simply use the Previous and Next buttons (

) to flip through the related information as if you are turning pages in a book.

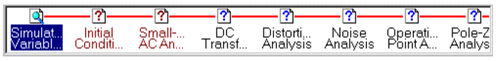
Selecting a Browse Sequence



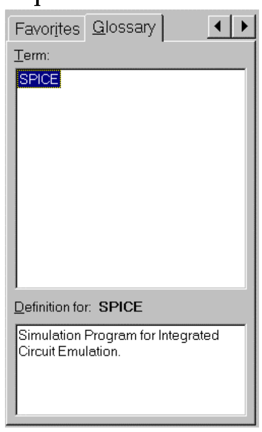
You can choose from the defined Browse Sequences by clicking the drop-down list and making a selection. This list is found just to the right of the Navigation Pane in the Help window.

After you make your selection, the Information Window will display the first topic in the selected sequence.

Using the Browse Sequence Map



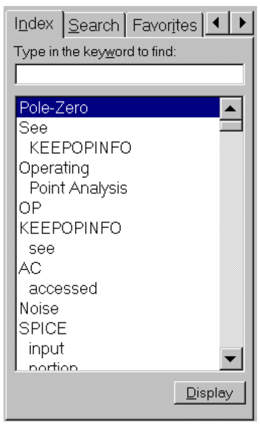
The Browse Sequence Map shows you where you are within a specific sequence. You can open a topic by clicking it on the map. If the Browse Sequence contains a lot of topics, use the arrows at either end of the map to navigate through the entire sequence.



Using the index

The index pane allows you to browse through a list of keywords and find the topics that will contain relevant information.

You can find a keyword by scrolling through the list or typing it in the **Type in the keyword to find:** field. Then double-click the keyword or select it and click the Display button. If the keyword applies to just one topic, the topic it will be displayed immediately otherwise if the keyword applies to more than one topic, you'll be prompted to choose which one you would like to see.



Saving and viewing your favorite topics

The favorites tab holds all you favorite or frequently used topics.

The Glossary

The Glossary is a list explaining important terms, acronyms, and technical jargon.

If you come across a word or acronym in the Help Topics that you don't understand, then check for a definition in the Glossary. The topic you are reading in the Information pane of the Help window will remain visible while you are searching for it.

Printing the help information

Click the Print button in the Navigation bar.

Installation

AutoTRAX is distributed either on a CDROM or as a downloadable executable (.exe) file from www.kov.com.

AutoTRAX requires that your video card be 16-bit (high color) or higher. It must also have a minimum resolution of 800x600. Right clicking on your desktop and selecting the Settings tab can change video setting.

To install AutoTRAX:

Insert the CD-ROM. The installation program should start automatically (unless auto-start is disabled on your PC). If the installation does not start, open your Explorer and double-click on the SETUP.EXE file on the CD-ROM.

Follow the on-screen instructions. Most of the instructions can simply be confirmed by clicking the Next button. If you wish, you can customize the installation directory or the location of the applications icon in the Start menu. The icon normally appears under the Programs sub-menu.

AutoTRAX setup will automatically set all ATX files to display the AutoTRAX document icon. Any files which have the icon assigned can be opened in AutoTRAX by double-clicking on the file.

Operating Systems

The developers of AutoTRAX have found Windows 2000 to be far superior in performance and reliability to Windows 98 and in fact strongly discourage the use of Windows 98.

The Mouse

To get the best out of AutoTRAX you are strongly advised to purchase a 3-button mouse with a thumbwheel.

Video Card

AutoTRAX takes full advantage of hardware acceleration of the current range of graphics card to give you real-time true WSIWG (what you see is what you get) viewing.

It is recommended to use 24bit color and a resolution 0f 1024x768 or 1280x1024. Using a larger resolution may slow down the speed of the graphics as more pixels need to be drawn. Using a lower resolution makes it more difficult to manage floating toolbars and dialog boxes.

Multiple Monitors

Multiple Monitors

With the advent of Windows 98 and Windows 2000 you now have the capability of adding additional monitors to your computer. AutoTRAX can be used with multiple monitors and the floating toolbars and dialogs can be positioned on these additional monitors. AutoTRAX remembers their position when it restarts.

All you need is an additional graphics card and monitor. See the Windows documentation of setting up the additional monitor.

For Windows 2000

Turn off your computer.

Insert your additional Peripheral Component Interconnect (PCI) or Accelerated Graphics Port (AGP) video adapter into an available slot.

Plug your additional monitor into the card.

Turn on your computer. Windows 2000 will detect the new video adapter and install the appropriate drivers.

Open Display in Control Panel.

On the Settings tab, click the monitor icon that represents the monitor you want to use in addition to your primary monitor.

Select the Extend my Windows desktop onto this monitor check box, and then click Apply or OK.

Selecting the Extend my Windows desktop onto this monitor check box allows you to drag items across your screen onto alternate monitors.

Or, you can resize a window to stretch it across more than one monitor. To use the multiple monitor support feature, you need a PCI or AGP video adapter for each monitor. If you have an onboard video adapter (one that is not a plug-in card but is part of the motherboard) that you want to use as part of a multiple-monitor configuration, it must be set as VGA.

Memory

The more memory you have the better. You are recommended to have at 256 Mbytes.

Glossary

analysis

The result of a SPICE circuit simulation.

arc

A curved line.

bmp

Bitmap, a rasterized graphic format in which the image is represented as a combination of dots. The TIFF and JPEG formats are variants of the bitmap file type.

cancel

To place your cursor over an object, toolbar button, menu, or other screen item and press the left mouse button.

cut

copy 1 or more objects to the clipboard and then delete them (not from the clipboard). They can then be pasted.

dominant object

The dominant object is used as a reference point for a command that affects multiple objects. For example, if you choose the Make same height command, every object that is selected will become the same height as the dominant object.

fill color

The internal color of a two-dimensional object.

focus

An object or a segment of the user interface has focus when it will be the target of your next action. If you click the Gallery, the Gallery has focus. If a dialog box is open and you use the TAB key to highlight the OK button, the OK button has focus.

grid

A design aid that helps you place objects on your sheet. The Snap to Grid function, when enabled, causes objects to "jump" to the nearest grid point. This function can also help you draw perfectly straight lines and round circles.

ground

Electrical grounds. This is node 0.

group

Arrange a collection of objects so that the act as a single object. The move together, are all selected together.

guide

A horizontal or vertical line or a point that objects will snap to.

hover

To place your cursor over an object, a toolbar button, or a menu item without actually clicking.

instrument

An object that can be added to a design that represents an electrical test instrument. If it is clicked then it will simulate the circuit and present the results.

jpeg

Joint Photographic Experts Group, a rasterized graphic format.

left click

To place your cursor over an object, toolbar button, menu, or other screen item and press the right mouse button.

model

A description of the electrical properties of a part.

node

A set of wires between part terminals. The electrical signal is the same at all points and terminals on the node.

object

Any item that can be added to a sheet, including graphical elements and parts.

OK

The OK button on a dialog box confirms any changes or commands you have initiated.

part

An electronic part or component

paste

to copy/add (paste) objects from the clipboard onto a sheet.

png

Portable Network Graphic, a 48-bit graphic format designed especially for online viewing and endorsed by the World Wide Web Consortium (W3C).

polyline

A series of connected line segments

properties

The characteristics that define an object.

right click

To place your cursor over an object, toolbar button, menu, or other screen item and press the right mouse button.

select

To make an object the target for a subsequent action, such as resizing, moving, or changing its properties. Normally, you select an object by clicking it with the cursor or dragging a marquee around it.

sheet

a design is made up of 1 or more sheets. A sheet is the same as a page.

shortcut

A keyboard combination that acts as a 'shortcut' for a command.

simulate

run a computer model of an electrical circuit

snap

force a point to 'snap' to a regular grid

SPICE

SPICE stands for Simulation Program with Integrated Circuit Emphasis.

terminal

An electrical connection point for a part

thumbwheel
The rotating wheel in the middle of some mice.
tiff
Tagged Image File Format, a rasterized graphic format.
toolbar
A group of buttons on a dialog box
wmf
Windows MetaFile, a 16-bit graphic format typically used for vector drawings.

License Agreement

THIS KOVAC SOFTWARE LICENSE AGREEMENT ("LICENSE AGREEMENT") IS A LEGAL AGREEMENT BETWEEN YOU AND KOVAC SOFTWARE, FOR THE SOFTWARE PRODUCT. BY USING THE SOFTWARE PRODUCT, YOU ARE AGREEING UNCONDITIONALLY TO BE BOUND BY THE TERMS OF THIS LICENSE AGREEMENT, EVEN IF THIS LICENSE AGREEMENT IS DEEMED A MODIFICATION OF ANY PREVIOUS ARRANGEMENT OR CONTRACT. IF YOU DO NOT AGREE TO THE TERMS OF THIS LICENSE AGREEMENT, DO NOT USE THE SOFTWARE PRODUCT. INSTEAD, YOU MAY RETURN THE SOFTWARE PRODUCT TO THE PLACE YOU OBTAINED IT FOR A FULL REFUND (IF APPLICABLE).

Software Product License

The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold.

For purposes of this License Agreement, "Software Product" refers to the computer software and associated media, printed materials, and "online" or electronic documentation, including without limitation any and all executable files, add-ons, stencils, templates, databases, tutorials, help files and other files, that accompany the Kovac Software product identified above or in the accompanying documentation; "Use" means storing, loading (whether into temporary memory (i.e., RAM) or into permanent memory (e.g., hard disk, CD-ROM or other storage device)), installing, executing or displaying the Software Product; and "You" means the company, entity or individual whose funds are used to pay the license fee or who has otherwise acquired the Software Product.

1. Grant of License

Software Product. Kovac Software grants You the non-exclusive, non-sublicensable, limited license to Use one copy of the Software Product on a single computer, subject to the terms and conditions of this License Agreement.

Second Copy. The primary user of a computer on which a copy of the Software Product is installed may make a second copy for his or her exclusive Use on either a home or portable computer, but not both.

Storage/Network Use. You may also store or install a copy of the Software Product on a storage device, such as a network server, for the purpose of Using the Software Product on Your computers only over an internal network; however, You must acquire and dedicate a license for each separate computer on which the Software Product is Used from the storage device. A license for the Software Product may not be shared or Used concurrently on different computers. You shall inform all users of the Software Product of the terms and conditions of this License Agreement.

License Pack. If You have acquired this License Agreement in a Kovac Software License Pack, You may make the number of additional copies of the computer software portion of the Software Product authorized on the printed copy of this License Agreement, and You may Use each copy only in the manner specified in this License Agreement. You are also entitled to make a corresponding number of second copies for home or portable computer Use only in the manner specified in this License Agreement.

2. Ownership

Title, ownership rights and intellectual property rights in and to the Software Product shall remain in Kovac Software and its suppliers and are protected by US and international copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. There is no transfer to You of any title to or ownership of the Software Product and the license granted under this License Agreement should not be construed as a sale of any right in the Software Product. You must treat the Software Product like any other copyrighted materials (e.g., a book or musical recording) except that You may make one copy of the Software Product solely for backup or archival purposes (with the inclusion of all copyright and other proprietary notices), only as long as You otherwise comply with the terms and conditions of this License Agreement. You may not make additional copies of the Software Product. You may not copy the printed materials accompanying the Software Product. All rights not specifically granted under this License Agreement are reserved by Kovac Software.

3. Description of Other Rights and Limitations

Limitations on Reverse-Engineering, Decompilation and Disassembly; Other Restrictions. You acknowledge that the Software Product in source code form remains a confidential trade secret of Kovac Software and therefore You agree not to reverse-engineer, decompile or disassemble the Software Product, or make any attempt to discover the source code to the Software Product, except and only to the extent that such activity is expressly permitted by applicable law notwithstanding this limitation. Except as expressly permitted in this License Agreement, the Software Product may not be Used, copied, translated, redistributed, retransmitted, published, sold, rented, leased, marketed, sublicensed, pledged, assigned, disposed of, encumbered, transferred, altered, modified or enhanced, whether in whole or in part, nor may You create derivative works from or based on the Software Product. You may not remove any proprietary notices, marks or labels from the Software Product.

Separation of Components. The Software Product is licensed as a single product and its component parts may not be separated for Use on more than one computer.

Transfer of Software Product. You may transfer all Your rights under this License Agreement on a permanent basis only, provided You retain no copies, You transfer the License Agreement, the corresponding serial number (if applicable) and all the Software Product (including without limitation all component parts, media and printed materials, and any upgrades) and the recipient agrees to all the terms and conditions of this License Agreement. If the Software Product is an upgrade product, any transfer must include the latest release, all prior versions and any prior products used to obtain the Software Product.

Termination. This License Agreement is in effect until terminated. You may terminate it at any time by destroying the Software Product and all copies You have made. Unauthorized copying or duplication of the Software Product will result in automatic termination of this License Agreement. Without prejudice to any other rights, Kovac Software may terminate this License Agreement if You fail to comply with any term or condition of this License Agreement. Upon termination of this License Agreement, You agree to destroy the Software Product and all copies You have made.

Academic Software Product. If the Software Product is identified as "Academic", You must be a Qualified Educational User (as defined by Kovac Software) to Use the Software Product. If You are not a Qualified Educational User, You have no rights under this License Agreement. To determine whether You are a Qualified Educational User, please contact Kovac Software - Academic Sales at the applicable address set forth below.

Not For Resale ("NFR") Software Product. If the Software Product is identified as "Not For Resale" or "NFR", then, notwithstanding any other terms and conditions of this License Agreement, You may not, for value or other consideration, distribute, resell, dispose of, or transfer the Software Product.

Support Services. Kovac Software may provide You with customer and technical support services related to the Software Product ("Support Services"). Use of Support Services is governed by the Kovac Software policies and programs described in the user manual, in "online" or electronic documentation, and/or in other Kovac Software-provided materials. Any supplemental software code provided to You as part of the Support Services shall be considered part of the Software Product and subject to the terms and conditions of this License Agreement. With respect to any technical information You provide to Kovac Software as part of the Support Services, Kovac Software may use such information for its business purposes, including product support and development. Kovac Software will not utilize such technical information in a manner that personally identifies You.

4. Upgrades

If the Software Product is identified as an "upgrade", You must be properly licensed to Use a product, whether from Kovac Software or another supplier, that is specified by Kovac Software as being eligible for the upgrade in order to Use the Software Product. A Software Product identified as an upgrade replaces or supplements the product that formed the basis for Your eligibility for the upgrade. Notwithstanding any other terms and conditions of this License Agreement, You may transfer the Software Product only in conjunction with the product that formed the basis for Your eligibility for the upgrade, unless You destroy such product. If the Software Product is an upgrade from a component of a package of software programs which You licensed as a single product, the Software Product may be Used or transferred only as part of that single product package and may not be separated for Use on more than one computer.

5. Dual-Media Software

You may receive the Software Product in more than one medium; for example, in disk media, on a CD-ROM or in both media. Regardless of the type or size of medium You receive, You may Use only one medium that is appropriate for Your computer. You may not Use the other medium on another computer. You may not distribute, transmit, sell, loan, rent, lease or otherwise transfer the other medium to another user, except as part of the permanent transfer of the Software Product pursuant to the terms and conditions of this License Agreement.

6. Reports and Audit Rights

You shall institute reasonable measures to ensure compliance with the terms and conditions of this License Agreement. Upon Kovac Software's reasonable request, You agree to provide reports relating to Your Use of the Software Product as necessary to demonstrate Your compliance with the terms and conditions of this License Agreement. You further agree that Kovac Software has the right, upon reasonable prior notice, to audit Your records and inspect Your facilities to verify Your compliance with the terms and conditions of this License

Agreement.

7. US Government Restricted Rights

The Software Product and documentation are provided with RESTRICTED RIGHTS. Use, duplication or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1) and (2) of the Commercial Computer Software-Restricted Rights at 48 CFR 52.227-19, as applicable. The contractor/manufacturer is Kovac Software, Kovac Software. Fair Oak, Skeath Lane, Sandon Bank, Stafford ST18 9TD, England.

8. Export Restrictions

You may not export or reexport the Software Product or any underlying information or technology except in full compliance with all United States and other applicable laws and regulations. In particular, but without limitation, none of the Software Product or underlying information or technology may be exported or reexported (a) into (or to a national or resident of) Cuba, Haiti, Iran, Iraq, Libya, Serbia, Montenegro, North Korea, Sudan or Syria or (b) to anyone on the US Treasury Department's list of Specially Designated Nationals or the US Commerce Department's Table of Deny Orders, as such countries, lists and orders may be amended or modified from time to time. By Using the Software Product, You are specifically agreeing to the foregoing and You are representing and warranting that You are not located in, under the control of, or a national or resident of any such country or on any such list.

9. Entire Agreement; Governing Law

This License Agreement constitutes the entire agreement between Kovac Software and You with regard to the subject matter hereof and supersedes any and all prior agreements, understandings and representations, whether written or oral, concerning the subject matter of this License Agreement. This License Agreement shall not be modify except by a written agreement executed by a duly authorized representative of each of Kovac Software and You.

Nothing in this License Agreement is intended to exclude, modify or restrict the operation of any applicable statute or other law, the provisions of which cannot lawfully be excluded, modified or restricted. If any court of competent jurisdiction determines that a provision of this License Agreement is illegal, invalid or unenforceable in any jurisdiction, then such provision shall be deemed modified to the minimum extent necessary to make it comply with the applicable statute or law of such jurisdiction, and the remaining provisions of this License Agreement shall continue in full force and effect. Any such modification shall not effect any provisions of this License Agreement in any other jurisdiction where this License Agreement governs the Use of the Software Product.

If You acquired the Software Product in Canada, You agree to the following: Each of Kovac Software and You hereby confirm, desire and agree that this License Agreement has been and shall be written solely in the English Language.

IF YOU ACQUIRED THE SOFTWARE PRODUCT IN THE UNITED STATES, THIS LICENSE AGREEMENT IS GOVERNED BY THE LAWS OF THE STATE OF WASHINGTON, USA, EXCEPT FOR THAT BODY OF LAW DEALING WITH CONFLICTS OF LAW, WITHOUT REFERENCE TO THE 1980 UNITED NATIONS CONVENTION ON THE INTERNATIONAL SALE OF GOODS.

IF YOU ACQUIRED THE SOFTWARE PRODUCT OUTSIDE THE UNITED STATES, THIS LICENSE AGREEMENT IS GOVERNED BY THE LAWS OF THE REPUBLIC OF IRELAND.

Software Product Limited Warranty

To the original customer only, Kovac Software provides the following warranties:

Limited Warranty. KOVAC SOFTWARE WARRANTS THAT (a) FOR A PERIOD OF SIXTY (60) DAYS FROM THE DATE OF ORIGINAL PURCHASE ("WARRANTY PERIOD") AS EVIDENCED BY YOUR RECEIPT OR OTHER PROOF OF PURCHASE (i) THE SOFTWARE PRODUCT, UNLESS MODIFIED OR OTHERWISE ALTERED BY YOU, WILL PERFORM SUBSTANTIALLY IN ACCORDANCE WITH THE ACCOMPANYING WRITTEN MATERIALS, AND (ii) THE MEDIA ON WHICH THE SOFTWARE PRODUCT IS FURNISHED WILL BE FREE FROM DEFECTS IN MATERIALS AND WORKMANSHIP UNDER NORMAL USE; AND (b) ANY SUPPORT SERVICES PROVIDED BY KOVAC SOFTWARE SHALL BE SUBSTANTIALLY AS DESCRIBED IN APPLICABLE WRITTEN MATERIALS PROVIDED TO YOU BY KOVAC SOFTWARE. Kovac Software does not warrant that the Software Product will meet Your requirements or that Use of the Software Product will be uninterrupted or error-free. Kovac Software is not responsible for problems caused by changes in the operating characteristics of computer hardware or computer operating systems which are made after the release of the Software Product, nor for problems in the interaction of the Software Product with non-Kovac Software software products. Some jurisdictions do not allow limitations on duration of an implied warranty, so the above limitation may not apply to You. The Limited

Warranty gives You specific legal rights. You may have others, which vary by jurisdiction.

Exclusive Remedy. Kovac Software's entire liability, and Your exclusive remedy, shall be, at Kovac Software's option, either (a) replacement of the defective media, (b) repair or replacement of the Software Product that does not meet Kovac Software's Limited Warranty, or (c) return of the price paid, if any, and termination of this License Agreement. This remedy is subject to return of the Software Product to Kovac Software with a copy of Your receipt within the Warranty Period or, solely for Software Product that was obtained electronically via "electronic software distribution", to delivery to Kovac Software of a Kovac Software-approved "certification of destruction" together with proof of purchase within the Warranty Period. This Limited Warranty is void if failure of the Software Product has resulted from accident, abuse or misapplication. Any replacement Software Product will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer. Outside of the United States, neither these remedies nor any Support Services are available without proof of purchase from an authorized source.

No Other Warranties. THE ABOVE WARRANTIES ARE EXCLUSIVE. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, KOVAC SOFTWARE AND ITS SUPPLIERS DISCLAIM ALL OTHER WARRANTIES AND CONDITIONS, EITHER EXPRESS OR IMPLIED, INCLUDING, WITHOUT LIMITATION, IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, TITLE, AND NON-INFRINGEMENT, AND THOSE ARISING OUT OF USAGE OF TRADE OR COURSE OF DEALING, CONCERNING THE SOFTWARE PRODUCT, AND THE PROVISION OF OR FAILURE TO PROVIDE SUPPORT SERVICES. NO ORAL OR WRITTEN INFORMATION OR ADVICE GIVEN BY KOVAC SOFTWARE, ITS AGENTS, DEALERS, DISTRIBUTORS OR EMPLOYEES SHALL INCREASE THE SCOPE OF THE ABOVE WARRANTIES OR CREATE ANY OTHER WARRANTIES.

No Liability for Damages. REGARDLESS OF WHETHER ANY REMEDY SET FORTH HEREIN FAILS OF ITS ESSENTIAL PURPOSE, TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL KOVAC SOFTWARE OR ITS SUPPLIERS (OR THEIR RESPECTIVE AGENTS, DIRECTORS, EMPLOYEES OR REPRESENTATIVES) BE LIABLE FOR ANY DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, CONSEQUENTIAL, INCIDENTAL, INDIRECT, SPECIAL, ECONOMIC, PUNITIVE OR SIMILAR DAMAGES, OR DAMAGES FOR LOSS OF BUSINESS PROFITS, LOSS OF GOODWILL, BUSINESS INTERRUPTION, COMPUTER FAILURE OR MALFUNCTION, LOSS OF BUSINESS INFORMATION OR ANY AND ALL OTHER COMMERCIAL OR PECUNIARY DAMAGES OR LOSSES) ARISING OUT OF THE USE OF OR INABILITY TO USE THE SOFTWARE PRODUCT OR THE PROVISION OF OR FAILURE TO PROVIDE SUPPORT SERVICES, HOWEVER CAUSED AND ON ANY LEGAL THEORY OF LIABILITY (WHETHER IN TORT, CONTRACT OR OTHERWISE), EVEN IF KOVAC SOFTWARE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES, OR FOR ANY CLAIM BY ANY OTHER PARTY. YOU ACKNOWLEDGE THAT THE LICENSE FEE REFLECTS THIS ALLOCATION OF RISK. In any event, if any statute implies warranties or conditions not stated in this License Agreement, Kovac Software's entire liability under any provision of this License Agreement shall be limited to the greater of the amount actually paid by You to license the Software Product and Five United States Dollars (US$5.00), or, in the case of Support Services, providing such Support Services again or refunding the cost thereof. Because some jurisdictions do not allow the exclusion or limitation of liability for consequential or incidental damages, the above limitation may not apply to You.

Kovac Software. Fair Oak, Skeath Lane, Sandon Bank, Stafford ST18 9TD, England.

D

R

S

T

U

V

W

Z