

**nVision**

Version 4.40<sub>c1</sub>

**Getting Started**



# Getting Started



# **nVision System**

Version 4.40  
6 August 2003

Document Revision 6.0

This document and the software product which it describes are copyright, and are licensed to:

**nCode International Ltd**  
**230 Woodbourn Road**  
**SHEFFIELD**  
**S9 3LQ**  
**ENGLAND**

The information contained herein is the property of nCode International Ltd, and is supplied without liability for errors or omissions, and no part may be reproduced, used or disclosed, except as authorised by contract or other written permission. The copyright of the foregoing restriction on reproduction, use and disclosure extend to all the media in which this information may be embodied.

1996-2003 - nCode International Ltd.



# Contents

NVISION SYSTEM	5
<b><u>INTRODUCTION</u></b>	<b>1</b>
<b>OVERVIEW</b>	<b>1</b>
DEFINITION	1
WHAT IS NVISION?	1
WHAT CAN NVISION DO FOR ME?	2
<b>ABOUT THIS MANUAL</b>	<b>3</b>
<b><u>INSTALLATION</u></b>	<b>5</b>
<b>OVERVIEW</b>	<b>5</b>
<b>COMPUTER SYSTEM REQUIREMENTS</b>	<b>5</b>
<b>INSTALLING NVISION SOFTWARE</b>	<b>6</b>
BEFORE YOU START	6
PERFORMING THE INSTALLATION	6
ADDITIONAL INFORMATION	9
<b>ONCE YOU HAVE INSTALLED NVISION...</b>	<b>10</b>
IF YOU HAVE A HARDWARE KEY	10
IF YOU DO <u>NOT</u> HAVE A HARDWARE KEY	10
WHICH SECTION DO I READ NEXT?	10
<b><u>WINDOWS 95 BASICS</u></b>	<b>11</b>
<b>OVERVIEW</b>	<b>11</b>
<b>WHAT IS MICROSOFT WINDOWS 95?</b>	<b>11</b>
<b>FEATURES OF <i>MICROSOFT WINDOWS 95</i></b>	<b>12</b>
FEATURES OF A WINDOW	12
FEATURES OF THE POINTER	13
FEATURES OF THE TASKBAR	13
FEATURES OF ICONS	14
FEATURES OF MENUS	14
<b>BASIC <i>WINDOWS</i> TECHNIQUES</b>	<b>15</b>
MOUSE TECHNIQUES	15
WINDOW TECHNIQUES	15
ACCESSING MENUS	15
ACCESSING THE CONTROL MENU	16
DIALOG BOXES	17
WINDOWS HELP	18
<b><u>NVISION BASICS</u></b>	<b>19</b>

<b>OVERVIEW</b>	<b>19</b>
<b>FEATURES OF nVISION</b>	<b>19</b>
APPEARANCE ON START-UP	20
BASIC SYSTEM WINDOWS	21
TOOLBARS	22
THE DATA WINDOW	23
DATA OBJECTS	23
<b>BASIC nVISION TECHNIQUES</b>	<b>24</b>
OVERVIEW	24
EXECUTING MODULES	24
OPENING FILES	25
QUICK LOADING OF OBJECTS	26
GETTING INFORMATION ABOUT DATA OBJECTS	26
SAVING DATA OBJECTS	26
REMOVING DATA OBJECTS FROM THE SYSTEM	27
DRAG AND DROP	28
GETTING HELP	29
THE nVISION WORKSPACE	29
TO SAVE THE WORKSPACE LAYOUT	29
CHANGING THE SHAPE OF THE TOOLBARS	30
WORKING WITH REGIONS	31

---

<b>GUIDED TOURS</b>	<b>33</b>
---------------------	-----------

<b>BEFORE YOU BEGIN</b>	<b>33</b>
EASY INTRODUCTION	33
WHICH "TOUR" SECTIONS DO I FOLLOW?	33
SESSION PERSISTENCE	33
<b>BASE 1 TOUR</b>	<b>35</b>
OVERVIEW	35
IDENTIFYING AND LOADING THE DATA	35
TRANSFERRING DATA INTO APPLICATIONS	36
DISPLAYING DATA IN GRAPH	37
EXPERIMENTS TO TRY OUT IN GRAPH	38
DISPLAYING THE ALTERNATE SLICE IN GRAPH	39
HOT-LINKING	39
MODIFYING DATA VALUES	40
<b>BASE 2 TOUR</b>	<b>41</b>
OVERVIEW	41
DISPLAYING DATA IN CONTOUR	41
IDENTIFYING A <i>REGION</i>	42
SYNCHRONISED VIEWS	44
OVERVIEW OF A LARGE DATA SET - MAP	46
<b>TOOLS TOUR</b>	<b>47</b>
OVERVIEW	47
USING TOOLS TO PERFORM ARITHMETIC OPERATIONS BETWEEN MATRICES	47
TOOLS OPERATIONS INVOLVING SCALAR VALUES	48
USING THE UNDO MODULE TO PERFORM WHAT/IF CALCULATIONS	50
SYNCHRONISING DATA OBJECTS	52
<b>DATABASE TOUR</b>	<b>53</b>
OVERVIEW	53
RUNNING THE PROJECT DATABASE	53
CONFIGURING THE DATABASE	53

STORING RECORDS	54
QUERYING THE DATABASE	55
LOADING DATA OBJECTS	56
CREATING USER FIELDS	56
ADDING DATA TO USER FIELDS	58
UPDATING A RECORD	58
QUERIES WITH CRITERIA	59
<b>ANIMATION TOUR</b>	<b>61</b>
OVERVIEW	61
LOADING A SAMPLE ANIMATION	61
DISPLAYING A SAMPLE ANIMATION IN ANIMATE	62
EDITING AN ANIMATION IN SPLICE	62
<b>AUTOMOTIVE TOUR</b>	<b>65</b>
OVERVIEW	65
ENGINE RUN-UPS AND WATERFALL DISPLAYS	65
EXTRACTING AN RPM PROFILE FROM TACHO DATA	66
EXTRACTING ORDER INFORMATION	68
GENERATING A FREQUENCY SPECTRUM WATERFALL	69
KALMAN FILTERING	70
KALMAN ORDER EXTRACTION	71
<b>PC SOUND CARD TOUR</b>	<b>73</b>
OVERVIEW	73
DATA ACQUISITION USING THE SOUND CARD	73
PLAYBACK THROUGH A SOUND CARD	75
STORING SIGNALS IN nVISION OBJECTS	75
<b>WAVELET ANALYSIS TOUR</b>	<b>77</b>
OVERVIEW	77
FREQUENCY CONTENT OF TRANSIENT SIGNALS	77
CALCULATING THE WAVELET TRANSFORM	81
<b>INTENSITY TOUR</b>	<b>82</b>
OVERVIEW	82
FEATURES	82
LIMITATIONS	82
CONVENTIONS	83
VACUUM CLEANER PROJECT	83
STEP BY STEP	84
LOUDSPEAKER PROJECT	91
THINGS TO TRY	91
<b>DSP TOUR</b>	<b>92</b>
OVERVIEW	92
FEATURES	92
GENERATING A TEST SIGNAL	92
VIEWING THE TEST SIGNAL	93
CREATING A POWER SPECTRUM OF THE TEST SIGNAL	94
VIEWING THE POWER SPECTRUM	95
DESIGNING A BANDPASS FILTER	96
APPLYING A FILTER TO THE SIGNAL	98
COMPARING POWER SPECTRA	99
THINGS TO TRY	100
<b>IMPORT TOUR</b>	<b>102</b>
OVERVIEW	102
HELP ON IMPORTING FILES	102
FILE IMPORT - UNIVERSAL FILES	102
EXCHANGE	104

XLDRIVER - <i>MICROSOFT EXCEL</i> DATA EXCHANGE	112
<b>REPORT WRITING TOUR</b>	<b>118</b>
OVERVIEW	118
INTRODUCTION	118
USING nVISION WITH OTHER APPLICATIONS	118
INCORPORATING GRAPHS	119
HIGHLIGHTING POINTS OF INTEREST ON GRAPHS	121
USING NOTES FIELDS	122
INCLUDING STATISTICAL ANALYSIS	123
EMBEDDING SOUNDS	123
<b>APPENDIX A</b>	<b>124</b>
<hr/>	
<b>HARDWARE KEY</b>	<b>124</b>
OVERVIEW	124
INSTALLATION	124
UPGRADING YOUR HARDWARE KEY	125
UPGRADE PROCEDURE STEP BY STEP	126
<b>APPENDIX B</b>	<b>128</b>
<hr/>	
<b>ADDITIONAL SAMPLE DATA</b>	<b>128</b>
DO I NEED THE ADDITIONAL DATA?	128
INSTALLATION OVERVIEW	128
<b>INDEX</b>	<b>130</b>
<hr/>	

# Introduction

---

## Overview

### Definition

**Acquisition, Visualisation and Data Analysis for Microsoft Windows**

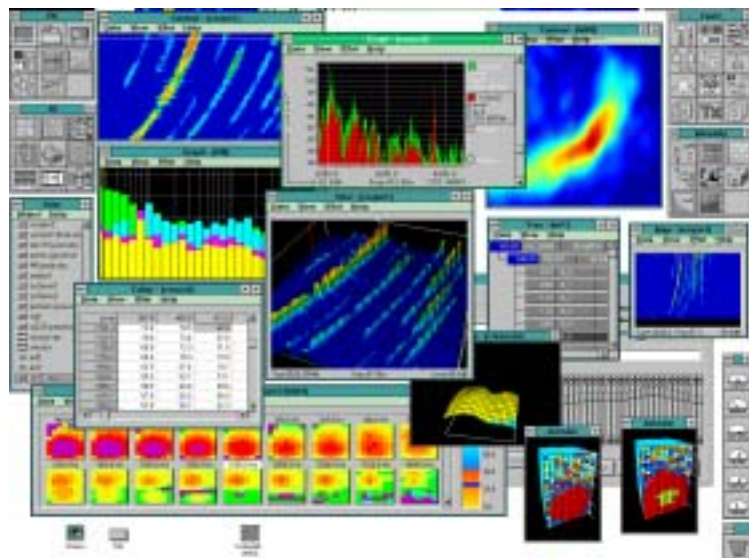
**Envision** (enVi-z-ohn), v. 1820 [- Fr. envision, f en- En-1 + Vision VISION ] 1. trans. To look in the face of; also fig. 2. To set before the mind's eye; to contemplate 1837.

### What is nVision?

**nVision** is a data acquisition, data processing and visualisation tool designed specifically for scientists, researchers and engineers who are tasked with the acquisition, management, analysis and presentation of large amounts of technical data.

High-speed data manipulation, comprehensive analysis, visualisation, and the rapid production of results are essential components of the overall “Visual Data Analysis” solution provided by **nVision**.

*nVision in action*



## What can nVision do for me?

Once, only “supercomputers” were capable of dealing with vast amounts of data and of providing the extensive but expensive number-crunching capabilities required.

Next came the workstations: capable, but still expensive and not user-friendly.

Finally came the affordable PC which, because of its low cost, is now to be found on almost every engineer’s desk. **nVision**, combined with the user-friendly Microsoft Windows environment, makes the PC the ideal platform for VDA (Visual Data Analysis) work in a field, research or laboratory setting.

Here are just some of the things you can do with **nVision**:

- Gather data from an increasingly wide range of portable instruments, acquisition cards, test installations and file formats - including standard sound cards built into many PCs today.
- Explore data using a wide variety of display options. Dozens of interactive display methods give you all the facilities you'll need to gain insight from your data.
- Interact with multiple dynamically linked views of your data, following trends, selecting significant events or making on-the-fly comparisons.
- Listen to your data, using your PC sound system.
- Gain easy access to your data, visually explore complicated data relationships, focus on significant points, make rapid assessments, then quickly create presentation-quality graphical reports from the results. nVision eliminates the obstacles that used to slow you down.
- Expand the system by adding your own applications using the programmer’s interface to Visual Basic or C/C++.
- Work with object-oriented data sets using ‘drag and drop’.

**nVision** does more than just create graphical views of your data sets - it lets you work interactively with them. **nVision's** views are dynamically linked, so you can see the results of changes in all views right away. For example, if you click and drag the mouse to highlight data in one graph, then that data will be automatically zoomed in related graphs. The same goes for cursor positioning and three-dimensional viewpoints. The system works with you. Stored workspaces remember how you have used the system, reducing tedious configuration and set-up procedures.

**nVision** is a major technological breakthrough in the field of PC-based “Visual Data Analysis”.

---

## About this Manual

This manual describes how to install **nVision** and how to quickly get the system up and running. For each major component it provides an introductory “tour” to help you quickly gain familiarity with **nVision** and its flexible capabilities.

Overall the manual contains the following chapters:

- Chapter 1 describes the system requirements for **nVision** and provides a step-by-step guide to installing the **nVision** software and the sample data.
- Chapter 2 gives novice users of *Microsoft Windows* a brief tour around the *Windows* environment, quickly and simply showing the most important parts of the *Windows* interface. If you are already familiar with the *Windows* interface and environment then you can skip this chapter.
- Chapter 3 provides a quick overview of the major components of the **nVision** system.
- Chapter 4 is a series of “Guided Tours” which explain the basic principles of the various **nVision** modules, grouped by the toolbar.



# Installation

---

## Overview

This chapter describes the hardware and software required for the use of **nVision** and the procedure you must follow in order to install the software components of the **nVision** system.

---

## Computer System Requirements

**nVision 4.40** requires the following *minimum* computer configuration :

- An IBM Personal Computer, or 100 percent compatible, running *Microsoft Windows 95/98/2000/NT/ME/XP*.
- A better than VGA Monitor. (A resolution of at least 800x600 is recommended with a minimum of 256 colours, 1024x768 with 16bit colour would be preferable.)
- An Intel Pentium processor running at 800MHz. (266MHz Pentium II recommended minimum with a maximum of as fast as you can afford!)
- 32 MB of system memory. (512MB of memory is recommended where larger data sets are to be handled.)
- A hard disk drive with 40+ megabytes of free disk space for the **nVision** system, in addition to that required for *Windows*.
- A CD-ROM drive to load software.
- A printer or other output device such as a plotter, if you want to produce hard copy.
- A mouse, digitising tablet or other form of pointing device which is supported by *Windows*.
- A Soundblaster-compatible sound card if you wish to replay or record audio signals within **nVision**.

*For further information on the hardware that you need to run Microsoft Windows, please refer to the Microsoft Windows documentation.*

---

# Installing nVision software

## Before you start

The **nVision** set-up program is a *Windows* application, so you will need to have installed *Windows 95/98* prior to the installation of **nVision**.

*NOTE:* Installation of **nVision** is achieved through its set-up program. The installation cannot be performed by the manual copying of files from the distribution CD, since these are held in compressed form.

## Performing the installation

- 1 **nVision** can be installed from floppy discs or CD-ROM. These instructions assume that the software is being loaded from disc drive **A:**. If you are loading software from CD-ROM then substitute the appropriate drive letter (usually **D:** or **E:**) whenever the instructions refer to 'drive **A:**'.



*floppy disc installation*

Insert the disk labelled “#1/Setup” into Drive **A:**.



*CD-ROM installation*

Insert the CD-ROM in the drive. This is usually drive **D:** or **E:**.

The CD-ROM drive letter can be found by clicking on the My Computer Icon, and finding the CD-ROM icon.

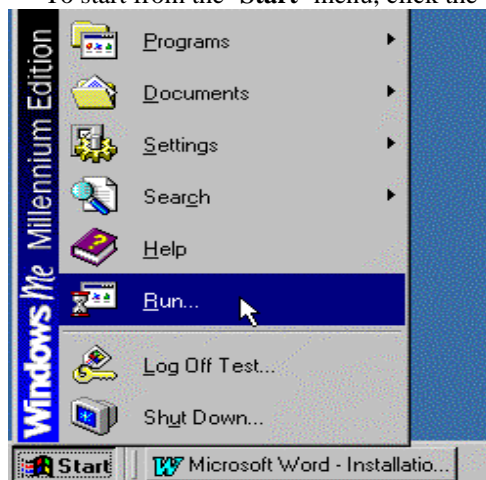


- 2 You can start the set-up program either from Windows Explorer *or* from the ‘**Start**’ Menu. To start from Windows Explorer, skip to section 5. The CD will also auto-start a setup program if Windows is configured to auto-play CDs.

To start from the ‘**Start**’ menu, click the ‘**Start**’ button and choose ‘Run...’.

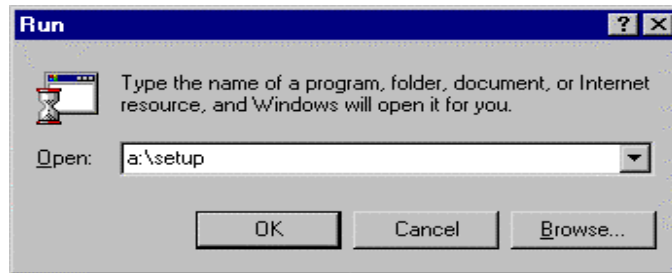


*Running the set-up program from the ‘**Start**’ menu.*




3 When the 'Run' dialog box appears, type "a:\setup", as shown below.

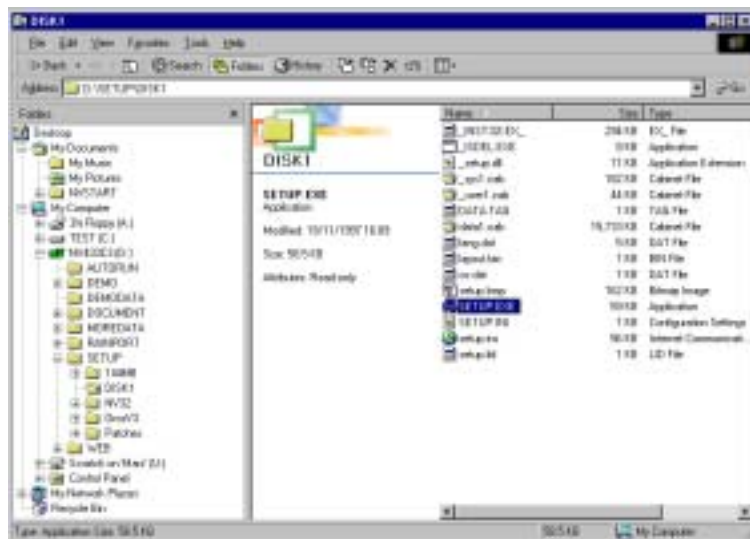
*Run Dialog box from Start menu.*



4 Press Enter or click on the 'OK' button. Now jump to step 6.

 *Running Set-up from Windows Explorer*

To run the set-up program from the Windows Explorer, select the distribution disk drive letter, and double click on the "setup" file:



6 The nVision initialisation dialog will appear. After several seconds the nVision set-up screen will appear with the set-up dialog box:

*nVision set-up program*



7 You can change the displayed installation directory as required.

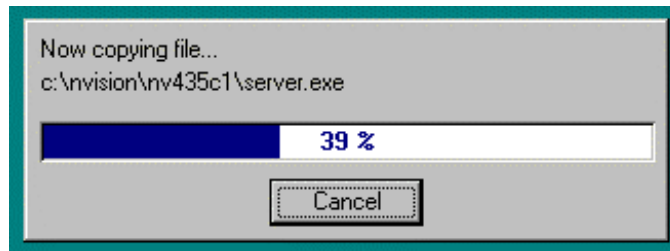
*nVision installation path*

We recommend that you make a note of the path where you are going to install **nVision** for future reference.

8 Click on the 'Next' button.

After a few more questions, the installation of **nVision** will begin with the status dialog being displayed, which shows the percentage of the installation completed.

*nVision status dialog*



When installing from floppy disks the set-up program prompts you for each disk in turn. Place the requested disk in drive **A:** and click on 'OK'.

When set-up has installed, it will create an **nVision** folder in the program menu similar to the one shown below.

*nVision Program Folder*



Finally the installation complete dialog will appear, as shown below. Click on 'OK' to complete the installation.

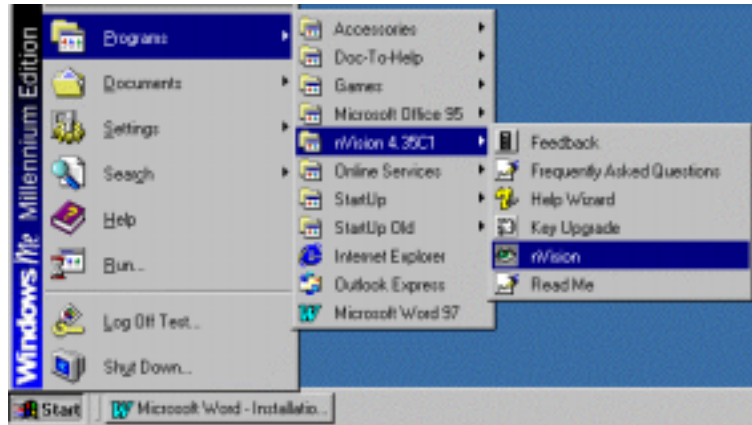
*Set-up complete dialog*




## Additional information

Once the system has been successfully installed, the **nVision** program folder will show two files that contain additional information about the current release. The files also appear in the '**Start**' Menu. Simply click on the icons to open the files:

Start Menu



 Read Me

README.WRI contains information about new features, hardware and technical issues, and any other information, which post-dates the manual.


 Frequently Asked Questions

*Frequently Asked Questions* (FAQ.WRI) contains a list of frequently asked questions customers have about **nVision**.

The program folder also contains the following entries :

 nVision

The *nVision* server - which is used to start up **nVision**.

 Key Upgrade

The "key upgrade" icon - which allows the user to perform a field upgrade of the hardware key - see Appendix A for details.

If the set-up program detects that you have Microsoft Visual BASIC installed on your system, you will also see the *Sample Visual Basic Program* icons - example Visual Basic programs to help you customise **nVision**.

---

## Once you have installed nVision...

### If you have a hardware key

If you have already purchased **nVision** and have a hardware key, then at this point you may wish to install the hardware key. This procedure is described in Appendix A.

The hardware key is not essential to follow this tutorial, since without it the software will still run, albeit in demonstration mode (see below). You may therefore install the hardware key later on. One benefit of installing your key now is that you will avoid the demo dialog appearing during your tutorial session (see below).

### If you do not have a hardware key

You can still follow the tutorials shown in this “Getting Started”, but you will **not** be able to print, save data, or use data other than special data sets marked as demo data.

A demonstration session of **nVision** is also time-limited, and will stop running with a warning message once this time limit is reached.

If you have installed the full release, complete with hardware key, the system will function continuously without the demo message, as long as the key is left in place.

*Demonstration mode Dialog*



A floating 'Demo Mode' dialog informs you of the time remaining.

## Which section do I read next?

### **Windows Basics**

If you are not familiar with the *Microsoft Windows* environment, you should read the chapter entitled “*Windows 95 Basics*”, which gives a brief overview of some of the skills needed to get you started. If you are happy using *Microsoft Windows* then you can skip this chapter.

### **nVision Basics**

If this is your first exposure to **nVision** then you should read this chapter, which gives a brief overview of some of the elements and features of the system. If you are an experienced **nVision** user, you can proceed directly to the tour that interests you.

### **Specific tours**

To find out more about the various parts of **nVision**, you can work through those tour sections that relate to your installed system. Remember: if you are working with a key attached then the system will display only those modules that you have purchased. To work through tours that you have *not* purchased, simply remove the key and restart **nVision** in demo mode!

# Windows 95 Basics

---

## Overview

If you are not familiar with the *Microsoft Windows* environment, this chapter shows you where to find help about using Windows.

---

## What is Microsoft Windows 95?

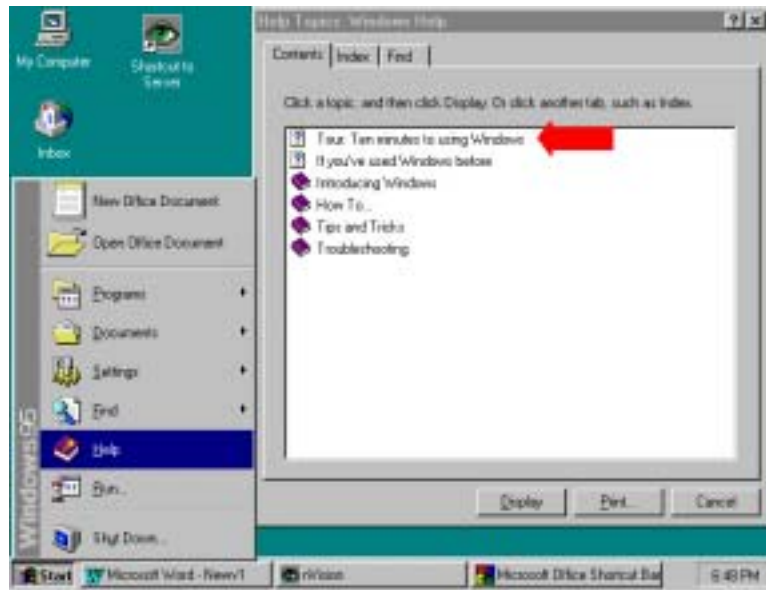
*Microsoft Windows 95* is a Graphical Operating Environment used with IBM PC and compatible computers. It has the ability to run more than one application at a time and transfer information between applications.

*Microsoft Windows 95* is what is known as a WIMP system - WIMP stands for **Windows Icons Menus Pointers**, which are the four main features of *Windows*. Once you have mastered the basic use of mouse operation all *Windows* applications work in the same way.

If you have never used *Windows* before, you are strongly advised to make use of the tutorial guide “Ten minutes to using windows” which is found on the *Windows 95* CD-ROM.

If you have previously used *Windows 3.x* there is also a help section to help you make the transition to *Windows 95*.

To read these guides click on ‘Help’ in the ‘**Start**’ menu.



## Features of Microsoft Windows 95

### Features of a Window

#### Features of a Window



- The Control Menu is used for re-sizing, moving, maximising, minimising and closing windows.
- The Title Bar shows the name of the application and is a different colour for the currently active application.
- The Window Title is usually the name of the application, but can be combined with the name of a document or file.

- The Menu Bar lists the available menus from which you can carry out various actions.
- The scroll bars are used to view unseen portions of the contents of the window.
- The 'Close' button is used to end an application when you have finished with it.
- To change the window's size to fill the screen, click on the 'Maximize' button with the mouse pointer (or cursor). After maximising, this button is replaced with the restore button. Clicking on the restore button will restore the size of the window to its previous size.
- The 'Minimize' button can be clicked to reduce the window into an icon.
- The window border is the outside edge of the window. Clicking and dragging over this area will allow you to change the size of the window.
- The window corner can be used to change the size of two adjoining window borders at the same time.
- The insertion point shows where you are in a document, and where the next character will appear.

## Features of the Pointer

The mouse pointer (or cursor) is the main control device within *Microsoft Windows*.

You move the mouse pointer around the screen using the mouse and can perform various operations with it such as clicking, double-clicking, dragging around the desktop and selecting. These actions will have different effects depending on what is being pointed to at the time.

## Features of the Taskbar

The taskbar is used in *Windows 95* to switch between different programs that are running at any one time. Because it is a multi-tasking environment, you can run lots of programs at once and switch between them using the taskbar.

Simply click on the rectangle that corresponds to the program to which you want to switch. *Windows 95* then brings that application to the front of your desktop.

The '**Start**' button is always present, and this can be used to get to the program you want to run, to find a file, get help or shut the computer down.

The time is displayed in the bottom right corner, along with various symbols that depend on your own machine's configuration.

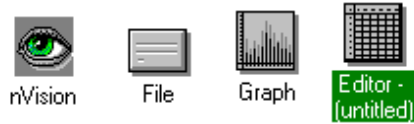
*The Taskbar*



## Features of Icons

Icons within *Microsoft Windows 95* are used to represent a file or application on the desktop or in a folder's file list.

### *Sample Icons*



- Double clicking an icon on the desktop or in Explorer will launch that specific application or file.
- You can drag icons around the desktop.
- You can create shortcuts to applications and files that are represented by icons on your desktop.

## Features of Menus

A window's menu bar is located immediately below the window's title bar.

Menus allow the user to select an item that performs a specific operation. You will see several standard menus, such as 'File', 'Edit' and 'Help' in quite a few applications.

Simply click on the chosen menu option to perform it.

---

# Basic *Windows* Techniques

## Mouse Techniques

There are four basic *Windows* techniques using a mouse:

**Clicking** Involves a quick press and release of a mouse button. Mainly used for selecting and moving the insertion point, selecting menu options and using the taskbar. The right mouse button can often be clicked to bring up a menu of relevant actions.

**Double Clicking** Involves clicking the mouse button twice in rapid succession. Mainly used for launching applications by double clicking the left mouse button over icons.

**Dragging** Involves holding down the mouse button while moving the mouse. Mainly used for 'Drag & Drop' operations which involve picking up an object such as an icon, dragging it to somewhere else on the desktop and dropping it by letting go of the mouse button.

**Pointing** Involves moving the mouse pointer on the screen until it is over the item of your choice.

## Window Techniques

You can perform several operations on a window:

**Moving** To move a window you need to **drag** the window's title bar.

**Sizing** To resize a window, point to either a corner or a border and **drag** it.

**Minimising** To reduce a window to the taskbar, **click** on the window's Minimise button.

**Restoring** To restore a window to its previous size when it has been minimised, you need to **click** the relevant button on the taskbar.

**Maximising** To maximise a window to the size of the desktop you need to **click** on the Maximise button.

**Closing** To close a window (to quit an application) you need to **click** the window's Close button.

## Accessing Menus

To access options on a menu in *Windows 95* you need to perform the following operation:

**Point** the mouse pointer at the name of the menu on the menu bar you wish to access.

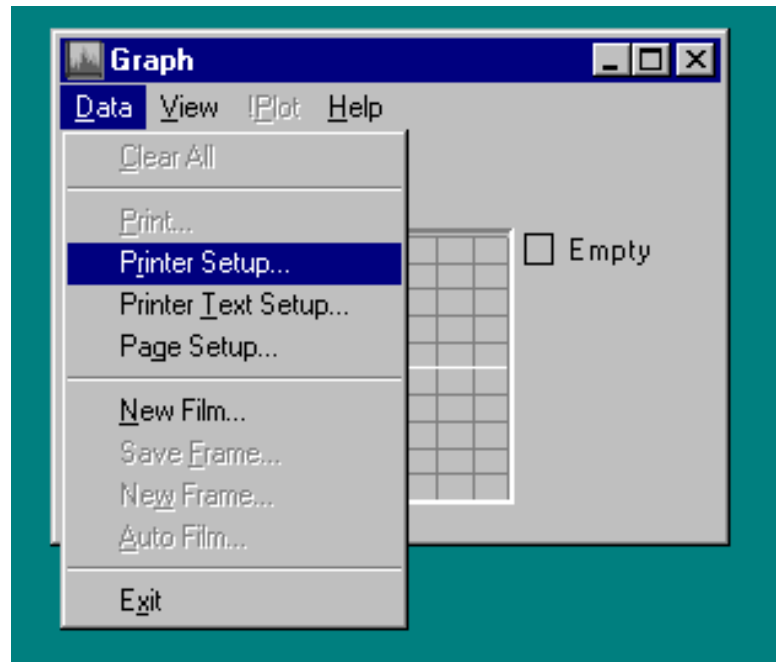
**Click** the left mouse button to open the menu.

**Move** the selection cursor down until the menu item you want to select is highlighted and then **click** the left mouse button.

To close a menu **click** on the menu name or anywhere outside the menu.

Items on a menu which are 'greyed out' are not available for selection.

## Sample Menu



## Accessing the Control Menu

To access the control menu in *Microsoft Windows 95* you need to click the left mouse button on the icon in the top left corner of the window's title bar. Alternatively click the right mouse button anywhere along the window's title bar. This will display the control menu from which you can drag the selector bar over the required option. Generally there are 6 options on the control menu.

Restore - used to restore the size of the window.

Move - used to move the window.

Size - used to resize the window.

Minimize - used to minimise the window.

Maximize - used to maximise the window.

Close - used to close the window.

## Control Menu

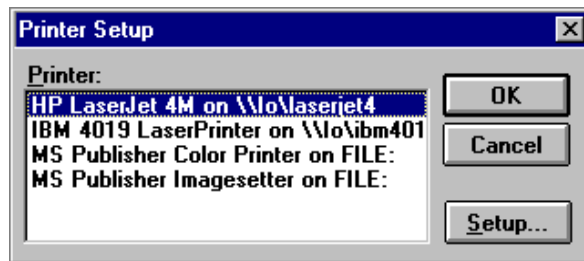
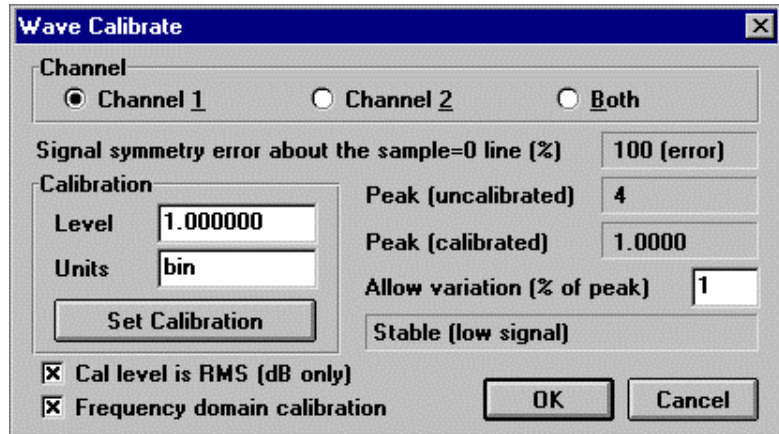


You can select a minimised window's control menu by pointing at the relevant button on the taskbar and clicking the right mouse button.

## Dialog Boxes

Windows 95 uses “dialog boxes” to request information from the user. A menu option indicates that it will result in a dialog box being displayed by suffixing the option name with an ellipsis (...).

Sample dialog boxes



### Command Button



There are six basic controls commonly found on dialog boxes :

You click on a command button to perform an action. ‘OK’, ‘Cancel’ and ‘Help’ are common command buttons

### Text Box



You use a text box to enter textual data such as filenames or numbers etc.

### List Box



A list box is used for selecting an item from a number of choices.

### Drop-down List



The same as a list box except you first have to click on the control before the list becomes visible. You can only see the selected item when the list is not visible.

### Radio Button



Used for selecting one option from a group of mutually exclusive options.

### Check Box



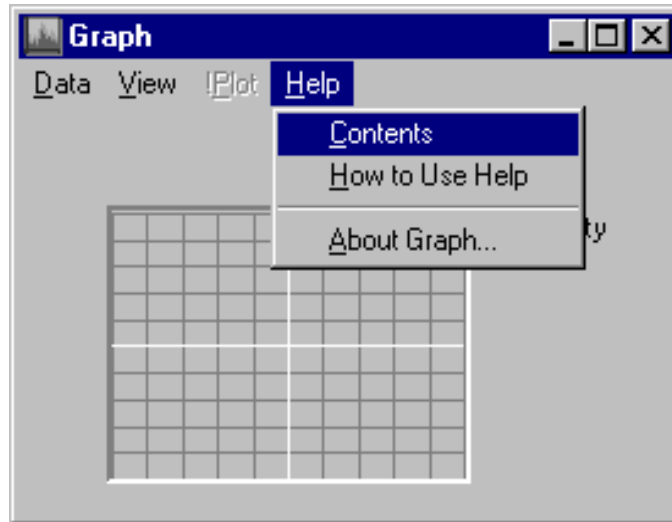
Used for selecting or deselecting an option independent of other options.

## Windows Help

*Microsoft Windows 95* has its own on-line help system from which you can look up information about the program it is attached to. This feature is generally available wherever you see a 'Help' command button or a 'Help' option on a menu - usually the menu labelled 'Help'.

In most of the **nVision** applications that have a help file, you start up the help system by selecting the 'Contents' menu option from either the 'Help' menu or from the control menu.

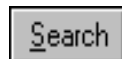
*Help menu options*



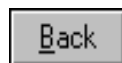
Once you have selected a help option you will be presented with the standard *Windows* help window. In this window, there is a row of command buttons just below the menu, which allow you to access the features of *Windows* help. These buttons are described below.



Displays the Contents of the current help file.



Accesses the search dialog for finding a specific topic.



Backtracks through previously displayed help topics.



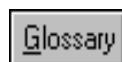
Prints the current help topic.



Views the previous related topic in the help file.



Views the next related topic in the help file.



View a glossary of terms used in *Windows 95* help.

# nVision Basics

---

## Overview

If you are not familiar with nVision, this chapter is aimed at giving you a brief overview of some of the systems features, and a few of the skills needed to get you started.

---

## Features of nVision

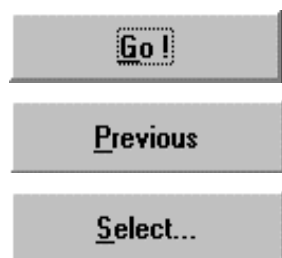
We'll start by running the **nVision** server and having a look at some of the system's visible features.

Double click on the **nVision** icon (the green eye) in the **nVision** Program Folder, or select it via the *Start* menu. The main start-up dialog will appear, allowing you to select from a number of start-up options.

*nVision start-up dialog box*



The exact appearance of this dialog will depend on the system that you have installed, and whether or not you have run the system before, but the 'Go!' button should always be enabled.



'Go!' starts nVision using a default "workspace layout", that is, the server reloads the windows layout, data and server options stored in a DEFAULT file

'Previous' starts nVision in exactly the same workspace configuration as the last time you used it, including applications and data objects that were loaded.

'Select...' allows you to start **nVision** with a workspace configuration that you have previously stored. (See section on Workspaces.)

## Appearance on start-up

For now, just click on the 'Go!' button, and we're in business.

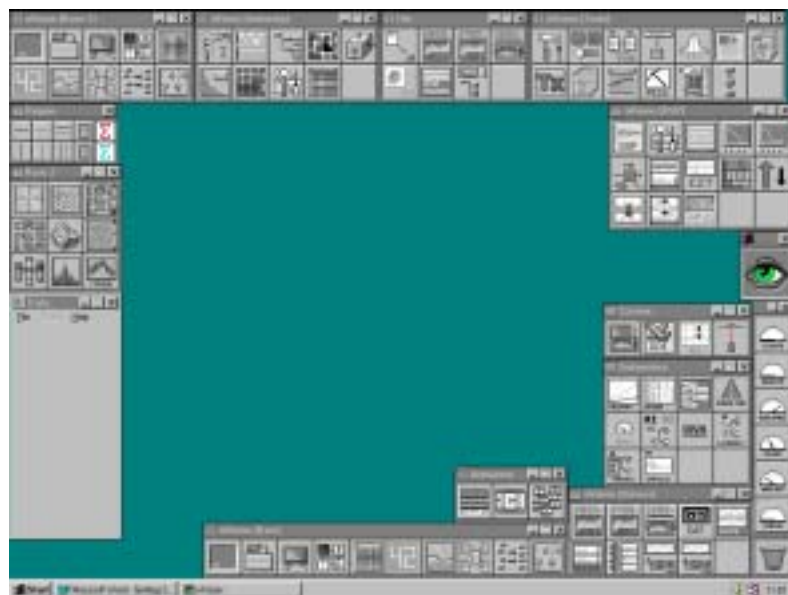
*The exact screen layout will vary from system to system, but **nVision** should look something like this when you first start up.*



The exact layout you adopt in general usage will vary according to your personal taste and the capabilities of your graphics system. We generally find that it's easiest to work with the toolbars and other control windows arranged on the edges of the desktop, leaving the central area free for data windows.

*This somewhat busier screen shows **nVision** configured to display ALL of the system windows and toolbars (except the tabbed toolbar).*

*If you want to switch on more toolbars, access the 'Windows...' menu option on the Server command menu.*



## Basic system windows

Now that the system is up and running, we need to become familiar with the names of the basic system windows which have appeared. These are not the only system windows, but they're the only ones that are essential for easy operation.

*The nVision server icon.*



Starts the **nVision** server, and controls loading modules. Once started, this icon should appear on the taskbar.

*A toolbar.*



The toolbars contain buttons to start each application program. Programs may be started by *double clicking on the required toolbar entry*, or by dropping data objects onto the icon.

*The “Trashcan”.*



This window may be used to dispose of unwanted data objects by dragging them to the trashcan and dropping them there.

*The “Data” window.*



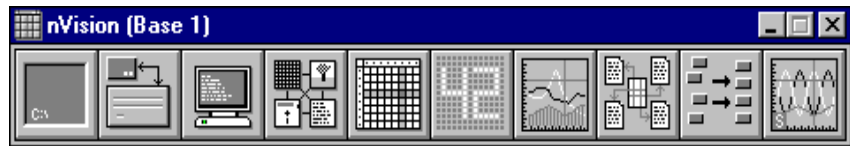
Data objects appear here when loaded from file or created by an external driver.

These four objects (at least) should appear by default, unless of course the system has been otherwise permanently configured. We will meet other windows later.

If none of these windows appears at this stage, refer to Appendix C, which describes how to set system options, and make sure that the ‘Tabbed’, ‘Trashcan’ and ‘Data’ entries are checked in Server’s ‘Options’ / ‘Windows’ menu.

## Toolbars

*Toolbars provide an easy way to execute **nVision** modules*



Toolbars offer an easy way to execute **nVision** modules. Each major division of **nVision** (Base 1, Base 2 and so on) has its own toolbar that contains the appropriate modules. Double clicking on the icon starts the module.

*The tabbed toolbar*

Alternatively, there is a ‘tabbed’ toolbar, which contains the modules for the entire system in one compact toolbar.

Each division of **nVision** can be selected by clicking on one of the tabs:



For the sake of clarity, we’ll use the tabbed toolbar for the tutorials. For a rundown of the contents of each separate toolbar, please refer to the “Icon Index” at the back of this guide.

If you have difficulty remembering which toolbar icon is related to which module there are two ways to find out :

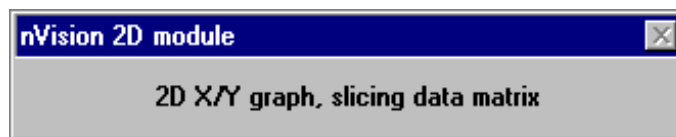
*ToolTips help identify the program modules*

When the mouse pointer is left over an icon, “ToolTips” help will appear to identify the program name.



*Pop-up help describes the function of each module.*

Holding down the left mouse button for a short time on the toolbar icon will display a pop-up box with a short description of the module.



On some displays, the width of the tabbed toolbar may be wider than the screen. In this case, the toolbar displays an arrow as the final icon; clicking on this arrow scrolls the icons across to reveal the other modules that are not displayed.

*Scrolling the tabbed toolbar on a narrow display.*



## The Data Window

Before we can view or manipulate data, it must be loaded into memory. The Data Window is simply a convenient place to represent the data objects that are currently loaded in the system, like a central storage area. It's from here that the user determines where the objects are to be displayed or processed, by dragging the object across to a graph or a toolbox. (See the Drag and Drop description later.)

When you drag data from the window into an application, the data isn't actually moved; we simply create a "link" between the object and the target application. Some of these objects are described below

*The Data Window lists the data objects currently loaded.*



## Data objects

Entries in the Data Window are referred to as data objects. These objects may represent numeric data - such as a time series signal acquired from a tape recorder - or numeric results such as a frequency spectrum matrix. They may even represent system instructions, such as colour scaling, scaling ranges, zoom levels and so on.

When loaded into the Data Window each object type is represented by its own miniature icon:



Matrix

Real-valued numerical array -e.g. a run-up containing spectra at different RPM values.



Complex Matrix

A complex-valued numeric array - such as a complex FFT result.



Film / Animation

A sequence of frames or movie animation.



Frame

Individual frame image.



Signal

A time series, e.g. a stereo audio recording.



Intensity

Intensity measurements job / part / area.



Scalar

A scalar quantity - e.g. a calibration factor.



Palette

Colour palette, e.g. colour scale for a contour plot.



Level

Level object; used for contour scales.



Viewpoint

The viewpoint location, e.g. for viewing a waterfall.



Zoom

The area of data to be viewed.



ASCII

An ASCII text file such as a command script.



---

# Basic nVision techniques

## Overview

**nVision** is an extensive system, and to gain a thorough understanding of it all takes time and patience. However, with an appreciation of some basic principles it's easy to make a start on the tutorial sections in this manual. These provide an excellent foundation, based upon which the curious user can explore and become familiar with the entire system.

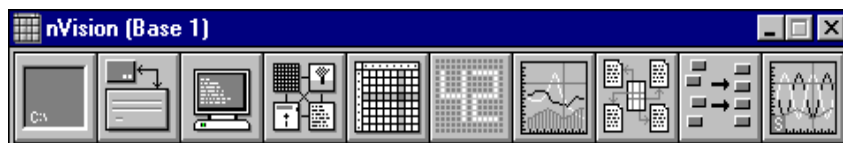
*We recommend that you move quickly through this section to gain a little familiarity with the ideas, and then move onto the specific tours in Chapter 4. Once you have completed the tours that interest you, you can then return to this section with a deeper understanding.*

## Executing Modules

There are two ways you can execute modules. The first way is to execute them from one of the toolbars.

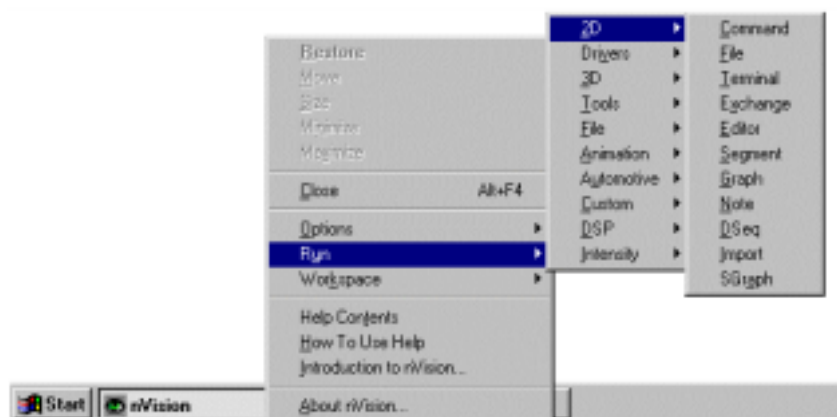
*Toolbars provide an easy way to execute nVision modules.*

Toolbars offer an easy way to execute **nVision** modules, and this method will be used throughout this tutorial. To execute a module from the toolbar, you need to click or double-click on a module entry. The number of clicks required depends on the status of the 'double click to launch' option on the 'Options' / 'Program execution...' menu option.



*Running modules from the server control menu.*

The second method is to use the 'Run' option from the server control menu (use the right mouse button over the icon on the taskbar to access this menu).



## Opening Files

*Opening files*

Go to the Base 1 toolbar and identify the entry that represents the FILE application.



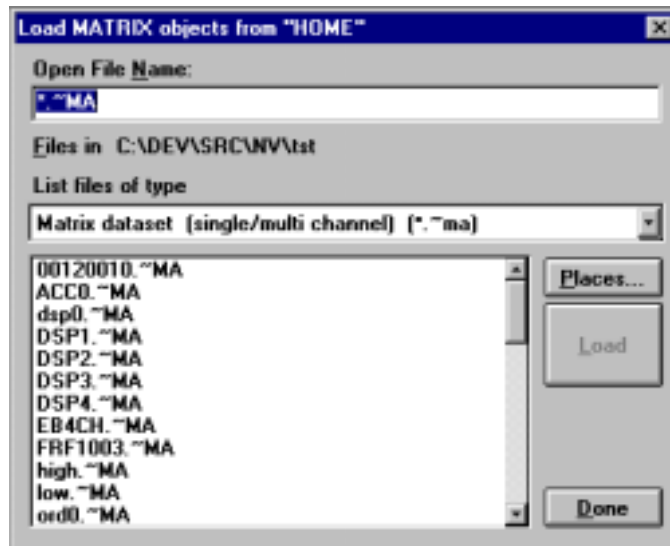
Run this module in order to load data. Double click on the FILE entry, and wait for the FILE window to appear minimised at the bottom of the screen. Click the restore button on this window to bring up the file dialog.

Click on restore button



Alternatively, select the 'Open' command from the 'File' menu of the Data Window.

*Loading files from the LOAD dialog.*



The files supplied fall into a number of categories, as identified by the Data Type list box. For now we will load a **MATRIX** object, which is simply a two-dimensional matrix of numeric values.

Identify the type of object to be loaded in the combo box entitled 'List files of type'. For example, a matrix object as shown. Each data object type is identified by its own unique file extension - for example the matrix files have "**~ma**" extensions.

The available matrix data sets will appear in the window of the box. To load a data set, **either** click on the filename to highlight it, then press 'Load', **or** double click on the file name to perform the load immediately.

Either method loads the data from disk, and creates a matching data object in the Data Window. While the data is being read from disk, the pointer will change to a book.

Load more files if required, and click the 'Done' button when finished to close the dialog box.

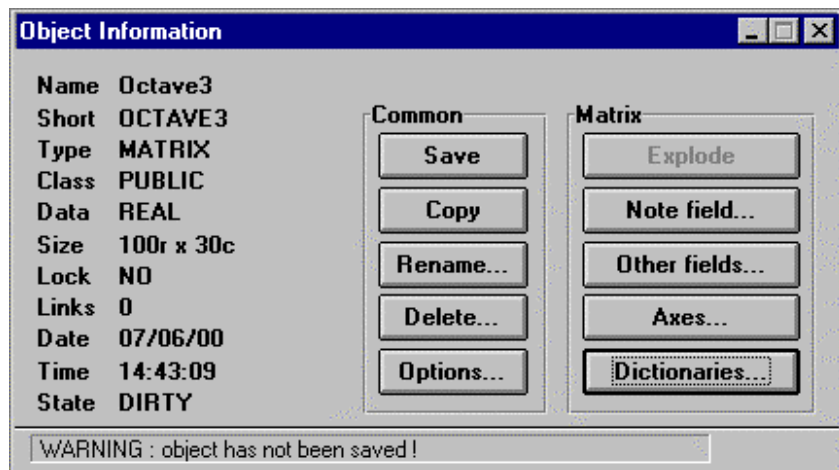
## Quick loading of objects

You can reach the load dialog more directly by clicking on the 'File' menu entry located on the Data Window and selecting 'Open...'.

## Getting information about data objects

Data objects loaded into memory are shown in the Data Window. You can find out more information about objects visible in the data window by double clicking on the object in the Data Window. This calls up the Object Information dialog:

*The Object Information dialog*



In addition to showing various information for the selected object, the Object Information dialog also has buttons that allow you to Copy, Rename or Delete the object. If the object has changed since it was last saved then you can save it by clicking on 'Save'.

## Saving data objects

There are four ways to save data objects in **nVision**.

- 1 You can drag the object from the data window and drop it over the minimised FILE window.
- 2 You can select 'Save' from the 'File' menu in the Data Window.
- 3 You can go into the Object Information dialog for the object and click on the 'Save' button.
- 4 When exiting **nVision** you will be asked whether you want to save any data objects that have not been saved.

## Removing data objects from the system

### *The nVision Trashcan*



The easiest method for disposing of data is to drag the data object from the Data Window and drop it over the Trashcan. This will remove that data object from memory.

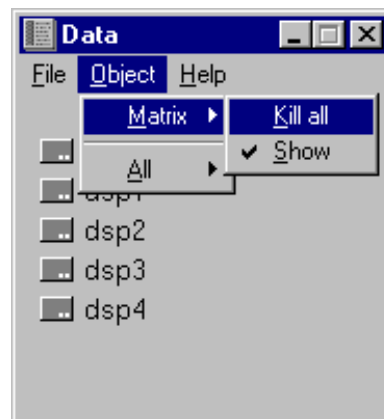
Depending on the 'Confirm Trashcan Deletions' option in the 'Data' dialog in server's options, you may be asked if you are sure that you want to delete the object.

There are two other ways to remove data objects from the system:

In the Object Information dialog for the object, click on 'Delete...'

To remove multiple objects from the system: From the Data Window's 'Object' menu, select which type you want to remove, or 'All' to remove all types. Then select 'Kill All' from the pop-up menu.

*Removing all the matrix objects from nVision. (This does not delete them from the disk.)*



### **“Clean” and “Dirty” objects**

Objects that have not yet been saved are marked as “dirty”, and if you try to remove them from memory, a warning will be given that your actions will actually destroy data.

Once saved to disc the object is “clean” and no warning is given as it can be reloaded on demand.

System options allow these warnings to be overridden, but for safety's sake we advise that you leave the warnings active, at least until you are familiar with the operation of the system.

## Drag and Drop

**nVision** uses a “**Drag and Drop**” system to create links between data objects and program modules.

A Drag and Drop operation consists of three steps:

- 1 With the mouse pointer over the relevant object in the data window, press and hold down the left mouse button.
- 2 With the left mouse button held down, **drag** the mouse pointer - which is now holding the data object - over to the module where you wish to use it.
- 3 With the mouse pointer over the module you want to use, release the left mouse button. This will **drop** the object into the module.

This explains the mechanical process required to “Drag and Drop”, but the question which remains is “What can you Drag and where can you Drop it in **nVision**?”

You will be Dragging and Dropping data objects. As you already know once you have loaded an object into the system, it appears in the Data Window. From this point, it needs to be linked to one or more **nVision** modules.

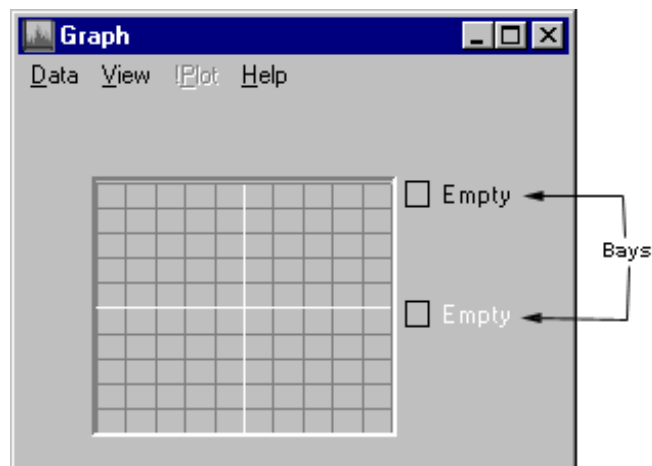
If the module into which you are placing data is already loaded, then you can just drag the data over that window’s client area and drop it.

If the module is not loaded then you can drag the object from the data window over the toolbar icon of that module and drop it there. If the module can work with that type of object then it will be executed and the data object will be automatically linked to it.

### **Working with “bays” when using drag and drop**

Some program modules work with simultaneous links to multiple data objects - for example GRAPH on the Base 1 toolbar can display eight different data objects, and so supports eight simultaneous links. Each link is represented by a *bay*, where data objects can be dropped.

*The GRAPH module showing two of its eight bays.*



If you drop an object over a module with bays but you don’t make the drop into a specific bay then the object will be linked to the first available (empty) bay. If you drop an object over a specific bay, the object will be linked to the chosen bay, and any existing link will be replaced.

## **Drag and Drop from Windows Explorer**

You can also load files into the system using the *Windows Explorer* application. To do this, highlight the file(s) you want to load in the Explorer Application, then drag them over to the **nVision** Data Window and drop them.

The native **nVision** files in the selection will be loaded into the system, whilst any which are not **nVision** files will be ignored.

## **Getting help**

The **nVision** system has a number of built in help files that use the **Windows** help system. Depending on the type of module you are using you will generally find the option to access help on either:

A 'Help Contents' option on the 'Help' menu of the application.

A 'Help Contents' option on the application's control menu.

The help files are grouped in terms of toolbars, so selecting help for GRAPH will take you to the help section for GRAPH in the Base 1 help file. The same goes for the Base 2, Tools, Drivers, DSP, Intensity and Custom toolbars.

## **The nVision Workspace**

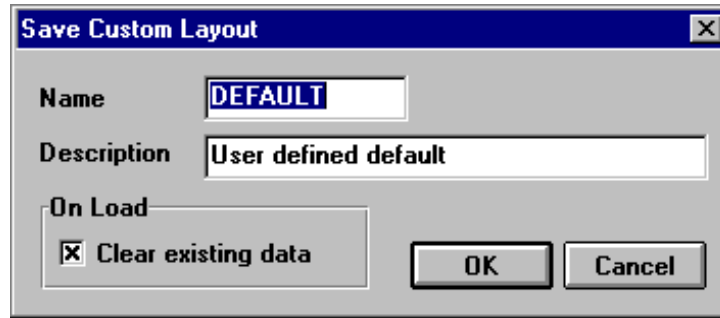
The nVision Workspace comprises of the layout of your windows, the active client modules and the loaded data objects. Storing the workspace saves time and effort in carrying out your analysis. For example, you may wish to have one screen layout suitable for data acquisition and validation, and another for subsequent analysis.

## **To Save the Workspace Layout**

- Move the toolbars and other modules to where you would like them on the screen.
- Click the right mouse button on the server's icon on the taskbar to get the system menu.
- Next, select 'Workspace' and 'Save...'. The 'Save Custom Layout' dialog will appear. The name should say DEFAULT. If not, type it in the name field.
- Type "User defined default" in the description and press 'OK'. Next time you open **nVision** and click 'Go!', the system will start up with the screen layout you just saved.

Of course, you can use other names and other screen layouts. You may want certain data objects and graphs always to appear. You can define as many of these layouts as you want. To recall a screen layout, other than DEFAULT, use 'Select' on the 'Start-up Options' window.

Save Custom Layout dialog.



At the end of a session the current workspace is automatically saved under the heading “**Previous**”, so it’s always possible to return the system to the previous Workspace state, i.e. as it was when you last closed down.

You can save the current workspace under your own heading and later restore that state on demand, either during system start-up or even in the middle of another session.

The server system menu contains a number of workspace entries:

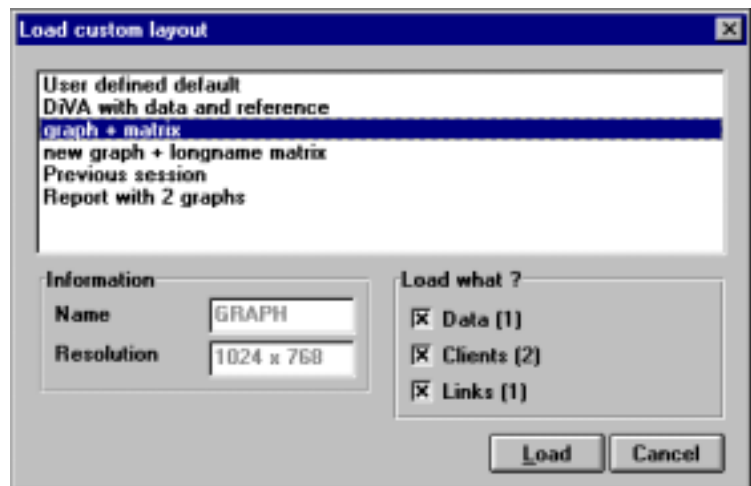
‘**Workspace**’ / ‘**Backdrop**’ removes all non-**nVision** windows from the desktop. Selecting this again redisplayes them.

‘**Workspace**’ / ‘**Hide Windows**’ removes all **nVision** windows from the desktop. Selecting this again will redisplay all **nVision** windows.

‘**Workspace**’ / ‘**Save...**’ Saves the current Workspace including window locations, and which modules and data objects are loaded.

‘**Workspace**’ / ‘**Load...**’ Loads a previously saved layout. Any data or modules that were loaded at the time will be loaded.

Load Workspace Dialog



**Hint:** Since you will almost always need to load or save data objects make sure that your **DEFAULT** workspace includes the FILE module.

## Changing the shape of the toolbars

To enable you to create a screen layout that suits you, the shape of the toolbars can be changed (except the Tabbed toolbar). The ‘Format’ option from the toolbar’s control menu changes the toolbar shapes; each time this is selected the toolbar toggles to the next shape, eventually returning to the original format.

Changing the toolbar format.

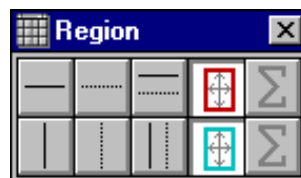
The Base 1 toolbar showing three of the possible configurations.



## Working with Regions

The REGION Window allows you to identify a range of useful data in a matrix. For example, you might want to determine which area of a data set is processed by a particular function.

Region Window



As an example, you could copy a subset of a data object to a new object:

- Plot the data using CONTOUR.
- Use the mouse to zoom in on the area of data that you want to keep.
- Now select the buttons with the solid rectangle in the upper and lower rows of the REGION Window, as in the table above.
- Open the 'Object Information' dialog by double clicking on the data object in the Data Window.
- Press the 'Copy' button.

A new data object, consisting of the subset of data in the zoomed area is created. The system automatically provides a name for the new object by adding a letter 'C' (for copy) to the front of the name. You can rename it if you wish.

*Hint: By using Region, it is possible to copy data from a single line in either axis, a single point, the area between the two cursors or the zoomed area.*



# Guided Tours

---

## Before you begin

### Easy introduction

The following guide uses the various features of the system in the simplest possible way, but in the interests of brevity does not necessarily explain the why's and wherefore's at each stage. Introducing **nVision**'s more powerful features here would tend to mask the basic simplicity of the system.

### Which “tour” sections do I follow?

Since **nVision** is a modular system, there is no guarantee that every user will purchase every module, and consequently there will be sections of this guide that may not be relevant to you.

If you have a hardware key attached, this will limit the modules visible on the toolbars to those that you have purchased. If you are operating without a key then most of the modules will be available but, obviously, they will only be running in demo mode.

If your system supports only Base 1 and Base 2, for example, you need only work through the “Basic Principles” section and then the “Base 1 Tour” and “Base 2 Tour”. However, should you be interested in investigating the other sections, you can do so even if your key doesn't currently support them. You may be thinking of extending your system to include these modules, or you may simply be curious. Just remove your hardware key and restart **nVision** in demo mode!

### Session persistence

Sometimes during these tours, the examples that appear on your screen may not match exactly with the examples shown. Don't be too distracted by this. The reason is that **nVision** has a long memory, and if you or someone else has been using the system prior to your tour, system defaults may have been changed. Though this feature may seem a bit of an inconvenience during your tour, you will find that it is a major benefit when you get down to some real work.



---

# Base 1 Tour

## Overview

This tour will use some of the simpler facilities available from within **nVision**, to give you some basic “hands-on” experience. We’ll identify and then load one of the sample data sets, display its contents in a number of different ways, and then explore the use of zooming and cursors, and how they can assist in the analysis procedure.

Some of the basic techniques are repeated here where necessary; but as we proceed through the various tours, this “hand-holding” will decrease.

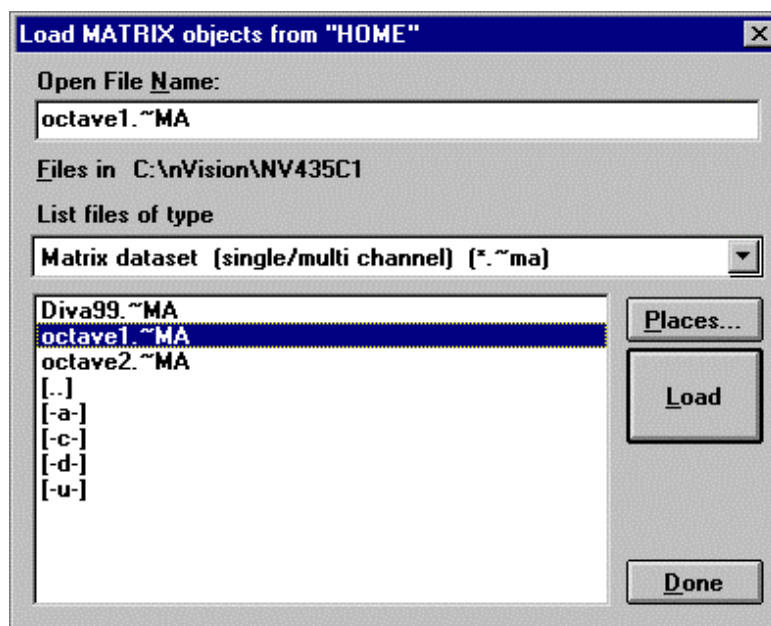
## Identifying and loading the data

Go to the Data Window and select the ‘Open’ command from the ‘File’ menu. This brings up the file dialog box, from where we can load any of the test data supplied with the system.

Select “Matrix dataset” from the data type list if it is not already selected, and a list of available matrix files will be displayed in the scrolling list box.

A file may be loaded by clicking on its name in the list box and then pressing the ‘Load’ button, or by double clicking on the list box item. If the file you want isn’t visible, you can move it into view using the scroll bar.

*File module ‘Load’ dialog - selecting a specific file corresponding to the selected object type, here we’re choosing a matrix object*



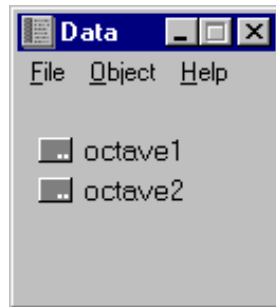
Select and load the file “**octave1.~ma**” now, using either the single click or double click method. During loading, the cursor will change into an open book, indicating that the system is reading from disk. (When writing data, the cursor changes to a fountain pen.) When this load operation is complete, also load the file “**octave2.~ma**”.

When this file has been loaded, signified by the cursor changing back into an arrow, hit the ‘Done’ button on the File Load dialog box. This closes the file menu, as we don’t wish to load any more data at this point.

We should now have two new entries called “**octave1**” and “**octave2**” present in the Data Window.

Experienced *Windows* users might also like to know at this point that **nVision** supports full file drag and drop, not only internally but also from *Windows Explorer*, so **nVision** data files can be selected by dragging them from the *Explorer* window and dropping them in the Data Window.

*The two test data sets were generated using a Yamaha SY55 polyphonic synthesiser feeding a Larson Davis LD3200 real time spectrum analyser.*

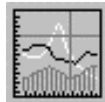


All data in the system is represented in the Data Window, which can hold a maximum of 896 items. Size the Data Window so that it occupies a reasonable amount of screen space, in preparation for the next stage. If you are using a VGA screen for your demo it is fairly important that the windows are kept relatively small, whereas users of 1600x1200 graphics systems need not worry too much!

## Transferring data into applications

Once data has been loaded into the data window, we need to transfer it to other applications in order to view and process it, but as yet we have no display applications running.

*Displaying data the Graph module*



Load the GRAPH module by double clicking on its icon in the Base 1 toolbar. The module should now appear with a blank frame.

When GRAPH appears, transfer the data from the Data Window to the module using **drag and drop** (used throughout the system as the means of transferring data into program modules) as follows :

1. **Position the cursor** over the “**octave1**” object in the Data Window.
2. **Click** the left mouse button and continue to hold it down.
3. **Drag** the object cursor over the GRAPH application.
4. **Release** the mouse button, to ‘drop’ the object into GRAPH.

*Data objects must match the module function for drag & drop to work*

## Module/Data object matches

There are a number of different possible data object types, and a large number of client modules that can display or modify that data, but not every client module accepts all data object types. (In fact none of the applications handles all types.) When experimenting with drag and drop, don't be surprised if the target module beeps, or issues a message rejecting the data you are attempting to drop. The manual contains a full discussion of which data objects are compatible with which client modules.

## Displaying data in GRAPH

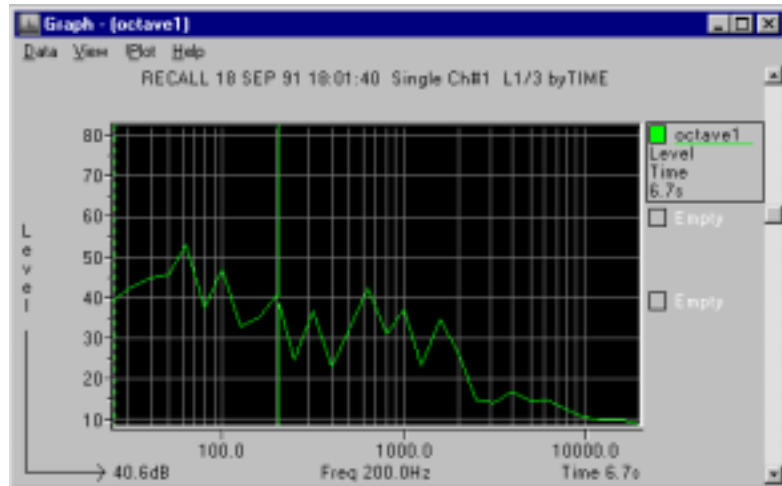
Once the drag and drop operation is complete, GRAPH reacts to the drop operation by linking to the data object, which can be confirmed by observing the title bar, which should now read “**GRAPH - (octave1)**”. In addition to the change in title, the X- and Y-axis entries are displayed, in this case Frequency versus Time.

2D Graph



As yet, no data is visible, (the graph area shows just a grey field with guidelines), so hit the '!Plot' menu option to display the graph. Most of the display applications have this '!Plot' option, since some redraw operations are time consuming, and are not always required immediately. A "switchable" plot allows the user to choose when redraw is required.

Once '!Plot' is selected, a representation of the data appears in the graph area. Frequency appears on the X-axis, and Amplitude in dB is on the Y-axis. Also shown is the cursor, positioned at 6.7 seconds. The current values for time and frequency are displayed at the bottom of the screen, with the corresponding value of the amplitude at this point.



The slice through “**octave1**” shown above is across the Frequency axis, showing Amplitude values at a Time of 6.7 seconds. By clicking on the scroll bar on the right hand side, the slice, or cross-section, can be moved in “Time”. If your graph doesn’t show the Time as 6.7s, move it now, until this value appears. Likewise, you can move the Frequency cursor to 200 Hz if it is not already in the correct position, by using the keyboard arrows.

## Experiments to try out in GRAPH

The data displayed (Level versus Frequency) is a two-dimensional slice through a three-dimensional map. Move through the other slices by using the scroll bar at the right of the screen. This controls the third axis slice, which in this case is Time.

Position the solid cursor by clicking in the graph area with the left mouse button, and observe the changing “pick-off” values as the cursor is moved. Position the second (dotted) cursor by clicking in the graph area with the right mouse button.

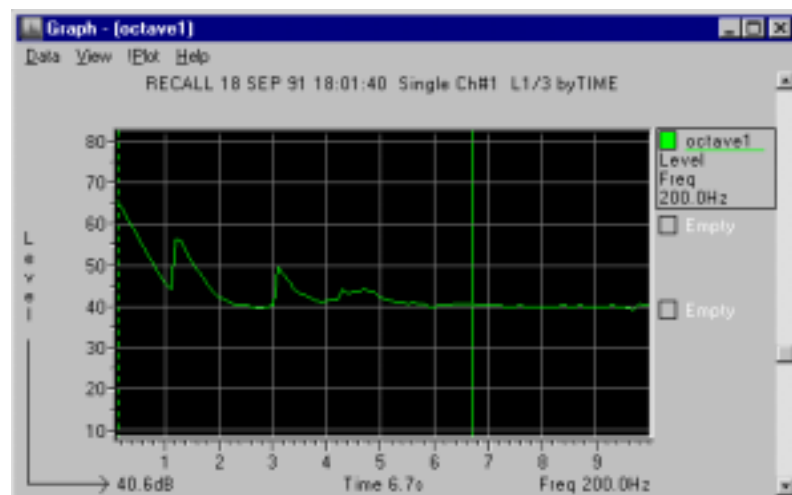
Zoom in to show greater detail by clicking and dragging an area within the graph. Zoom out by selecting ‘Unzoom’ or ‘Undo Last Zoom’ from the ‘View’ menu.

Switch the slicing direction by choosing ‘Level versus Time’ in the ‘View’ menu. (The scroll bar now moves us through Frequency.) Remember to switch back to ‘Level versus Frequency’ before the next stage.

Change the display method by double clicking the right mouse button on the bay at the right hand edge of the graph, where “**octave1**” is displayed. There are numerous display options to be tried here under the ‘Graph Style’ group heading. Note that there are eight of these bays, so we can simultaneously display and compare information from other data objects, or use multiple drops of the same object to compare slices from within the same data set.

## Displaying the alternate slice in GRAPH

In the previous example, using GRAPH, we accepted the default slicing direction (Level versus Frequency) but we may also want to see the other slice. Leave the existing GRAPH module, and start another copy of GRAPH by double clicking on the GRAPH symbol on the Base 1 toolbar. Transfer in the “octave1” data set as before, select the ‘View’ menu option and choose the slice labelled ‘Level versus Time’. This should give a graph something like the one shown below. Note that the Frequency and Time labels have exchanged places, confirming that we have indeed flipped our viewpoint to the alternative slice.



## Hot-Linking

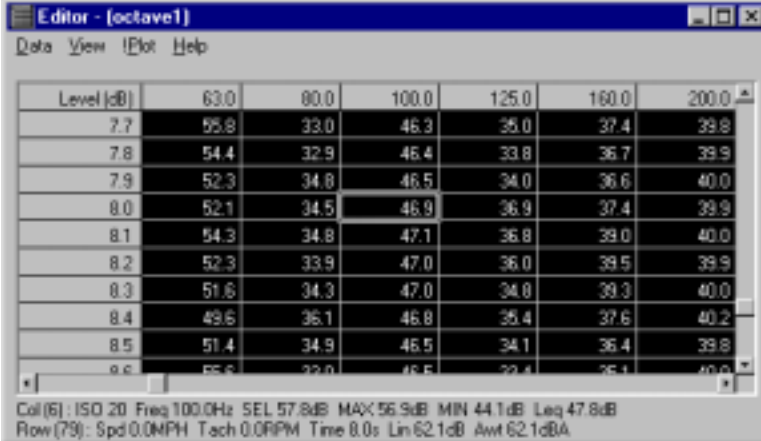
The “Hot-Linking” principle is central to the design philosophy of **nVision**.

If you manipulate the screen contents so that you can see both of the GRAPH applications, you can now move around the data, zooming in or out and moving the cursors, observing the interaction between the two applications. Try moving the cursor viewpoint from within one of the GRAPH applications and watch the slice move in the other GRAPH. We call this **Hot-Linking**.

The applications co-operate like this because they have a common interest; that is, they are connected to the same piece of data. As you move around the data, the Server keeps track of your actions, and tells other applications that are connected to the same data set what you are doing. Each application is known as a Client, and the co-operative system as a whole is known as a Client-Server system.

## Modifying data values

The EDITOR module provides a method of manually altering data values, or even creating data sets from scratch, for comparison purposes. The data set is loaded as before, and appears in a spreadsheet-style grid as floating point values.



The screenshot shows a window titled "Editor - [octave1]" with a menu bar containing "Data", "View", "Plot", and "Help". Below the menu bar is a spreadsheet grid. The first column is labeled "Level (dB)" and contains values from 7.7 to 9.6. The subsequent columns are labeled with frequency values: 63.0, 80.0, 100.0, 125.0, 160.0, and 200.0. The data values in the grid are as follows:

Level (dB)	63.0	80.0	100.0	125.0	160.0	200.0
7.7	55.8	33.0	46.3	35.0	37.4	39.8
7.8	54.4	32.9	46.4	33.8	36.7	39.9
7.9	52.3	34.8	46.5	34.0	36.6	40.0
8.0	52.1	34.5	46.9	36.9	37.4	39.9
8.1	54.3	34.8	47.1	36.8	39.0	40.0
8.2	52.3	33.9	47.0	36.0	39.5	39.9
8.3	51.6	34.3	47.0	34.8	39.3	40.0
8.4	49.6	36.1	46.8	35.4	37.6	40.2
8.5	51.4	34.9	46.5	34.1	36.4	39.8
8.6	55.6	33.0	46.5	33.4	36.1	40.0

At the bottom of the window, there is a status bar with the following text: "Col(6): ISO 20 Freq:100.0Hz SEL:57.8dB MAX:56.9dB MIN:44.1dB Leq:47.8dB Row(79): Spd:0.0MPH Tach:0.0RPM Time:8.0s Lin:62.1dB Awt:62.1dBA".

Cursors can be moved from within the application, and as with all other modules, these changes are reflected throughout the system, wherever the same data set is present. Similarly, blocks may be marked using click and drag (when not in 'Edit Mode'), and when numeric values are edited, the change is updated simultaneously in other applications.

With "octave1" loaded, move the cursors to 1600 Hz and 4.8 seconds, where the value should be 79.8dB. Go to the 'View' menu, and make sure that 'Edit Mode' is checked, then type in "0.0", or some other easily recognisable value. When you hit <Enter> to enter the new value, the GRAPH and EDITOR applications update their displays to show the new value.

Modifying data values manually is fine if there are only a couple of alterations, but for heavier tasks we need slightly more power. The TOOLS modules provide everything we need, and are covered in a separate tour.

---

## Base 2 Tour

### Overview

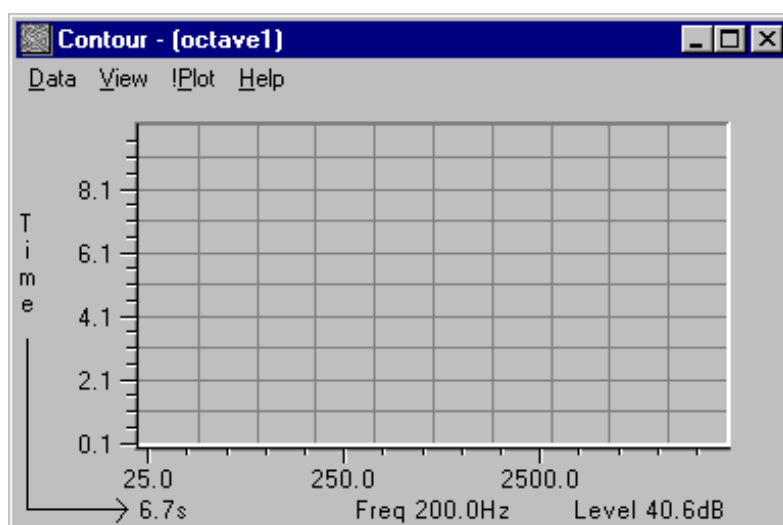
Now we're ready to have a look at some of the display modules available in the Base 2 toolbar. We've already covered the basics of identifying data and drag-and-drop, so all you need to know at this point is that we need to drag the "octave1" data set into the contour module.

*Base 2 - the CONTOUR module.*



### Displaying data in CONTOUR

Once the drag-and-drop operation is complete, CONTOUR reacts to the drop operation by linking to the data object, which can be confirmed by observing the title bar, which should now read "Contour - (octave1)". In addition to the change in title, the X and Y-axis entries are displayed, in this case Frequency versus Time.

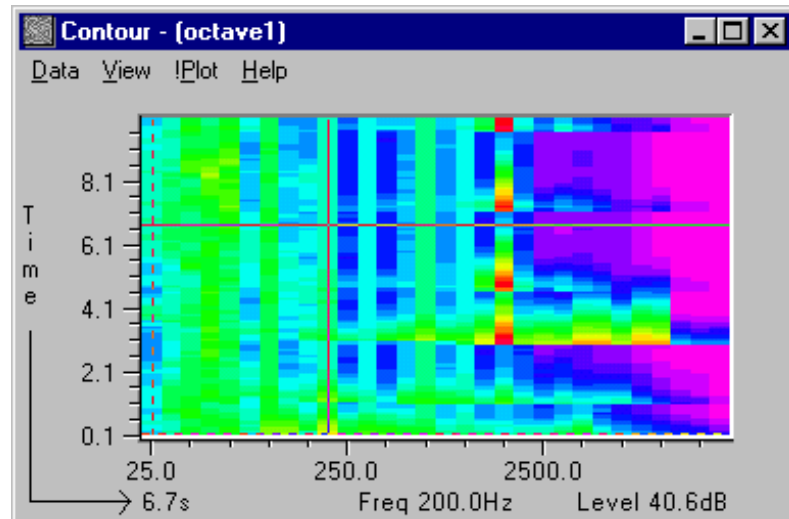


As yet no data is visible (the graph area shows just a grey field with guidelines) so hit the '!Plot' menu option to display the graph. Most of the display applications have this '!Plot' option, since some redraw operations are time-consuming, and not always required immediately. A "switchable" plot allows the user to choose when redraw is required. You can interrupt the plotting of the graph by clicking the mouse anywhere over the graph area.

Once '!Plot' is selected, a representation of the data appears in the graph area. Frequency appears on the X-axis, Time on the Y-axis, and Amplitude in dB is on the Z-axis, which, since we're looking "from above" is represented by the colour of the data point. This matches a contour level key (not shown here) which shows the assignment of colours to contour bands. There are various styles of contour plot available, such as conventional contour lines, (weather

map style), filled interpolated plots, or as a coloured patch (which is the faster plotting default used here).

Also shown are the cursors, one for each axis, positioned at 6.7 seconds in the Time axis, and 200 Hz in the frequency axis. The corresponding values are displayed at the bottom of the screen, together with the Z value at the intersection.



Try moving the cursors around in the graph area, using the arrow keys on the numeric keypad. There are four peak values, as indicated by the reddest areas, all occurring in the 1600 Hz band, at 3.1s, 4.8s, 7.3s and 9.8s, with values of 81.4dB, 79.8 dB, 77.7 dB and 82.3 dB respectively. Try practising moving the cursors now, to see if you can locate each of these values. You can move the cursors more directly by clicking with the left mouse button on the graph at the position you want to highlight.

The two cursors in the example above each represent a slicing plane through the data set, which would yield a pair of conventional X/Y graphs. One graph is Frequency versus Amplitude, and the other is Time versus Amplitude. We can display these slices by taking the same data set and dropping it into the GRAPH application, which will render the data in one of these formats.

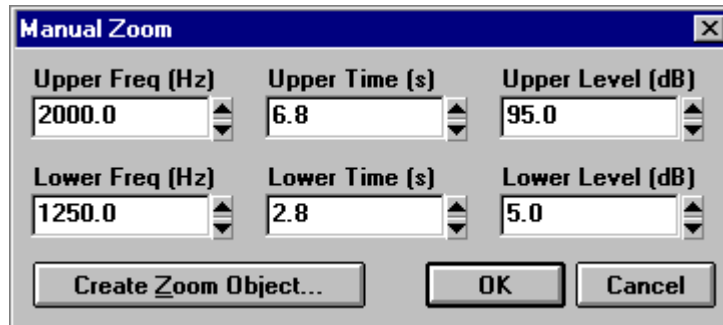
## Identifying a *region*

### ***Click and Drag***

The main area of interest in this data set appears to be those peaks occurring in the 1600 Hz band, which can also be inspected more closely by zooming in using click and drag. Practise this now, in the CONTOUR application, by clicking and holding down on the left mouse button and dragging the mouse right and downward (still holding the button down). As you move, an area is highlighted in inverse colour, which we call a *region*. When you let go of the mouse button, this 'region' will expand to fill the graph area. It may take a few trials before you get the hang of this, so it's as well to know that if you go to the 'View' menu and select 'Unzoom', the data set will return to its original (zoomed-out) state.

## Manual zoom

The click and drag method is ideal for marking a rough region, but for exact positioning, a manual zoom option is also provided. In CONTOUR, click on the 'View' menu item, and select 'Manual Zoom...' to obtain the dialog shown:

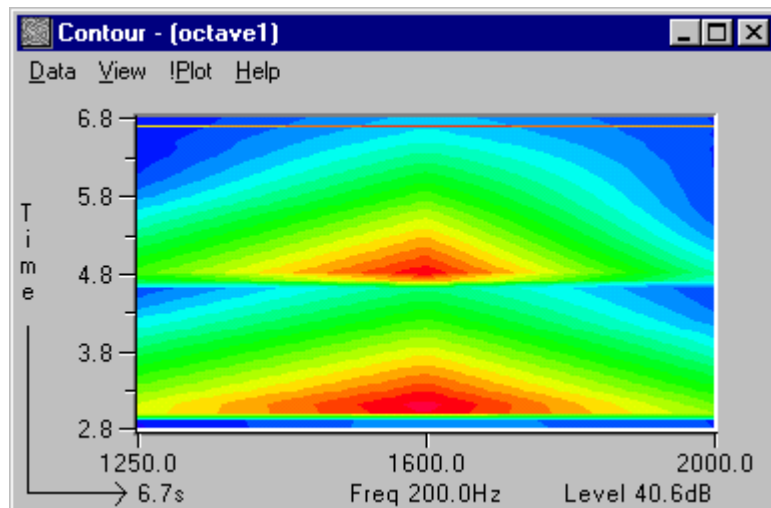


Use the spin controls in the Frequency axis to select a lower frequency of 1250 Hz and an upper frequency of 2000 Hz. This range covers just the 1600 Hz band we're interested in, and one band on either side.

Now select 2.8s to 6.8s on the Time axis, which will give us the peaks that occurred at 3.1 seconds and 4.8 seconds.

The Upper and lower levels in the Z-axis should remain unaltered, giving us a dialog that looks like the one shown above. When we click on 'OK', the dialog disappears, and CONTOUR will zoom in to the new Region of Interest.

*Zoomed-in contour plot of "octave1". The graph style has been changed to 'Filled Contour', using the 'View', 'Options..' menu commands.*



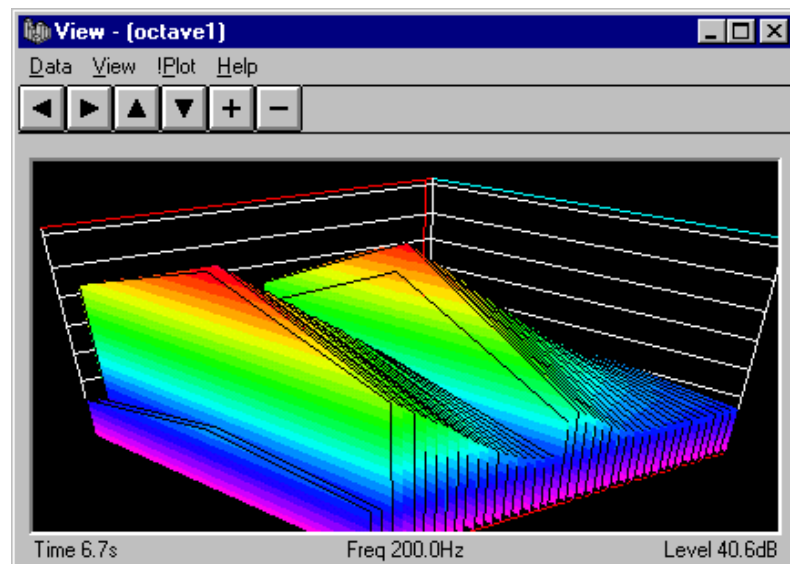
## Synchronised Views

### Overview

As we have seen in the previous examples, by dropping the same data object into different applications, multiple synchronised views of the same piece of data can be viewed. Try dropping the “**octave1**” data object over the VIEW module’s icon in the Base 2 toolbar. The module starts with the title bar changed to:

“View - (octave1)”

Hit ‘!Plot’ on the menu bar to view a surface plot of the “**octave1**” data. Assuming that the cursor positions were left in the positions they assumed after the manual zoom operation, the surface should look something like the one below, which shows the two peaks at 3.1 seconds and 4.8 seconds.

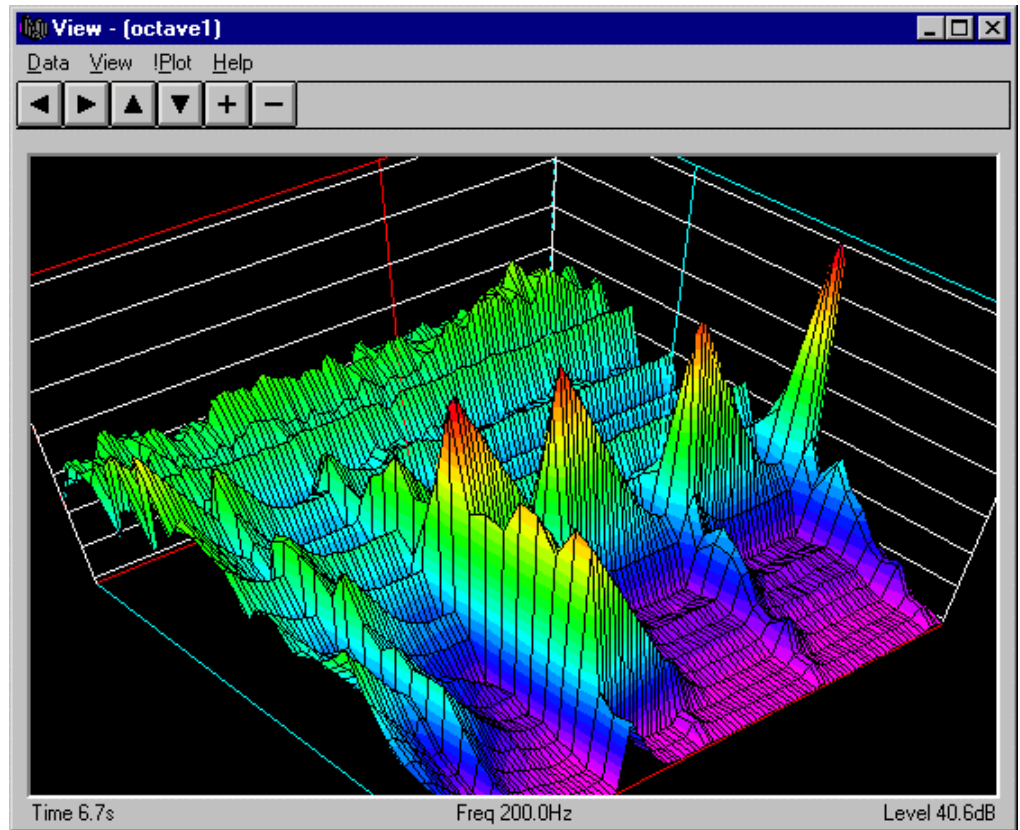


Each data object has its own private cursors (and many other attributes as well), so when the VIEW application was started and linked with the “**octave1**” data set, it showed the current region rather than the whole data set.

Depending on the capabilities of your hardware this may be a good thing, since plotting a surface from a large data set can be a time-consuming task, involving many millions of floating point co-ordinate transformations. This is one reason why a fast processor is recommended.

The VIEW application is one where the ‘!Plot’ option is particularly useful, since if your graphics system is particularly slow, or the data set is particularly large (or both if you’re really unlucky), manipulating the view can be quite time-consuming. Try experimenting with the arrow and +/- keys whilst ‘!Plot’ is switched off, and observe the rotation of the framework, which shows how the data will be presented when ‘!Plot’ is turned back on.

*The View module.*



## Overview of a large data set - MAP

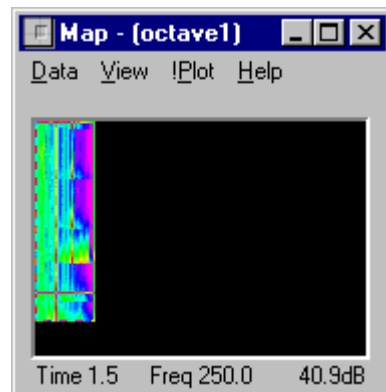
**nVision** was designed with the handling of reasonably large data sets in mind. This was originally for use with instruments capable of producing 16 MB of data in a single pass, perhaps lasting only a few seconds, and then providing the capability of examining subsets of that data in detail.

In order to decide which areas of the data set are worthy of further examination, it's necessary to get an overview of the data set. **nVision** features a number of methods for doing this, one of which is to view a pixel map in the MAP application.

The data set is dropped into an existing MAP application or onto the MAP icon on the toolbar as before, and '!Plot' is clicked to display the data. In order to accommodate potentially large data sets, each data point is assigned a single pixel in the display area, the colour of which is related to the contour band of the associated floating point value. This is very similar in concept to the CONTOUR application, but with a number of significant differences:

- There is only one display style - one pixel for each value.
- If the image extends beyond the current window borders, scroll bars are made available to bring areas of interest into view.
- When a click and drag operation is performed, a rectangular box, which matches the dragged out area marks the region, but the region does not zoom up to fill the available display area.

*Map display*



The "**octave1**" data set isn't particularly large (100x30), but for a data set such as 2000 timed samples from a 3200 line FFT spectrum analyser, it would provide an extremely useful tool for examining the resulting 6.4 million data points.

---

# Tools Tour

## Overview

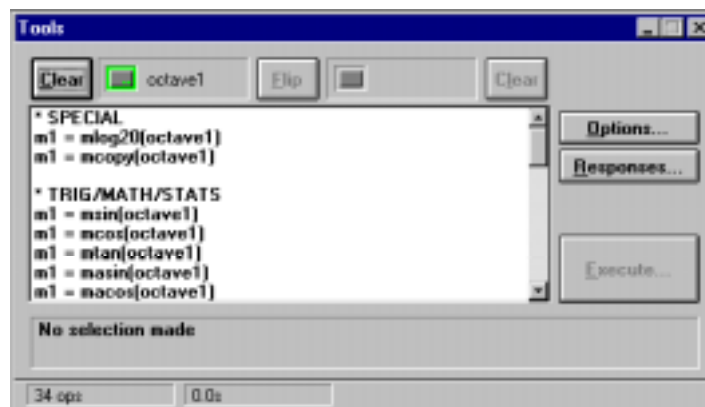
Data sets are linked to the TOOLS module by dragging and dropping in the normal way. The system analyses the data characteristics and builds a list of compatible operations or functions. These functions are displayed in a list box, taking the same form as the functions that are available in command mode.

## Using TOOLS to perform arithmetic operations between matrices

Execute the TOOLS module in the normal way, from the main toolbar. The two data sets, “**octave1**” and “**octave2**” should still be visible in the data window; if not, use the FILE module to load them now.

The TOOLS module has two *bays*, which are areas on the window designated as dropping zones for data. They are the two rectangular areas in the top left corner of the window. Take the “**octave1**” data set from the data window, and drag it into the left-hand bay. The name should appear inside the bay, as shown below. Notice that the characteristics of the data set are analysed by the TOOLS module, and a list of operations that suit the data is built up. Before continuing, use the scroll bar to have a look through the list, which is arranged into categories, and examine the operations available so far. Note that this list will continue to grow as the nVision system expands, and provides an increasingly powerful and flexible source of data manipulation.

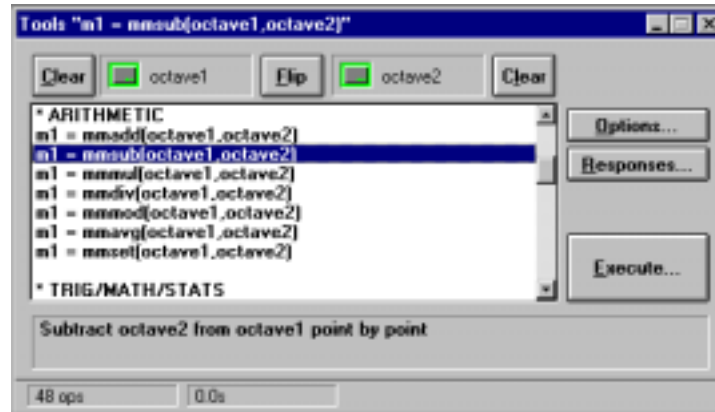
*Tools MODULE: operations for one matrix object.*



You will probably observe that all of the operations in the list feature just the single data set “**octave1**”. This is because so far, we have only linked this one data set to the TOOLS module, and the list of possible operations is based purely on what we can do with a single matrix.

Now drag the matrix “**octave2**” into the right-hand bay. The data set name appears in the right hand rectangle as before, and the list is expanded to show the list of operations that are possible with two data sets. Look through the list using the scroll bar, and you will observe that it has been expanded to include calculations that use two data sets.

Tools MODULE: operations between two matrix objects.



By clicking on one of the list items, we can select the matrix operation we want to perform - in this case, to subtract the contents of “octave2” from “octave1” on a point-by-point basis. We can proceed to the execution phase by clicking the ‘Execute’ button, or double clicking directly on the entry in the list box.

The processing will now be carried out according to the default options. These options include which channels of the matrix to process, whether to create a new object, how much of the data to process, etc.

This is the last stage in the procedure, and following a short delay (depending on the capability of your host machine) a new data set, “m1”, will appear in the data window.

## TOOLS operations involving scalar values

In the previous example we saw how to obtain the difference between two matrix objects, but what if we simply want to apply a scalar modification to a matrix? Alternatively, we may wish to extract a scalar result from a matrix by performing some analysis, for example, the average value.

### Creating a SCALAR object

Scalar values exist as objects within the system, just like the matrices we have already observed. They can be created in a number of ways, as we’ll see shortly, but for now we can create a scalar from within the TOOLS module as follows:

- Hit the ‘Clear’ button on the right-hand bay in the TOOLS module.
- Double click the mouse inside the rectangle that represents the bay.
- When the ‘Set Value’ dialog appears, enter the scalar value “2” and click on OK.

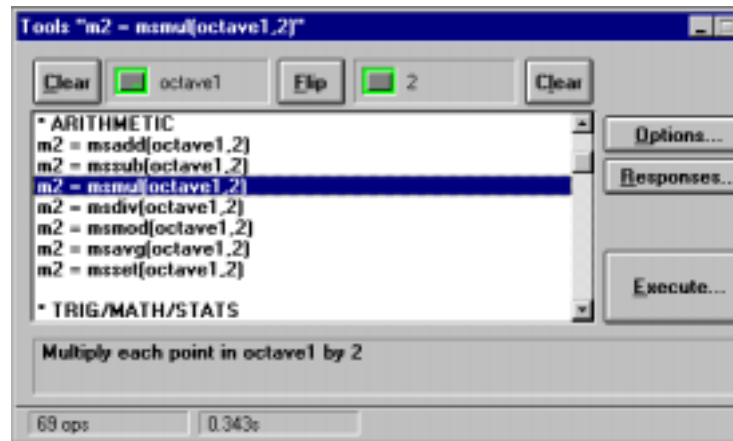
The system will now create a new SCALAR object having a value of two. The object appears in the Data Window as before, and is also automatically loaded into the TOOLS module in the appropriate bay.

Because the objects loaded into TOOLS have changed, the list of available operations is updated to show legal combinations of matrix and scalar operations.

### Performing the calculation

To make the calculation, select the appropriate entry in the list box, and click on ‘Execute’ as in the previous example. For this example we choose multiplication.

*Scalar multiplication operation.*



Now we can multiply each value in the matrix “**octave1**” simply by highlighting the option:

**m2 = msmul(octave1,2).**

At this point, the description “Multiply each point in octave1 by 2” will appear in the description panel, immediately below the list box. The calculation is performed by pressing the ‘Execute...’ button.

## Using the UNDO Module to perform What/If calculations

The UNDO module is a simple program that gives the facility to restore a data set to its original state following the application of a calculation.

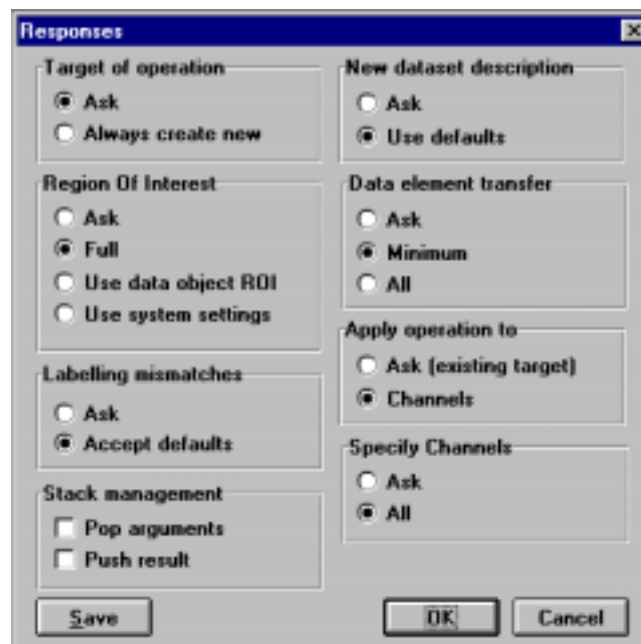
*The UNDO module.*



Drag the data set “**octave1**” into the UNDO module, which will automatically select the ‘Capture’ check box.

Now we will perform a calculation on the data set “**octave1**” using the TOOLS module, for example, an increase in the data values by 50%. To do this, first change the scalar value “2” to “1.5” by double clicking on the right-hand bay and entering the new scalar value. Then click on ‘Responses...’ and select ‘Ask’ in the first group of controls, which is titled, ‘Target of Operation’.

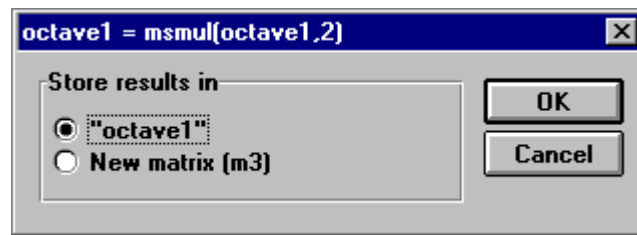
*Responses dialog allows choice of various default processing options.*



Now click on ‘OK’ to close the dialog. In changing this option, we have allowed the possibility of writing the new data over the old data in the “**ocatave1**” matrix, as opposed to creating a new matrix object. You can experiment with the other options later, and observe how they give you more freedom over various processing parameters.

Select the scalar multiplication entry from the operations list box and press ‘Execute...’. We are now able to choose our target matrix:

Choose the target matrix for the new data.



Choose “**octave1**” as the target matrix for the new data, and click ‘OK’. The “**octave1**” data points have now been increased in value by 50%.

Now go back to the UNDO module:

*UNDO* module: some commands are now enabled.



In the example above, the multiplication was performed across the entire data set, which is 100 rows x 30 columns or 3000 data points in all. The undo stack size is therefore set to  $U = 3000$ . Because *undo* is now possible, some of the controls have been enabled.

Assume that we now perform some other analysis using this data, and once the analysis is complete, we wish to restore the original data.

We can undo all 3000 values by clicking on ‘All’, or undo them one by one, by clicking on ‘Last’. Alternatively, the undo data can simply be thrown away using the ‘Dump...’ button, which effectively makes the changes permanent. (Although if the data is in a file, you can still get the original data back by throwing the modified data set in the Trashcan and reloading it).

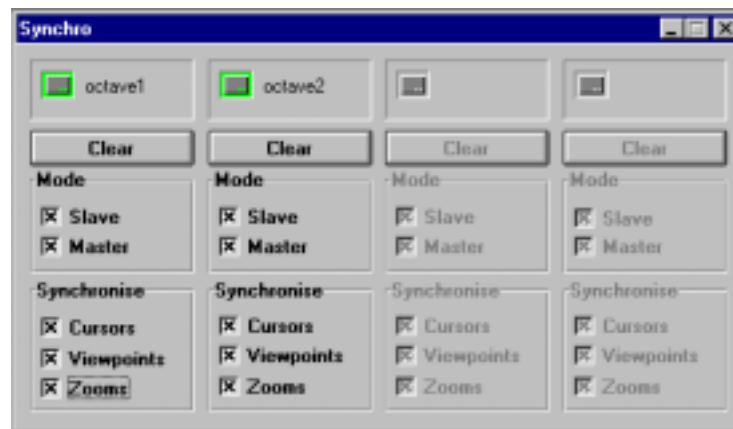
If you want to perform a series of staged calculations, and undo them individually, you can use the ‘Drop’ button, which puts a marker in the undo stack at the current position. Then you can retrace to each of the markers you have dropped by using the ‘Retrace’ button. The “M = ?” value keeps a track on the number of markers available.

## Synchronising data objects

In the examples used in the Base 2 tour, we demonstrated the convenience of loading the same data set into more than one display application. We had the facility to track cursor movements and zoom levels across all of the modules (this was called “*hot-linking*”).

Given the similarity of the two data sets, “**octave1**” and “**octave2**”, it would be even more useful to be able to map these movements so that movements in “**octave1**” affect “**octave2**” as well. The SYNCHRO module allows us to relate up to four data sets, and then to select if any or all of the Cursor, Viewpoint and Zoom operations are repeated in each data set.

The data sets are dragged into SYNCHRO in the normal way, into one of the four bays. Checking the ‘Slave’ option means that the data set will follow movements in objects selected as master, but will not cause others to do so. If ‘Master’ is checked, other data sets will follow. If both are checked, the data set will cause change in others as well as reacting to change.



Try loading the two data sets into SYNCHRO, as shown above, and then load each of them into a separate CONTOUR module. Observe how clicking in one CONTOUR affects the other, even though they hold different data. Similar effects can be observed in display applications such as VIEW, GRAPH and EDITOR. Additionally in VIEW, the angle and orientation of the data can be synchronised, so that two different data sets may be observed at identical rotations. The VIEWPOINT option only affects data seen in the VIEW module.

Note: you cannot specify the same data set more than once (even if you run more than one copy of SYNCHRO), since this would effectively setup an infinite loop.

---

# Database Tour

## Overview

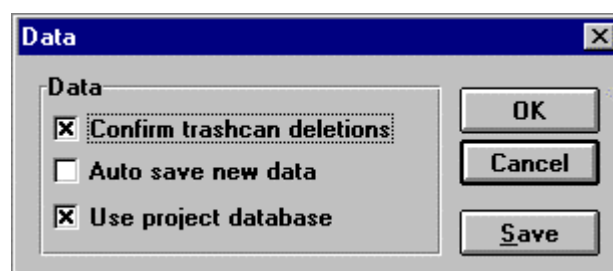
The nVision Project Database stores information about the data objects in the nVision system. The user can also create his own fields to store alongside the information held within an Object. Powerful queries can be posed to the database to track down specific data objects. Once you have created the records of the objects, you can view them in full, archive, delete or load them into the system.

This tour will look at some of the features of the nVision project database. In particular we will look at storing, searching for and loading records.

## Running the project database

The use of the nVision project database is optional. By selecting the "Use project database" option from the "Data" dialog box from Servers "Options" menu it can be permanently enabled. Selecting this option will inform the database every time a data object is saved, which will allow the database to add or update its entry for that data object. Alternatively you can add/update an entry to the database by dropping the data object directly over the database. For the purpose of this tour we will enable the "Use project database" option and you should select that now if its not already selected.

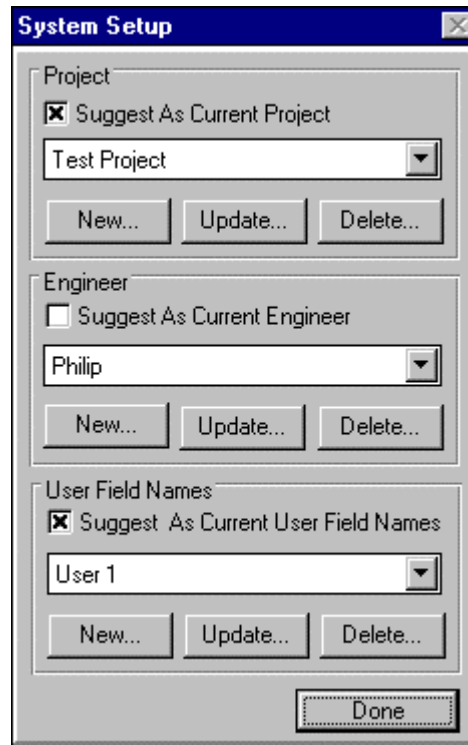
*The Data dialog box to permanently enable the use of the database.*



## Configuring the database

The nVision database allows a data object to be associated with a project and engineer. These associations help when entering queries to find previously saved data. The System Setup dialog box is accessible from the setup menu, select it now. In the project group box press the "new" button. In the resulting dialog type the name of the project for this tour we will use "Test Project" type this now and press OK. Enable the check box entitled "Suggest As Current Project". This will force the current project to default to "Test Project" every time a new record is added. Repeat these steps for the engineer box inserting your name as appropriate. The "System Setup" dialog also allows you to enter "User Field Names" but we will deal with these later.

*The System Setup dialog box. Use this dialog to setup current projects and engineers and user field names.*



## Storing Records

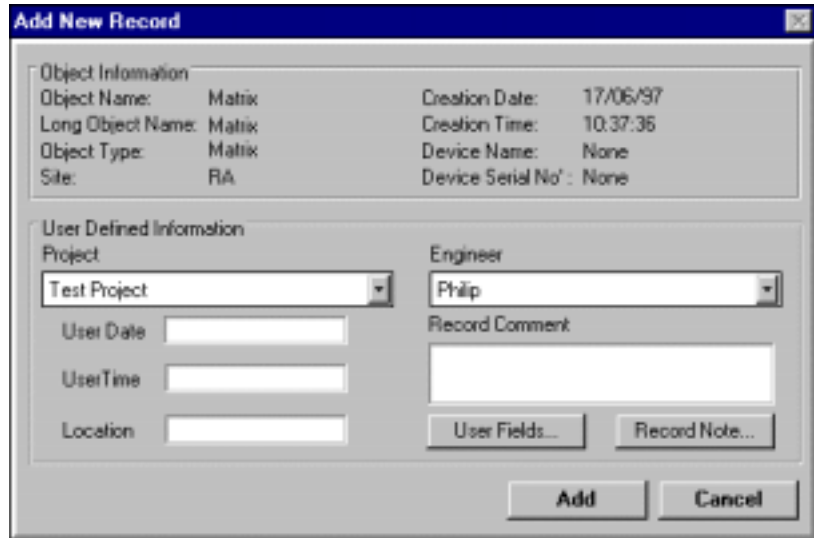
A record stores information about a saved nVision data object in such a way that makes it possible to pose queries to isolate individual records. These records can then be viewed or their associated objects loaded into the nVision system.

A record for a data object can be entered when the data object is first saved. Saving the object subsequently will allow the record to be updated.

Use the “Create Matrix” option from editor’s “Data” menu to create two matrices, **matrix** and **matrix2**. Drag and drop the matrix object from the Data list over the file icon. The Database will be automatically started (if it isn’t already running) and the “Add New Record” dialog will be displayed. If the Database isn’t started automatically then ensure that the option “Use project database” is selected as described in the section “Running the project database”. The object could have been dropped directly over the database application instead of over file. Both methods have the same effect.

For now we will just accept the default information that is shown in the “Add New Record” dialog so press the “Add” button. The record will now be entered and the object saved.

The “Add New Record” dialog



## Querying the Database

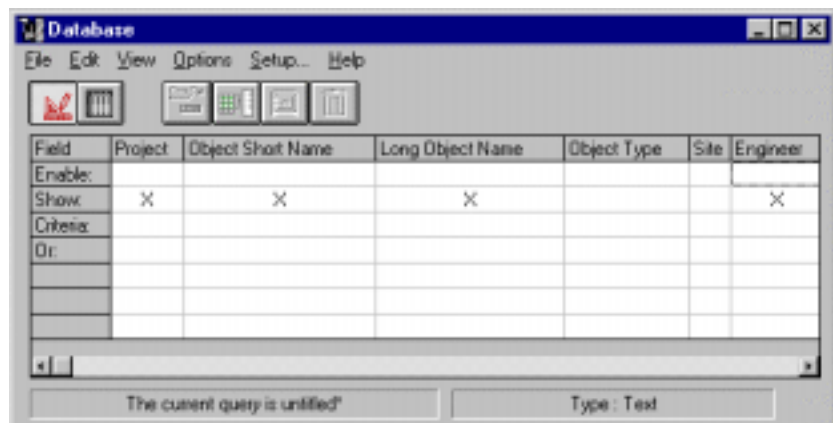
The nVision database offers facilities for creating and manipulating query's which can be used to select various records from the database, given a set of search criteria. These facilities allow for the combination of logical operators in constructing queries with more than one search criteria.

Queries are entered using the query grid on the front panel of the database. The query grid is arranged with field names running across the top and search criteria running down the left-hand side. Double-click in the “Show” field of Project, Object short and long name and Engineer so the query grid looks like that shown below. Now press the “Query Results” button or select “Query Results” from the View menu.

The “Query Results” button



The Database front panel showing the query grid.



The results of the query are now shown. The result grid will replace the query grid.

The “Results” grid showing all the records that match the criteria laid down in the current query.



## Loading Data Objects

Data objects can be loaded from the results grid by selecting the record or records whose associated data object you want to load and pressing the “load objects” button or by selecting “Load Objects” from the Options menu.

The “Load objects” button



In the data list delete the “**Matrix**” object. Now press the “Load objects” button to reload it.

## Creating User Fields

The nVision project database contains 16 fields whose names can be defined by the user. These fields allow the user to customise the database to suit their individual needs. These field names are placed together in a group so the user can create multiple field names to suit his application.

User field names are setup from the system setup dialog box. The System Setup dialog box is accessible from the setup menu, select it now. In the User Field Names group box press the “new” button. This will bring up the “Create New User Field Names” dialog. For the purpose of this tour we will assume that we are involved in tests involving tyres. Now enter the following information: -

### Group Name

Tyre

### Text Names

Field 1 Make

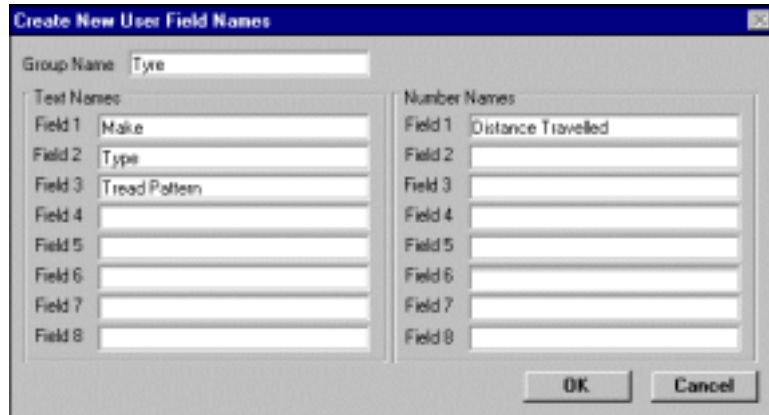
Field 2 Type

Field 3 Tread Pattern

### Number Names

Field 1 Distance Travelled

The “Create New User Field Names” dialog box is used for entering the names of the user defined fields.



Press OK and select Tyre in the list of User Field Names and check “Suggest As Current User Field Names” the System Setup dialog should now look like that below.

The “System Setup” dialog as it should now look



Press “Done” on the System Setup dialog to return to the front panel. If you use the scroll bar to move the query grid to the right you will see the names of the user fields that you typed in as shown below.



Make JP1  
Type Magnum  
Tread Pattern QX23

**User Numerical Fields**

Distance 500

Press OK to finish the user defined fields and then press Update to complete the record.

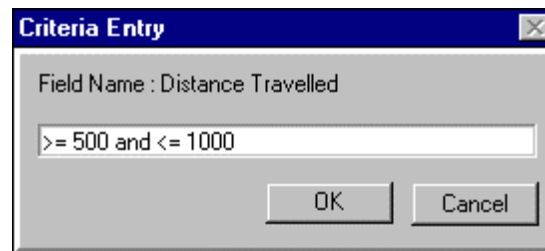
## Queries with Criteria

The previous query we performed did not have any criteria that is to say that every record in the database matched. You can filter out unwanted records by imposing criteria that every record must have to be included in the results.

Scroll the query grid to the right until the distance-travelled field comes into view. Double click the box at the intersection of Distance travelled and criteria. The criteria entry box will appear. We want to track down all the sets of tyres that have done between 500 and 1000 miles. Enter the following: -

$\geq 500$  and  $\leq 1000$

*The criteria entry dialog.*



We need to tell the database which fields we want to be displayed as a result of our query. Double clicking in the show box of the entries we want to include does this. For our query we want to see the Project, Object short name and engineer fields so place a click in their show boxes now and then press the “Query results” button. The query should match the two records we entered earlier.

It is now possible to go back to the query design form and alter the query to filter our results. It’s not really necessary when you only have two results but it will serve as a good illustration.

*The query design button*



The query results grid showing the results of the query Distance  $\geq 500$  and  $\leq 1000$

Rec No'	Project	Object Short Name	Engineer	Distance Travelled
1.	Test Project	Matrix	Philip	500
2.	Test Project	Matrix2	Philip	1000

Press the query design button or choose “Query” from the View menu. We will now narrow our search to tyres that have travelled between 500 and 1000 miles and have a Make beginning with J.

Enter the following in the Criteria box of the Make field.

J\*

The asterisk indicates that all subsequent letters match, its proper name is a wild card. Press OK in the Criteria entry box and then double click Makes show box because we want to see the Make in the result list. The query grid should look as it does below.

The updated query grid.

Field	Make	Type	Tread Pattern	User Text4	User Text5	User Text6	User Text7
Enable:	X						
Show:	X						
Criteria:	J*						
Or:							

Press the “Query Results” button. Now only the record for “**Matrix**” will be shown.

---

# Animation Tour

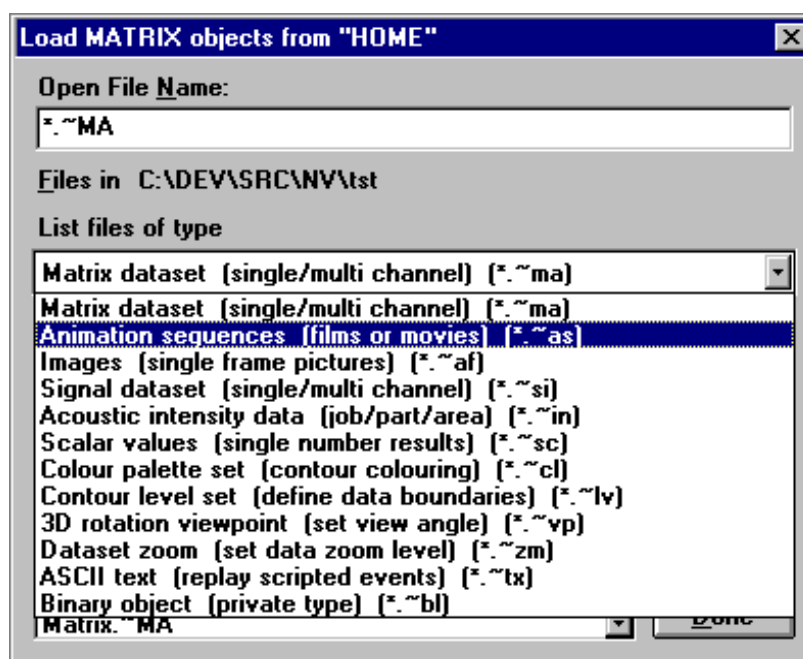
## Overview

Animations generated by **nVision**'s display modules are stored in the *film* object. The film object stores a number of still *frames* which form an animation when played in sequence. With the **nVision** system there are two modules used for dealing with animations. The ANIMATE module is used to view animations and the SPLICE module is used to edit animations.

## Loading a sample animation

First, we need to load a sample animation (film object). To do this select 'File' / 'Open' from the Data Window menu, or restore the FILE window.

*File Data Types available from the File Type list box, showing the selection of the film type.*



Select the 'Animation sequences' option from the data type list if it is not already selected, and a list of available film files will be displayed in the scrolling list box. A file may be loaded by highlighting a selection with a single click and then pressing the 'Load' button, or by double clicking on the list box item. If the file you want isn't visible, you can move it into view using the scroll bar.

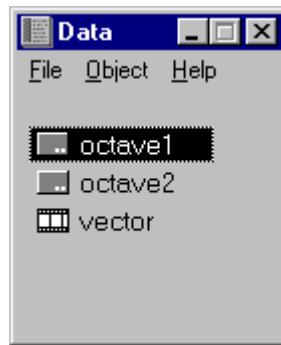
Select and load the sample file "**vector.~as**", using either of these methods. During loading, the cursor will change into an open book, indicating that the system is reading from disk, (when writing data, the cursor changes to a fountain pen).

When this file has been loaded, signified by the cursor changing back into an arrow, hit the 'Done' button on the File dialog box, as we don't want to load any more data objects at this point.

You should now see a new entry called "**vector**" in the Data Window.

The film “vector.as” shown loaded in the Data window.

An animation is held as a series of frame objects, each of which holds a single bitmap.



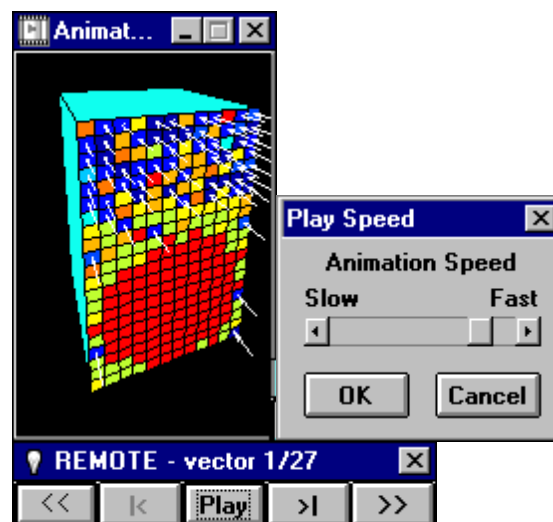
The “vector” animation was generated from a series of 3D views generated from an acoustic intensity test on a hi-fi loudspeaker. Each of the frames represents the measured intensity for a different octave band. As the animation progresses, the crossover from woofer to midrange to tweeter can be clearly observed. Intensity direction is indicated by arrows pointing into or out of the speaker baffle.

## Displaying a sample animation in ANIMATE

Now that we have the sample animation in the Data Window, we can display it using the ANIMATE module. Drag the “vector” film icon from the Data Window, and drop it over the ANIMATE icon on the ‘Animate’ toolbar.

To start an animation, select the ‘Play’ option from the ANIMATE control menu. Alternatively, bring up the remote control panel by clicking on the ‘Remote Control...’ option on the ANIMATE control menu and click on ‘Play’. To change the playback speed of an animation you need to bring up the play speed panel by clicking on the ‘Play Speed’ option on the ANIMATE control menu and alter the scroll bar.

The Film “vector” loaded into the ANIMATE module, remote and play speed dialogs.



## Editing an animation in SPLICE

You can edit an animation using the SPLICE module. Splice displays an animation in its component frames and lets you cut and paste them in any order. You can also drag FRAME objects from the Data Window or another instance

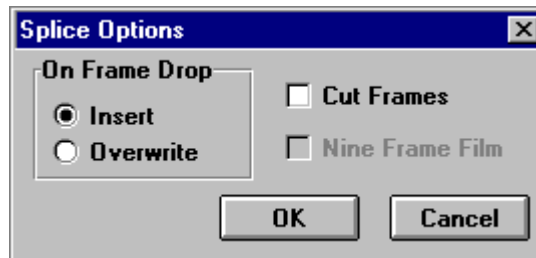
of SPLICE and drop them over a specific frame in splice to add them to the animation.

*The film “vector” loaded into the SPLICE module.*



To copy a frame in SPLICE you need to move the mouse pointer over the frame you want to copy and hold down the right mouse button. This will change the cursor into a frame and you can then drag the copied frame to another position and drop it. Depending on the Insert/Overwrite setting in the SPLICE options dialog SPLICE will either insert the frame at that point or overwrite the frame you dropped it on.

*The SPLICE options dialog.*



To cut frames from the animation you need to make sure the ‘Cut Frames’ option on the SPLICE options dialog is checked and then you can cut a frame by double clicking the left mouse button over it.



---

# Automotive Tour

## Overview

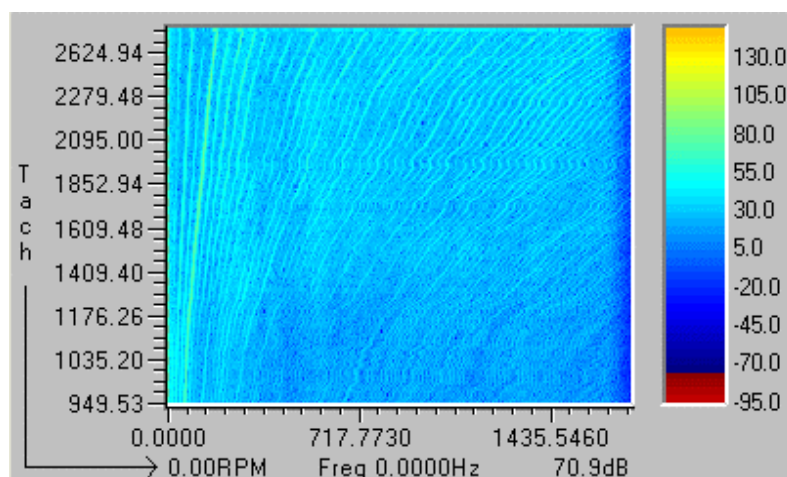
This tour will use some of the tools available within **nVision** for dealing with automotive data, and in particular measurements from engine run-ups. We will look at waterfall data, calculating RPM values from a tacho pulse train, calculating order data and extracting order related effects using Kalman filtering.

## Engine Run-ups and Waterfall Displays

We will consider how to view and analyse data from three types of engine measurement. The first is the type of waterfall measurement typically performed by spectrum analysers that produce frequency spectra at intervals during an engine run-up.

Load the matrix **“runup freq”** (from the ‘DEMODATA’ folder on the CD) and view it in the CONTOUR module.

*Viewing “runup freq” data in CONTOUR.*

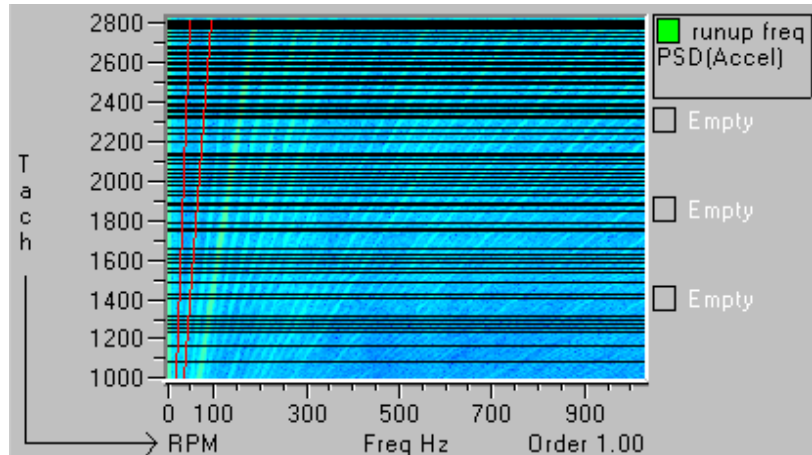


The data in the contour window clearly shows the orders produced during the acceleration. Remember that by viewing the same object in a 2D window you can use the “hot linked” cursors to view individual RPM or frequency slices.

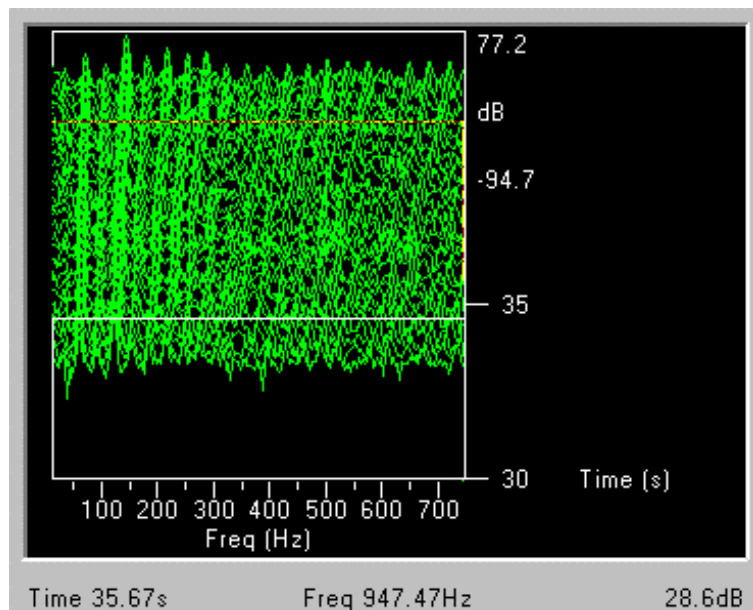
Because each measurement is plotted equally spaced, orders do not appear as straight lines. By viewing the data in the PLOT module (Base 2 toolbar), the data is drawn spaced according to the measured RPM values so orders appear as straight lines. Additional order cursors can be switched on by the menu entry ‘View’, ‘Order Cursors...’.

**Tip:** whenever **nVision** is looking for RPM values to use in a calculation, it expects a dictionary to be available called “Tach” in the data object. If you have valid tacho data but order cursors are not present, or modules refuse to accept the data, check that the RPM data is called “Tach” in the object Information dialog. It can be manually changed if necessary.

Order cursors highlighting orders 1 and 2 of the “runup freq” data.



WATERFALL display, with viewing angle set to 90 degrees

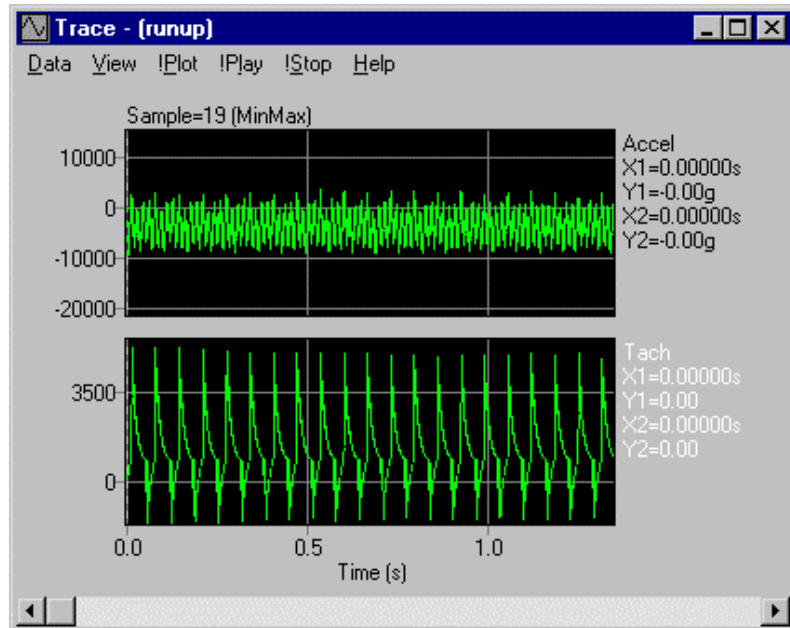


## Extracting an RPM profile from tacho data

Now let us consider how to process continuous time series containing run-up data. This could have come, for example, from a measurement using a sound card or a DAT recorder.

Load the signal called “runup” (from the ‘DEMODATA’ folder on the CD). This contains two channels of sampled data. One is an acceleration signal; the other is a pulse train with a half pulse per rev tacho signal. In order to take an order cut we need to know what the RPM value is.

Acceleration and tacho pulse train signal



The order analysis setup

The screenshot shows the 'Order - (runup)' window with various configuration options. The 'Data Channels' section lists 'Accel' and 'Tach'. The 'Process' section has radio buttons for 'All' (selected) and 'Between Cursors'. The 'Cursors' section shows 'Dotted' and 'Solid' options with a time range from 0.0 to 53.210s. The 'Tach Channel' section has a dropdown menu set to 'Tach' and buttons for 'Setup...' and 'Process...'. The 'Trigger' section includes input fields for 'Start RPM' (1000), 'End RPM' (5000), and 'Interval' (10). The 'Max. Order' is set to 20 and 'Order Step' is 0.1000. The 'Scaling' section has radio buttons for 'Lin' and 'Log' (selected), with a 'dB Reference' of 1.000e+000. There are buttons for 'Options...', 'Execute...', and 'Cancel'. The status bar at the bottom shows 'Ready...' and '0%'.

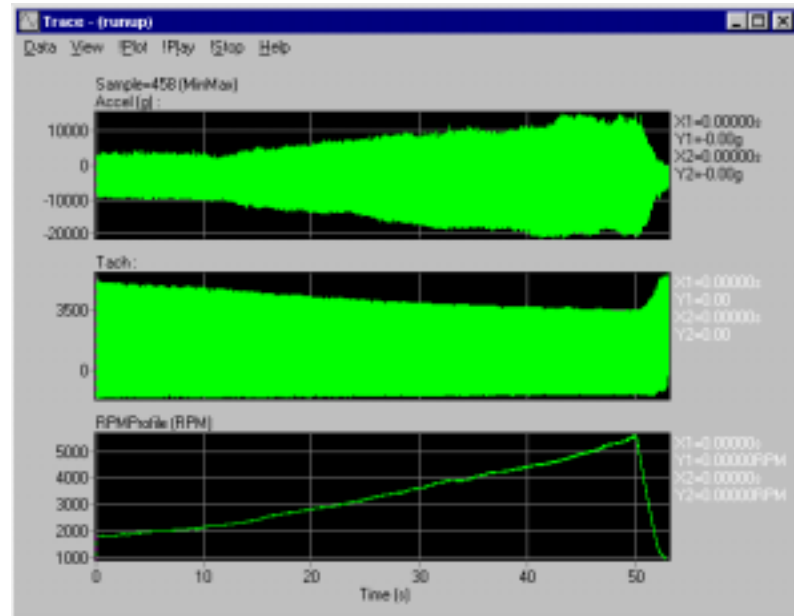
Calculate the RPM profile

The order analysis module allows you to extract the RPM profile from the tacho pulse train. This profile shows how the RPM value varies with time. Click on 'Setup...' to enter the hysteresis trigger parameters that the program uses to detect the pulse train, and remember to set the number of pulses per engine revolution. In the case of the runup demo data, this is half (0.5) pulse per revolution.

The hysteresis levels let you set the threshold voltages the signal must cross for a pulse to be detected. With this data the signal crosses 0 volts.

Process the tacho channel. The RPM profile appears as a new channel in the runup object, which you can view in the trace display.

*Extracted RPM profile from tacho pulse train*



## Extracting Order Information

*Calculating an Order Map.*

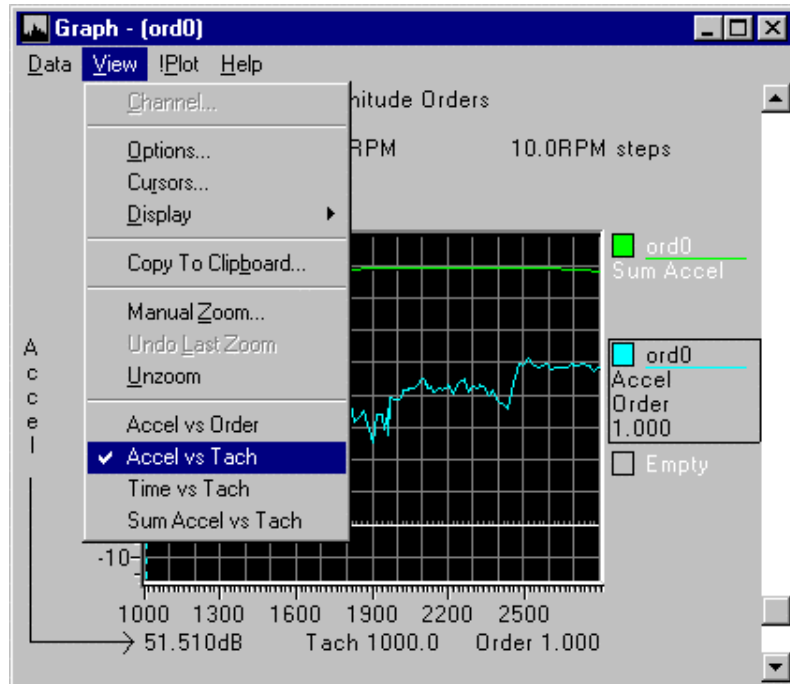
The order module allows you to calculate an order map. Having produced the RPM profile channel, select the "Accel" channel, leave the other parameters on their default values and click 'Execute...'. This will create a new object which contains order cuts up to the maximum order requested (default is 20<sup>th</sup>). The overall order summation is also calculated.

*Overall level and order section.*

By placing the order object in two bays of the GRAPH window, you can overlay the overall level summation (menu option: 'View', 'Sum Accel vs. Tach') with individual orders ('View', 'Accel vs. Tach') to examine their relative contribution to the total. The scroll bar on the right of the display allows scrolling through each order in turn.

The data can be viewed as order sections against tacho RPM value (set 'View', 'Accel vs. Tach') or as Order slices at each measured RPM (set 'View', 'Accel vs. Order').

Of course you can also view the data in 3D graphs such as CONTOUR to see all extracted orders at once, using the "hot linked" cursors to examine sections in the 2D GRAPH module.

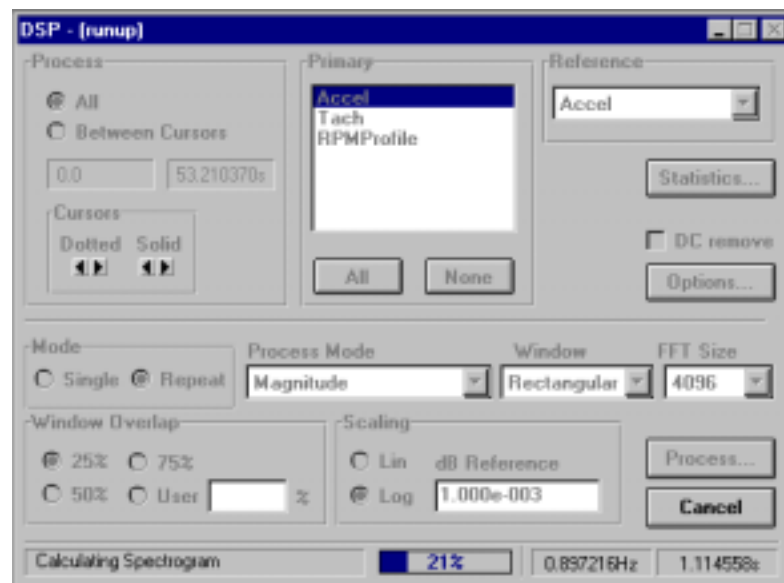


## Generating a Frequency Spectrum Waterfall

To view the frequency components of our signal we can use the DSP module to generate magnitude frequency spectra. The DSP tool breaks the time series up into blocks of the specified FFT size, calculates the Fast Fourier Transform of each block and stores the result in an object matrix.

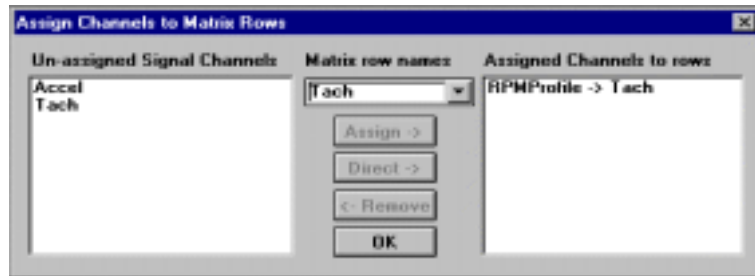
Drag and drop our run-up signal in the DSP module. This module is located on the DSP toolbar. Select “Accel” (the acceleration data channel) and click ‘Process’. You will then see the ‘Assign Channels to Matrix Rows’ dialog.

*Calculating a waterfall of spectrums from the time signal*



*Assigning the measured RPM profile to the Tach matrix row*

Now we have an RPM profile channel available and we wish to use this channel to create a row of RPM values in our matrix object - so that each frequency spectrum will be tagged with the appropriate measured RPM value.

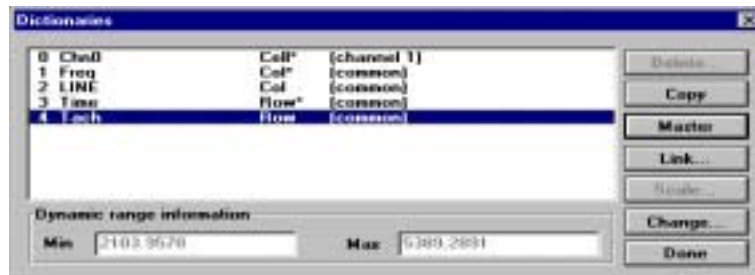


“RPMProfile” should appear in the unassigned channel list. Highlight it and click the ‘Assign’ button. This will assign the RPMProfile channel to the “Tach” matrix row.

Display the resulting frequency data in the PLOT or WATERFALL modules, plotting frequency against Tach value. Note how the significant orders stand out.

*Making Tach the master display row*

Hint: If the plot shows a map of frequency against time and you require frequency against Tach, you can change to this view by double clicking the Data Object in the Data Window, selecting the Dictionary from the Object Information, and ensuring that Tach is the Master row.



## Kalman Filtering

Kalman filtering provides a form of digital tracking filter that enables orders to be extracted from an acquired time series signal, provided that a tacho signal is also measured. The tacho signal can be a pulse train, or a signal proportional to RPM (RPM profile). In the case of a pulse train, the module can calculate an RPM profile.

The Kalman filter module produces a number of outputs:

- User selectable order traces
- A “Mixer” output combining these extracted orders (including the ability to attenuate or amplify individual orders).
- An RPM profile.
- Unlike FFT based order extraction these order traces have the same resolution and number of samples as the original measured signal. This enables true addition and subtraction of component orders. The results of recomposition can be played back through a sound card.

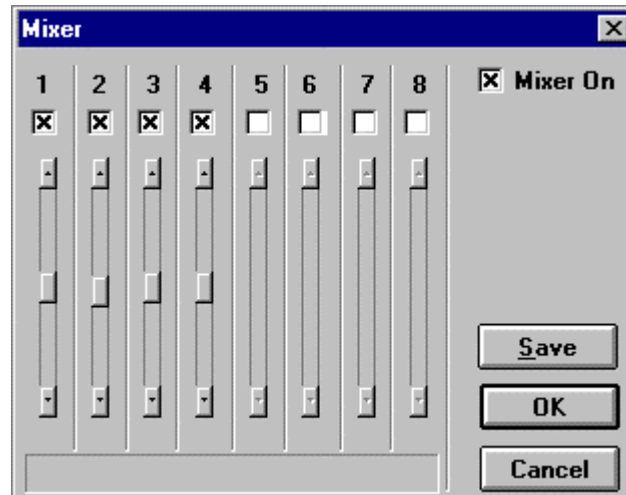
## Kalman Order Extraction

Start the KALMAN module (in the Automotive toolbar). Using the runup signal object, select “Accel” as the data channel and “Tach” as the Tach channel. The Data channel setup allows you to select which orders you wish to extract. From our examination of the runup signal earlier in this tour, you should have found that orders 1,2,3,4 are the dominant ones.

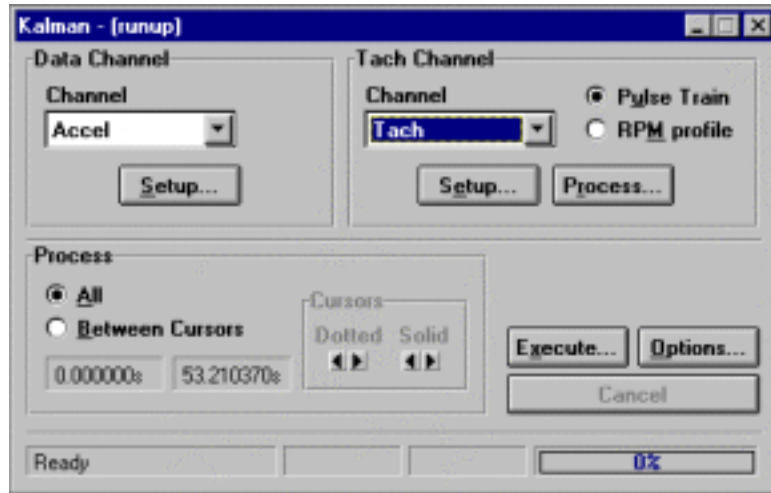
Setup the tach channel processing as before with 0.5 pulses per revolution and click on ‘Process...’ to produce the RPM profile channel.

Select the orders to be extracted. This is done in the ‘Filter Setup’ dialog. This dialog is brought up by clicking on the ‘Setup...’ button in the ‘Data Channel’ group. Here you should enable the filters of the orders you require. Also in this dialog is a button labelled ‘Mixer...’. Clicking this button will bring up the mixer dialog. The Mixer output is the summation of any set of specified orders from the extracted orders. The amounts of each order to be added can be varied via the sliders.

You can overlay the mixed version on top of the original runup channel in the Base 1 SGRAPH module, and see just how closely the signals match.



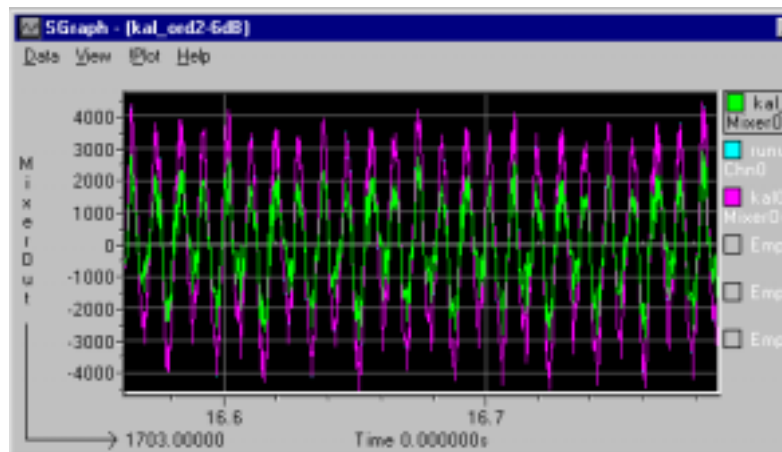
Press ‘Execute...’ to perform the filtering operation, and view the resulting signals in the DSP TRACE module.



*Changing the Mixer values*

Now try re-running the filter but this time change the Mixer settings. Each of the slider bars controls the attenuation or amplification of the signals. Filter 2 corresponds to order 2 in our setup so try attenuating this by 3 dB and calculate the result.

*Comparing original and filtered mixer outputs*



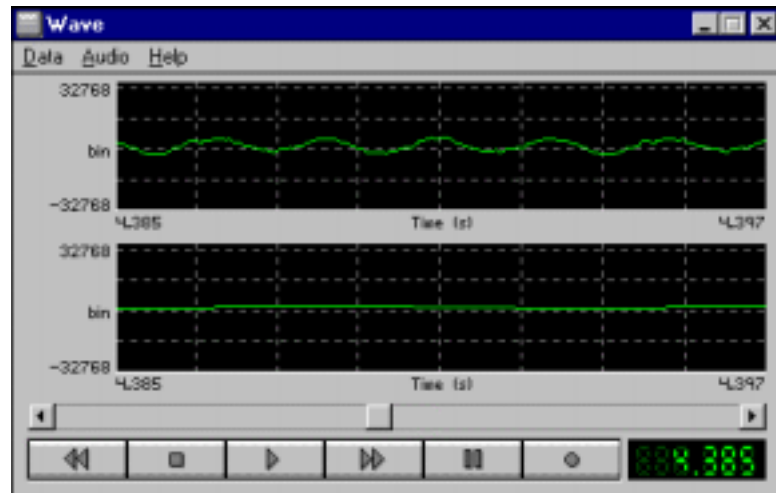
The original data and mixer output of orders 1+2+3+4 overlay nearly completely.

The smaller trace is the Mixer output with order 2 attenuated 6 dB.

If you have a sound card try listening to the difference by pressing '!Play' in the TRACE module.



*Monitoring in the time domain.*

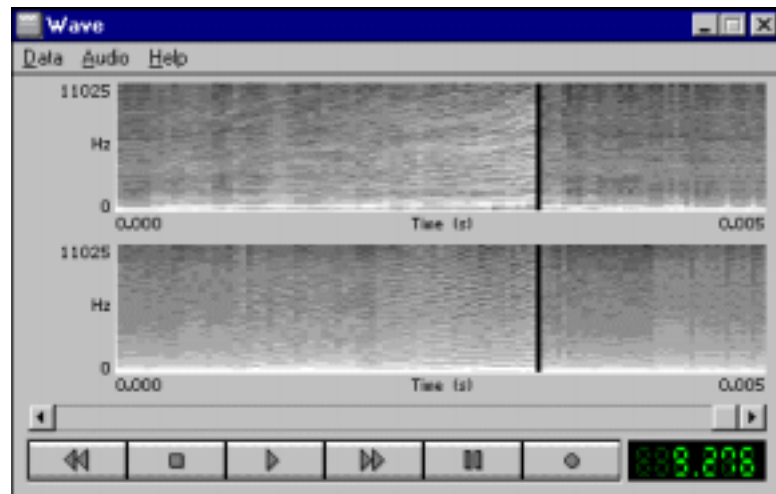


*Monitoring in the frequency domain.*

You can also display data in octave or third octave format during acquisition and playback. The display format is selected in the 'Audio', 'Display...' menu.

*The spectrogram display during acquisition.*

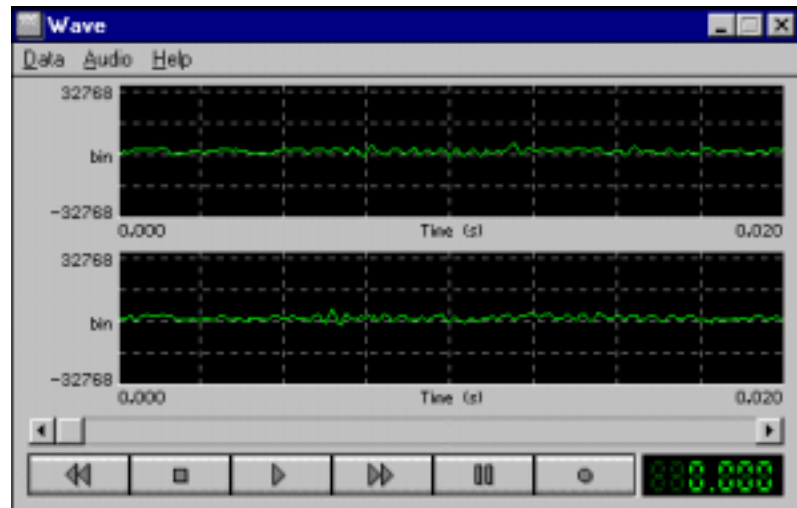
In the spectrogram display, the acquisition draws vertical stripes in grey scale to represent the frequency content of the data.



## Playback through a Sound Card

*Playing signals through the sound card.*

The data in the WAVE module can be played out through the sound card by clicking on the play button at the bottom of the window.



*Storing measured data in .wav format files*

Data can be stored in WAVE (~.wav) format files which can be played back on any PC sound system, or even embedded in Word documents for reports you can listen to. (See Reports Tour.)

To store wave files use the menu option 'Data', 'Save as...'

## Storing Signals in nVision Objects

*Storing data in signal objects for manipulation in nVision*

To pass acquired data to other **nVision** modules you need to create an **nVision** data object from the acquired data. To do this simply select menu option:-

'Data', 'Create Signal...'

After a dialog to define which channels are required, a new signal object will be created in the data window, which you can then use in modules such as DSP.

Remember that sound playback is available for any signal displayed in the TRACE window - just press '!Play' to playback the signal between cursor positions, and '!Stop' to stop the playback.



---

# Wavelet Analysis Tour

## Overview

This tour will demonstrate the use of the Wavelet analysis tool to analyse the frequency content of short-duration time series data, comparing the results with Fourier analysis techniques.

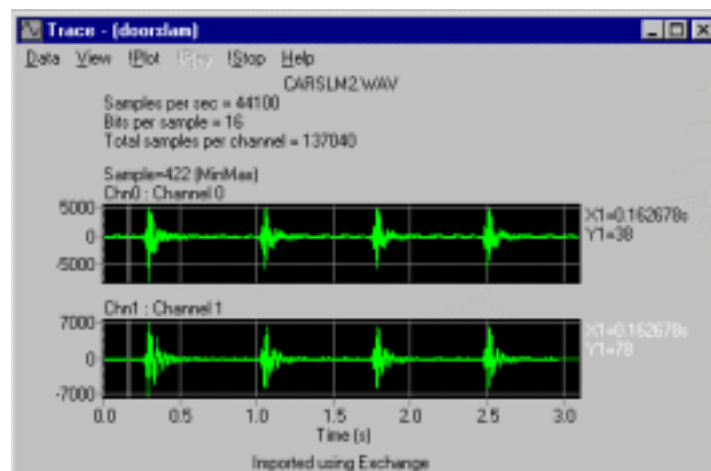
## Frequency content of transient signals

The demo data file, “**door slam.~si**” contains a stereo recording using microphones in the driver’s ear as a car door is slammed shut four times. In the three seconds’ duration of the recording, some 137000 samples have been recorded. This was sampled at 44.1kHz using a PC Sound Card, and imported from the WAVE file format using the EXCHANGE module.

Within these short transient impact events, a wide range of frequency components are excited, which we would like to analyse to determine what causes them and how the sound quality of the closure can be improved. Wavelet analysis allows us to identify frequency components during these short, transient events, where a conventional Fourier transform analysis would give only an “average” answer over the whole event.

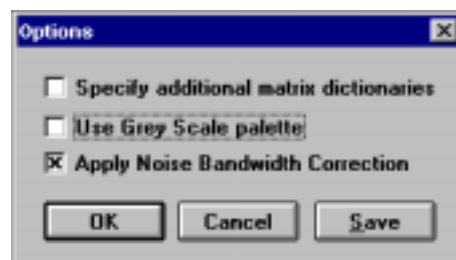
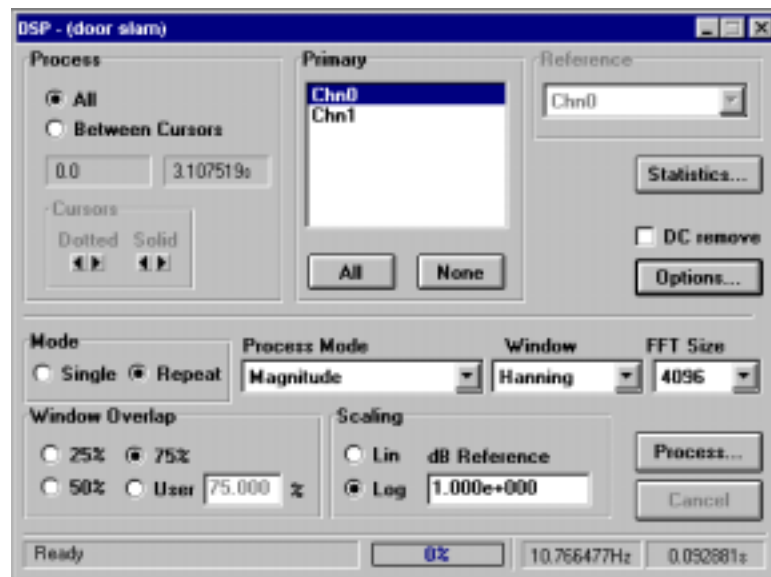
Load the signal data object “**door slam.~si**”, and view it in the TRACE module, located in the DSP toolbar.

*Door slam data in the time domain.*



Using the Fourier Transform to view the data.

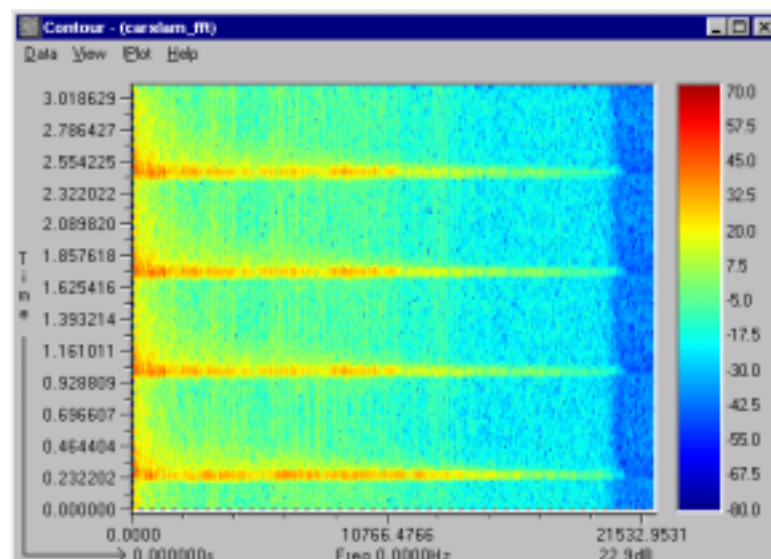
First, let's see what the data looks like using the Fourier transform:



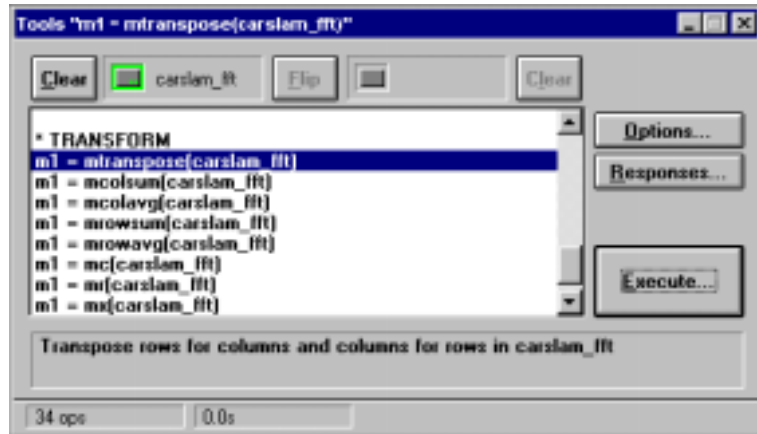
Drop the “door slam” object on the DSP module, and calculate the magnitude FFT of the data, using the above options. Call the result “carslam\_fft”, and display it in the CONTOUR module.

Displaying data in log magnitude format.

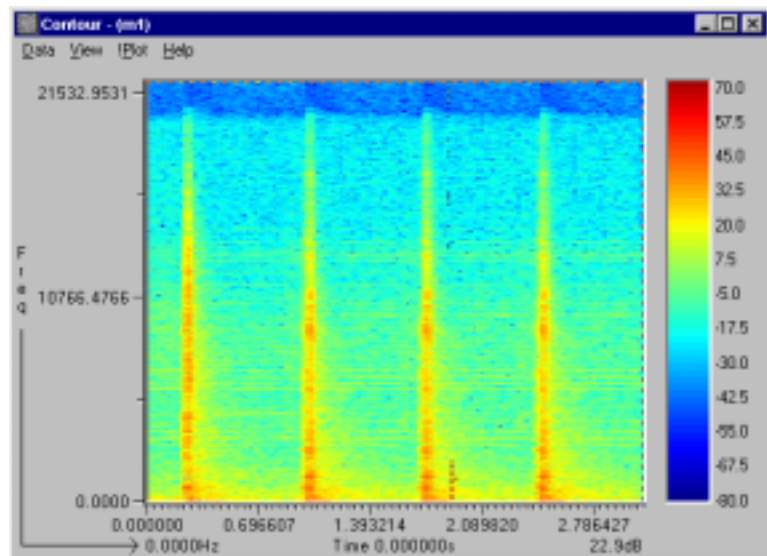
Display the FFT data in a CONTOUR graph. Your display should look something like the one below.



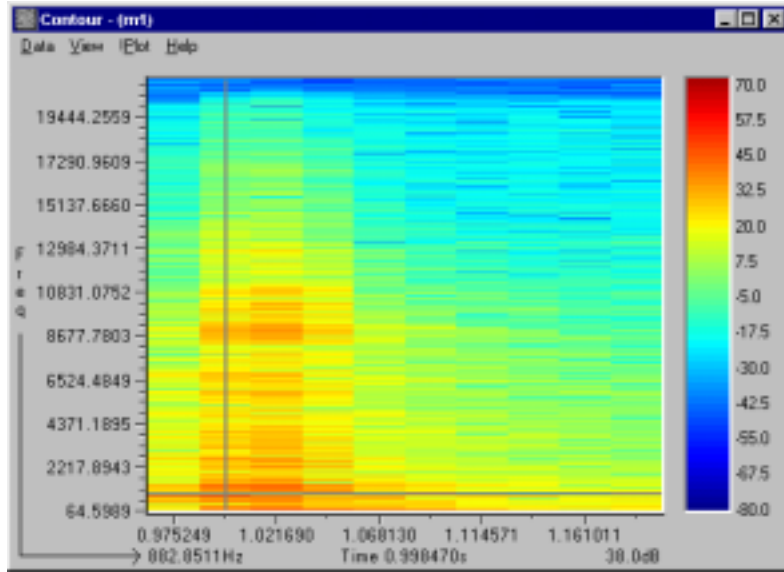
To make the data easier to interpret, drop “carslam\_fft” into the TOOLS module on the Tools toolbar. Select the operation of transposing the matrix, so that we can see time along the X-axis and frequency up the Y-axis.



Drop the resultant matrix, “m1”, into CONTOUR. The X-axis should now be time, and the Y-axis frequency:



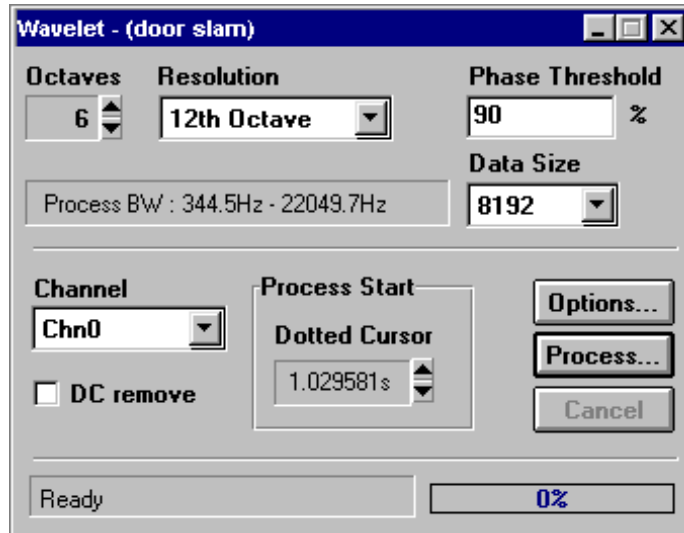
While the FFT calculation does show the frequency content of the impacts, it is impossible to gain much insight into the frequency variation during the impact, and so we cannot identify the stages of the impact and their effects. This is because we can only divide the three seconds into 134 separate FFT calculations. If we zoom into one of the impacts we see that there are only ten FFT spectra within the impact, and the frequency information is smeared out across them so it is impossible to determine exact events within the impact.



## Calculating the Wavelet Transform

*Calculating the wavelet transform.*

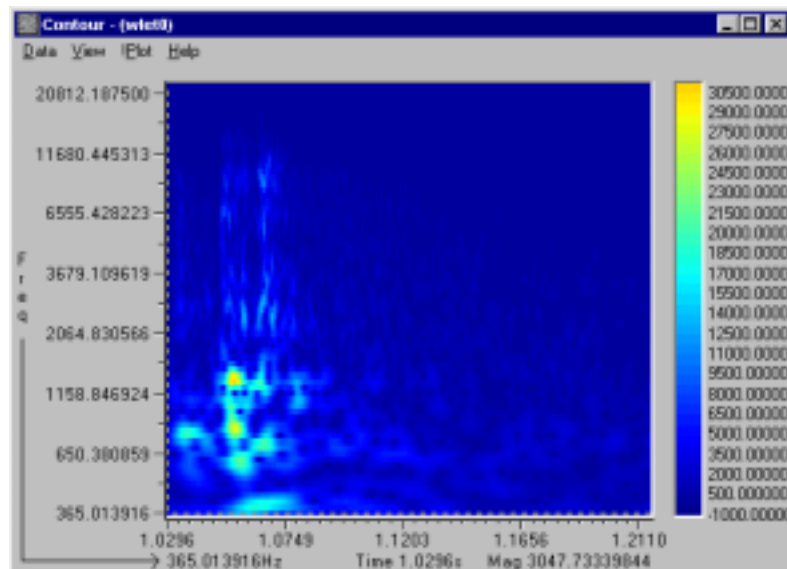
Drop the car slam time data into the WAVELET module. The wavelet calculation starts at the dotted position. The calculation is numerically intensive, so restrict it to the minimum number of data points.



The resulting transform shows the frequency content 8192 times through the 0.2-second impact event.

Distinct impacts exciting different frequencies can be seen, and the type of impact determined from the frequency ranges excited - and therefore linked to mechanical causes within the mechanism of the door closure and lock.

*Wavelet transform of a single door closure.*



---

# Intensity Tour

## Overview

Perhaps more than 80% of the Intensity measurements made are for research and development. The remainder is for source identification and location *in situ*, sound power measurements, and comparison of products' noise output to standard requirements.

The **nVision** Intensity system is intended primarily as a research and development tool, but can be used equally well as a noise-reduction analysis tool for *in situ* measurements. Using the powerful tools available in the **nVision** Intensity system, noise sources can be quickly identified and ranked, and WHAT/IF analyses can be made to determine what noise reductions may be possible.

## Features

- The Intensity system is designed to operate with instruments capable of measuring acoustic Intensity, mainly in 1/3 octaves, but just as easily in other fractional octaves or FFT bandwidth; however fractional octave synthesis from FFT is NOT provided.
- Source ranking for quick and easy identification of highest sources.
- WHAT/IF analysis to quickly determine what would happen if changes were made in the spectrum.
- Colourful contour plotting with vectors.
- 3D plotting.

## Limitations

It's just as important to outline any obvious limitations of a system, as well as features, so although the system is constantly being improved, these are also outlined here:

- Only supports planar surfaces.
- Cylindrical and spherical co-ordinate systems are not supported.
- No support for data quality indicators. Most manufacturers of Intensity measurement instrumentation provide these calculations inside the instrument.
- No support for three-dimensional Intensity. Three-dimensional Intensity measurements are made by very few people. It can be interesting but for most applications provides little more information. The technical problems and large errors associated with rotating the probe with respect to the three axes, and the very high cost and limited frequency range of three-dimensional probes makes this type of measurement impractical.
- Does not merge multiple spacer measurements. This is a feature intended for a future release. Most R&D measurements are made with a single spacer since the problem energy is contained within a limited bandwidth usually covered by one spacer setting.

## Conventions

### **Job, Part and Area**

**nVision** Intensity uses Job, Part and Area format to describe the project and its measured components. This method provides a good way of summing the various parts of a noise source where we have a hierarchy consisting of a single job, with a number of parts, each consisting of a number of areas.

In cases where the hardware supports it, the job, part and area descriptions are entered into the analyser during the measurement process. If the data is being imported from another system that does not support job, part and area directly, then the import procedure will allow the information to be entered at the file translation stage.

### **Area (physical)**

This term is also reused to describe the physical size of each measurement area.

## Vacuum cleaner project

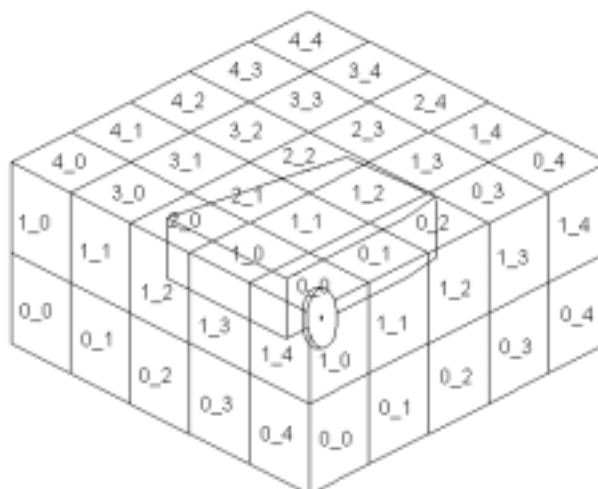
To demonstrate how **nVision** Intensity can be used, we will look at the demo data object “**int2**”, which is a test conducted on a household vacuum cleaner:

For the purposes of this and many similar tests, an imaginary five-sided parallelepiped was constructed around the unit under test, which was sitting on a reflecting surface.

The five surfaces were labelled “Front”, “Back”, “Left”, “Right” and “Top”, and these were entered into the analyser as parts. Each of the four sides had ten measurement areas - two rows by five columns - and the top had 25 measurement areas - five rows by five columns.

Each area was labelled with its location on each surface, starting from (0, 0) to (1, 4) for the sides and from (0, 0) to (4, 4) on the top. The purpose of this project was to determine the value of sound power, identify the highest frequencies and to conduct a WHAT/IF analysis to see what might happen to the overall sound power if the highest spectrum was altered.

*Imaginary Parallelepiped  
- showing the labelling  
convention used for the  
test*



## Step by step

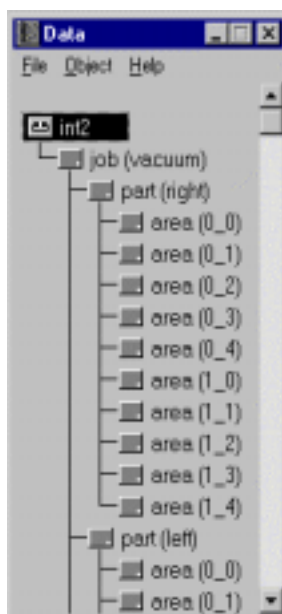
### Load the data

- 1 Go to the File load dialog and change the data type selection to “Intensity”. You should see (at least) two Intensity data sets, named “**int2**” and “**int4**”.
- 2 Select and Load the “**int2**” data object from disk.

#### NOTE:

When “**int2**” is properly loaded, the Data Window will show the data in a tree like structure, showing the JOB, PARTS and AREAS. Only the data object “**int2**” can be dragged into the Intensity modules, since this is a complete “Intensity Object”. However, it is composed of a number of spectra stored as matrix objects, and these individual parts and areas can be dragged into those modules which support matrix objects.

*Intensity object displayed in the Data Window - the component parts and areas will only be displayed if the relevant options are checked*



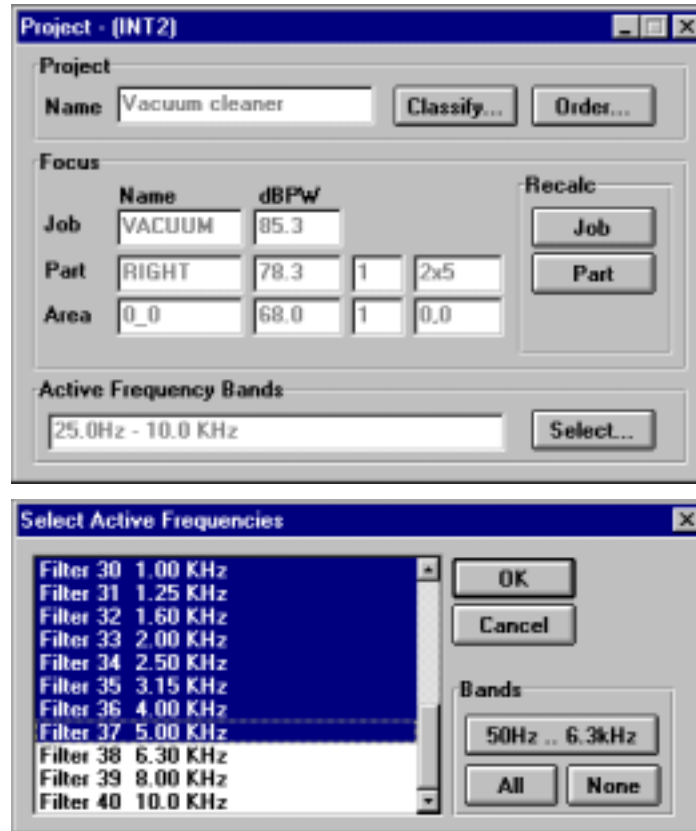
Now drag the “**int2**” data object (subsequently referred to as “the data”) to the various Intensity modules described below:

### Project

- Drag the data to the project module on the Intensity toolbar.
- Observe the summary of the data, and note that you can set the frequency range of interest. The default frequency range of the host instrument was 25 Hz to 10 kHz, but we are only interested in the frequency range of 100 Hz to 5 kHz.
- Press the ‘Select’ button next to the Active Frequency Bands field.
- Now select the ‘None’ button to deselect all the filters.
- Observe that the dark area around the filter and frequencies changes to white - these filters are now deselected.
- Select all of the 1/3 octaves of interest, or filters 20 - 37, (100 Hz - 5 kHz). They should change to a darker colour when selected.

- Press OK. At this point, you will be notified that the bands have been changed and asked if you want to recalculate the data set.
- Answer YES. The data set will now be recalculated so that the Sum and A-wt values take account only of the bands between 100 Hz and 5 kHz.

The PROJECT module.

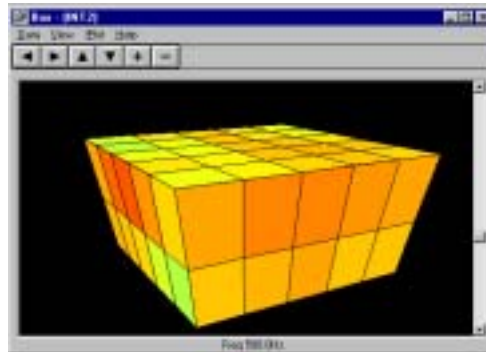


This module will not be needed again so you can now close it down.

## Box

- Drag the data into the BOX module. This module assigns the 65 measurements to their respective places on the parallelepiped. Only one frequency is displayed at a time, with the amplitude being represented by the colour.
- Change the displayed frequencies by moving the scroll bar on the right hand side of the window, and observe the movement through the octave bands.

*The BOX module.*



Box is intended to provide a visual representation of the relative amplitudes on at least three sides at a time. The more areas there are the better resolution there will be.

This module will not be needed again so you can close it down.

## Tree

TREE is a very quick way to view a large group of parts or areas to see the contribution of the various sources at a glance.

- Drag the data into the TREE module.
- Select 'View' and look to see if the 'Show Areas' selection has a check besides it. If it does, click on 'Show Areas' to deselect it.
- Observe that you can see the percentage of contribution of all Parts at a glance.

*The TREE module.*

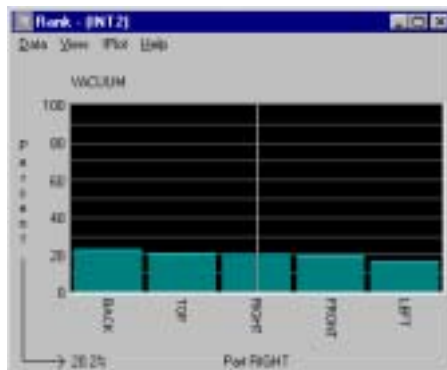
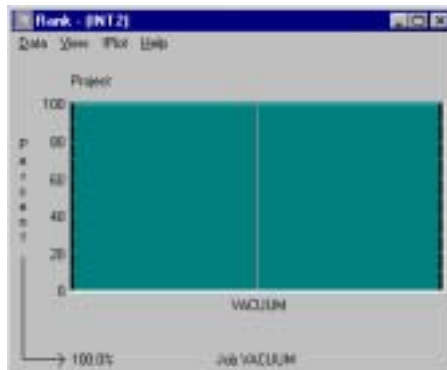
A screenshot of a software window titled 'Tree - (INT 2)'. The window displays a hierarchical tree view of data contributions. At the top, there is a summary row: '100.0%' | '100.0%' | 'VACUUM' | '85.2dBpW' | '80.3dBAPW' | '2.600m²'. Below this, there are five rows representing different areas, each with a blue bar indicating its percentage contribution. The rows are: '20.2%' | '21.2%A' | 'RIGHT' | '78.3dBpW' | '74.2dBAPW' | '0.400m²'; '16.5%' | '19.4%A' | 'LEFT' | '77.4dBpW' | '73.8dBAPW' | '0.400m²'; '19.4%' | '19.0%A' | 'FRONT' | '78.1dBpW' | '73.7dBAPW' | '0.400m²'; '22.8%' | '18.5%A' | 'BACK' | '78.8dBpW' | '73.6dBAPW' | '0.400m²'; '21.0%' | '21.9%A' | 'TOP' | '78.5dBpW' | '74.3dBAPW' | '1.000m²'. The window has a menu bar with 'Data', 'View', and 'Help' options.

This module will not be needed again so you can now close it down.

## Rank

- Drag the data into the RANK module.
- Observe that the left-hand scale should be scaled in Percent from 0% to 100%. At first, only the overall project will be shown and it will show that it represents 100% of the sound power.
- Select 'View' and then 'Parts'. The five parts will be displayed in order of their contribution to the overall.
- Click the cursor to the bar labelled "BACK" so that the vertical cursor is on "BACK".
- Now select 'View' and select 'Areas'. Now the graph shows the ten areas with the one with the highest contribution on the far left. The highest area is 1\_2.
- Move the cursor to that bar.

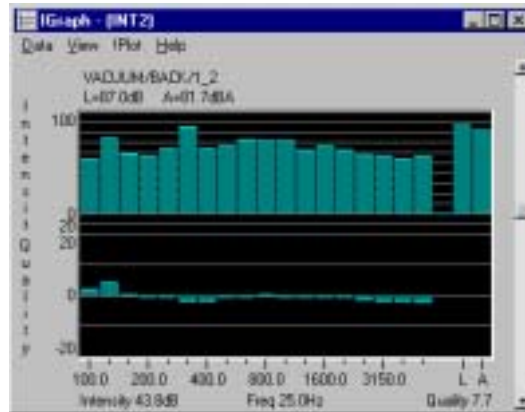
*Various View modes in the RANK module.*



## Igraph

- Now drag the data into the IGRAPH module. In this module, you will see the spectra data as a conventional spectrum plot with the quality spectrum below. The entire spectrum is not shown, but just the values between 100 Hz and 5 kHz, as previously selected.
- Minimise IGRAPH for the time being.

The IGRAPH module.



## What/If

- Drag the data into the WHAT/IF module. In this module, you will see the data presented in a tabular format.
- Observe that only the data from 100 Hz to 5 kHz are shown, as previously selected in the project module.
- Observe the various parts of the display - the top line, in light green, is the sum of all 65 measured areas. Just below that, in dark green, is the sound power for the first part - the right side. Directly under that is the sound power for the ten areas, 0\_0 to 1\_4, of the right side. These are summed upward. By scrolling the screen upward you can see that the data of the remaining four sides is shown in a similar format.
- Select 'View' and then click on 'Show Areas' to deactivate the area data. You will now see that only the Part and Project power levels are shown.

The WHAT/IF module

Power [dB]	100Hz	125Hz	160Hz	200Hz	250Hz	315Hz	400Hz	500
Job Total	53.4	55.1	57.2	57.2	59.5	61.7	61.7	60.1
RIGHT	43.1	48.1	54.0	53.8	54.0	74.9	42.4	
LEFT	49.1	48.4	52.7	57.9	54.9	74.4	50.7	
FRONT	46.8	48.4	53.8	55.4	58.6	78.9	61.8	
BACK	46.9	48.4	53.4	50.2	57.1	77.4	57.7	
TOP	40.7	48.3	54.9	58.8	60.0	75.0	42.1	

- Now reselect 'Show Areas' from the 'View' menu.

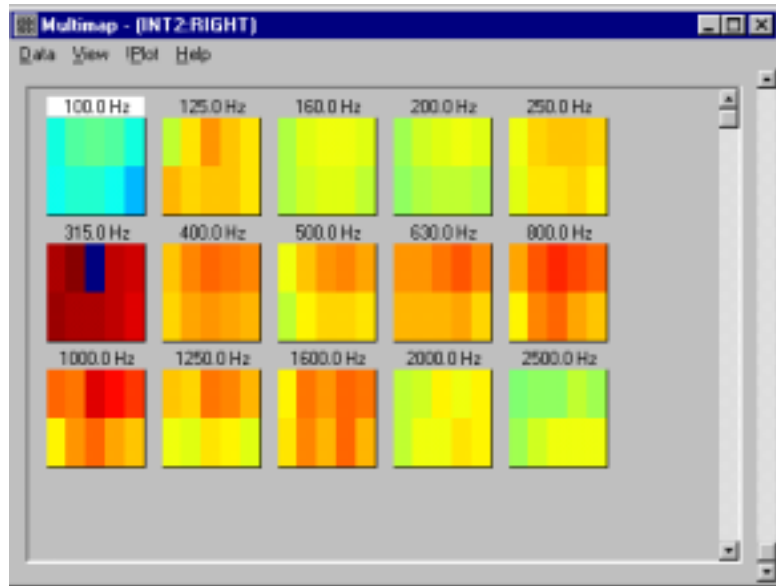
*NOTE: If the data shown are Intensity but the text describes power, then the system may have been reconfigured. To display power, select the 'View' menu and then select 'Channel'. Select power in the displayed combo box.*

- Minimise WHAT/IF for the time being.

## Face

The FACE module is not appropriate for displaying this particular data. There are too few measurement areas to give a meaningful display. The FACE graph is similar to BOX except it only displays one face or surface at a time; however, the FACE module provides smoothing and line-contour plotting whereas BOX does not.

## Multimap



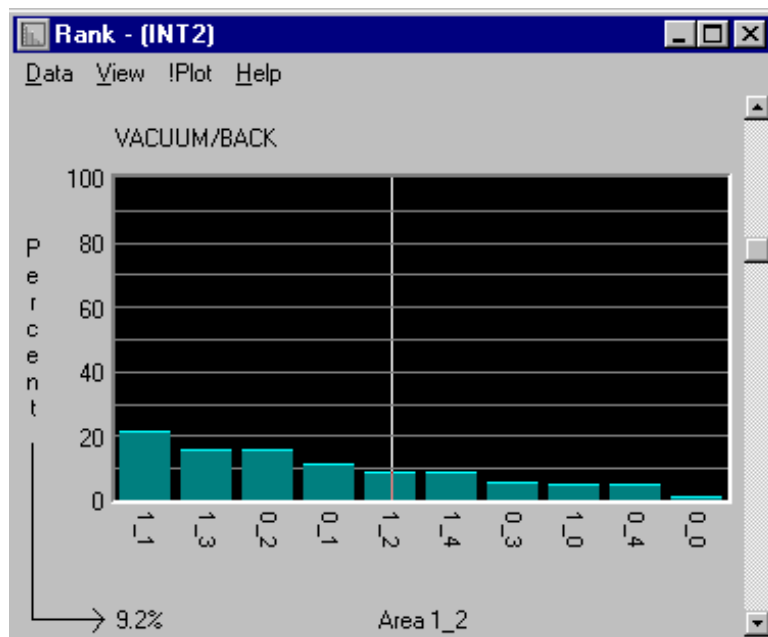
## Putting it all together

As with the standard modules, it's possible to observe the data in more than one Intensity module simultaneously, and because of this powerful feature, you can quickly locate and modify the problem frequencies or sources. The system links activities in the various modules so that cursor movements, zooms, etc., which occur in one module are reflected in other modules that are linked with the same data.

- Move the three module windows, WHAT/IF, IGRAPH and RANK, so that they do not overlap each other. This is so that we can observe the data in each, without the need to manipulate the windows themselves.
- Move to RANK and select 'Parts' from the 'View' menu.
- Move the vertical cursor on to the highest Part, which should be Back.
- Select 'Areas' from the 'View' menu, and again move the vertical cursor to the highest area, which should be 1\_2. This now represents the area with the highest power level. The highest spectrum will be displayed in IGRAPH.
- In the IGRAPH module, move the vertical cursor to the highest peak, 315 Hz. You will see that the WHAT/IF module will highlight the intersection of the 315 Hz and the 1\_2 area. In this example, this is the highest peak in the entire project. At this point we can edit this single peak and recalculate the entire data set.

- In the WHAT/IF module, select 'View' and 'Modify Marked Block'. A small menu will open from which you will indicate the change to be made.
- Click on the highlighted rectangle, 72.0 in the 315 Hz column. This activates the 'Apply' key in the 'Modify Marked Block' dialog. The default is to apply a 10-dB reduction to the highlighted cell.
- Press 'Apply'. The data set will be recalculated and the various modules will be updated. Notice the reduction in the 315 Hz band in IGRAPH and the change in the ranking of 1\_2 in RANK. It is now the fifth highest.
- Change RANK to show the Parts' ranking. The Back is now the fourth highest. These changes in the WHAT/IF module can be applied to a single frequency at a single point or any series of frequencies or areas.

Power (dBm)	100 Hz	125 Hz	160 Hz	200 Hz	250 Hz	315 Hz	400 Hz	500 Hz	L2
1_2	25.4	27.2	45.1	47.8	52.2	67.8	53.5	54.0	
1_3	28.1	29.0	44.7	46.8	48.4	63.9	51.7	53.0	
1_4	37.5	37.6	42.8	43.9	45.9	68.6	49.5	52.0	
BACK	46.9	49.4	51.4	52.2	57.1	75.1	57.7	58.3	
3_6	32.9	38.5	38.6	37.2	41.8	44.7	43.2	43.4	
3_7	35.8	39.4	41.8	39.6	46.5	67.2	46.2	47.8	
3_2	36.8	37.8	40.2	40.7	48.4	68.1	48.7	47.9	
3_3	33.9	35.7	41.1	39.3	48.4	64.2	46.1	45.6	
3_4	31.2	33.0	38.3	37.9	42.4	63.8	46.0	42.2	
1_8	36.4	43.1	39.7	38.1	44.6	59.9	44.4	47.5	
1_1	33.7	43.0	42.8	41.4	48.1	63.4	50.5	52.2	
1_2	40.6	40.2	44.8	43.1	58.9	62.3	58.9	53.4	
1_7	30.4	27.6	41.1	41.1	46.1	71.1	41.0	33.5	



## Loudspeaker project

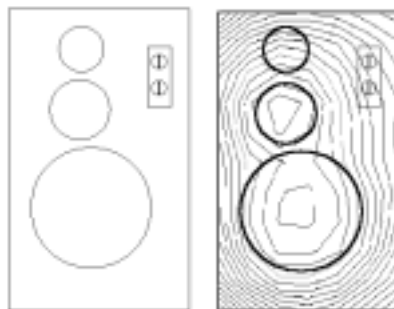
The purpose of this project was to make a contour map of the front face of a loudspeaker, consisting of one bass woofer, one midrange driver and one tweeter. The idea was to observe the crossover frequencies and to see how Intensity modules are capable of mapping the locations of the major drive components.

A grid was laid out over the surface of the speaker consisting of 77 points, in 11 rows by 7 columns, and the Job was labelled as “int4”. There was a single Part labelled ‘Front’, and the 77 points or areas were labelled with the co-ordinate location of the measurement point, running from 0\_0 to 10\_6.

The stationary point method was used to take the data, which can be briefly explored as follows:

- If you followed the previous example, drag the “int2” Intensity object into the trash can, (in order to conserve system memory) and load the “int4” object.
- Drag “int4” into the FACE module so that we can see the contour map. Depending on the smoothing mode selected you will see either a contour map that looks like a series of blocks or one which appears as a smooth map. Block mode provides the fastest plotting.
- This module displays the contour map at a single selectable frequency.
- Move the slider bar to the right of the graph until 500 Hz is shown. You will see that this is the crossover point between the woofer and midrange. At this time, you might want to change the smoothing to filled contour or line to improve the look of the display.
- To see a 3D graph, select the ‘Data’ menu and then the ‘Create Matrix...’ option. Accept the matrix default name of “int4”. You will see that a new matrix has been created in the Data Window.
- Move the newly created “int4” Matrix to the VIEW module on the Base 2 Toolbar.
- Select ‘!Plot’ and you will see a 3D map of the 500 Hz contour map.

*Speaker and Speaker with Intensity Contour Map (at 500 Hz).*



## Things to try

Combine what you have learned in the Animate and Intensity tours by automatically creating a movie of the loudspeaker response in the FACE module and showing the result in the Animate module. This will show a 3D map of the loudspeaker baffle for every third octave band.

---

# DSP Tour

## Overview

This tour will use some of the simpler facilities available from the **nVision** DSP Toolbar to give you some “hands-on experience”. The tour will start by creating a sine wave, which is buried in white noise. The main objective of the tour from that point on will be to recover the sine wave from the noise by application of a suitably designed filter. This process will involve the use of most of the basic modules provided by the DSP Toolbar.

## Features

The DSP toolbar consists of a suite of modules designed to perform signal processing on time-domain signals. The main functions include:

- Display and Playback of signal traces

- FFT Toolbox (functions include Power Spectral Density, cross spectrum, coherence, correlation and transfer functions).

- Wavelet analysis

- Fractional Octave analysis

- FIR Filter Design.

- Waveform generation

## Generating a Test Signal

Execute the WAVEFORM module by double clicking on its icon in the DSP toolbar. The following window should appear.



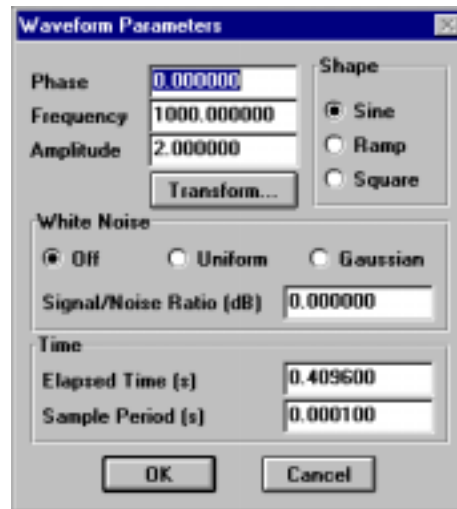
Click on the ‘Signal...’ button to invoke the following dialog:



The dialog will show the name of the signal object that WAVEFORM intends to create; it will be of the form “waveX” where X is a unique number. If no other signal objects are currently loaded into **nVision**, the name of the signal to be

created should be “**wave0**”. For this tour, we only require a single channel of data so we may accept the default settings by clicking on the ‘OK’ button.

Now we must define the parameters of the signal to be created. To do this click on the ‘Parameters...’ button. The following dialog will be displayed:



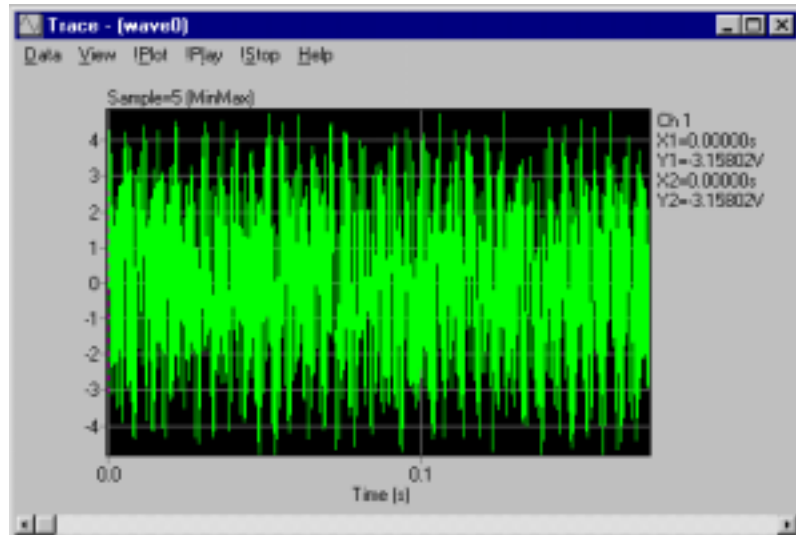
From this dialog, we wish to define a 200 Hz sine wave corrupted by uniform white noise with a S/N of -10 dB. Click on the ‘Frequency’ edit box and enter a value of 200. Next click on the ‘Uniform’ radio button to enable the noise generator and specify a value of -10 in the ‘Signal/Noise Ratio (dB)’ edit box. The remaining options may be left with their default values. Click on the ‘OK’ button when you have completed the changes.

We are now ready to create the test signal, which is done simply by clicking on the ‘Create’ button. The new signal object should be visible in the **nVision** data window with the name “**wave0**”.

## Viewing the test signal

Load the TRACE module by double clicking on its icon in the DSP toolbar. The module should appear with a blank frame.

When TRACE appears, transfer the signal “**wave0**” from the Data Window to the module using drag and drop (described in the “**nVision Basics**” section). TRACE should react by changing its title bar to read “**Trace - (wave0)**” and automatically plotting the signal data. The trace module shows a representation of the time data, where time appears on the X-axis and signal level on the Y-axis. For the noise corrupted sine wave, we should obtain a display similar to the one below.



Trace includes a pair of cursors, one solid and one dotted, which may be used to take readings from the channel. At the top right of the display are four values: X1, Y1, X2 and Y2. X1 and Y1 represent the position of the solid cursor, while X2 and Y2 represented a dotted cursor. When a signal is first created the cursors are set to the first data point in the channel (i.e.  $X1 = X2 = 0$ ). To change the position of the solid cursor, simply move the mouse pointer over the point of interest in the signal and click the left mouse button. You can then use the arrows keys on the numeric pad to move to the exact location. A similar approach is used to change the position of the dotted cursors but this time the right mouse button must be used to select the approximate position before the arrow keys may be used.

### ***Identifying a region***

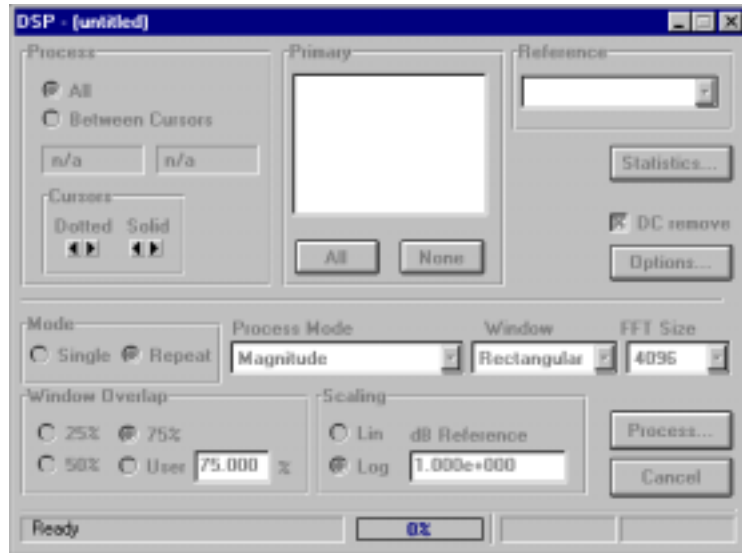
Trace will allow you to zoom in on a region of interest in the data set by using click and drag. Practise this now by first selecting 'Unzoom' from the 'View' menu. The signal should have a duration of 0.4096s. Zoom in on an area of the signal around 0.2s by clicking and holding down the left mouse button and drawing the mouse right and downward (still holding the button down). As you move, an area is highlighted in inverse colour. When you let go of the mouse button, the highlighted region should be expanded to fill the trace display.

### ***Signal Playback***

If you have a PC equipped with a suitable sound card, TRACE will have enabled the '!Play' button on its menu. Playback will commence from the earliest cursor position and end on the latest one. If you have a sound card, practise this now: first, unzoom the signal and set the cursors at the signal's start and end positions. Then simply hit '!Play' to hear the noisy sine wave.

## **Creating a Power Spectrum of the Test Signal**

Load the DSP module by double clicking on its icon on the DSP toolbar. The following window should appear:



When DSP appears, transfer the signal “**wave0**” from the Data Window to the module using drag and drop (described in the “**nVision Basics**” section). DSP should react by changing its title bar to read “**DSP - (wave0)**” and enabling many of the DSP options. In this tour we want to generate the power spectrum of our test signal so that we can later compare it with the signal that has had the noise suppressed. To obtain the Power Spectrum of the test signal, perform the following steps:

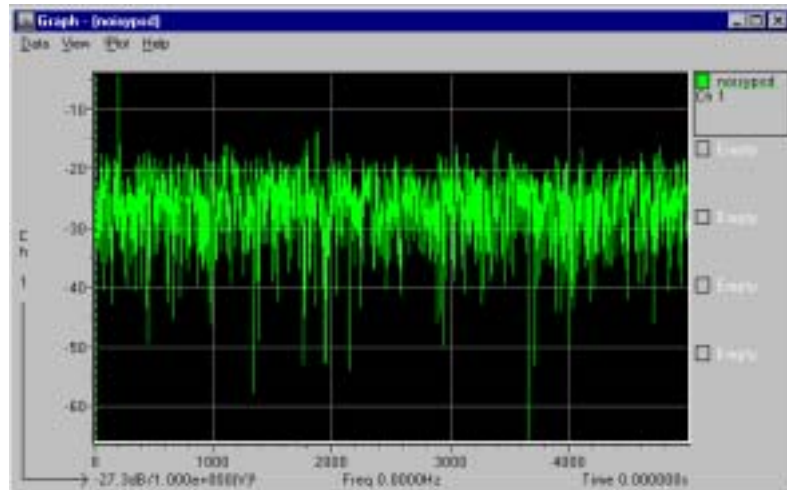
- Click on the ‘Single’ radio button in the ‘Mode’ group.
- Select ‘Power Spectrum (PSD)’ from the ‘Process Mode’ list box.
- Select ‘Blackman’ from the ‘Window’ list box.
- Select ‘4096’ from the ‘FFT Size’ list box.
- Select the ‘Log’ radio button in the ‘Scaling’ group. Also, enter a value of 1.0 in the ‘dB Reference’ edit box if it is not already set to this value.
- Click on the ‘All’ button in the ‘Primary’ channel group to select the channels to process. The ‘Process...’ button should now be enabled.

Once DSP is configured, click on the ‘Process...’ button to start the analysis. You will first be presented with a dialog prompting you for a matrix name. Enter the name “**noisypsd**” in the edit box and click on ‘OK’.

Depending upon the settings in DSP’s ‘Options’ dialog, you may be presented with a dialog with a title “Assign Channels to Matrix Rows”. If this dialog appears, simply respond by clicking on ‘OK’.

## Viewing the Power Spectrum

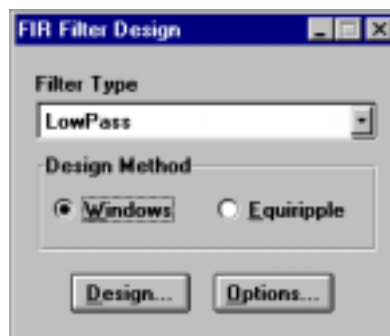
Once DSP has completed its calculations, a new matrix object called “**noisypsd**” should appear in the Data Window. To view this data set, launch the GRAPH module from the Base 1 Toolbar, then drag and drop the “**noisypsd**” object into the module. More detailed instructions on using GRAPH are available in the “Base 1 Tour”. Try zooming in on the data set from 0 to 2000 Hz. Using the cursors you should be able to determine that the peak level of the 200 Hz sine wave is -4.0 dB, and from visual inspection it is clear that the noise level is around -20 dB.



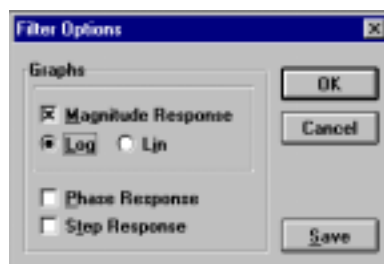
## Designing a Bandpass Filter

The signal to noise ratio of the test signal can be improved drastically by passing the test signal through a bandpass filter, which has been designed to pass the 200 Hz sine wave unmodified, but to suppress all other components. In this part of the tour, we will design one such bandpass filter using the DSP filter design module.

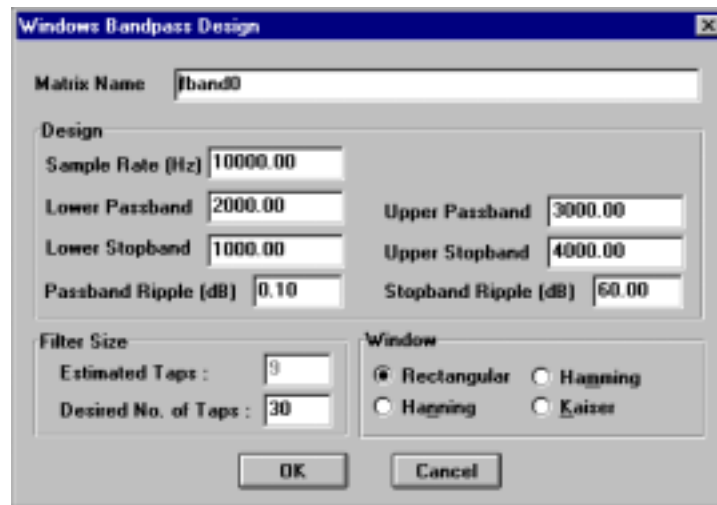
Load the FILTDES module by clicking on its icon in the DSP toolbar. The module should appear as shown below.



First, click on the 'Options...' button to invoke the following dialog. These options allow the user to select which of the Magnitude, Step and Phase responses are generated automatically whenever any filter is created. For this tour, we are only interested in the Log Magnitude Response. Therefore, ensure that only the Magnitude Response check box is checked and that the Log radio button is selected. Click on the 'OK' button to return to the main panel.



From the main FILTDES panel, select 'Bandpass' from the 'Filter Type' list box and click on the 'Windows' radio button in the 'Design Method' group box. We are now ready to design the bandpass filter, so click on the 'Design...' button to bring up the dialog for bandpass filter design.

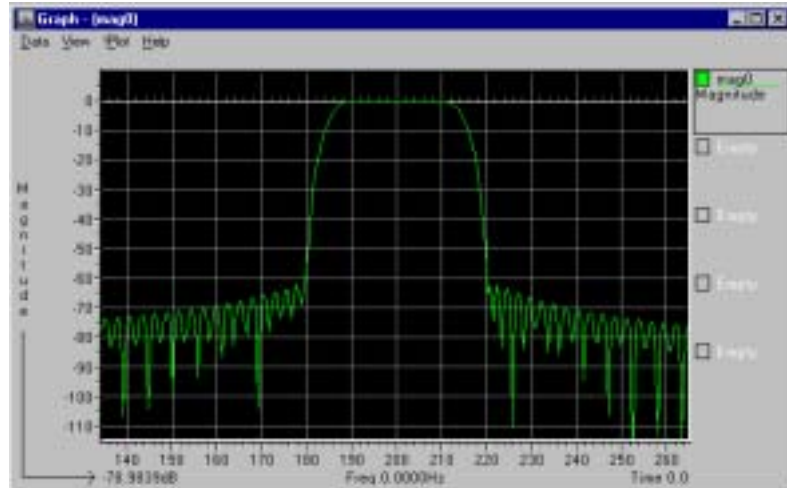


We require a filter that will allow our 200 Hz sine wave to pass unchanged, but will suppress as much of the noise as possible. Unfortunately, there is not enough time within this tour to explain the mathematics used in the design of such filters; this is discussed in many good DSP text books. For the purposes of this tour, we will design a filter with a pass band from 190 Hz to 210 Hz and stop band frequencies of 180 Hz and 220 Hz. Enter the following values:

- Select 'Kaiser' from the 'Window' group.
- Enter "190" into the 'Lower Passband' edit box.
- Enter "210" into the 'Upper Passband' edit box.
- Enter "180" into the 'Lower Stopband' edit box.
- Enter "220" into the 'Upper Stopband' edit box.
- Finally, the 'Estimated Taps' edit box should be displaying a value of 3625. This is the minimum number of taps necessary to realise a filter with the specified parameters. Use this value by entering it into the 'Desired No of Taps' edit box.

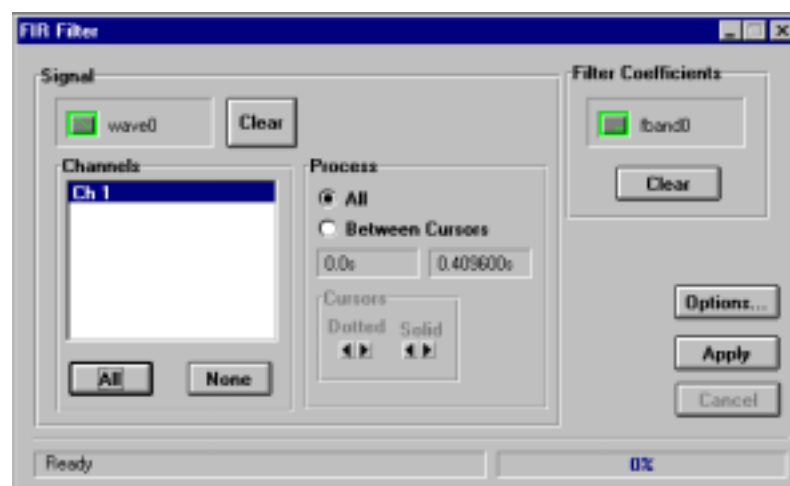
Now that the filter design parameters have been set, click on the 'OK' button to initiate the design. FILTDES will create two new matrix objects, which should appear in the Data Window. The data set "**fband0**" contains the filter coefficients while "**mag0**" contains the log magnitude response of the filter.

The magnitude response may be viewed by dragging and dropping "**mag0**" into the GRAPH module. By zooming in around 200 Hz, you should see a response similar to the one shown below.

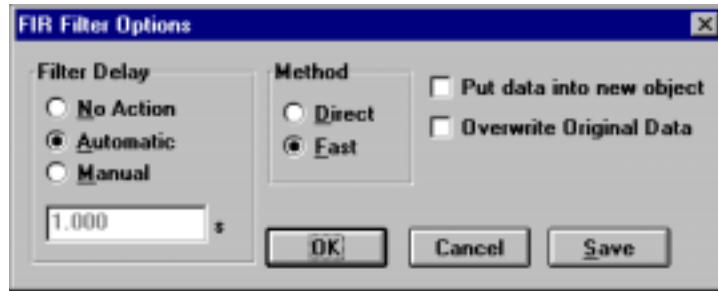


## Applying a Filter to the Signal

The next stage in this tour is to take the filter we have just designed and apply it to the noisy 200 Hz sine wave in order to improve its signal to noise ratio. The DSP Toolbar includes a Finite Impulse Response (FIR) filter module called FIRFILT, which performs this operation. Execute the FIRFILT module from the DSP Toolbar in the normal way. The module has two *bays*, which are areas on the window designated as dropping zones. The left-hand bay accepts the signal object that is to be filtered, while the right-hand bay accepts the matrix object that contains the filter coefficients. Take the “**wave0**” data set from the Data Window and drag it into the left-hand bay. The name of the data set should appear in the bay, as shown below. Next, take the “**fband0**” data set and drag it into the right-hand bay. Select the channel “Ch 1” by pressing the ‘All’ button in the ‘Channels’ group. The ‘Apply’ button should now become enabled, indicating that the module is ready to apply the filter.



Before the filter is applied, we may need to change some of the FIRFILT options. Click on the ‘Options...’ button to bring up the following dialog:

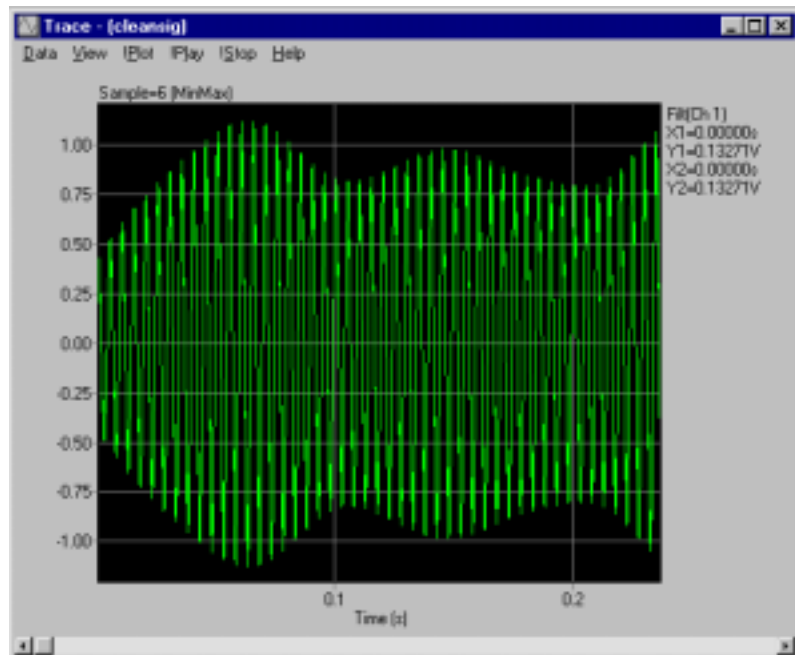


Change the settings on the options dialog so that they match the one shown above. 'Filter Delay' should be 'Automatic', 'Method' should be 'Fast' and the 'Put data into new object' check box should be checked. Once these options are set click on 'OK'.

From the FIRFILT main window, click on the 'Apply...' button. A dialog will appear prompting for a matrix name for the filtered signal. Enter the name, "cleansig" into the edit box and click on 'OK'. The FIRFILT module will proceed to filter "wave0" and the result should appear in the Data Window as a new signal object called "cleansig".

### Viewing the filtered result

Take the data set "cleansig" and drag and drop it into trace. You should see a result similar to the one shown below. The 200 Hz sine wave is now clearly visible above the noise.

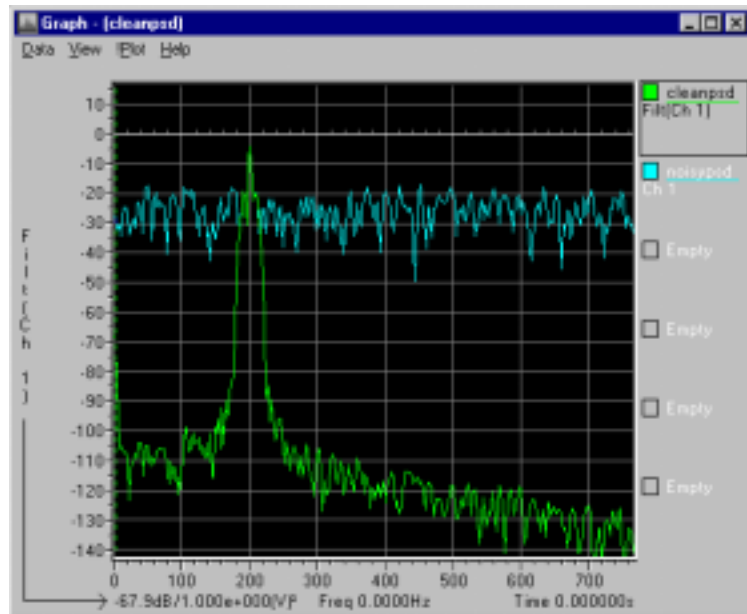


### Comparing Power Spectra

Generate a power spectrum for the filtered signal "cleansig" using the same procedure described in the above section "Creating a Power Spectrum of the Test Signal". The only difference will be in the name of the generated spectrum; use the name "cleanpsd" for the new data.

By dragging and dropping both "cleanpsd" and "noisypsd" into a copy of GRAPH, we can directly compare the results of the filtering process. Again,

zoom in around 200 Hz to produce a graph similar to the one shown below. It should be clear from these plots that the noise level has been reduced by over 60 dB outside the pass band of our bandpass filter, while the 200 Hz sine wave remains unchanged in level.



## Things to try

Try producing a two-channel signal object in which each channel contains a sine wave of the same frequency, but where both are corrupted by different types of noise (first channel uses Uniform while second channel used Gaussian). Ensure that you specify a signal/noise ratio which buries the sine wave (-10 dB should be sufficient). Using the DSP Toolbox coherence operation between these two channels, you should be able to identify that the only coherent component is the sine wave.



---

# Import Tour

## Overview

In this tour, you are introduced to the range of data import options available within **nVision**. **nVision** includes a wide range of device and file drivers, dynamic data exchange links with *Microsoft Excel* and a flexible exchange module that allows import and export of data to user-defined formats.

## Help on Importing Files

Even with the minimum **nVision** system, you can access the EXCHANGE module, from which you can import data. You can find out more information about this module by accessing its on-line help. To do this you need to select the 'Contents' option from the 'Help' menu. Also, see the "Exchange" Tour, which follows this section.

If you have the Drivers toolbar enabled in your **nVision** system, you can find out more information about importing data by accessing the Drivers help from the Drivers toolbar's control menu. Selecting the 'Help Contents' option will display the **nVision** Drivers' help file.

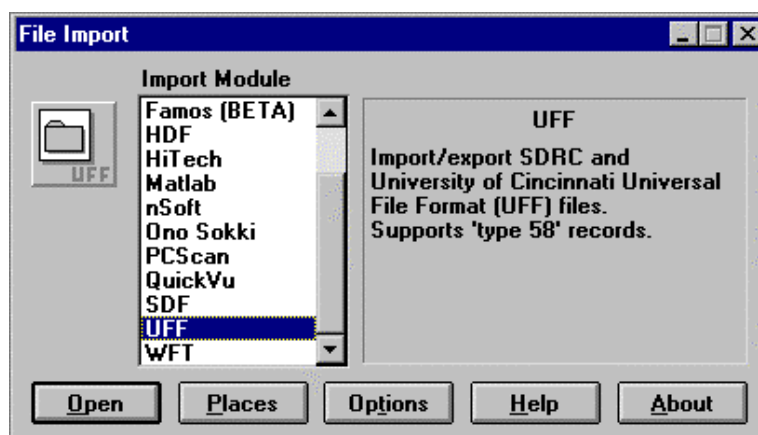
You can also access help for a specific driver module by selecting the 'Contents' option from the 'Help' menu of the specific module. If it does not have a help menu then select the 'Help Contents' option on the module's control menu.

## File Import - Universal Files

The File Import menu supports Specific file formats, exclusive from the EXCHANGE module. One example is the SDRC Universal file format.

An example universal file is "**crash3.uff**".

*File Import menu*



Select the file format.

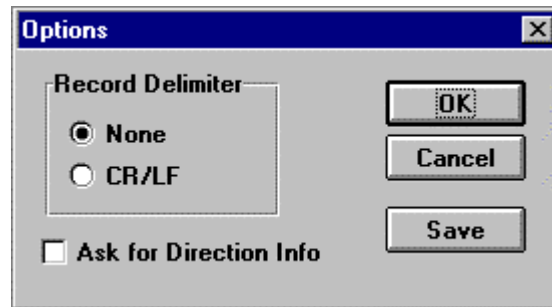
*Chose the appropriate file delimiters*

Choose the appropriate import filter options, by clicking on the 'Options' button.

Universal files are ASCII (text) files. However, each line of the file

may have a record delimiter at the end of it. Whether this is present will depend upon the operating system of the computer used to create the file, and the transfer method used to move it to your PC. A UNIX based system, for example, is likely to include CR/LF (carriage return/line feed) characters at the end of each line.

Tip: if the file read fails to find any records in the file, try the other option. Once you have identified the correct setting, click on Save in the options menu to set this as the default value.



Read the file by clicking on the 'Open' button, and entering the filename.

The data will appear in the Data Window. In the case of the car crash data, this is a time series. It would make a good data source to try Wavelet analysis.

## Exchange

### Introduction

The EXCHANGE module is capable of importing and exporting data in a variety of file formats, and is also capable of learning new formats, through the 'intelligent' macro facility.

### Binary file import

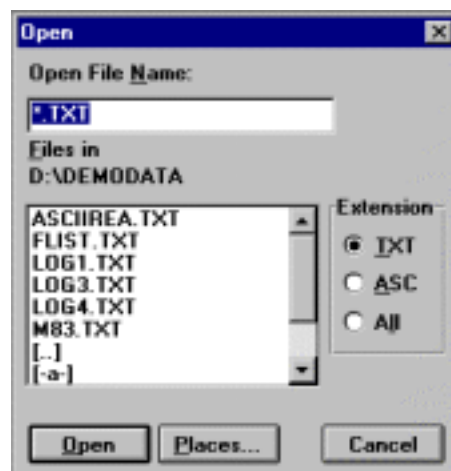
Binary files are much simpler to import than ASCII files, but they also require the user to understand exactly how the source file is laid out in order to import data correctly. Unlike an ASCII file, binary files are not self-describing, and it's not possible to have a peek into them to get some clue as to their layout. Consequently, the importing and exporting of binary data is covered in depth in the manual.

### ASCII file import

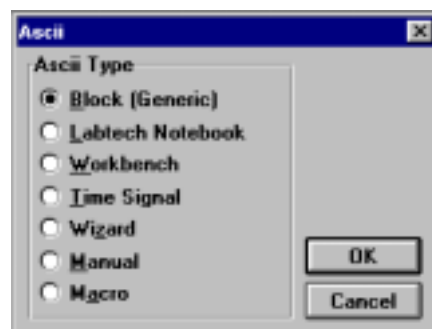
The following procedure describes the steps necessary to import an ASCII file, that is, a normal readable text file.

We will import a file called **"log4.txt"**, which as we shall see, contains two channels of data.

Start the EXCHANGE module from the Base 1 toolbar. The first step is to choose the 'ASCII' option from the 'Import' menu, at which point we will be presented with the dialog shown below:

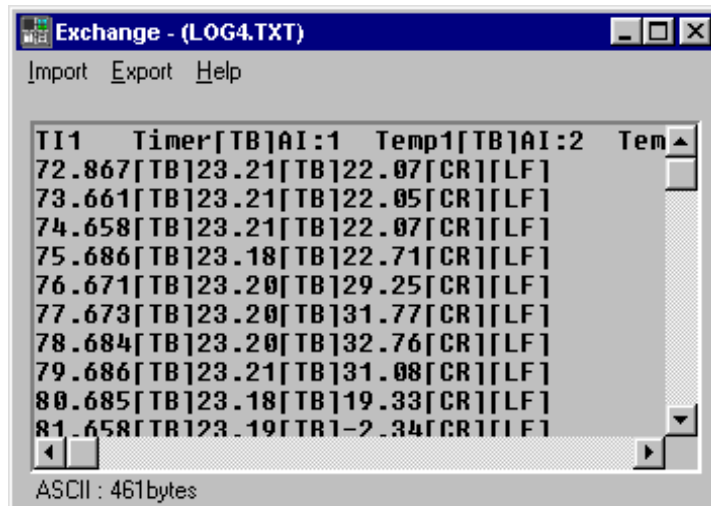


Highlight the file **"log4.txt"**, and click on the 'Open' button.



We are going to perform a wizard import, where each step of the process is under the control of the user. The program automatically monitors the steps we take during the import procedure, and later we'll get the chance to save those steps as a macro, which can then be used to import a similar file automatically.

Select the 'Wizard' option and click on 'OK'.



The program reads the "log4.txt" file and automatically analyses it, so that it can offer the user a reasonable idea of what the file contains. Due to the varied nature of ASCII files, this can be a difficult job, but the program knows about the more common methods. The file shown here has two channels (columns) of time stamped data, separated by tabs, with lines separated by linefeed characters.



The analysis offered automatically by the program can be accepted at this point, or if the display doesn't look right the user can specify alternative line and column separators, and then try these out using the 'Reformat' button.

When the data layout displayed in the client area of the program is acceptable, click on 'OK' to accept the formatting and the program will perform the first regular trial. Our file should now look something like the display shown below. The program has organised the data into regular rows and columns, and we can now clearly see the Time stamps, and the two channels of data, A1:1 Temp1 and A1:2 Temp2.

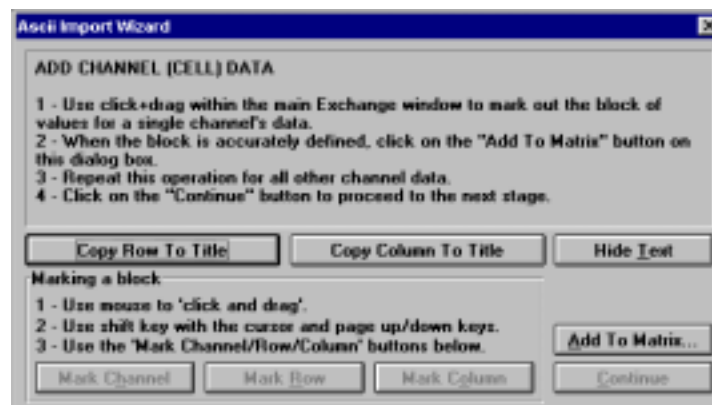
	1	2	3
1	T11 Timer	Al:1 Temp1	Al:2 Temp2
2	72.867	23.21	22.07
3	73.661	23.21	22.05
4	74.658	23.21	22.07
5	75.686	23.18	22.71
6	76.671	23.20	29.25
7	77.673	23.20	31.77
8	78.684	23.20	32.76

ASCII : 461 bytes

The data as displayed is obviously more readable than the original, but in order to create a data object we need to take further steps to refine the layout. From the first trial format above it's clear that the numeric data is mixed up with information that describes the data.

In common with the EDITOR module the EXCHANGE module regards central white grid portion of the display as data, and the surrounding grey buttons as the parts which display information about the rows and columns of the grid.

Using this principle as a guide, we can move the top line of the display so that it occupies the column description area. This is achieved very simply, using the import wizard dialog.



The top row of the display should already be highlighted, but if it isn't, click on the '1' button on the left-hand side of the row. Now click on the 'Copy Row To Title' button on the wizard dialog. This copies the text from the selected row onto the grey buttons above, but leaves the original text intact. Although this step is not necessary for importing data, you will find that copying the appropriate row and column titles onto the buttons will help when you are moving around in larger data sets.

Exchange - (LOG4.TXT)  
 Import Export Help

	Tl1 Timer	Al:1 Temp1	Al:2 Temp2
1	Tl1 Timer	Al:1 Temp1	Al:2 Temp2
2	72.867	23.21	22.07
3	73.661	23.21	22.05
4	74.658	23.21	22.07
5	75.686	23.18	22.71
6	76.671	23.20	29.25
7	77.673	23.20	31.77
8	78.684	23.20	32.76

ASCII : 461bytes

Having moved the first row upwards into the column title area, we can also move the first column containing the time, so that it appears as the row title. Highlight the column by clicking on the grey area marked 'Tl1 Timer' at the top of the first column.

Exchange - (LOG4.TXT)  
 Import Export Help

	Tl1 Timer	Al:1 Temp1	Al:2 Temp2
1	Tl1 Timer	Al:1 Temp1	Al:2 Temp2
2	72.867	23.21	22.07
3	73.661	23.21	22.05
4	74.658	23.21	22.07
5	75.686	23.18	22.71
6	76.671	23.20	29.25
7	77.673	23.20	31.77
8	78.684	23.20	32.76

ASCII : 461bytes

Click on the 'Copy Column To Title' button to use the highlighted column as the row title. The display should now appear something like that shown below.

	TI1 Timer	AI:1 Temp1	AI:2 Temp2
TI1 Timer	TI1 Timer	AI:1 Temp1	AI:2 Temp2
72.867	72.867	23.21	22.07
73.661	73.661	23.21	22.05
74.658	74.658	23.21	22.07
75.686	75.686	23.18	22.71
76.671	76.671	23.20	29.25
77.673	77.673	23.20	31.77
78.684	78.684	23.20	32.76

ASCII : 461bytes

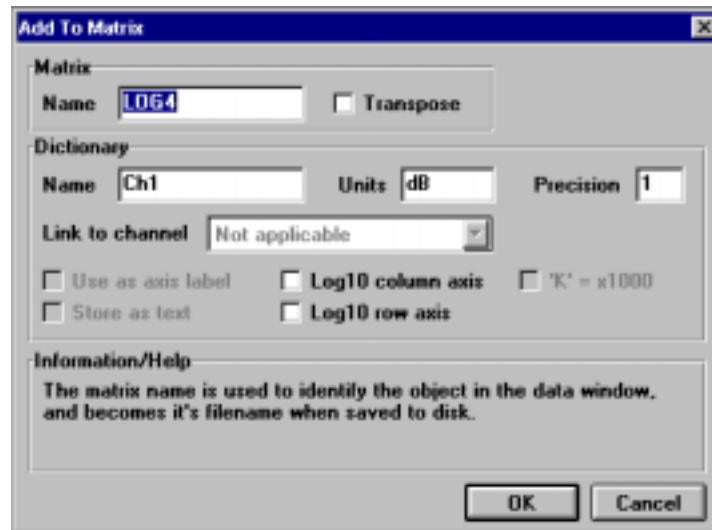
The display is now in a readable form, with the data positioned in the grid, and row and column titles correctly established. We're now ready to proceed to the next stage, where a data object is created.

Now we need to mark out the numerical data values that apply to one channel. In this case, we start a click and drag operation at the "23.21" value immediately below "AI:1 Temp1". Drag vertically downward to the last value (also "23.21") which is on the row labelled "92.671".

	TI1 Timer	AI:1 Temp1	AI:2 Temp2
AI:1 Temp1	AI:1 Temp1	AI:1 Temp1	AI:2 Temp2
86.685	86.685	23.21	-11.26
87.685	87.685	23.19	-5.34
88.673	88.673	23.19	-14.56
89.683	89.683	23.20	3.75
90.669	90.669	23.20	6.06
91.657	91.657	23.20	15.69
92.671	92.671	23.21	16.80

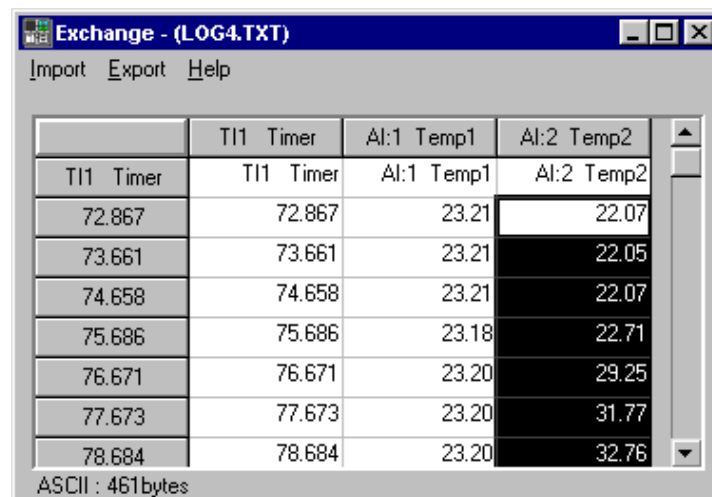
ASCII : 461bytes

On the import wizard dialog box, click the ‘Add To Matrix...’ button.



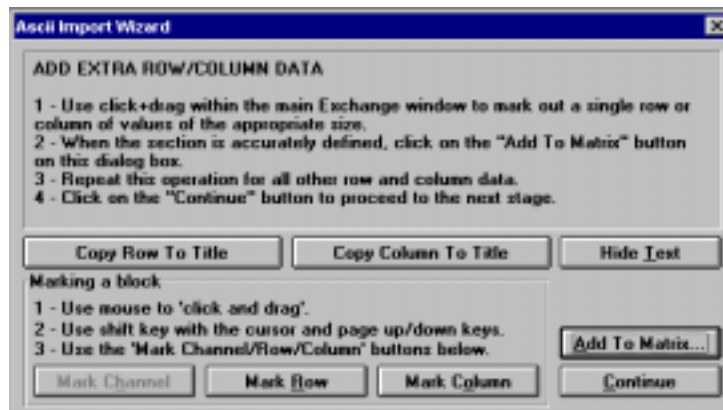
Click on ‘OK’ to perform the operation and return to the table of values.

We have now added one channel of data and can continue by adding the second. Click on the “22.07” value immediately below “A1:2 Temp2”, but this time do not drag out the values for this channel. By adding the first channel, we effectively told the program how many rows and columns each channel occupies. Now we can let it do the work for us by selecting the first value for a channel and clicking on the ‘Mark Channel’ button.

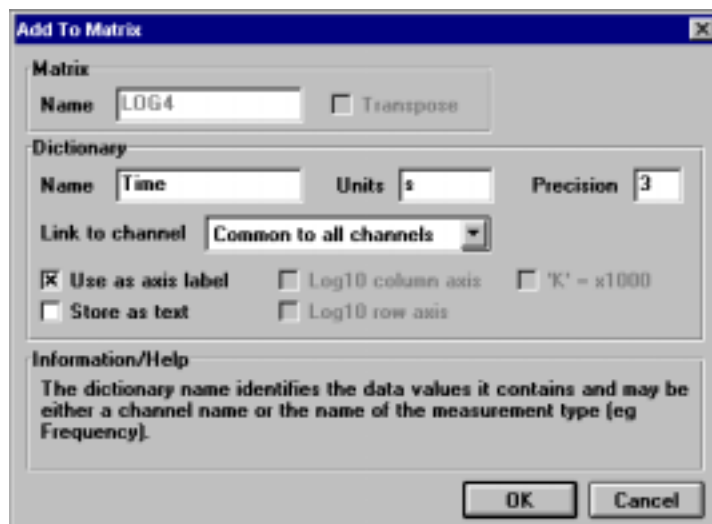


Select ‘Add To Matrix...’ and click on OK.

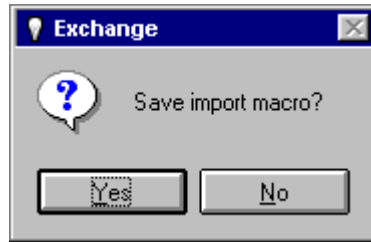
We have now finished adding channel data values, so click on the 'Continue' button to move on to the next stage.



Click on the first "time" value (72.867) in the grid and click on the 'Mark Column' button on the dialog box. Select 'Add To Matrix...' and change the information to match that shown below, then press 'OK'.

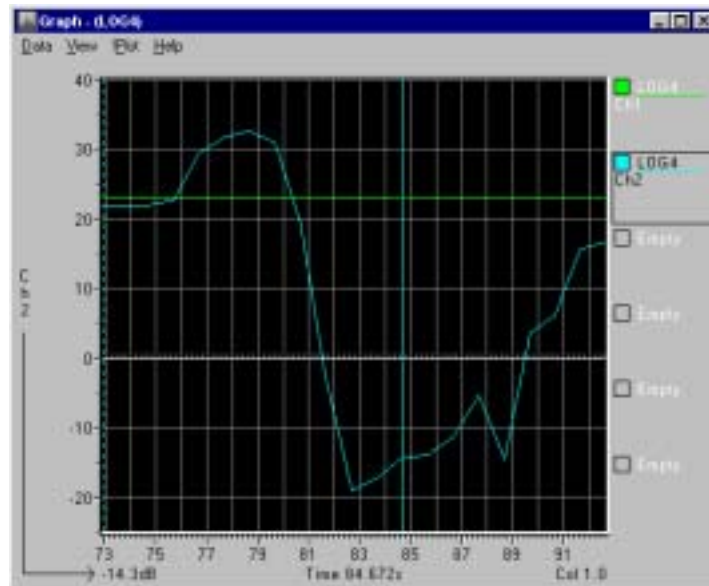


We do not need to add any further information to the object, so click on the 'Continue' button and then the 'Done' button (this is actually still the continue button, but the text changes).



At this point, you will be asked whether or not you want to save the import macro, and under normal circumstances, you probably would want to. However, as **“log4.txt”** is one of the demonstration files supplied with the system, it will probably be of no further use.

*Viewing the resultant matrix in the GRAPH module.*



# XLDriver - Microsoft Excel Data Exchange

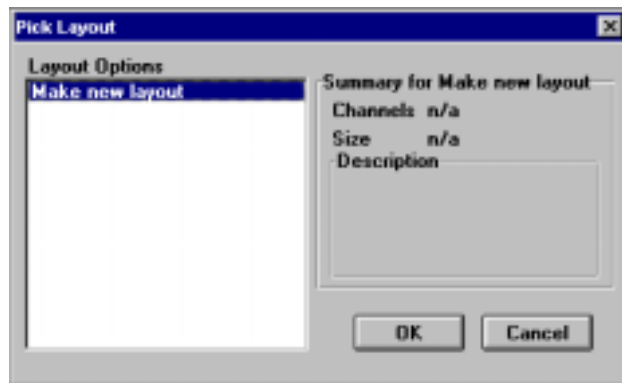
## Introduction

The XLDRIVER module allows bi-directional data linking to Microsoft *Excel* using DDE (Dynamic Data Exchange). XLDRIVER uses layouts to specify the position of data items in *Excel*. Layouts are easy to create using the XLDRIVER layout wizard, which takes you through all the steps necessary to create a layout; once created the layout can be stored for later use.

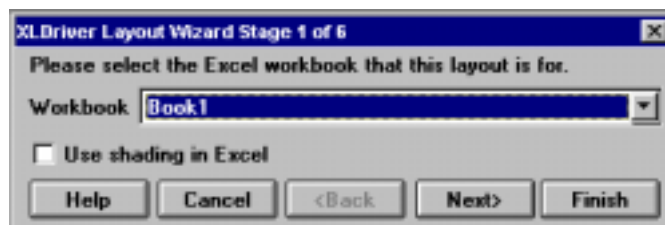
## Importing Data

To import data into **nVision**, the XLDRIVER needs to know where to look for the information in *Excel*. To do this we use a layout. Layouts are created using the layout wizard. In this example, we'll import a 10x10 channel of data from *Excel* along with its row and column labels.

Execute the XLDRIVER module from the Custom toolbar, and click on the 'Import...' button. If *Excel* is not currently running you will be asked if you want to start the program now. We are now asked to pick a layout. If you had previously defined layouts, they would appear in the list ready to be used again. Select 'Make new layout' and press 'OK'.

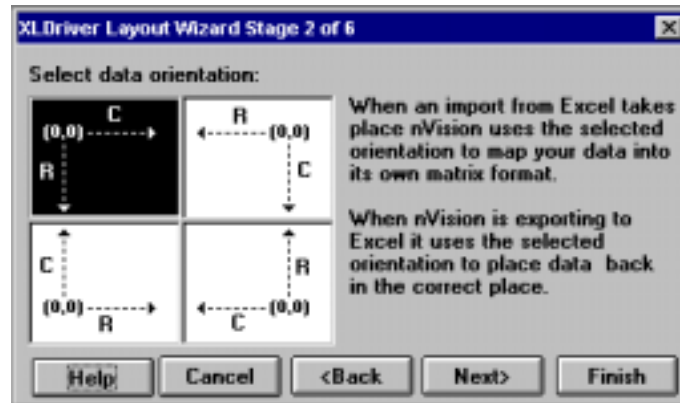


You will now be placed in stage 1 of the layout wizard. From here, you select the *Excel* workbook from which you will import data. If you select the 'Use shading in Excel' option then whenever you select an area in *Excel* to import (or export), the area will be highlighted, so you know exactly what you are importing. This is not recommended for large areas.



Each stage of the layout wizard uses the same five buttons, Help, Cancel, Back, Next and Finish. 'Back' and 'Next' allow you to look at the previous stage or next layout stage respectively.

You can 'Finish' using the layout wizard after the third stage, using default values for the remaining questions. If you press 'Finish' before then you will be warned that insufficient information has been entered and given the choice to return to the layout wizard.



Stage 2 of the layout wizard lets you select the orientation of your data. This stage need not concern us since we are just starting, so we will just select the default setting offered. More information on this stage can be found in the on-line help for XLDRIVER.

Stage 3 of the layout wizard allows you to define channels of data. You add a channel to the layout by highlighting the area in *Excel* and then pressing the 'Add to Layout' button. XLDRIVER always uses the currently marked area in *Excel* when adding information about layouts. The current selection in *Excel* is shown in the 'Current Range' box.



Once you have added the first channel subsequent additions can be sized automatically by using the 'Auto size' or 'Drill from master' option. (See on-line help for more details.)

For our example, type some numbers into a 10x10 area in *Excel*. Now highlight this block in *Excel*; the current range should be similar to the one shown above. Now press 'Add to Layout'.



The channel description box will now appear. This box lets you give the channel a name and select its units. Accept the default name and type something in for the units, then press 'OK'.

You will now see the information about the newly entered channel in the list box on the right-hand side of the stage 3 layout wizard. We only wanted one channel of data, so we can now move onto the next stage by pressing the 'Next>' button.



In stage 4, we define any row & column data that we need to import. If you click in *Excel*, you will notice that the current selection is automatically sized. This allows you to enter the row & column labels very quickly. The type of label that is to be added depends upon the selection in the 'Label Type' group. Once you have defined the row/column that you require, the procedure to add it is the same as it is for adding a channel.

The remaining stages of the layout wizard define the text you can associate with individual channels & the data set as a whole. We are not concerned with them here so choose the 'Finish' button.

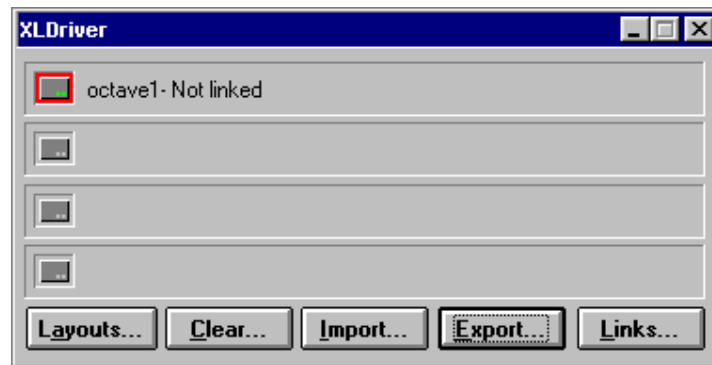
You will then be asked if you want to save the layout; choose 'No'. Then you will be asked for a name for the matrix that will be created; accept the default name by clicking 'OK'.

A new matrix will have appeared in the Data Window. If you drop it in the editor module, you will see that the values match those that are in *Excel*. You can test the dynamic linking by changing values in *Excel* and **nVision**.

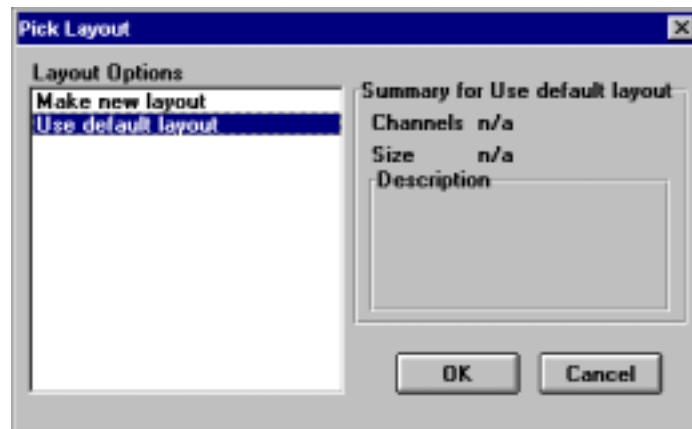
### Exporting Data

In this example, we'll export a data set to *Excel*, and create a hot link, so that changes made in **nVision** are automatically updated in the *Excel* sheet.

Execute the XLDRIVER module, and load "octave1" in the first bay.



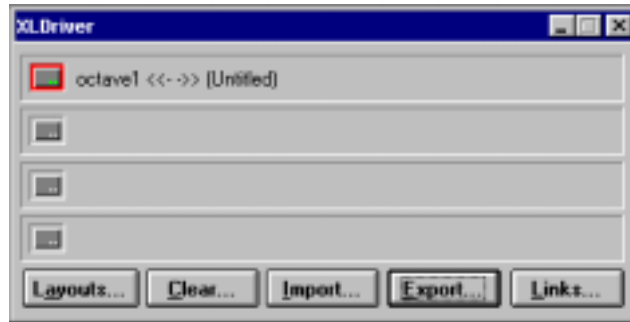
Once the data set is loaded, click on the 'Export' button to start the export operation. If *Excel* isn't currently running, XLDRIVER will ask if you want to execute it. Presumably, we do, and if *Excel* is on the path, it should run.



We are now asked to pick the layout we want. Layouts are created using the Layout Wizard. This is a series of six dialog boxes that guide you through the process of creating a layout. We will select the default layout; this is one created by XLDRIVER to fit the matrix we are going to export.

Select 'Use default layout' and press 'OK'. The first stage of the layout wizard will appear. Select a book from the list box and press 'Next>'. Since this is the default layout, all the other stages will have been done already, so press 'Finish'. The layout wizard will only let you press the 'Finish' button when it is safe to do so.

You will now be asked if you want to save the layout. If you had a layout that was going to be used again, you would answer 'Yes'. Since we have used the default layout, there is no need to save it. The data is now transferred to *Excel*.



Now that the link is established, you can make changes to the “**octave1**” data in an **nVision** module, and the changes will be reflected in the “Sheet1” sheet in *Excel*.



---

# Report Writing Tour

## Overview

This tour will demonstrate how to integrate the results of analysis and measurement in **nVision** with reports written in *Microsoft Word* (or similar word processor).

## Introduction

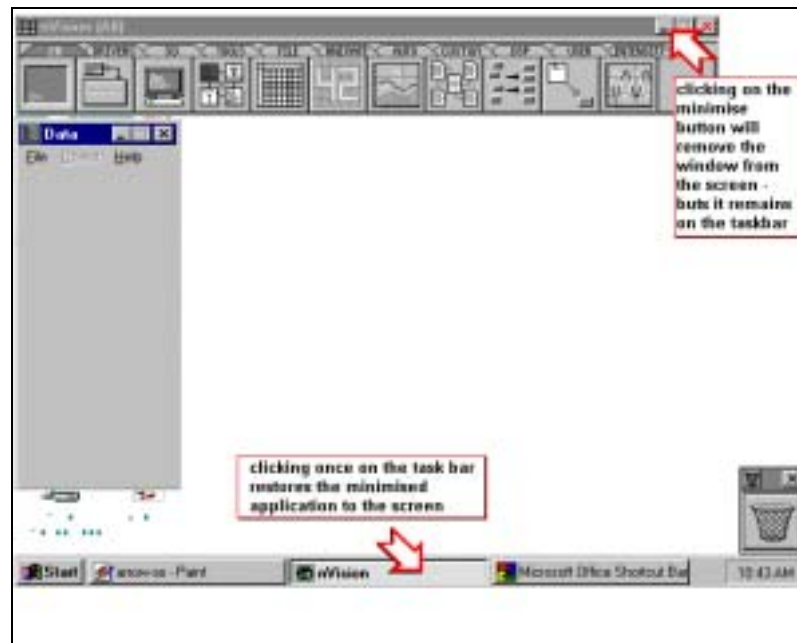
Results, tables and graphs from **nVision** can be easily incorporated into word processor documents enabling reports to be written as the measurement and analysis proceeds. This significantly shortens the time to complete the report, and minimises the opportunities for mistakes that can occur when reports are written after the event.

This tour focuses on the use of *Microsoft Word*; however, any *Windows 95* word processor would enable these techniques to be used.

## Using nVision with other Applications

**nVision** results can easily be transferred to a word processor such as *Word* using cut and paste techniques. While you are acquiring or analysing data in **nVision**, you can keep your word processor program open, ready to accept results and make conclusions as you work. However to keep the screen from getting too cluttered it's a good idea to minimise windows that are not in use. These applications can easily be restored from the taskbar.

*Using the taskbar to move between nVision and other applications*



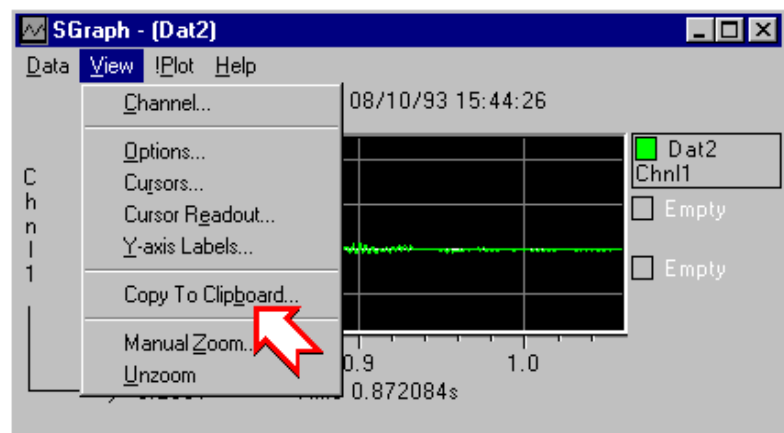
## Incorporating graphs

*Copying graphs to the clipboard*

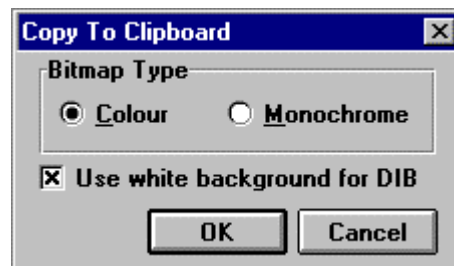
Graphs displayed in the modules from the Base 1 and Base 2 toolbars can be copied to the *Windows* clipboard, and then pasted into other *Windows* applications.

In the **nVision** window, click on 'View', 'Copy to Clipboard...' to copy the window. A dialog box will give you some options on how to manipulate the image that is copied.

Once on the clipboard, you can paste the image into your document using *Word*'s 'Edit', 'Paste Special...' menu options.



*Copy to clipboard dialog*



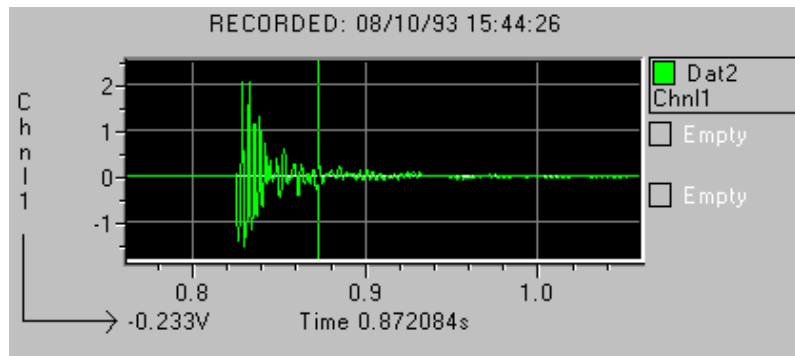
The copy allows a number of image-processing options. These are designed to ensure that the graphs are legible on paper. Remember that colours that look good on the screen may not look good on paper. For example:

When printed on a black and white printer dark backgrounds may obscure the data.

Light coloured text on dark backgrounds may not show up well on a colour printer output.

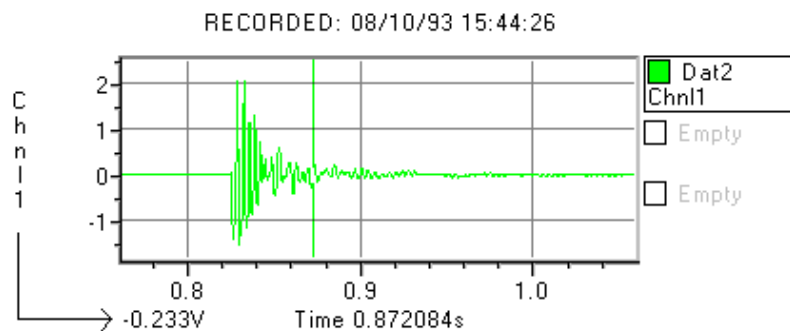
*Colour*

Selecting colour output will reproduce the window just as it is on screen, without the borders or menus:



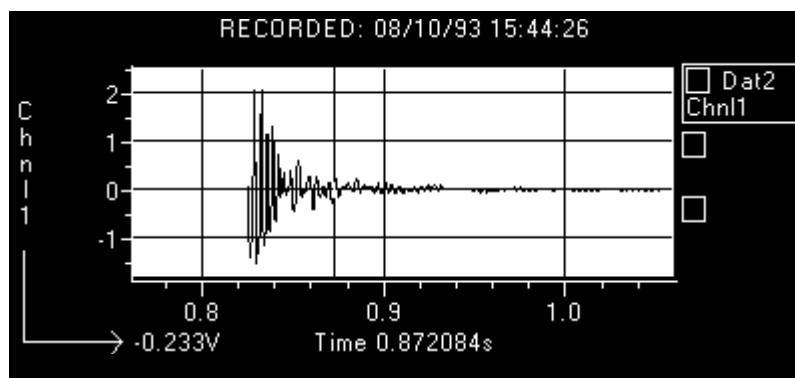
*Colour, Use white background for DIB*

Clicking on 'Use white background for DIB' will make the background white. This will normally emphasise the data and text on printed output.



*Monochrome*

The monochrome option will convert the clipboard to black and white only.



(Tip - remember Paint can swap black and white areas if you prefer.)

## Tips

Remember you can easily manipulate the graphs either in *Word* or by using *Paint* to:-

- change colours and backgrounds to highlight data,
- add distinctive arrows, boxes, or clipart to accentuate data features,
- delete unwanted text, and
- make your report more readable - and more fun

Remember that ANY Window can be copied to the clipboard:

Use the **Print Screen** key to copy the entire screen to the clipboard.

Use **Alt-Print Screen** keys together to copy the current, active window.

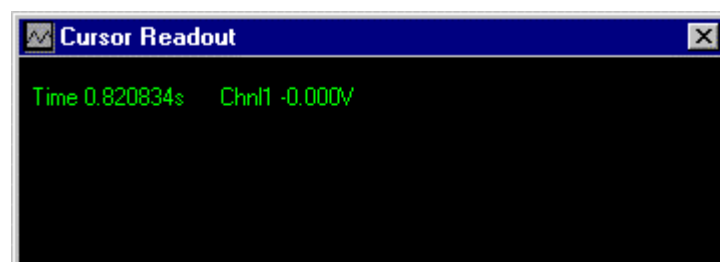
## Highlighting Points of Interest on Graphs

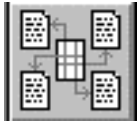
*Highlight points of interest in nVision data*

Whatever cursors are currently in the **nVision** graph displays will be copied to the clipboard too. Note that the copy will copy ANY window that overlaps the window border - so if you have a cursor readout window onscreen it will be copied provided it is within the borders of the copied window.



Of course, the cursor readout box could also be copied separately by clicking on it to make it the active window then using the keyboard command **Alt-Print Screen** to copy it to the clipboard.





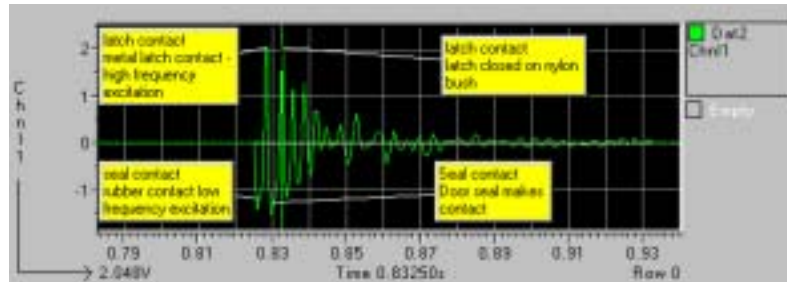
Using *nVision* notes fields to annotate data features.

## Using Notes Fields

The **nVision** Notes command allows text descriptions of data features to be stored with the data object itself. The descriptions can be tied to data samples using the cursor, or a range of samples.

Notes can be displayed in the graph display (select menu option 'View', 'Display', 'Notes') - and when displayed can be copied to the clipboard to include in reports.

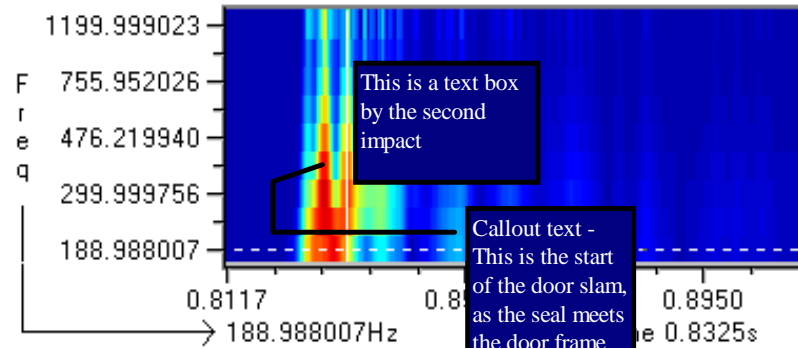
The text boxes can be repositioned and resized by the user, and since the labels are stored with the data, they appear whenever the data is displayed.



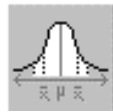
Using *Word* to annotate data

Of course, once an **nVision** display has been copied to the clipboard you can use *Word*, *Paint* or any *Windows* application to add text, markers and highlight important data features.

Double clicking on the picture in *Word* will allow you to edit it. This includes the ability to add lines and arrows, text boxes and "callout" - text boxes joined to features of the picture by lines.



*Including statistical analysis results*



## Including statistical analysis

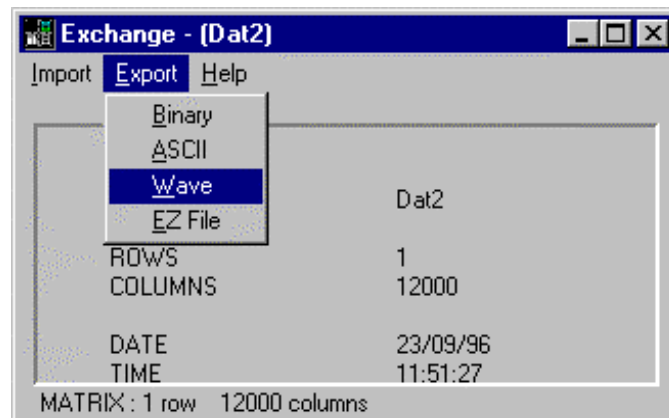
The STATISTICS module outputs tables of statistical analysis of the data, and includes a write to clipboard option in its menu for easy pasting of results into reports.

## Embedding Sounds

If the recipients of your report have sound cards in their PCs, then you can include sound clips as objects in your report. Then the readers of your report can hear your conclusions for themselves. Sound playback can be a useful way of distinguishing the features of any time signal within the audio frequency range, not just sound recordings - for example acceleration data.

*Creating a WAVE file using nVision EXCHANGE*

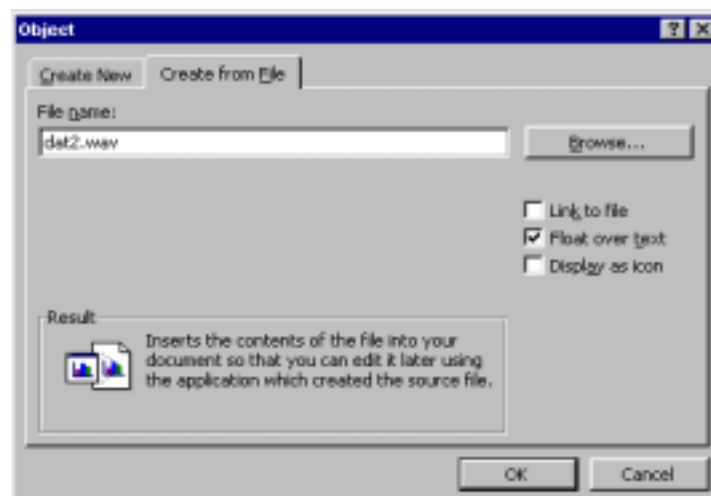
Any signal or matrix can be exported into a WAVE format sound file using the EXCHANGE program.



*Embedding a WAVE file object into a Word document*

In *Word*, the sound file can be embedded as an object. From the *Word* menu, select 'Insert', 'Object', and select the 'Create from File' tab in the dialog box. Entering the filename of the wave file created in **nVision** will embed it in your document, represented by an icon.

Double clicking on the icon will cause the sound to be replayed.



# Appendix A

---

## Hardware Key

### Overview

The “Hardware Key”, sometimes referred to as a “dongle” is a small device that plugs into the computer’s parallel port, optionally the USB port. Its purpose is to determine the validity of the user’s licence, and to restrict access to the software modules according to the user’s particular licence.

Without a hardware key, **nVision** will run in demonstration mode where some of the system facilities will be disabled or completely unavailable.

The hardware key also contains customer name information, which appears on the system ‘About’ dialog box, and details of the particular software modules with which it will work.

The hardware key will need a driver programme to be installed on your PC this programme is operating system dependant and can be found on the installation CD under the Rainport directory, please read the notes provided in this directory.

### Installation

#### Precautions



WHEN CHANGING CONNECTIONS AND CABLING, TURN OFF POWER TO THE COMPUTER AND OTHER EXTERNAL DEVICES. FAILURE TO OBSERVE THIS SIMPLE PRECAUTION COULD RESULT IN PERMANENT DAMAGE TO THE EQUIPMENT

#### *Connecting the key to a PC*

To connect the hardware key to your PC, you should follow these steps:

- Switch off the power to the computer and all peripheral devices such as printers and plotters.
- Locate the parallel printer port, LPT1 (a 25 pin D-Type connector). If in doubt, identify this port connection with the help of the hardware manual supplied with your computer.
- Connect the hardware key to this connector. There are arrows engraved on the case of the key that indicate which end goes into the computer.

- If you previously had a peripheral connected to LPT1 then connect this to the socket on the available end of the hardware key. (The key is a transparent device, so does not affect peripheral operation.)

## Upgrading your Hardware Key

### *What is a field upgrade?*

The hardware key can be upgraded to enable you to access other elements of the **nVision** system, without needing to return the key for modification. To do this you need to contact your **nVision** agent who can supply you with an upgrade code, usually by email or fax.

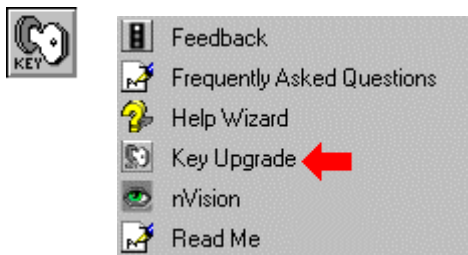
### *When can I use a field upgrade?*

If you simply want to release additional features in an existing installation, then all you need is an upgrade code. If you want to use new features, which have been added to the system in a later release of the software, you will first have to obtain and install a new set of software. Your new set of software disks will be supplied with a new code for your key.

### *How to use an upgrade code*

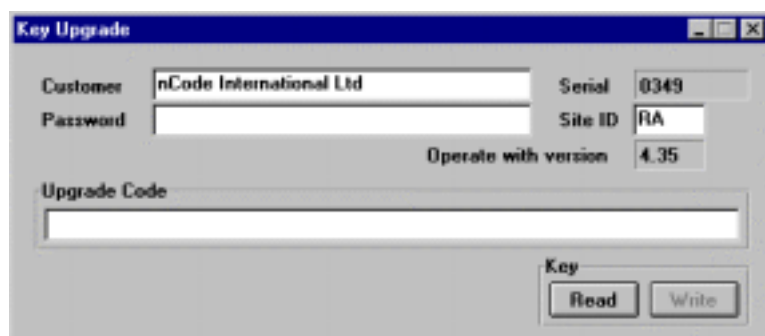
Once you have received your upgrade code, it is a simple task to go about upgrading the key. To upgrade the key you need to use a program called “Keyuser” which will have been installed in your program manager group or program folder for **nVision**.

*Keyuser Key Upgrade Icon*



To start Keyuser, select its menu entry in the **nVision** group from the ‘Start’ menu ‘Programs’ option.

*The Key Upgrade Program*



If the hardware key is NOT connected correctly, the Keyuser program will give an “Error opening key” message on start-up or pressing ‘Read’.

## Upgrade procedure step by step

To upgrade your key, follow these steps:

- Make sure the hardware key you are going to upgrade is connected to your machine.
- Run the Keyuser program. Click on the 'Read' button to read in the details from the key if it hasn't already done this automatically.
- If the details in the 'Customer' text box are not correct or you want to change them then type in either your own or your company's details (maximum 35 characters).
- If the 'Site ID' text box details are not correct or need changing then type in the site ID for the key (maximum 2 characters).
- Finally and most importantly you need to type in the upgrade code into the 'Upgrade Code' text box. The code is a 56-character string consisting of the numbers '0' to '9' and the letters 'A' to 'F'. On the upgrade code sheet it is broken up into 14 blocks of 4 characters separated by dashes for ease of reading. When you enter the code, you must type it in *without* the dashes.

When you have typed all 56 characters correctly, the 'Write' command button should become enabled, the 'Write' button label changing from light grey to black. When this occurs simply click on the 'Write' button to write the new details to the hardware key. If the 'Write' button does *not* become enabled then you should check that the code has been typed in correctly and is actually for the key number displayed in the 'Serial' box. If none of this enables the 'Write' button then there is a problem with the upgrade code and you should contact your **nVision** agent for a new code.



# Appendix B

---

## Additional Sample Data

### Do I need the additional data?

This step is optional, since all of the sample data required to follow the more basic tutorials is present on the standard distribution disks. However, once the tutorials have been completed, you may wish to explore the system in detail, using a more varied set of data.

### Installation overview

There is no installation for the **nVision** sample data, since the files can be loaded as required from the **nVision** CD. Windows Explorer can be used to copy any or all of the sample data to your hard drive from the 'DEMODATA' and 'MOREDATA' folders.



---

# Index

---

## *A*

Animate · 92  
ASCII file · 105, 106

---

## *B*

Bay · 38, 47, 48, 116  
Binary file · 105  
Box · 87

---

## *C*

Calculations · 47  
Client-Server · 39  
coherence · 101  
Contour level · 41  
Cursor · 35, 36, 38, 44, 52

---

## *D*

Data · 5, 1, 2, 21, 23, 25, 27, 28, 29, 31, 36, 37, 47, 48, 61, 62, 85, 92, 94, 96, 99, 100, 113, 129  
  displaying · 37, 41  
Data files · 36  
data objects · 19, 21, 23, 26, 27, 28, 29, 30, 37, 38, 52, 61  
Data window · 31, 36, 47, 48, 62  
DDE · 113  
demonstration · 10, 112, 125  
Dialog box · 7, 35  
Dongle · 125  
Drag and drop · 36  
Driver · 36, 37, 103  
DSP · 29, 93, 94, 95, 96, 97, 98, 99, 101  
Dynamic Data Exchange · 113

---

## *E*

Editing · 40, 52, 62, 107

Editor · 40, 52, 107  
Embedding Sounds · 124  
Excel · 113, 114, 115, 116, 117  
EXCHANGE · 103, 105, 113, 124  
Executing Modules · 24  
Export · 116

---

## *F*

Face · 90, 92  
File · 6, 7, 26, 30, 35, 36, 61, 85, 92  
File Import · 103  
Film · 23, 61, 62  
Filter · 93, 97, 98, 99, 100  
Functional demo · 112

---

## *G*

Graph · 28, 29

---

## *H*

Hardware · 5, 125, 126  
Hardware Key · 125, 126  
Help · 17, 18, 29, 103

---

## *I*

Igraph · 89, 90, 91  
Import · 103, 105  
Installation · 5, 125, 129  
Intensity · 23, 29, 62, 83, 84, 85, 92

---

## *L*

Layouts · 116  
log magnitude format · 78

---

## *M*

Manual Zoom · 43  
Map · 92  
Modules · 24

---

## *N*

Notes · 123

---

## *O*

object · 2, 15, 23, 25, 26, 27, 28, 31, 35, 36, 37, 38, 41, 44, 48, 61, 84, 85, 92, 93, 94, 96, 99, 100, 101, 107, 109, 111

---

---

## *P*

Paint · 122  
Plot · 37  
Power Spectrum · 95, 96, 101  
Previous Workspace · 30

---

## *R*

Rank · 88, 90, 91  
Regions · 31  
Rolling demo · 112  
RPM profile · 67, 68

---

## *S*

Save Custom Layout · 29  
Saving · 26  
Server · 20, 39  
Signal · 23, 93, 94, 95, 99, 101  
Software · 6  
Spectra · 101  
Splice · 62

---

## *T*

Toolbars · 22, 24  
Tools · 29, 47  
Trashcan · 21, 27, 51  
Tree · 87

---

## *U*

Universal file · 103, 104  
Upgrading · 126

---

## *W*

WAVE format sound file · 124  
Windows · 1, 3, 5, 6, 10, 11, 12, 13, 14, 15, 16, 17, 18, 20, 21, 29, 36, 98  
WORD · 119, 123, 124  
wordprocessor · 119  
Workspace · 29  
Workspace/Backdrop · 30  
Workspace/Hide Windows · 30  
Workspace/Load... · 30  
Workspace/Save... · 30

---

## *X*

XLDriver · 113

---

## *Z*

Zoom · 23, 38, 43, 52, 95

