Customer	:	**ESTEC**	**Document Ref**	:	**NW1.013.SUM.001**
Contract No	:	**11891/96/NL/GS CCN-13**	**Issue Date**	:	**21 August, 2002**
WP No	:	**WP3**	**Issue**	:	**2.0.7**

Title : **EnviView User Guide**
Version 2.0.7

Abstract : This document provides a comprehensive user guide to the EnviView tool produced for handling of Envisat Data.

Author :
Guy Brooker, Craig Ferguson,
Phil Goldsmith,
Marcus de Deus Silva
Lalit Jalota

Approval :
Guy Brooker
Senior Consultant

Accepted :
Pete Collings
Quality Engineer

Distribution :
Hard Copy File:
Network ID:

© The Copyright of this document is the property of VEGA Group PLC. It is supplied on the express terms that it be treated as confidential, and may not be copied, or disclosed, to any third party, except as defined in the contract, or unless authorised by VEGA in writing.

VEGA Group PLC
2 Falcon Way, Shire Park, Welwyn Garden City, Herts, AL7 1TW, UK
Tel: +44 (0)1707 391999
Fax: +44 (0)1707 393909
http://www.vega.co.uk

TABLE OF CONTENTS

1. INTRODUCTION 7
1.1 Purpose and Scope 7
1.2 Structure of the Document 7
1.3 Referenced Documents 8
1.4 Definitions of Terms 8
1.5 List of Acronyms Used in this Document 9

2. THE ENVIVIEW APPLICATION 10
2.1 Capabilities of EnviView 10
2.2 Limitations of EnviView 11
2.3 Envisat / PDS Data format 11
2.4 User Feedback & Bug Reports 12

3. GETTING STARTED QUICKLY WITH ENVIVIEW 13
3.1 How do I start EnviView? 13
3.1.1 Windows or Macintosh platform 13
3.1.2 Unix platform 13
3.1.3 Any java platforms : 13
3.2 Is there on-line help available? 14
3.3 What happens when EnviView starts? 14
3.4 How do I locate and load data files? 15
3.5 How do I examine a product's structure? 16
3.6 How do I find and open a data set? 17
3.7 How do I examine the fields in a data set? 18
3.8 How do I compare the data in different records? 18
3.8.1 Comparisons by Graphs 18
3.8.2 Comparisons in Tables 19
3.9 How do I view the raw data in a record or field? 19
3.10 How do I plot a graph? 20
3.11 How do I view an image? 21
3.12 How do I view Geolocation data? 21
3.13 How do I view Radar Altimeter waveform data? 22
3.14 How do I subset and calibrate Sciamachy level 1B products? 23
3.15 How do I export Envisat PDS data to HDF? 23
3.16 How do I see two views simultaneously? 23
3.17 Can I use EnviView functions from the command line? 24
3.18 How do I process multiple commands in a batch job? 24
3.19 How do I set preferences in EnviView? 25
3.20 How do I customise the appearance of EnviView? 26

4. DESCRIPTION OF THE USER INTERFACE 27
4.1 Views and Windows 27
4.2 Tabs 27
4.3 Menus 27
4.3.1 The File menu. 28
4.3.2 The *Edit* menu. 28
4.3.3 The *View* menu 29
4.3.4 The *Plot* menu 30
4.3.5 The *Window* menu 30
4.3.6 The *Help* menu 31

5. VIEWING DATA IN TABLES 32
5.1 Tabular View features 33
5.2 Creating a new view 34
5.3 Switching between Views 35

5.3.1 Changing the data set within the same view. 35
5.3.2 Closing a view 36
5.3.3 Using Multiple windows 36
5.4 HTML View 36
5.5 Hex Dump View 37
5.6 ASCII View 38

6. VIEWING DATA GRAPHICALLY 39
6.1 Introduction 39
6.2 What is the Simple Graph Viewer 39
6.2.1 Quick Start 39
6.2.2 Feature Summary 40
6.2.3 Array vs Scalar fields. 41
6.2.4 Selecting Data to Plot 41
6.2.5 Changing the display of graphs 43
6.2.6 Viewing Array Field data 44
6.3 Extended Data Extraction and Plotting 46
6.3.1 What is Extended Data Extraction 46
6.3.2 The Graphing Components 46
6.3.3 Opening the Plot Document 46
6.3.4 The Add Data Dialog 49
6.3.5 The Modify Graph Data Dialog 50
6.3.6 The Add Variable Dialog 51
6.3.7 How to plot a and extract the data for a graph 55
6.3.8 Configuring each graph type 56
6.4 Saving Graph configurations 61
6.4.1 Loading a Saved Graph 61
6.4.2 Deleting a Saved Graph 61

7. VIEWING ENVISAT IMAGE DATA 63
7.1 The Image Viewer 64
7.2 Moving around the Image 65
7.3 The Colour Bar 67
7.4 Magnifying the image display (zoom) 68
7.5 Geographical Co-ordinates 68
7.6 Pixel quantities 68
7.7 Overlaying Images 68
7.8 Image Statistics 69

8. VIEWING WAVE SPECTRA 71
8.1 Feature Summary 72
8.2 Selecting a data set to view 73
8.3 Selecting Records 73
8.4 Playing a movie 74
8.4.1 Rescaling behaviour 74
8.5 Changing the spectra display 74
8.6 Getting specific bin information 75
8.7 ASAR Wave Spectra display types & characteristics 76
8.7.1 Level 1B Spectra symmetries 76
8.7.2 List of level 1B spectra types 77
8.8 The ASAR Wave spectra display characteristics 78
8.8.1 Level 1B spectra characteristics 78
8.8.2 Level 1B Spectra Orientation 79
8.8.3 Level 2 spectra characteristics 79
8.8.4 Level 2 Spectra Orientation 80

9. VIEWING 2-D MAP DATA 81
9.1 Quick Start 81

9.1.1 Differences between an image view and a 2-D map view....81
9.2 Feature Summary....82
9.3 How EnviView selects Map View Data....83
9.4 Selecting a data set to view....84
9.5 Selecting records to view....84
9.6 Choosing a value to plot....84
9.7 Changing the scale of the map....85

10. THE ARRAY MOVIE VIEWER....87
10.1 Rules for viewing Arrays....88
10.2 Array Movie Features....89
10.2.1 Selecting a Data set and starting an Array Movie View....89
10.2.2 Selecting the fields & records to be viewed....89
10.2.3 Changing the speed of a Movie....89
10.2.4 Changing the look of the data displayed....90
10.2.5 Y axis automatic rescaling behaviour....91

11. SAVING AND EXPORTING ENVISAT DATA....93
11.1 Saving a Selected Region of an Image....93
11.2 Exporting Data in HDF....94
11.2.1 Organisation of Envisat data in HDF format....94
11.2.2 Selecting data for export....95
11.2.3 Selecting a Subset of Data for Export....96
11.2.4 Shortcuts for Exporting the Data to HDF....97
11.3 Creating a Sciamachy Calibrated (Level 1C) Product....98
11.3.1 Creating a Level 1C file....98
11.4 Saving a Data Set Record in HTML....101
11.5 Saving Individual Data Fields....101
11.6 Exporting to Printable Files....101

12. DECODING LEVEL 0 DATA....102
12.1 Introduction to the scripting language....102
12.2 Variable Properties....103
12.2.1 Name Property....103
12.2.2 Bitoffset Property....103
12.2.3 Size Property....103
12.2.4 Type Property....104
12.2.5 ElementCount Property....104
12.2.6 Units Property....105
12.2.7 Description Property....105
12.3 Statements....105
12.3.1 Basic Statement....105
12.3.2 If Statement....105
12.3.3 If Else Statement....106
12.3.4 Math Statement....107
12.4 User Level 0 Scripts....107
12.4.1 File naming conventions....107
12.4.2 Where to save script file....107

13. GETTING HELP IN ENVIVIEW....102
13.1 Documentation....109
13.2 On-line help....109

14. COMMAND LINE TOOLS AND ENVIVIEW....110
14.1 Maximising performance....110
14.2 Batch files....110
14.3 Tips for using the command line options....111
14.3.1 Arrow keys on command line under DOS and Windows....111

14.3.2 Arrow keys on command line under Unix. 111
14.3.3 Quoting arguments containing spaces. 111
14.4 Pds2Ascii 112
14.5 Pds2Hdf 115
14.6 PdsPixels 117
14.7 PdsPixelTrack 120
14.8 PdsSubset 123
14.9 SciaL1C 126

APPENDIX A INSTALLING ENVIVIEW 130

APPENDIX B .PROCESSING ENVIVIEW DATA WITH IDL 131
B.1 Introduction to examples 131
B.2 Load the utilities into IDL 131
B.3 Listing the datasets in an Envisat HDF file 132
B.4 Listing the contents of an Envisat HDF data set 132
B.5 Reading data from an SDS 133
B.6 Reading data from a Vdata 134
B.7 EnvisatHdf.pro 135

AMENDMENT POLICY

This document shall be amended by releasing a new edition of the document in its entirety. The Amendment Record Sheet below records the history and issue status of this document.

AMENDMENT RECORD SHEET

ISSUE	DATE	DCI No	REASON
2.0b1	26.09.2000	N/A	Initial Issue for EnviView 2.0 beta 1
2.0b2	12.04.2001	N/A	Update for EnviView 2.0 beta 2
2.0.1	14.02.2002	N/A	Launch release – EnviView 2.0.1
2.0.7	21.08.2002	N/A	Commissioning Review release – EnviView 2.0.7

1. INTRODUCTION

1.1 Purpose and Scope

This document provides a detailed guide to using the EnviView tool for viewing data from the Envisat mission. It explains how Envisat data can be extracted, decoded, and displayed using various visual representations, including images where appropriate, and exported in a variety of formats.

1.2 Structure of the Document

After this introduction, the document is divided into a number of major sections that are briefly described below:

- **Chapter 2 – The EnviView Application.** The EnviView tool is introduced and this chapter provides a brief introduction to the capabilities of EnviView.

- **Chapter 3 – Getting Started Quickly with EnviView.** This chapter provides a quick start introduction to using EnviView, giving summaries of how to perform typical tasks, by way of simple step-by-step examples. The following chapters describe the tool and its capabilities and use in more detail.

- **Chapter 4 – Description of the User Interface.** The basic design philosophy is explained here and a description of the user interface and concepts. This chapter explains how data is loaded, how new windows on the data are opened and how the data can be modified and saved.

- **Chapter 5 – Viewing Data in Tables.** Envisat data can be viewed in various forms of tables. This section discusses how to use the different capabilities in EnviView for examining the data in ASCII, Hex, or HTML formats.
- **Chapter 6 – Viewing Data Graphically.** This chapter describes how EnviView can be used to display data in various type of graph. It describes how to select data of interest, assign data to the X- and Y-axes, and customise the display properties.
- **Chapter 7 – Viewing Envisat Image Data**. This chapter describes how EnviView can be used to view, manipulate, and save image data taken by Envisat.
- **Chapter 8 – Viewing Wave Spectra.** This chapter describes how EnviView can be used to view ASAR wave spectra data.
- **Chapter 9 – Viewing 2-D Map data.** This chapter describes how EnviView can be used to view geo-location information.
- **Chapter 10 – The Array Movie Viewer.** This chapter describes a viewer used to display a selectable number of multiple records in a single plot, which can be used to easily compare data and track the development of features in subsequent records. With this tool it also possible to display a sequence of records that are updated continuously so that the plots are *animated*

- **Chapter 11 – Saving and Exporting Envisat data.** This chapter describes the functionality in EnviView for extracting and saving data to different file formats.

- **Chapter 12** – **Decoding Level 0 Data**. This chapter describes how to configure EnviView to read information from Level 0 source packets.

- **Chapter 13** – **Getting help on EnviView**. This chapter describes the help available in EnviView, both on- and off-line.

-

- **Chapter 14 – Command Line Tools and EnviView.** This chapter describes command line utilities for exporting data to various files formats and for calibrating Sciamachy data.

- **Appendix A – Installing EnviView.** This appendix points to instructions for installing EnviView.

- **Appendix B – Processing EnviView Data with IDL.** This appendix provides examples of analysing and further processing of Envisat data in HDF format by the commonly used commercial product IDL, or its derivative ENVI.

1.3 Referenced Documents

The following is a list of documents with a direct bearing on the content of this report. Where referenced in the text, these are identified RD.n, where 'n' is the number in the list below:

RD.1 ENVISAT : Mission & System Summary, ESA

RD.2 Envisat-1 Product Specifications, ESA-GS-2009.

RD.3 Envisat Payload to Ground Segment Interface Control Document (PGICD), PO-ID-DOR-SY-0032

1.4 Definitions of Terms

The following terms have been used in this report with the meanings shown.

Data Set	A collection of data set records in an Envisat product.
Data Set Record	A collection of data fields of certain sizes and data types.
Dialog	A window that displays information or presents options to the user.
Focus	The destination of keyboard input.
Java Runtime Environment	The software required to run a Java application
Product	An Envisat data file, in PDS format
View	A manner of visualising data. e.g a Graph View or an Image View.

Table 1.1 List of Terms Used in this Document

1.5 List of Acronyms Used in this Document

The following acronyms have been used in this document.

AATSR	Advanced Along Track Scanning Radiometer
ASAR	Advanced Synthetic Aperture Radiometer
ASCII	American Standard Code for Information Interchange
ADS	Annotation Data Set (time stamped processing data)
COTS	Commercial Off The Shelf Software
DSD	Data Set Descriptor
DDT	Alcatel Product Data Dictionary Tool
EOS	Earth Observation System (NASA)
ESA	European Space Agency
GADS	Global Annotation Data Set (not time stamped)
GIF	Graphics Interchange Format
HDF	Hierarchical Data Format
HTML	Hyper-Text Mark-up Language (web page format)
IDL	Interactive Data Language
IEEE	Institute of Electronic and Electrical Engineers
JPEG	Joint Photographic Expert Group (image format)
JVM	Java Virtual Machine (also Java VM)
MERIS	Medium Resolution Imaging Spectrometer
MDS	Measurement Data Set (time stamped measurement data)
MPH	Main Product Header
PDS	Envisat Payload Data Segment (systems processing and archiving data)
PPM	An image format common on Unix
SPH	Specific Product Header
TIFF	Tagged Image File Format
VM	(Java) Virtual Machine (used to run java software. Also JVM)

Table 1.2 The List of Acronyms used in this document

2. THE ENVIVIEW APPLICATION

The Envisat satellite carries a package of instruments to observe the Earth and its atmosphere from space in a synergetic fashion, addressing crucial matters such as global warming, climate change, ozone depletion and ocean and ice monitoring. Detailed information about the mission is available at http://envisat.esa.int or RD.1.

Data acquired in orbit are transmitted to ESA ground stations, and undergo a number of stages of processing, resulting in various data products. These data products are written in a data format exclusive to the Envisat Ground Segment (Payload Data Segment - PDS).

EnviView is a tool capable of opening and decoding Envisat data. It then displays the contents as tables, graphs, or as images as appropriate.

2.1 Capabilities of EnviView

EnviView is a simple tool providing a *first look* at Envisat data files in the PDS format. It has a simple to use graphical user interface for data exploration and visualisation. It is intended for the following purposes:

- To be able to open any Level 0, 1b, Level 2 or auxiliary PDS data file, section 2.3.
- To determine and display the contents of a file and the value of any given fields in any data set.
- To provide basic visualisation of data through simple 2D & 3D graphs and images.
- To visualise certain high level data sets in a meaningful manner where a graph conveys little information, especially
    - image data sets from AATSR, ASAR and MERIS.
    - ASAR Wave spectra.
    - RA2 Waveforms
- To provide a representation of the geographic coverage on a rectangular map projection.
- To export selected data to HDF, ASCII or binary data files, for analysis in COTS tools such as Mathematica, IDL, Excel, *etc*.
- To calibrate Sciamachy level 1B data
- To save the contents of a record as HTML text for inclusion in reports.
- To provide support for printing and system clipboard (cut/paste) where possible.
- To be highly portable and to reach as wide an audience as possible.
- To be guaranteed to run on Unix, Windows and MacOS and potentially other platforms too.

2.2 Limitations of EnviView

EnviView is not intended for detailed analysis, visualisation, and processing of Envisat data, other commercial and proprietary tools provide these facilities in great detail and with many specialised options. The difference between such tools and EnviView is that they are either highly generalised, or aimed at specific classes of data, such as images or atmospheric data, or even data for a particular instrument.

EnviView, however, is a single solution for all Envisat products. EnviView is intended as a stepping stone between the proprietary data format used for Envisat data and other tools.

Widely used commercial packages include: -

- IDL & ENVI http://www.rsinc.com
- Matlab http://www.mathworks.com
- Mathematica http://www.wolfram.com/
- Noesys http://www.rsinc.com
- PV-WAVE http://www.vni.com

2.3 Envisat / PDS Data format

To understand Envisat data files, one needs to know a little information about the structure of files in the PDS format.

There are five classes of data files: -

- Level 0 products - raw data source packets from the satellite, wrapped in a PDS compatible file structure.
- Level 1b products - geo-located engineering products, in which data has been converted to engineering units. Auxiliary data has been separated from measurements and selected calibrations have been applied to the data.
- Level 2 products - geo-located geophysical products, in which Level 1b data has been transformed through higher-level processing into geophysical quantities.
- Browse products - low-resolution overviews of Level 1b and level 2 image products provided for catalogue browsing.
- Auxiliary products - external data used at various stages of the ground processing, including instrument characterisation measurements, calibration settings, meteorological and orbit predictions.

All of these data products are written in the following structure: -

1. A Main Product Header (MPH)
2. A Specific Product Header (SPH) - including a number of Data Set Descriptors (DSD), a directory of the remaining data sets in the file.
3. Measurement Data Sets (MDS) – time tagged measurements
4. Annotation Data Sets (ADS) – time tagged processing parameters
5. Global Annotation Data Sets (GADS) – unrelated to measurement time

The format of the MPH is identical for all products. The SPH is specific to each product or group of related products. Both the MPH and SPH are written in plain ASCII text. The rest of the file is in binary format, except for a few products that also have ASCII ADS or MDS.

Each MDS, ADS or GADS consists of a sequence of Data Set Records, containing one or more fields of data. The format of these data sets is described in Product Specification Documents and is also available within EnviView from the Help menu.

A measurement data set contains data derived from instrument measurements, and related processing or instrument information. Every MDS has a time stamp indication the time of measurement. An ADS contains processing information related to measurements, such as Geolocation, or quality parameters, and is also time stamped with the time of related measurements. The GADS contain processing configuration data that apply to the entire product, and are not related to individual measurements, and hence have no time stamp.

2.4 User Feedback & Bug Reports

User feedback is essential for improving EnviView and comments and bug reports can be sent directly to the EnviView team via enviview@jw.estec.esa.nl, or via ESA EO Helpdesk eohelp@esa.int.

When making a bug report, **please include the following information**:

- From the **About EnviView** option in the **Help** menu:-
  - Operating System & Machine Type
  - Java version, vendor name and vendor specific
  - EnviView and data format version numbers
- Steps leading to problem
- Any text sent to the terminal

We would like to thank all those who were kind enough to send bug reports and feedback in the past. Every message helps to make the tool better for everyone in the future.

3. GETTING STARTED QUICKLY WITH ENVIVIEW

In the modern, fast-paced world of today, nobody, it seems, has time to *RTFM*[1]. The EnviView graphical user interface has been designed to be as intuitive and as easy to use as possible, but nevertheless some explanation of how to use it is necessary.

This chapter provides the basic information that is required to get started with EnviView and look into data files, or *products*, view the data therein in an appropriate manner, perform elementary processing of that data, and finally how to save the results. It is a *quick-start* guide and provides essentially tutorial style instructions to perform the tasks that are most commonly requested.

Each sub-section heading is in the form of a question, emphasising the quick start nature of the material in this chapter. These provide the basics for using EnviView in a productive and efficient manner. Users are then encouraged to explore and experiment with the possibilities provided by EnviView.

For a detailed understanding of the many features of EnviView, one can always *RTFM.*

3.1 How do I start EnviView?

The answer depends on the platform from where it is being run.

3.1.1 Windows or Macintosh platform

Start EnviView by double clicking on the application file icon

3.1.2 Unix platform

Start EnviView by running the shell script.

3.1.3 Any java platforms :

Start EnviView by executing the command :-

```
java -cp EnviView.jar envisat.EnviView
```

This requires that the Java 2 environment is already installed and set up correctly on the platform that EnviView is to be executed, so that the command java is recognised. The file EnviView.jar can be in any directory, but the path must be included in the above command. Alternatively, the environment variable for the classpath can be pre-defined.

For java 1.1, the syntax of the java command is slightly different, and one must also ensure that the swing library (version 1.1.1) is also present on the classpath.

```
java –classpath EnviView.jar:swingall.jar envisat.EnviView
```

Some java installations may use the 'jre' command to start a java application, instead of ‘java’. Please refer to the documentation with the java runtime.

---

[1] *RTFM* - politely defined as “Read The Fine Manual”. Other definitions also exist.

3.2 Is there on-line help available?

Yes, this manual is available from the help menu. Some user interface elements may also provide *tooltips* to explain their function. A tooltip is displayed when the mouse cursor is held above a component (such as a button or a text field), and provides information or instructions pertinent to that component and it's use.

3.3 What happens when EnviView starts?

When EnviView starts a new window is created. Until a data file is opened, it has the title *EnviView*, Figure 3.1

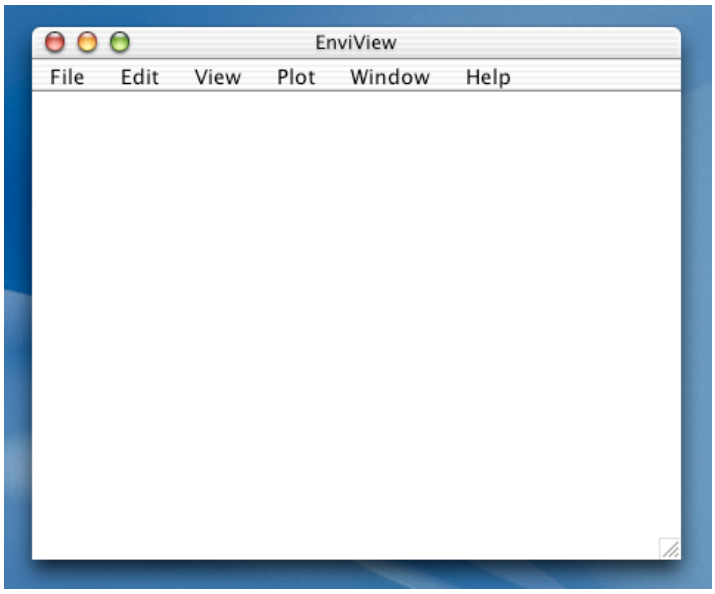


Figure 3.1 The opening window when EnviView starts

The window has menus but most of the options are disabled. At this stage it is possible to consult the on-line help, set user preferences (section 3.19), see the Envisat data formats (section 4.3.6) and of course *Open* and load a data file, Figure 3.2.

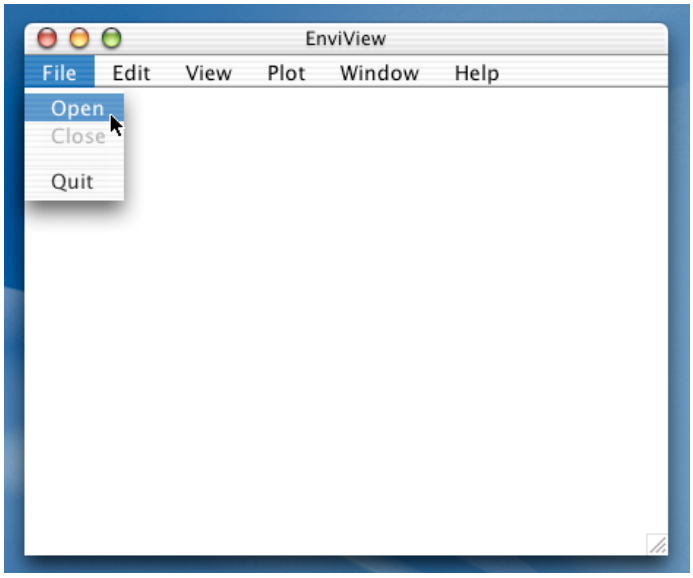


Figure 3.2 The opening window with the *open* option.

3.4 How do I locate and load data files?

A data file can be opened from any EnviView window, even if that window already has a data file loaded, through the *Open* option of the *File* menu (Figure 3.2 and section 4.3.1). Once selected, a *file chooser* is displayed, which allows the local directory structure to be navigated and the desired file found and selected.

Note that there is a bug in some versions of the Java environment where the default directory in the viewer does not necessarily correspond to the files listed which may be in a different directory.

When a file is selected – a new window is opened, with the same name as the data file, and more options in the menus are enabled, depending on the file type, e.g. Figure 3.3. Of most interest is the fact that it now possible to open *views* on the data i.e. inspect the data in different ways.

NOTE: for best performance, copy data files to the local hard disc before opening. Loading files from CD-ROM or from a network disk will result severely impair reaction time for displaying data.

3.5 How do I examine a product's structure?

When a file is opened, there is one view that is opened by default, namely the *File View*. This shows a list of all the data sets in a product. They are displayed in a *tree* or hierarchical layout showing the large-scale structure of a file and showing the headers (MPH and SPH) and what kind of data set it is, i.e. either measurement, annotation, or global annotation data sets, e.g. Figure 3.3.

One can examine the format if any Envisat product without opening a file, by selecting the *Product Formats* option, from the *Help* menu.

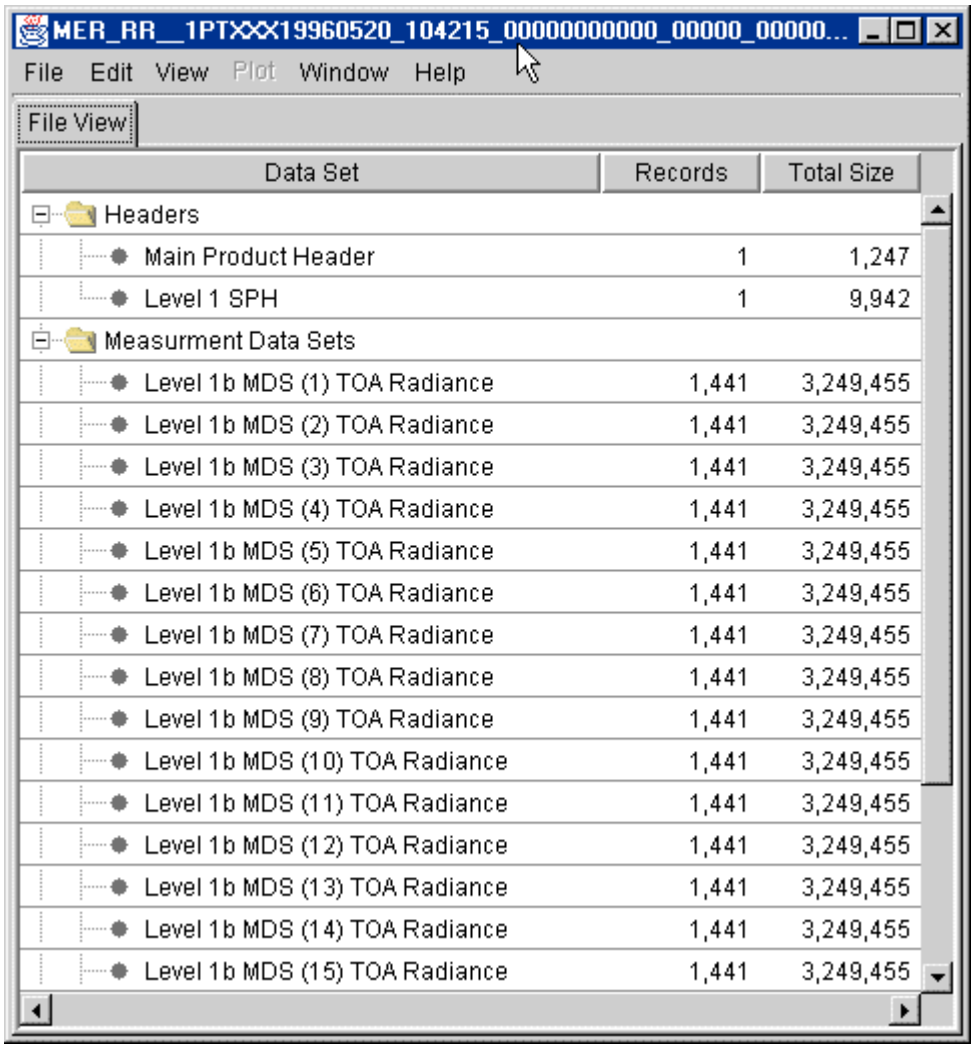


Figure 3.3 The *File View* in the EnviView window

The File View is a table with three columns that shows: -

- the data set description
- the number of records for a given data set
- and the total size of the data set in bytes

This view only shows the top-level elements of an Envisat file.

To see the contents of the individual fields and sub-fields in a data set, use the *Record View*. This is displayed by selecting the *New Record View* in the *View* menu (section 4.3.3). This provides a table containing the fields and their values of a single record in a data set. Each view has a named tab, at the top of the window. Views may be interchanged by clicking on the tabs. The data set displayed in a given record view may be changed through the use of a pop up list labelled *Data Set*. Any record in a data set may be displayed, by moving a slider, or typing the index of the record in a text box, see Figure 3.4.

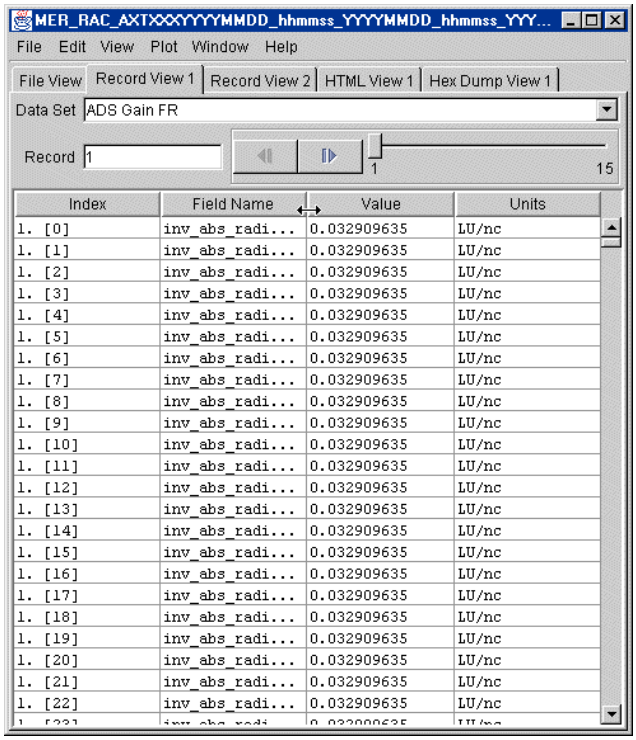


Figure 3.4 A Record View

3.6 How do I find and open a data set?

There are two ways, once a file has been loaded : -

- a *File View* is opened (Figure 3.3), which is a simple tree-like view of a products structure and it lists all the data sets in the file, including MPH and SPH. Once the desired data set has been located in the tree, it can be examined in more detail by double clicking on the data set name.

- Alternatively, the *New Record View* option from the *View* menu can be selected which creates a view of the MPH. This has a selection box labelled *Data Set* at the top of the window with the list of all the data sets in the file, from which the desired one can be selected and examined.

A more descriptive look at a records contents is provided by the *HTML View (*section 5.4). A description of the format of all Envisat data products may be browsed by selecting the *Envisat Data Formats* option from the *Help menu.*

3.7 How do I examine the fields in a data set?

First load the data file and select the data set, section 3.6. The result is a record view which displays the fields in the data set in a table of four columns : -

- the index column simply provides the index of the field,
- the second column gives the Field Name,
- the actual field value is in the third column,
- and the final column shows the units for the field, if there are any.

If the field name is highlighted in blue, holding the mouse pointer over the name will pop up a description of the field quantity. Likewise, if a field value is highlighted in blue, holding the mouse pointer over the value will pop up a window with an alternative representation of the values.

An optional fifth column, containing the raw bytes of a field in hexadecimal may also be displayed. This column is switched on or off from the *View* menu. Select *Show Raw Bytes Column* from the sub-menu *Record View Options*, to display the extra column. Select *Hide Raw Bytes Column* to return to the default set of columns.

The columns can be resized, simply position the cursor over the dividing line between the columns. When the cursor changes to a double-headed arrow, click and hold down a mouse button and adjust the column as desired. The column order can also be adjusted; click and hold down a mouse button over the column header, and drag the column over to the desired location.

The view will have a scroll bar at the right hand side to scroll through the data set enabling all the fields to be examined.

For fields which have multiple elements, both the field name and field index for each entry have an index appended to them enclosed in square brackets.

For fields with sub-fields, the name of the sub-field is appended to the field name separated by a full stop or decimal point and displayed in the Field Name column. The index of the sub-field is appended to the field index, again separated by a full stop, but note sub-field indices are given as alphabetical references not integers.

3.8 How do I compare the data in different records?

3.8.1 Comparisons by Graphs

The comparison depends on the type of data. For scalar fields, that is a single value per record, a plot of n records will show the trends in the data. For array fields, i.e. many points per record, a single record is displayed per plot. To look at subsequent records, EnviView allows the source record index to be updated without changing the field settings. To visually compare two records simultaneously, a second plot view can be opened (choose *New Simple Graph View* from the *View* menu). While this latter method has two plot views, by default, only one can be seen at a time. The solution is to open a new window on the data with the *New Window* option in the *Window* menu. One view may then be selected in one window, and the other view in the second window.

Alternatively, a new *Plot Window* may be opened via the *Plot* menu, and data from different records may be extracted an plotted on the same graph. See section 6 for full details.

3.8.2 Comparisons in Tables

First load the data set and open the *Record View*, Figure 3.4. Find the particular range of data of interest, then using the controls to change records, (see Figure 5.2) navigate through the records.

To visually compare two records views simultaneously, open a new window on the data with the *New Window* option in the *Window* menu, and a second record view from the *View* menu One view may then be selected in one window, and the other view in the second window.

3.9 How do I view the raw data in a record or field?

When working on I/O routines to access data files directly, or to understand how a data value is represented in an Envisat file, one may wish to view the raw bytes making up a field or a record.

To view the raw bytes of an individual field, the *Record View*, provides an optional fifth column, displaying the raw bytes of a field in hexadecimal. This column is switched on or off from the *View* menu. Select *Show Raw Bytes Column* from the sub-menu *Record View Options*, to display the extra column. Select *Hide Raw Bytes Column* to return to the default set of columns.

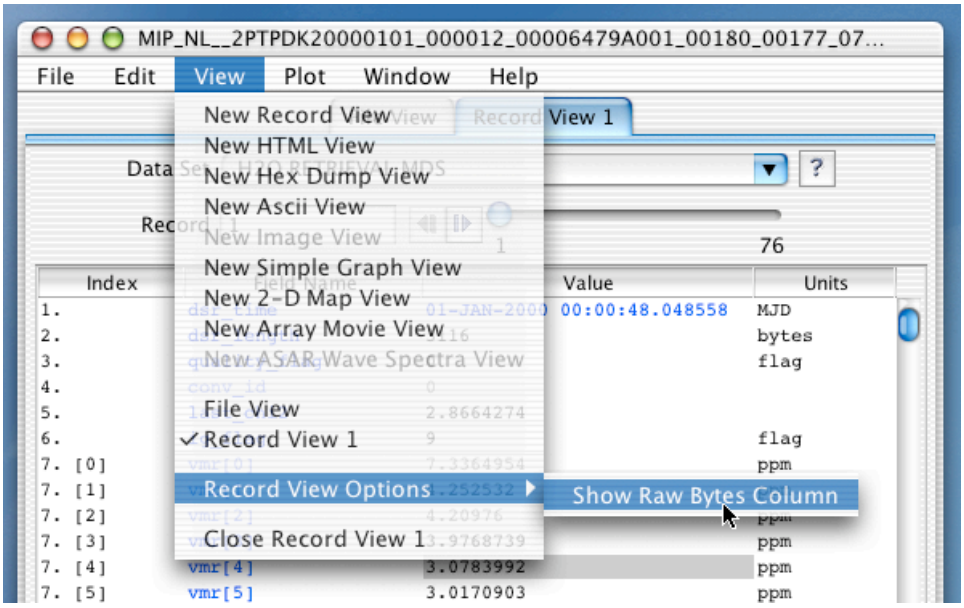


Figure 3.5 Showing Raw Bytes

The *Hex Dump View*, shown in Figure 3.6, displays the actual bytes making up the data in an entire record. It contains three columns. The first gives the byte offset, in hex, at the start of the row of data. The second column shows the hex value of each byte, with 16 bytes per row divided into four four-byte words. The third column gives the ASCII translation of the hex dump. Note that columns can be neither resized nor rearranged in the hex view.

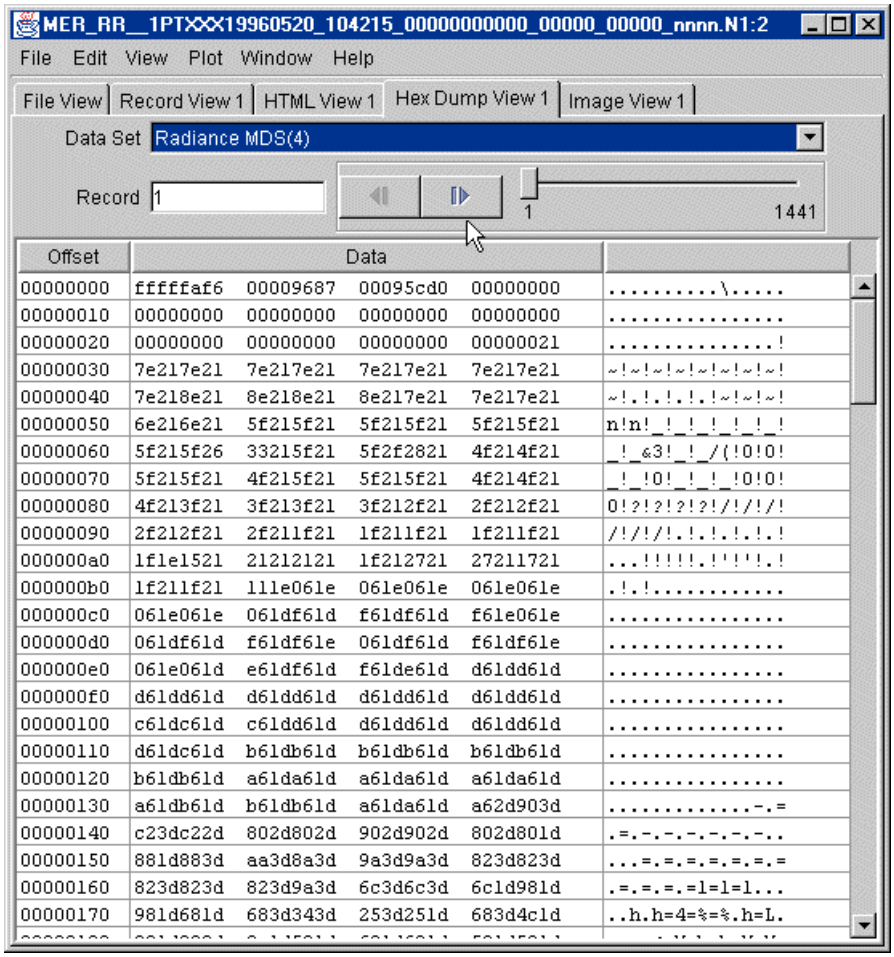


Figure 3.6 The Hex Dump View

3.10 How do I plot a graph?

See Chapter 6 for full details.

3.11 How do I view an image?

If an open product contains image data, a *New Image View* option is enabled in the *View* menu. An image view allows any quantity in any data set to be displayed, see Figure 3.7.

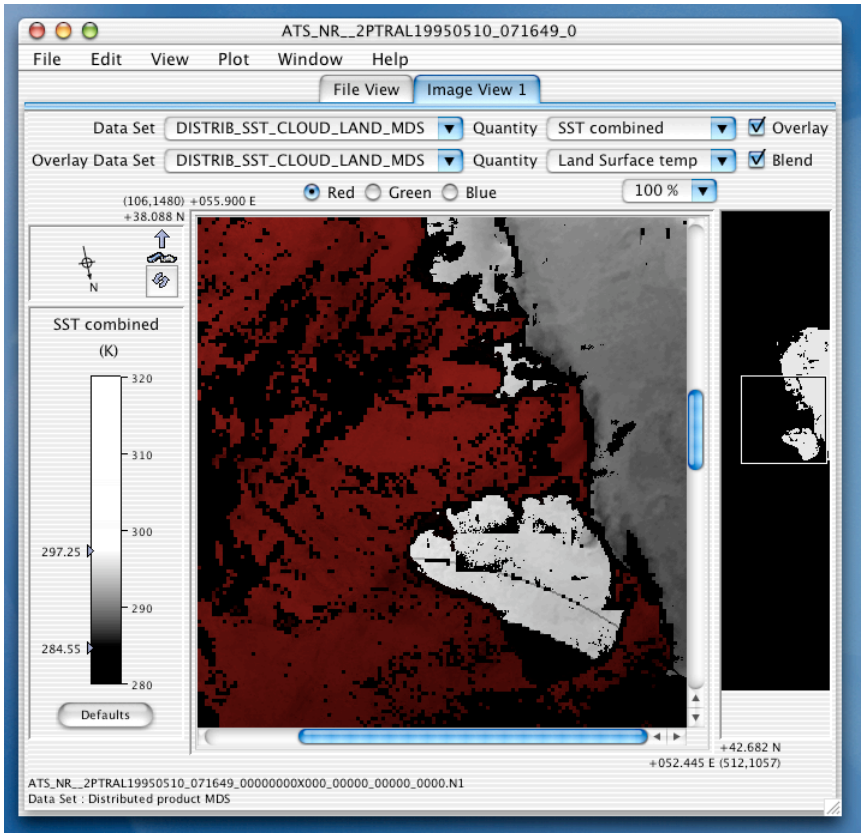


Figure 3.7 An example image view

If multiple quantities or data sets are present in a product, a second quantity may be overlaid on top the first for comparison. The colour scale may be changed to enhance the image's appearance. For more information, refer to Chapter 7.

The AATSR level 2 averaged surface temperature product does not contain image scan lines, and hence can not be displayed in the image viewer. It's data may be displayed using the 2-D map view (see Chapter 9 for details).

NOTE: for best performance, copy data files to the local hard disc before opening. Loading files from CD-ROM or from a network disk will result severely impair reaction time for displaying data.

3.12 How do I view Geolocation data?

If a file has a geolocation ADS, the option *New 2-D Map Viewer* will be enabled in the *View* menu. The map view displays a world map using a simple rectangular projection, see Figure 3.8. Any field in a data set containing geolocation information may be displayed on the map. The area displayed may be modified by clicking and dragging the mouse over the desired region, or modifying the axes ranges. Please see Chapter 9 for details about the functions in the viewer.

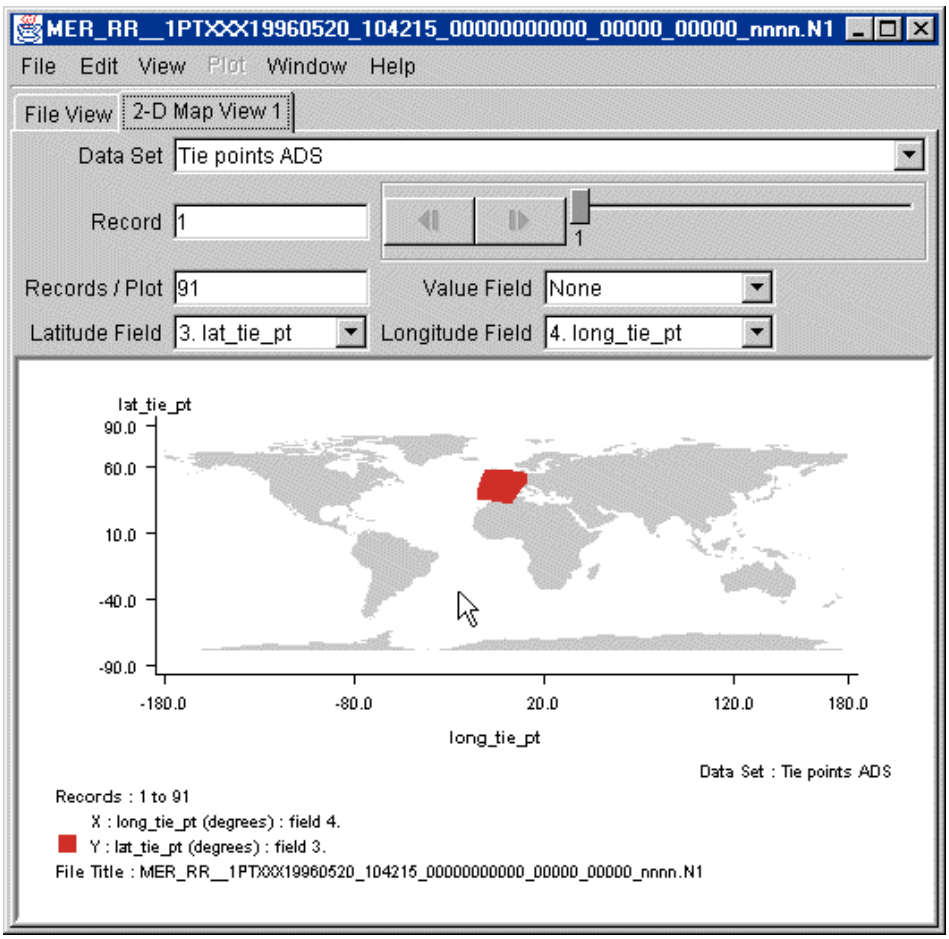


Figure 3.8 An example of a 2-D **map view**

3.13 How do I view Radar Altimeter waveform data?

From the *View* menu select the *New Array Movie Viewer* option. This option is available for any data sets with fixed length arrays. A selected array field (e.g. ave_ku_wvform_corr from the RA2_SCIENCE_LEVEL_1B data set) may be displayed from subsequent records, see Figure 3.9. Basic animation functions are provided for visualising the development of the waveform from record to record. See Chapter 10 for details about the view.

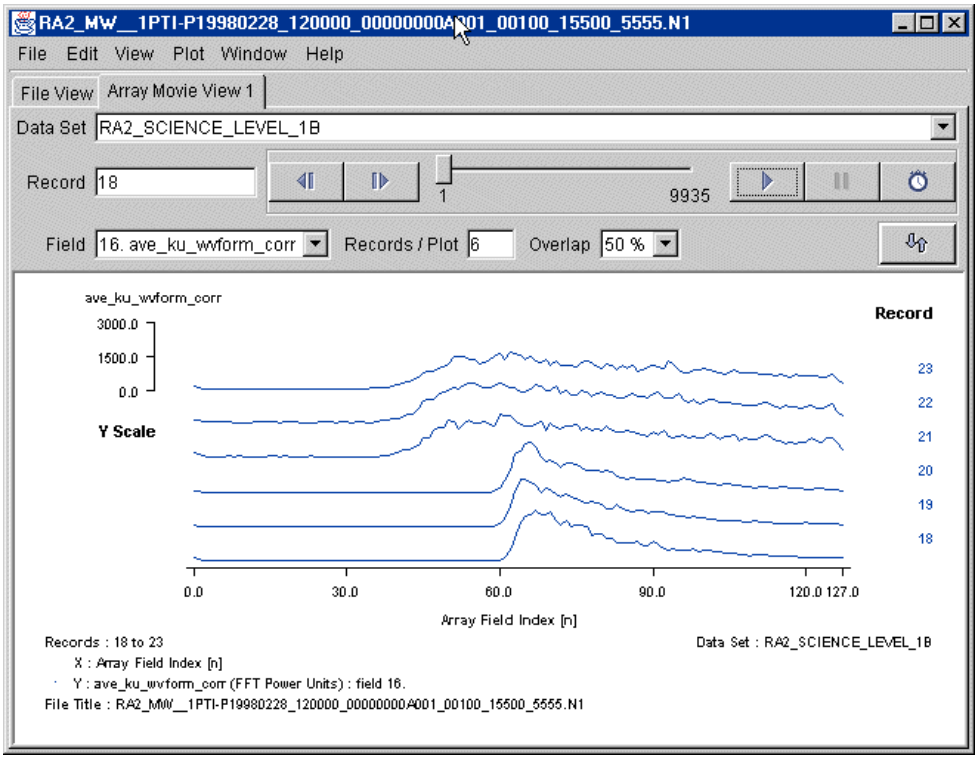


Figure 3.9 An example of the Array Movie View

3.14 How do I subset and calibrate Sciamachy level 1B products?

Either using the special Sciamachy calibration function from the file menu (see section 11.3) or by using the command line tool *SciaL1C (see* Chapter 14).

3.15 How do I export Envisat PDS data to HDF?

Use the Save as HDF function, described in chapter 11, or the command line tool *Pds2Hdf*, which is explained in more detail in chapter 14.

3.16 How do I see two views simultaneously?

To be able to see two or more *views* simultaneously, simply choose the New Window option from the *Window* menu.

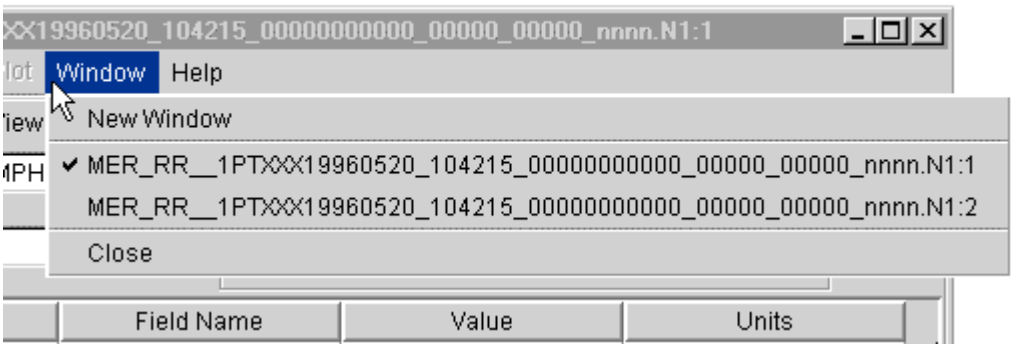


Figure 3.10 The window menu

3.17 Can I use EnviView functions from the command line?

The following tools are provided in the standard EnviView package (version 2.0.1).

- Pds2Ascii – extract field values to an ASCII text file
- Pds2Hdf – extract datasets (and subsets) to an HDF 4 file
- PdsSubset – extract datasets to a smaller PDS child product (i.e. Pds2Pds)
- PdsPixels – extract portions of image datasets to various image file formats
- PdsPixelTrack – extract individual pixels from an image data set, along a given track
- SciaL1C – extract and calibrate Sciamachy measurement data

All tools have a similar syntax for options and arguments on the command line. They are all described in detail in Chapter 14.

3.18 How do I process multiple commands in a batch job?

All EnviView command line tools can be run in a batch mode, *i.e.* with a set of commands in a text file that can be executed sequentially and automatically.

e.g. if they are written in a file called batch.txt, they can be executed so :-

```
> Pds2Hdf –batch batch.txt

> SciaL1C –batch batch.txt
```

The main advantage of using the batch command line argument, instead of a shell script or DOS batch file, is that the removal of the significant overhead to restart the java runtime for each command. i.e the batch command line argument is much faster.

More details are provided in Chapter 14.

3.19 How do I set preferences in EnviView?

Select *Preferences* from the *Edit* menu.

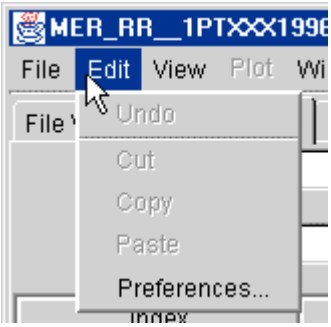


Figure 3.11 Opening the preferences dialog

The list of options that may be modified appears in the left hand column. This has three options, one to configure the Image View, one to save the current working directory for the next session, under the *Files* option, Figure 3.12, and one to select the “look-and-feel” of EnviView.

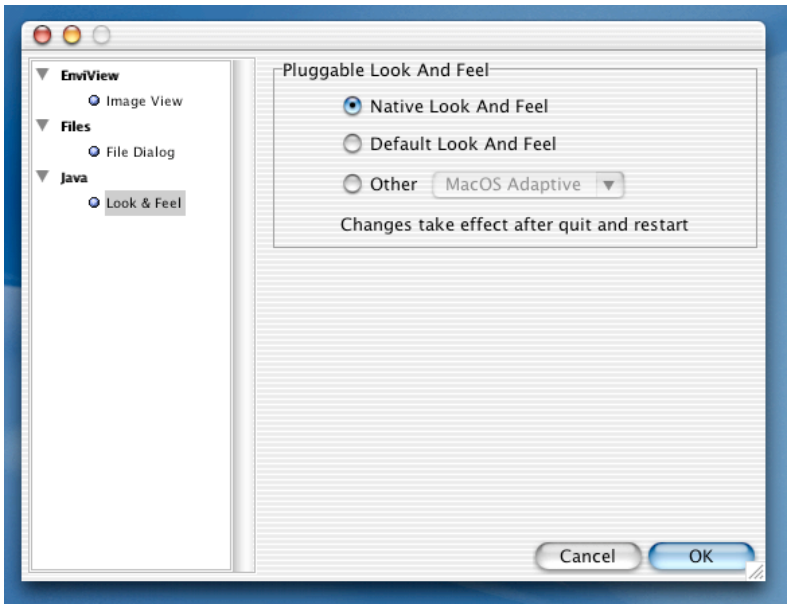


Figure 3.12 Preferences window

The File Dialog preferences allow the user to enable (default) or disable that EnviView remembers the last directory visited, each time it is quit. When EnviView is run again, if the option was enabled, the open dialog will default to the last visited directory.

3.20 How do I customise the appearance of EnviView?

Select *Preferences* from the *Edit* menu, and then *Look & Feel* in the list on the left. Three options are available

- Native Look and Feel – this uses the look and feel of the operating system upon which the application run.
- Default Look and Feel – this uses the java default look and feel for your platform. This is configured in the properties file in your java runtime (see your java runtime documentation for details).
- The final option is actually a selection box with any “look and feels” available on your platform. These may include : -
    - Metal – a design specific to Java.
    - CDE/Motif – the X Windows Motif appearance
    - Windows – only available when running under Windows
    - MacOS - only available under MacOS 9 or X.

If the default settings are not attractive, simply experiment with different settings. Note that the changes do not take effect until EnviView has been restarted.

The Mac OS 9 look & feel is not a perfect implementation, particularly the file dialog. Under Mac OS 9, EnviView works best with the Metal look & feel.

4. DESCRIPTION OF THE USER INTERFACE

This chapter explains some of the basic and common themes in the user interface for EnviView.

4.1 Views and Windows

The EnviView graphical user interface allows a user to display the data contained in a Envisat data file in a variety of ways, namely in textual, graphical, and image formats. The interface presents these *views* of the data together in a single window. The views are effectively overlaid on top of each other but each view can be accessed by a *tab* label attached to it, see Figure 3.4 for an example.

A new window is created for every data file opened and all views of the data are displayed in this window. Each window has the name of the data file that is displayed therein.

It is also possible to create duplicate windows for the same data file. The window titles are appended with a unique identifier, namely a colon followed by an index that is increased as duplicate copies are opened.

By opening two windows on the same data file, separate views may be examined side by side. For instance an image view could be open in one window, and a record view in the other, all from the same file.

These windows and views, of which there could potentially be many, are managed from the *View* and *Window* menus available in each windows menu bar. Essentially, the menu choices allow easy navigation between views and windows, the creation and closing of the different views themselves and opening of new products. The menu options are the same for each window opened. Certain options are enabled or disabled depending on the data file to which they refer. These are discussed in more detail in section 4.2.

4.2 Tabs

EnviView uses tabbed pages to organise the different data views created by a user. Each time a new view is opened, either by selecting an option from the *View* menu (section 4.3.3) or some other method (e.g. section 5.2), another tabbed view is added to the display. The tab is labelled with the view name. If more than one view of the same type is opened, a unique integer suffix is appended.

Simply clicking on the tab brings that view to the front, but views can also be brought to the front by selection the *View* name at the bottom of the *View* menu (section 4.3.3).

4.3 Menus

Clicking on one of the menu options at the top of each window (or screen) displays a list of options of which only one can be selected at a time. The items in each list are identical for each product. However, some of them are disabled, or “greyed out”, if they are not available or not appropriate. Each window keeps a record of which other windows are opened and which views are currently open. The different menu items, their options, and their functionality are discussed in more detail below.

4.3.1 The File menu.

This menu has 7 options;

- **Open** –opens a new data file. When selected a *file chooser* opens in a separate window. The details of the file chooser depend on the operating system under which the application is being run.

- **Close** –closes the current file. All duplicate windows for the same file are also closed. To close only one duplicate window, select *Close* from the *Window* menu. Naturally, this option is only enabled once a file has been opened or loaded.

- **Export to Hdf** –export an Envisat product to HDF 4. This option is only enabled once a file has been opened and if the HDF library has been correctly installed. See further explanation in Chapter 11.

- **Sciamachy Calibration** – calibrate and subset a Sciamachy level 1B product, creating a level 1C product. This option is only enabled once a SCI_NL__1P file has been opened or loaded. See further explanation in Chapter 11.

- **Page Setup** – this option is intended to open a page setup dialog to configure printing. It is only functional on certain platforms.

- **Print** – this option opens a platform specific print dialog, provided by your java runtime. This option is only enabled if the currently open view can be printed.

- **Quit** – this ends the current EnviView session and all views and windows are closed.

When EnviView is started, only the *Open* and *Quit* options are enabled. The window is simply labelled “EnviView”. The window title is updated as soon as a data file is loaded.

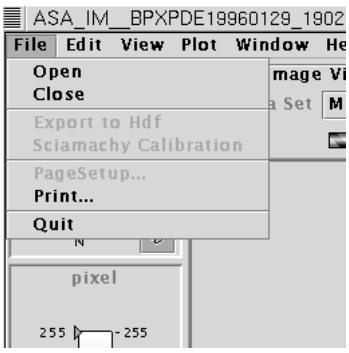


Figure 4.1The *File* menu

4.3.2 The *Edit* menu.

This menu has several options of which only the *Preferences* option is available at all times. The others, namely *Undo, Cut, Copy,* and *Paste*, are enabled when selected text may be copied to the System clipboard, for exchange with other applications. The *Cut* and *Paste* functions are naturally only enabled in editable fields. The *Copy* function however is available whenever any text is selected. The *Undo* function is not currently available.

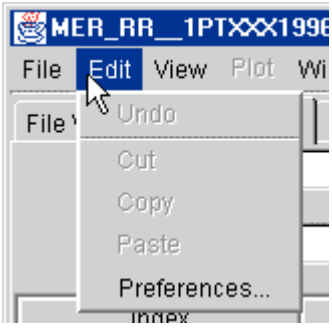


Figure 4.2 The *Edit* menu

The *Preferences* option allows the appearance of EnviView and some of its behaviour to be customised, see section 3.19 for more information.

4.3.3 The *View* menu

This menu has a number of sections to it, separated by lines for clarity.

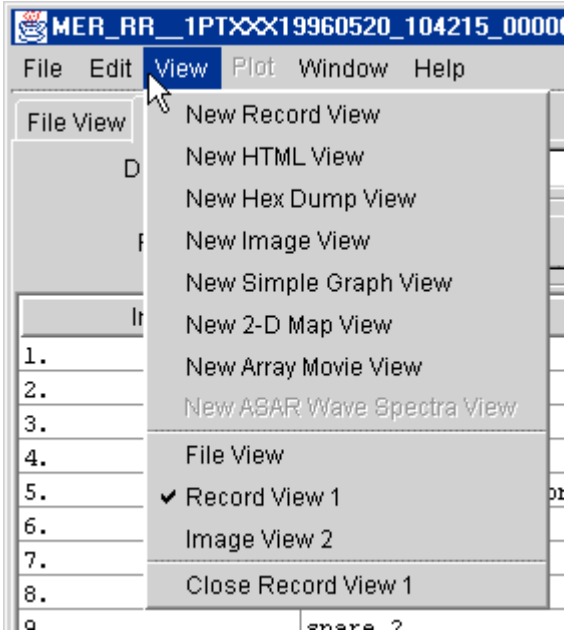


Figure 4.3 The *View* menu

- The first section allows new *views* to be created. Only views that are appropriate for the currently open file may be selected. These options are shown in Figure 4.3. If more than one type view is created, an integer will be appended to the view name so that they can be distinguished from one another. The views available are :-
  - New Record View – shows the contents of one record, section 5.1.
  - New HTML View – shows the contents of one record in a formatted HTML table, and includes a full description for every field, section 5.4.
  - New Hex Dump View – shows the contents of one record as a hexadecimal dump of the bytes, section 5.5.
  - New ASCII View – display the contents of ASCII data sets, section 5.6.

- New Image View – examine and manipulate image data, chapter 7.
- New Simple Graph View –a view where fields from a single data set can be plotted, chapter 6.
- New 2-D Map View –a simple graph to plot geolocation information, chapter 9.
- New Array Movie View –a graph for array data, in particular Radar Altimeter waveforms, chapter 10.
- New ASAR Wave Spectra View – opens a new view for ASAR wave spectra, chapter 8.

- The second section is a list of views already created for the current data file and is automatically updated as new views are opened. The view name is identical to the identifier used on the tabs and this provides an alternative method to switch views.
- An optional third section may appear for some views with a submenu for view options (e.g show raw bytes column for record view).
- The final section offers the option to close whichever *View* is currently open (note this does not close the *Window*).

4.3.4 The *Plot* menu

The *Plot* menu opens a new window allow greater flexibility in plotting data than the Simple Graph View. See Chapter 6 for more information.

4.3.5 The *Window* menu

Like the *View* menu this has a number of sections, see Figure 4.4. This example shows the *Window* menu pop-up when two files are already opened.

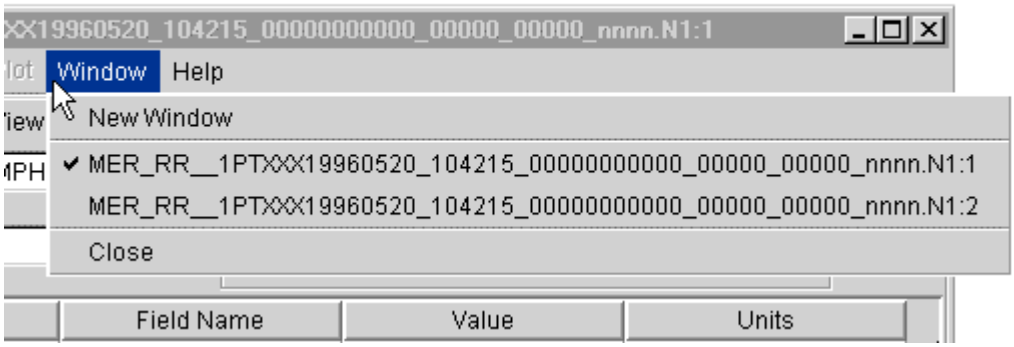


Figure 4.4 The *Window* menu

- The first is *New Window* that will make a duplicate of the present window with the same data file. If selected, the two windows have the same name appended by a unique integer identifier. Views created in either window will appear in both windows. This allows views to be easily compared side by side on screen.
- The second section contains a list of the opened windows. Selecting of one of these will bring the window to the front and make it active.

- The final option is simply *Close,* which closes this window. If no other windows are open for the current file, the file and all the views contained are also closed. Duplicate windows open on the same file are unaffected.

The list of open windows is automatically kept up to date.

4.3.6 The *Help* menu

This menu option currently offers two options.

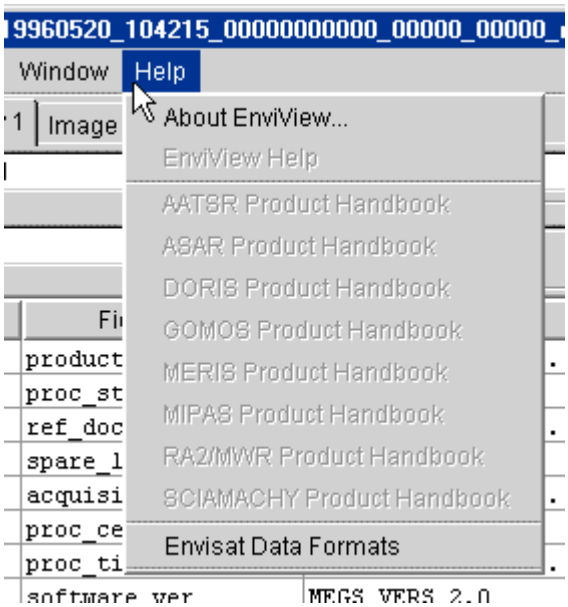


Figure 4.5 The *Help* menu

- *About EnviView,* which creates a new pop-up window with information about the current version of EnviView. It also displays information about memory being used.

- *Envisat Data Formats* – this opens a new window which allows the details about data formats for different products to be examined.

The other options will provide links to the detailed handbooks for the instruments on board Envisat. However, at the current time these are not available.

When reporting a bug or requesting support, please copy the contents of the About EnviView box into your e-mail. (Use Ctrl-C to copy the selected text).

5. VIEWING DATA IN TABLES

Table views show Envisat data as text. Any type of Envisat data can be viewed in a table format and this is the default way of viewing the data. Three table view formats are available, depending on how the data is to be used. Each of the views is optimised for a different use.

- Record View (Figure 3.4)
  This is the default view showing the fields of a single record as a table in a spreadsheet like way. Each row shows a single field with 4 columns showing the field index number, the field name, the value in scientific or engineering units and finally the units. This view is good for viewing and comparing the actual values of fields. If additional description is also available for a given field, the field name is highlighted in blue. Hovering the mouse pointer over the field name displays the additional description. An optional fifth column displays the raw bytes for each field.

- HTML View. (Figure 5.6)
  This view shows a record in a two column table. The first column gives the full field description whilst the second column gives the field value or array values and summary (only sample values if there are many array elements). This view is useful for viewing the descriptions of fields together with their values, which helps to understand the layout of a product type.

- Hex Dump View (Figure 5.7)
  This view shows the actual bytes making up the data in a particular record. It contains three columns. The first gives the byte offset, in hexadecimal, at the start of the row of data. The second column shows the data in hexadecimal with 16 bytes per row divided into four four-byte words. The third column gives the ASCII translation of the hex dump. This view may be useful when one is developing one's own decoding software.

- ASCII View (Figure 5.8)
  This view displays the contents of ASCII data set records. All products have an MPH and an SPH, and a few products have additional data sets that are also stored in ASCII format.

5.1 Tabular View features

From a *File View* of the data product, double click on the data set, which will create a new *Record View* containing the data in a table.

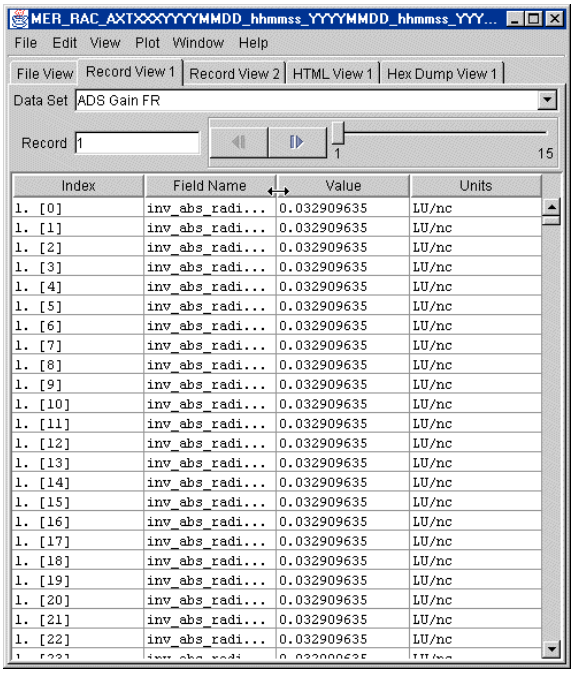


Figure 5.1: Table View Feature Summary

Every table, be it in a Record, HTML, Hex or ASCII view, contains a set of controls for navigating through and selecting the data being displayed, Figure 5.2.

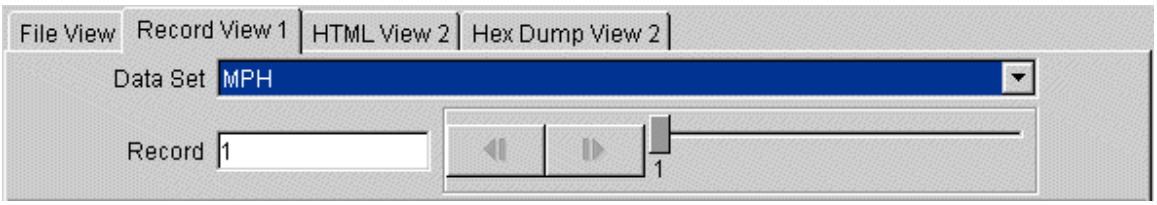


Figure 5.2 Table view controls

The controls are divided in two vertical sections. At the top is a selection box with a pop-up list of the data sets in the product that can be displayed in the current view. Selecting a data set loads the data for a single record into the table.

Below the data set list, on the left hand side is a text field containing the currently displayed record number. Any record number can be entered here, assuming it is a valid number and valid value for the data set. The record's data will be displayed after pressing enter. To change record, back and forward buttons can be used to increment or decrement the record counter by a single record, or for larger changes a slider can be moved left or right. The record index in the record number field is updated accordingly.

The following features are also available in all views: -

- Creating a new view of the same or different data by selecting the *New* view in the *View* menu.

- View tabs, for quick navigation between multiple open views within the same data product. Also the *View* menu may be used to switch between currently open views, section 4.3.3.

The following features are specific to record views

- Resizing column widths by using the mouse to drag the column divider to the left or right.
- Changing column order by dragging the column headers.
- Displaying extra field description by hovering the mouse pointer over a field name highlighted in blue.
- Displaying alternative value formats by hovering the mouse pointer over a value highlighted in blue.
- Optionally display an extra column containing the raw bytes for each field.

5.2 Creating a new view

To create a new view of the data, select the *View* menu and then a *New View* item (either Record, HTML, Hex or any of the other views available). The example in Figure 5.3 shows a how to create a new hex dump view.

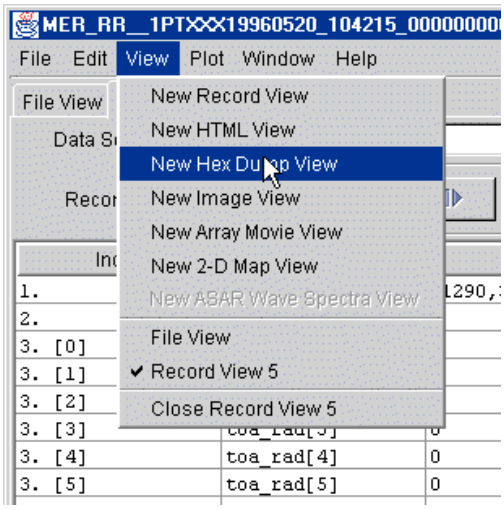


Figure 5.3 Creating a New View using the menu

The alternative way of creating a new record view is to double click a data set in the *File View*. Click the File View tab to select this view. Then double-click on the chosen data set and a new Record view of the data is created.

5.3 Switching between Views

Only one view of the data is displayed at a time per EnviView window. To switch between them use the tab selectors at the top of the window, below the EnviView menu bar. Clicking a tab brings the corresponding view to the front. Figure 5.4 shows how to switch from an HTML view to a Hex Dump view.

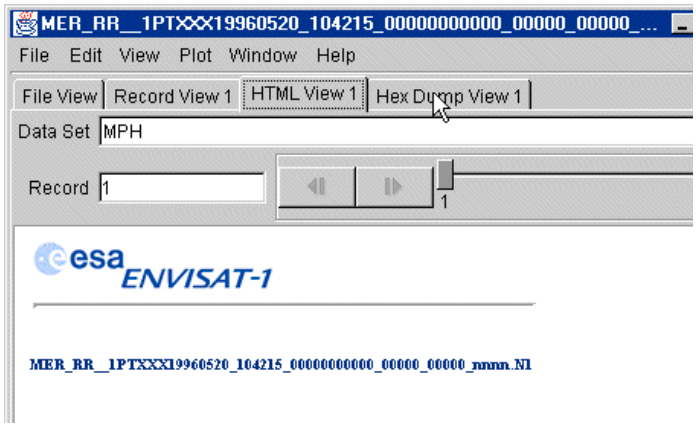


Figure 5.4 Switching Views using Tab selectors

An alternative way to switch views is to use the *View* menu. The View menu also contains a list of all the open views, see section 4.3.3. The currently displayed view is indicated with a tick. Select the view to display from the other views on the list. Figure 5.5 shows how to select a Hex Dump view from the view menu. This view switch is identical that shown in Figure 5.4.

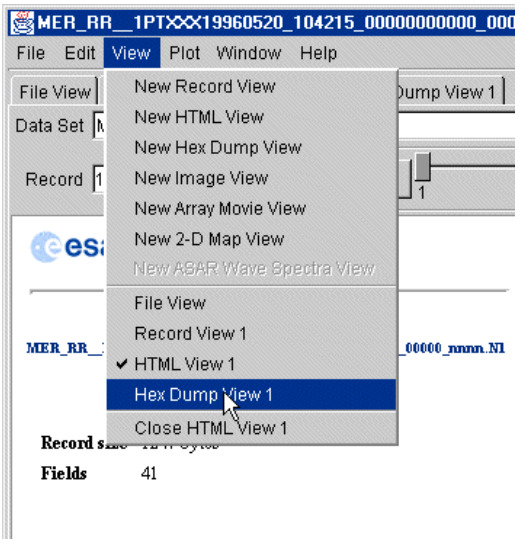


Figure 5.5 Switching Views using the View menu

5.3.1 Changing the data set within the same view.

To view a different data set (within the same data product) without opening a new view, select the data set from the Data Set popup menu, which shows a list of all the data sets that may be displayed. Selecting a different data set updates the information in the current view. For instance if a Hex view is displayed of a particular data set, selecting a different data set will display the contents of the selected record of this new data set.

5.3.2 Closing a view

Select the view to be closed either using the tab selectors or the view menu (e.g. Hex Dump View 1). Then from the *View* menu, select *Close Hex Dump View 1* item. Note that the close view menu item for the currently displayed view is always the last item of the View menu.

5.3.3 Using Multiple windows

It is sometimes useful to see different views of the same data or different data sets within the same product side by side. To do this, a new window must be created. Select the *New Window* item from the *Window* menu. A duplicate window opens with exactly the same views as the original. In each window, select the views of interest.

5.4 HTML View

Figure 5.6 shows an example of the HTML view. The data area shows an ESA/ENVISAT logo below which is the data product name, the number of records and the number of fields per record for the data set. Below this is a two-column table, which lists the field names in the first column with the value and units in the second column. Each row in the table corresponds to one field of a record.

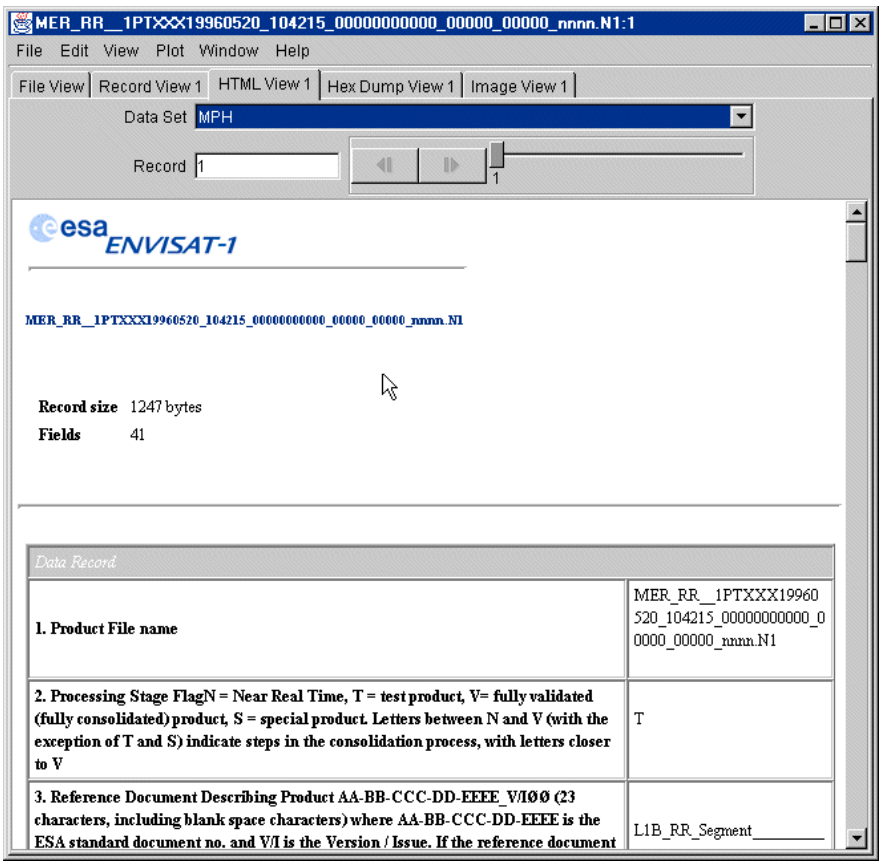


Figure 5.6 Example HTML view

This view provides a more detailed description of each field than the standard *Record View*, and may be saved to an HTML file for inclusion in reports. Note that long array fields are summarised in this view, to reduce the rendering time required. For full analysis of array fields, use the *Record View*.

5.5 Hex Dump View

This view shows the raw bytes of each data set record. It contains three columns. The first gives the byte offset, in hex, at the start of the row of data. The second column shows the hex data with 16 bytes per row divided into four four-byte words. The third column gives the ASCII translation of the hex dump. Note that columns can be neither resized nor rearranged in the hex view.

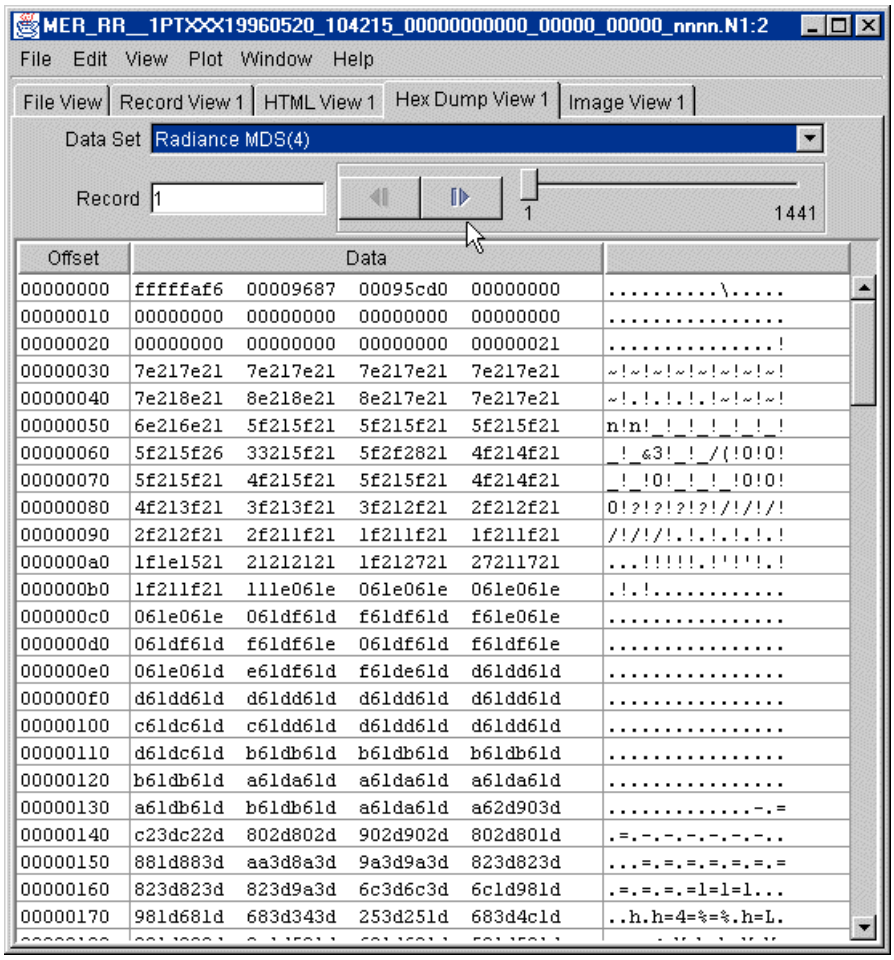


Figure 5.7 Hex Dump View example

This view may be useful for developers of software to read Envisat PDS format products directly, so that they may verify their results against those obtained in EnviView.

5.6 ASCII View

This view displays the contents of ASCII data set records. This view cannot display binary data sets, for which a Record View, HTML View or Hex Dump View are better suited.

All products have an MPH and an SPH which are always written in ASCII, and a few products have additional data sets that are also stored in ASCII format.

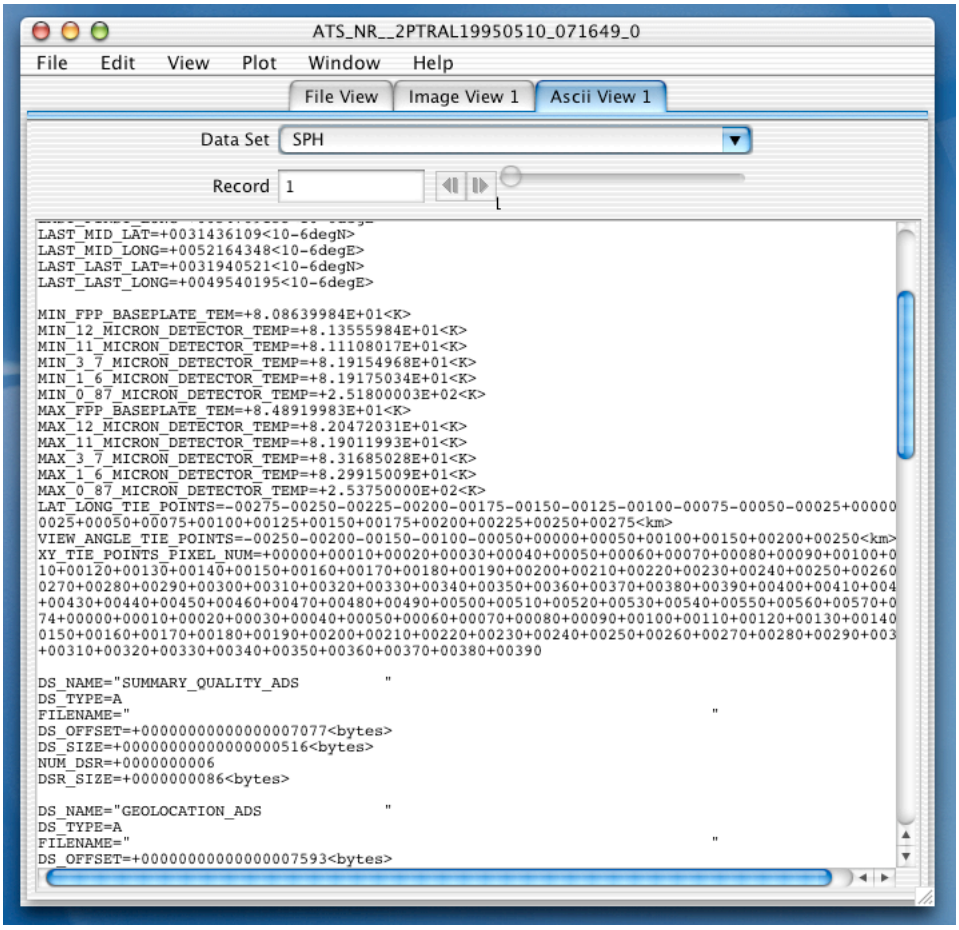


Figure 5.8 ASCII view example

6. VIEWING DATA GRAPHICALLY

6.1 Introduction

EnviView has two methods for displaying data Graphically. The simple graph viewer, see section 6.2, can be used to display fields from a single record in a quick and simple manner. For more advanced graph plots, and for combining data from multiple data sets and/or products, an extended data extraction and plotting facility is available, see section .

6.2 What is the Simple Graph Viewer

The Simple Graph Viewer displays one or two data set field values plotted against the record index or other field values, as line graphs.

6.2.1 Quick Start

Open the required data product. The *File View* opens showing a list of data sets. Select the *View* menu, *New Simple Graph View* menu item (as shown in Figure 6.1).

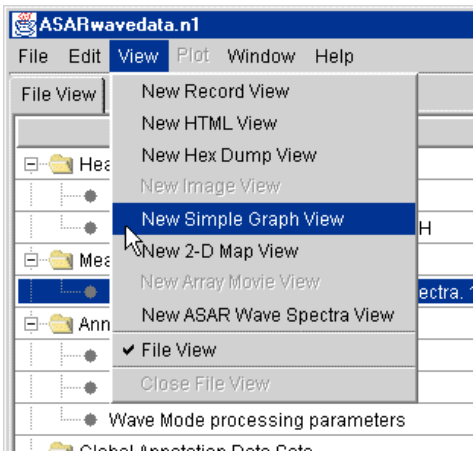


Figure 6.1 Starting a new simple graph view

A new simple graph view will open (similar to that shown in Figure 6.2). EnviView automatically chooses the first valid data set (for plotting) however no data is plotted. At least one Y field must be chosen to plot using either the **Y' field** or **Y field** pop-up menus.

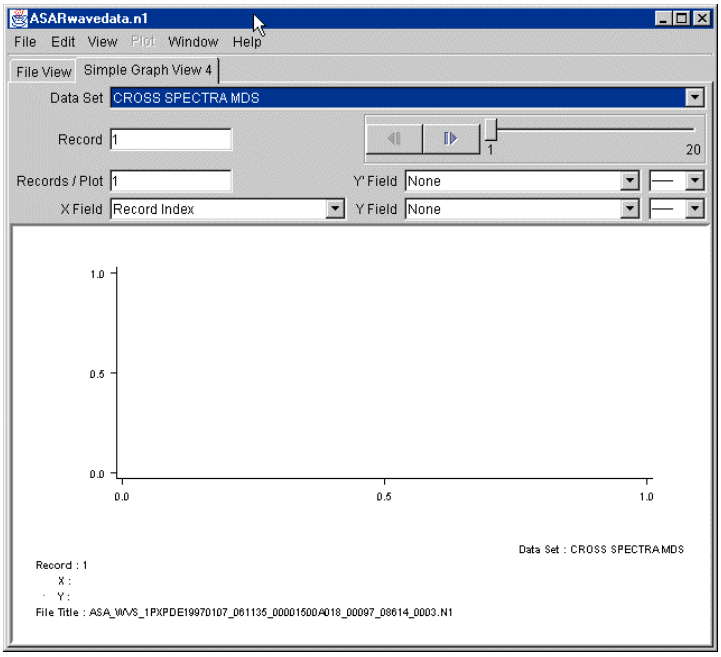


Figure 6.2 A new Simple Graph View

6.2.2 Feature Summary

Figure 6.3 shows an example of a Simple Graph View window showing the values of two fields plotted against record index number. This section briefly describes the features and controls available for Graph views. More details are available in the sections that follow.

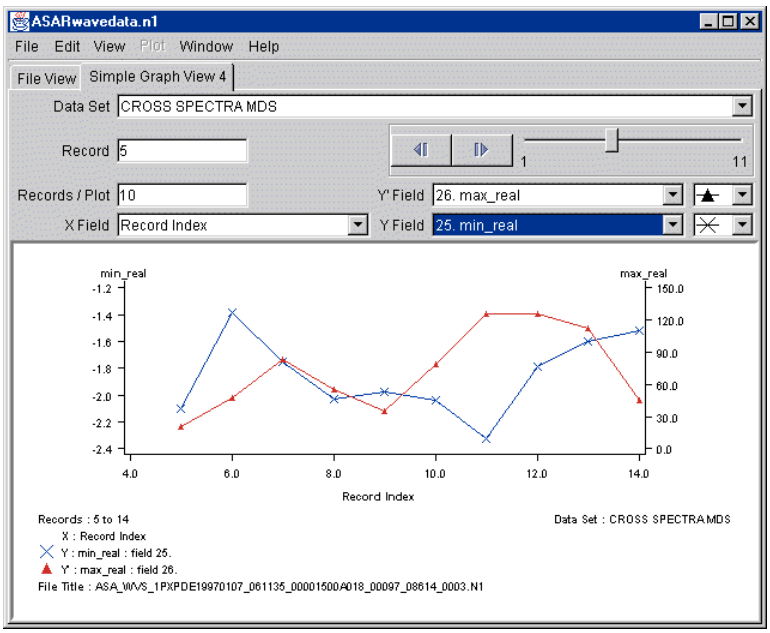


Figure 6.3 Simple Graph View example with data

The features available moving from the top towards the bottom of the window are;

- Menu bar and Menus.
  These are identical to those on other windows

- View tabs.
  These select other open views with a single mouse click.

- Data Set Selector.
  This displays the name of the current simple graph data set, in the data product, allowing selection of other data sets displayable by the simple graph viewer. See section 6.2.4.1 for more details.

- Record Selector Controls and indicators.
  These consist of the **Record** field which displays the record number of the lowest displayed record. A record number can be entered into this field. The nudge back and forward buttons allow movement through the data set one record at a time and the record slide control allows a quick location to a particular part of the data set. The **Records/Plot** field shows how many records have their data plotted on the current simple graph. This field can be edited. See section 6.2.4.2 for more details.

- Field Selector controls.
  These controls allow selection up to 2 field values to plot on the vertical Y axis and also selection of the X-axis field against which to plot the Y values. See section 6.2.4.3 for more details.

- Plot point style selector and indicator
  The point style pop-up menu selectors display the current symbols used to plot points. Use the pop-up menus to select different point styles. See section 6.2.5 for more details.

- Data Area
  The data area displays the plot, the vertical Y axes (or axis) and the horizontal X-axis.

- Identification and Information Area. See section 6.2.5 for more details.
  This area, at the bottom of the window, displays which records the data plotted comes from, the data set name, the X-axis field, the plot symbols and fields for the Y-axis and finally the File title (that corresponds to the data product).

6.2.3 Array vs Scalar fields.

It is important to appreciate that there are two types of field data that can be plotted in EnviView: -

- scalar fields: These are fields which have a single value per record in the data set. To be meaningful, a plot using these data will be of the data value for *multiple records*.

- array fields: These are fields with multiple values (i.e. arrays) per record. Therefore, each line on a plot has the data for a single record (usually there will be a single curve per plot, but more are possible).

Most of the functionality of the *Simple Graph Viewer* is independent of the field type, such as data set selection and field selection and is described in the following chapter. However, it is important to understand the difference so that the resulting plots can be understood.

6.2.4 Selecting Data to Plot

This section explains, in detail, how to select data to plot on simple graph view. The major steps are listed below. Note that once a data set is selected, the remaining steps can be carried in any order and then the settings can be interactively altered once the graph is being displayed;

- Select the data set which contain records to plot.

- Select the record from which the plot is to start.
- Select how many records should appear on the screen.
- Choose the X field (the default is record index).
- Choose the field within the record to plot on the Y scale. 1 or 2 fields can be selected.

6.2.4.1 Selecting a data set

The data set pop-up menu, towards the top of the window, contains a list of plottable data sets for the product. It displays the name of the data set that is currently being viewed. To select a different data set (if available), click and open the pop-up menu and select the data set required.

Data sets must contain 2 or more records in order to be viewable with simple graph viewer.

6.2.4.2 Selecting Records

Once a data set is selected, the range of records from which to display data can be chosen. The controls to do this are shown in Figure 6.4. Note, the figure has blurred the parts that are not relevant to this discussion.

Figure 6.4 Record Controls

Select the record from which the plot is to begin. This can be done in several ways;

- If the record number of the first record to plot is known, type it into the **Record** field. Select the record field and enter the value using the keyboard. Remember to press Return. The record slider, to the right of the record field, will move to the corresponding position of the record.
- Use the nudge back and forward buttons (between the Record field and the record slider) to move through the data set one record at a time.
- To browse through records or know the approximate position of the data, in the data set, drag the record slider to the corresponding position. If the data set contains a large number of records use the nudge back and forward buttons to move a single record at a time to the precise start record required.

Select the number of records per plot. Usually this will be the same as the number of points per line on the display. Follow the steps below;

- Select the **Record/Plot** field.
- Enter how many records are to be displayed per plot into this field.
- Remember to press Enter to see the changes applied

The number at the right hand end of the record slider (11 in the case of Figure 6.4) is the highest start record from which a plot can begin. This is the same as the number of records in the data set if only 1 record is plotted. In this example the data set contains 20

records;10 records are plotted (Records/Plot field) so the lowest start record is 11, because the number is inclusive, i.e.

Lowest start record = Number of records in data set – Records/Plot +1

Note that a graph can be displayed containing data from just one record so long as the field being plotted is an *array field*, see section 6.2.6 for more details.

6.2.4.3 Selecting Fields

To choose the fields to display on the simple graph viewer, there are controls to do this and to control the look of the plotted points, see Figure 6.5.

Figure 6.5 Field controls

The default X field (plotted on the horizontal X-axis) is the record index. However, this can be altered to any available data field in the record. This is useful to plot two data fields against each other.

Select the X-field pop-up menu to display list of available fields. Select the field to be plotted on the X-axis from this list.

Initially no data fields are selected for the Y axis. At least one Y field must be chosen to plot a graph. Up to two line graphs can be plotted simultaneously by the simple graph viewer. Both lines are plotted against the same X-axis field.

Select either the Y' field or Y field pop up menus to display a list of available fields. From the list displayed, chose the field to plot on the Y-axis. Note that record index is not available as a Y-axis field. Alternatively, another Y-value can be plotted chosen from the other Y or Y' field pop-up menu to display a second line.

6.2.5 Changing the display of graphs

The look of the graph being displayed can be modified in several ways, listed below;

- Point style selection
- Changing the axes scales
- Resizing the graph.

To alter the symbol of the plotted points associated with each line, use the pop-up menus to the right of the Y and Y' Field pop-up menus (see Figure 6.5). 7 styles are available starting with a plain line, having no point symbol, which is the default setting.

The scales of both the X, Y and Y' axes can be altered. EnviView scales the axes to maximise the screen area on which the graph is plotted by default. However, it also allows scales to be set manually, which is useful for instance to emphasis a certain region of the data. The process is similar for all 3 axes.

Double click the axis that to be changed. A dialogue box opens, similar to that shown in Figure 6.6.

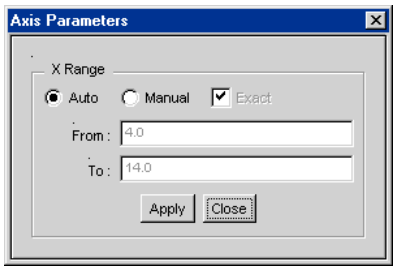


Figure 6.6 Changing the simple graph axes scale

To let Eniview choose the scale, maximising the axis length used by the plot make sure the *Auto* radio button is checked. To set the scale manually, set the *Manual* radio button and enter the values for the start and end points of the axis in the **From:** and **To:** text fields. Click the *Apply* button to see the changes take effect. If the *Exact* check box is left unchecked, EnviView may round the values entered.

This procedure is the same for each axis. Note that the Y' axis appears at the right hand side of the plot with the Y axis at the left hand side. Changing the X-scale range to be larger than the record range plotted does not increase the number of records plotted. This just displays the same number of records (defined in the Records/Plot field) in a smaller area.

To increase the screen area of the plot, increase the window size by maximising it or dragging a resize control. (The details of this depend on the Graphical User Interface being used). The plot will be resized to make use of the increased window area. Conversely decrease the window size to shrink the area of the graph plot.

6.2.6 Viewing Array Field data

One-dimensional field arrays can be viewed in simple graph viewer. These plots can plot the field array data from several records at a time. Figure 6.7 shows a simple graph view of a field array. Only one record/plot is shown and the X-axis shows Array Field Index [n] instead of Record number.

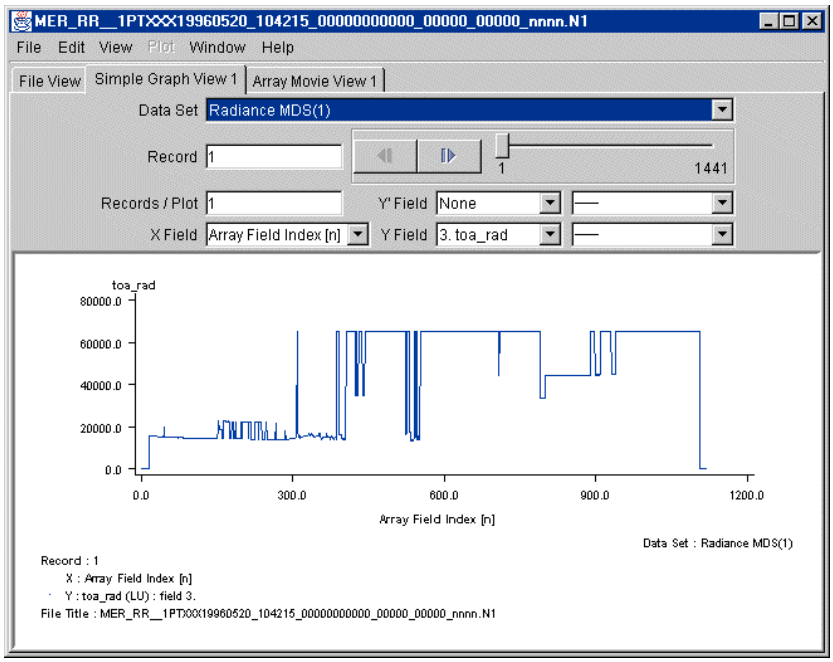


Figure 6.7. Simple graph view of a field array

These are the steps to display a field array, for just one record, on a simple graph view;

- Select a data set containing array fields.
  To find out whether a data contains array fields, view it in the HTML or Record views and check the field characteristics.
- Select the record to be viewed.
- Set the Record/Plot fields to 1.
- Select the X-field to be Array Field Index [n]
- Select the Y-field (or fields) to be the field array to be plotted

Navigating through the records, using the record controls, causes the plot to be re-drawn to show the field array values for the record selected.

6.2.6.1 Comparison with the Array Movie viewer

The Array movie viewer can produce a similar display to that of a field array in the Simple Graph Viewer. Figure 6.8 shows the same data as plotted in Figure 6.7 in an array movie viewer.

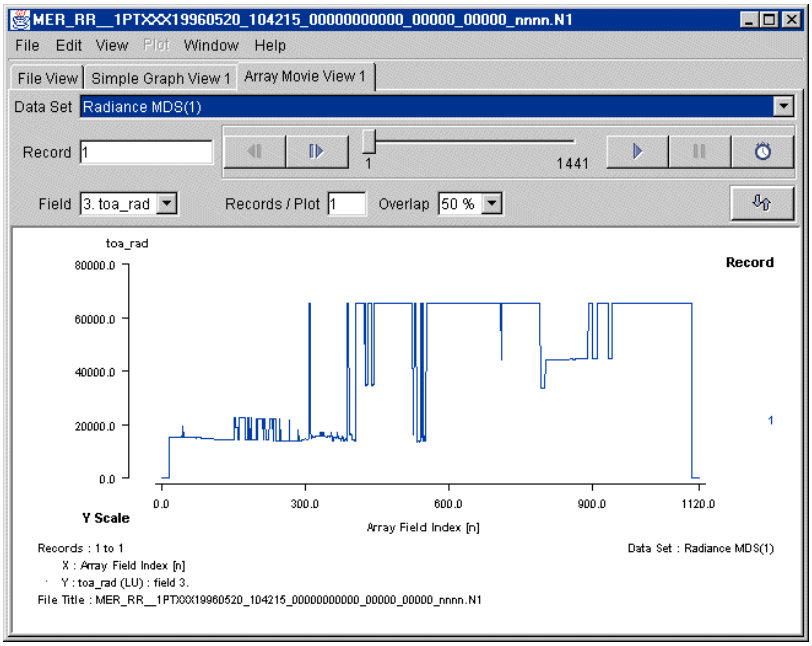


Figure 6.8 Array movie equivalent

These two different viewers are designed for different uses even though they can represent the data in similar ways. Generally the following guidelines apply to the choice of which viewer which should be used.

- To compare arrays, from different records, use the array movie viewer.
- To compare the array field with another field in the same record use the simple graph viewer

See chapter 10 for details about using the array movie viewer.

6.3 Extended Data Extraction and Plotting

The plotting and extended data extraction facility allows users to extract data from multiple locations and plot them in a number of different ways. Extended data extraction is explained in section 6.3.2. There are several ways to display the data, each of which is explained further in section 6.3.3.

6.3.1 What is Extended Data Extraction

The simple graph viewer allows data from a single data set to be examined quickly and easily. For more advanced data analysis, the extended data extraction facility provides a method for extracting data from various data set records, and even from multiple files. These variables can then be plotted in various ways. The following sections explain how to go from extracting data to plotting a graph.

There are two formats EnviView uses in the extraction and viewing of data. The first format is the "variable". A variable is a set of data extracted form a data set. It can be used in multiple graph data sets. The second format, which is a graph data set, is a set of variables configured for display on a graph type. For example a variable may be the "bad_pixel_mask" field from the PPG/ELATON" data set record in a Sciamachy L1B product. This can then be set against its index variable to be a line (graph data set) on a two-dimensional line graph.

6.3.2 The Graphing Components

There are two types of graphs, a set of pre-configured graphs for each product (discussed in section 6.4), and a set of blank graph types discussed below:

- *2D line graph:* This is a graph with multiple lines plotted on a two dimensional plane. Each line, representing a graph data set, can have an interpolation type, linear or spline. A marker line can be added to the graph to guide the eye.

- *3D line graph: This* is a graph with multiple lines plotted in a three dimensional plane.

- *Pie Chart:* This graph type can only be used to plot single one-dimensional data sets.

- *Bar Chart:* This is a graph which can represents multiple one-dimensional data sets. The bar sets can be colour configured and given a label.

- *Contour Graph.* This graph type can only be used to plot single two-dimensional data sets. Both the number of contours and colours are configurable.

- *Density Graph.* This graph type can only be used to plot single two-dimensional data sets. In this plot, the magnitude of the data is reflected by the colour intensity of each pixel.

- *3D Surface Graph.* This graph type can only be used to plot single two-dimensional data sets, where the z-plane reflects the magnitude of the values.

The configuration of each graph type is discussed further in section 6.3.8

6.3.3 Opening the Plot Document

To start using the extended data extraction tools, select the *New Plot Window* menu item from the *Plot* menu in any EnviView window (as shown in Figure 6.9).

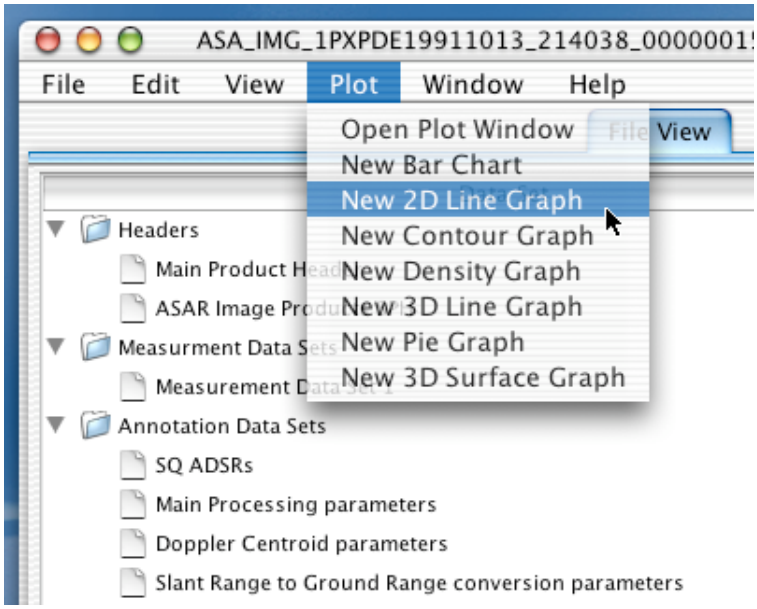


Figure 6.9 Starting a new plot window

The *Plot* menu contains several sets of preconfigured or saved graphs, and a set of graph templates. The description of the preconfigured graphs follows this section. Once a preconfigured graph or a graph template is selected, a plot window pops up displaying the selected graph type. If a further preconfigured graph or graph template is selected, it too is added to the plot window. The user can easily switch between graphs using the tabs at the bottom of the plot window.

6.3.3.1 The Graph Panel

Figure 6.10 shows the plot window displaying the two-dimensional line graph template.

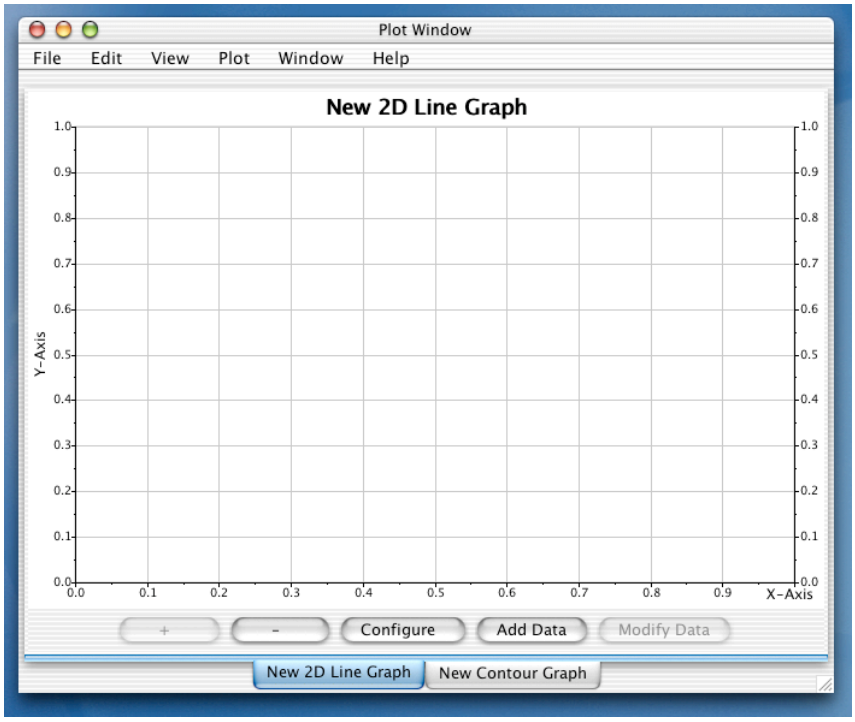


Figure 6.10 Plot window

At the bottom of the plot window is a tabbed panel, which can be used to switch between graphs. Each tab contains a graph and a *Graph Panel*, which provides the user with the following options:

- The *"+"* button allows the user to increment through the data and choose the next record or the next field as set on the *Variable Extraction Panel* (see section 6.3.6.1).
- The *"-"* button allows the user to decrement through the data and choose the previous record or field as set on the *Variable Extraction Panel* (see section 6.3.6.1).
- The *Configure* button brings up a dialog box which allows the user to change graph properties such as the title and axis labels etc. (see next section).
- The *Add/Set Data* button brings up the *Add/Set Data Dialog* which can be used to set the data for the selected graph (see section 6.3.4).
- The *Modify Data* button brings up the *Modify Data Dialog* described in section 6.3.5. Note that this button is only enabled if data has already been added to the graph.

6.3.3.2 Graph Configuration

Clicking on the *Configure* button brings up a dialog box similar to the one shown in Figure 6.11 below.

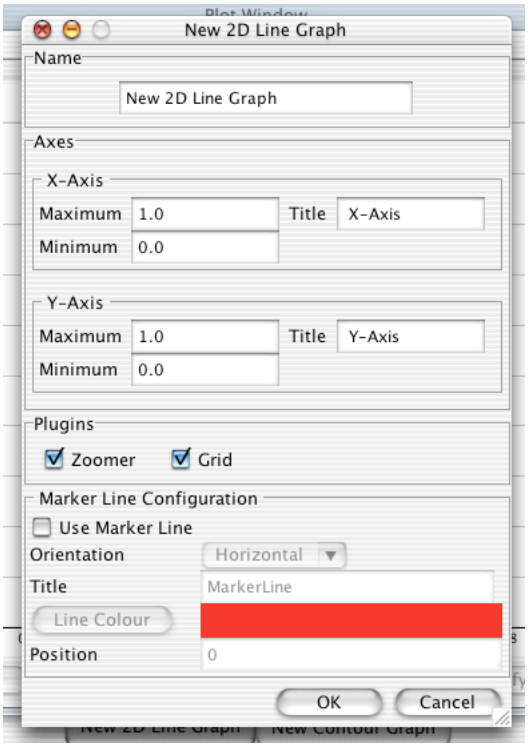


Figure 6.11 Graph configuration dialog box

The dialog box provides the user with the following options:

- The *name* text field allows the user to change the name/title of the graph.
- The X and Y axis title fields allow the axes labels to be changed.

- The X and Y axis maximum and minimum fields display the current X and Y ranges. Note that the maximum and minimum values are only enabled if their variation is not restricted by the graph type.

- Configurable options specific to certain graph types (see section 6.3.8) are displayed at the bottom of the dialog box.

- The *OK* button confirms the changes and closes the dialog box.

- The *Cancel* button ignores any changes and closes the dialog box.

6.3.4 The Add Data Dialog

As shown in Figure 6.12 below, the *Add Data* dialog provides the user with the following options:

- All graph data sets are given a default name by which they can be identified. The *name* text field allows the user to rename the graph data set to something more appropriate or preferential.

- Variable selectors for each axis or plane allow the user to select from a list of preconfigured variables.

- The *Get New Variable* button brings up the *Add Variable Dialog* (see section 6.3.6) and allows the user to choose new variables not included in the preconfigured variables list.

- Configurable options specific to certain graph types (see section 6.3.8) are displayed at the bottom of the dialog box.

- The *OK* button confirms the changes and closes the dialog.

- The *Cancel* button ignores the changes and closes the dialog.

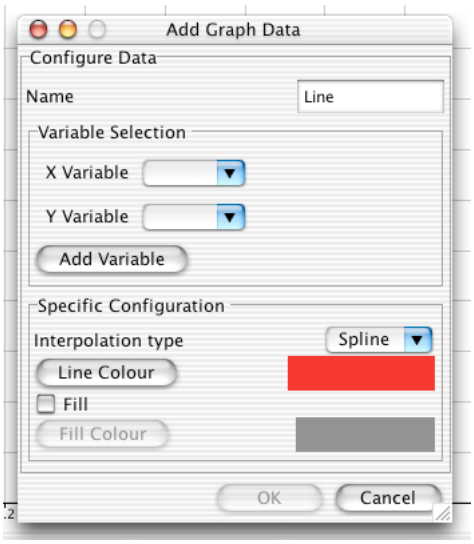


Figure 6.12 Add Data Dialog

6.3.5 The Modify Graph Data Dialog

The *Modify Data* button, which causes the *Modify Graph Data* dialog to be displayed, is only enabled if data has already been added to the graph using the *Add/Set Data* option discussed above. Hence, this option only allows users to change graph data type variables that already exist.

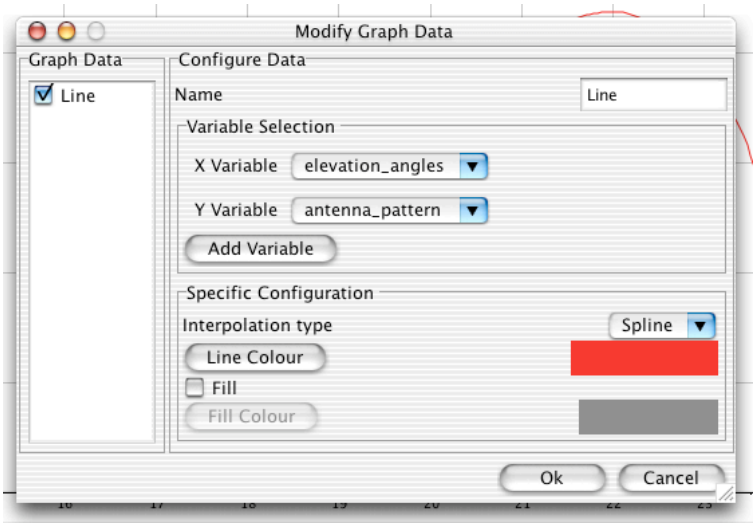


Figure 6.13 Modify Graph Data Dialog

The *ModifyGraph Data* dialog, shown in

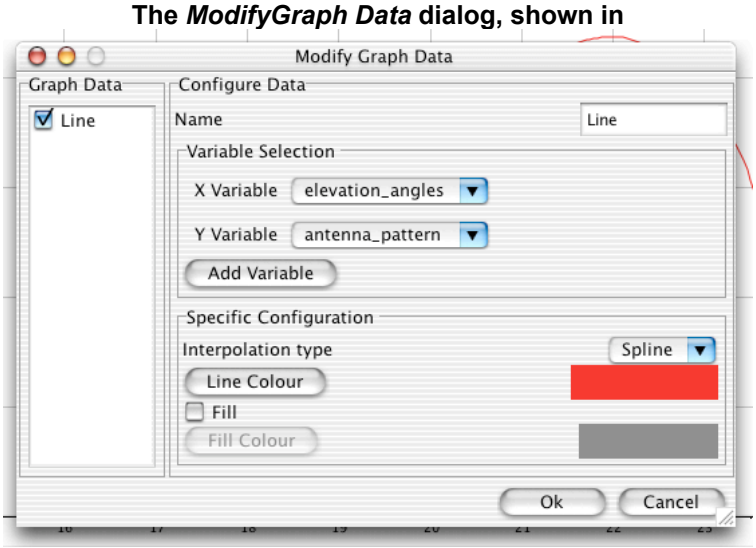


Figure 6.13 above, provides the user with the following options:

- A list of all the available graph data type variables is displayed on the left hand side of the dialog box. Variables already displayed in the graph are marked with a tick. The variables can be removed from the graph by clicking on them until their tick is removed. To change the configuration of any of the variables on the list, simply click on them until they become highlighted. Then make the necessary changes using the available options discussed below.

- All graph data sets are given a default name by which they can be identified. The *name* text field allows the user to rename the graph data set to something more appropriate or preferential.

- Variable selectors for each axis or plane allow the user to select from a list of preconfigured variables. Note the difference between a variable and graph type variable as discussed in section 6.3.1.

- The *Get New Variable* button brings up the *Add Variable Dialog* (see section 6.3.6) and allows the user to choose new variables not included in the preconfigured variables list.

- Configurable options specific to certain graph types (see section 6.3.8) are displayed at the bottom of the dialog box.

- The *Delete* button deletes the selected item from the list of available graph data variables.

- The *Update* button allows changes to be confirmed before moving on to make further changes to other graph data sets on the graph variable list.

- The *OK* button confirms the changes and closes the dialog.

- The *Cancel* button ignores the changes and closes the dialog.

6.3.6 The Add Variable Dialog

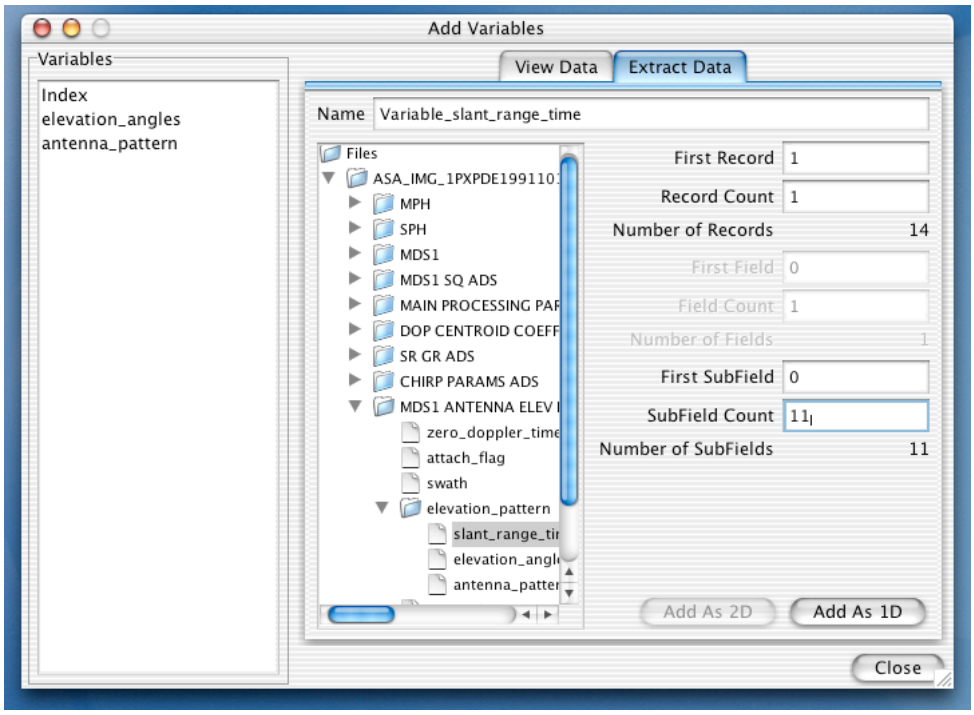


Figure 6.14 Add Variable Dialog

The *Add Variable* dialog, shown in

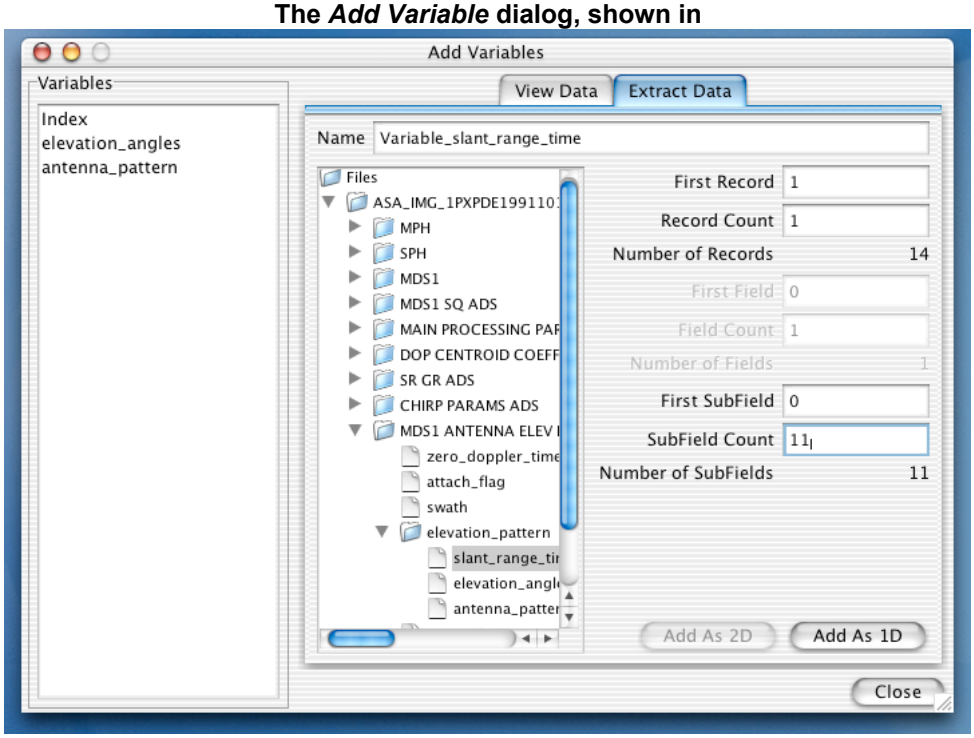


Figure 6.14 above, provides the user with the following options:

- A list of preconfigured variables, displayed on the left, allows the user to select between the variables. Upon selecting a particular variable on the list, a *View Data* panel is brought up displaying the data associated with the variable selected.

- The tabbed panel provides the user with three options:
  - A *View Data* option, which as discussed above, simply displays the data associated with a selected variable.
  - An *Extract Data* option, which is referred to as the *Pds Variable Extraction Panel*, is discussed further in the next section
  - A *Combine Data* option, which allows the user to transform or combine two or more variables is described in section 6.3.6.2.

6.3.6.1 Pds Variable Extraction Panel

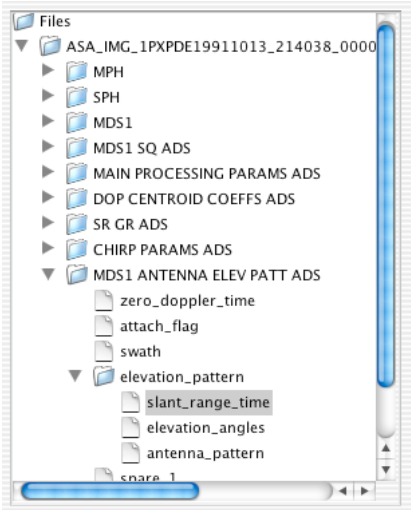


Figure 6.15 Pds Variable Extraction Panel

The *Pds Variable Extraction Panel*, shown in Figure 6.15 above, provides the user with the following options:

- The *element selection tree*, displayed on the left of the panel, shows all Envisat product files that have been opened by the running EnviView session. The selection of an element on the tree results in the selected element being expanded to display its contents of child elements. Hence, the selection of a product results in it expanding to show its list of data set records (DSDs). Similarly, the selection of a DSD results in it expanding to showing its list of fields. In the case of complex fields, they can be further expanded to show a list of their sub fields. When an extractable element on the tree is selected, hence, a field or a sub field, the *Add* button at the bottom becomes enabled.

- The *name* text field is by default set to have the same name as the selected field on the *element selection tree*. The user is able to change this name, but it must be noted that each variable name must be unique.

- The *First Record* and *Record Count* options are both user configurable and by default are set to 1. As its name suggests, the *First Record* option marks the record at which the data extraction should start. The *Record Count* option specifies the number of records to be extracted starting at the *First Record*. Below the two user configurable options is a label showing the total number of records available. The *Adjustable Dimension* check box allows the user to activate the "+" and "-" buttons discussed in section 6.3.3.1.

- The *First Field* and *Field Count* options are both user configurable and by default are set to 1. As its name suggests, the *First Field* option marks the field at which the data extraction should start. The *Field Count* option specifies the number of fields to be extracted starting at the *First Field*. Below the two user configurable options is a label showing the total number of fields available. The *Adjustable Dimension* check box allows the user to activate the "+" and "-" buttons discussed in section 6.3.3.1. Note that these options are only enabled if the element selected on the *element selection tree* is a field or a sub field.

- The *First* Sub*Field* and *SubField Count* options are both user configurable and by default are set to 1. As its name suggests, the *First* Sub*Field* option marks the sub

field at which the data extraction should start. The Sub*Field Count* option specifies the number of sub fields to be extracted starting at the *First* Sub*Field*. Below the two user configurable options is a label showing the total number of sub fields available. The *Adjustable Dimension* check box allows the user to activate the "+" and "-" buttons discussed in section 6.3.3.1. Note that these options are only enabled if the element selected on the *element selection tree* is a field or a sub field.

- The *Add as1D* button extracts the data selected into a one-dimensional array. Note that this button is only enabled if the data selected meet the extraction criteria.

- The *Add as2D* button extracts the data selected into a two-dimensional array. Note that this button is only enabled if the data selected meet the extraction criteria.

6.3.6.2 The Combine Data Panel

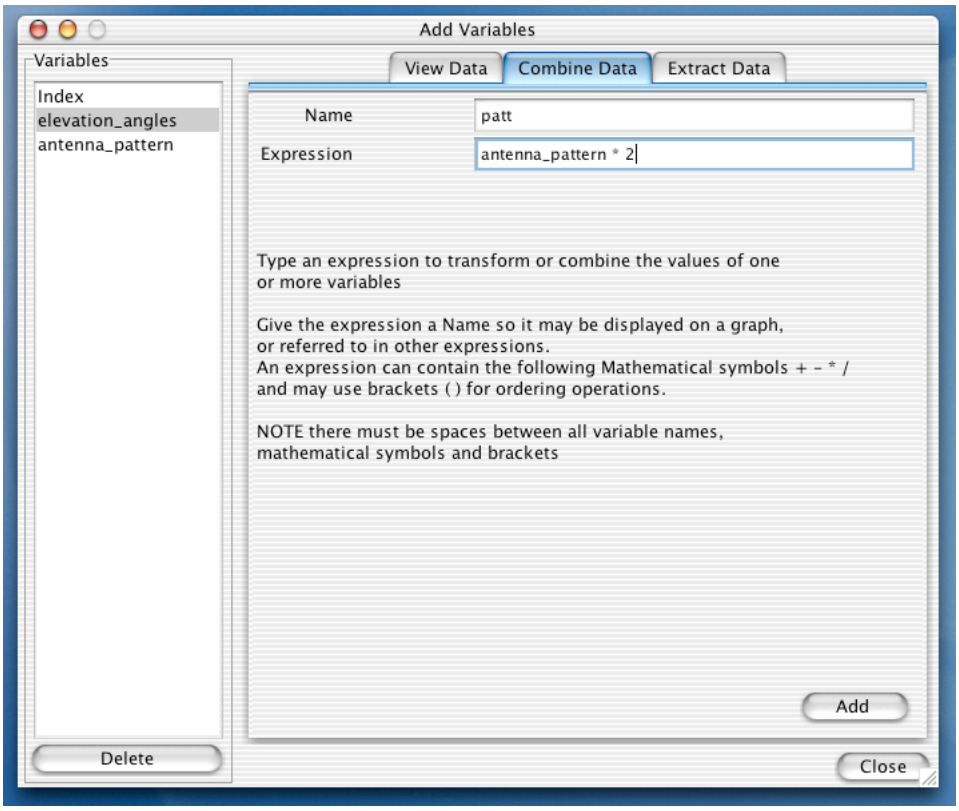


Figure 6.16 Combine Data Panel

The *Combine Data* Panel, shown in

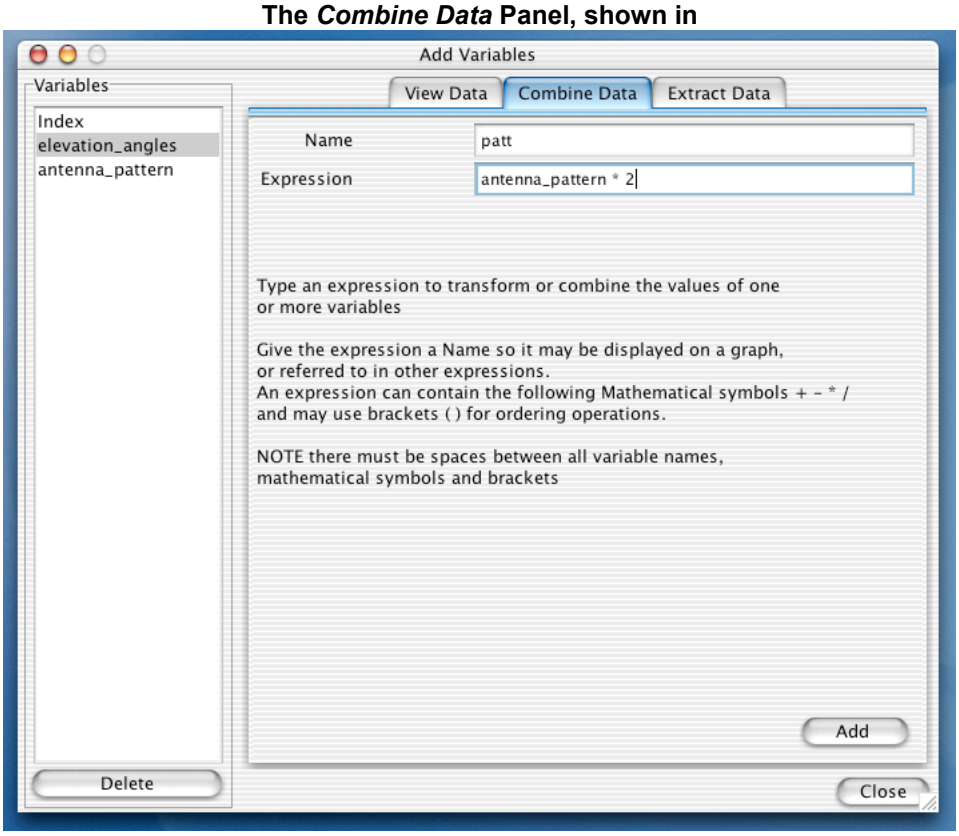


Figure 6.16 above, allows the user to transform or combine two or more variables using the following options:

- The *name* text field allows the user to label their expression, thus allowing the modified or combined data to be identified.
- The *expression* text field allows the user to transform or combine variables using the normal mathematical symbols and brackets for ordering operations. Note that all variable names, mathematical symbols and brackets must be separated by spaces.

6.3.7 How to plot a and extract the data for a graph

- Start EnviView and open a product file.
- Go to the *Plot* menu and select the graph type that you wish to plot e.g. the *2D Line Graph*.
- If the 2D line graph option is selected, the *Plot Window* appears with a blank 2D line graph. Similarly, if other graph types are selected, the *Plot Window* appears with their blank equivalent.
- Click on the *Add/Set Data* button to bring up the *Add/Set Data* dialog box.
- On the *Add/Set Data* dialog, click on the *Add New Variable* button to bring up the *Variable Creation* dialog.

- On the *Pds Variable Extraction Panel*, click on the *element selection tree* to select the fields/sub fields of interest. Then click on the *Add as1D* button to extracts the selected data into a 1D array. Finally, click on the *Close* button to close the *Variable Creation* dialog.

- Now on the *Add/Set Data* dialog, set the X and Y values. For example use the *X Variable* selector to set the X value to index and the *Y Variable* selector to set the Y value to the variable that you have just extracted.

- Click on the *OK* button and the data should now appear plotted on the graph.

- You can now label the X and Y-axes and change the graph title by clicking on the *Configure* button to bring up the *Configure Graph* dialog box.

- Click on the *Modify Data Button* . The *Modify Data Dialog* should appear. Change the colour of the line by clicking on the *Line Colour* button.

- Similarly, other configurable options may be changed by clicking on the *Modify Data* button to bring up the *Modify Data* dialog box.

6.3.8 Configuring each graph type

Each graph type is different and therefore has a slightly different configuration. The graph specific configurations for all the different graph types are described in the sub sections below.

6.3.8.1 The 2D line graph

- The 2D line graph has the ability to zoom into a particular area. By holding the *Shift Key* and pressing the left mouse button an area can be selected to zoom. To zoom out simply hold down the *Shift Key* and clicking on the left mouse button.

- In the *Graph Configuration* dialog there is the *Marker Line Configuration Panel.* Using this configuration panel, a marker line, which is simply a straight line parallel to the X or Y axes, can be set to guide the eye or mark a certain level. The *Marker Line Configuration Panel*, shown in Figure 6.11, has to following configurable options:

  - The *Use* check box allows the user to activate or deactivate the marker line.

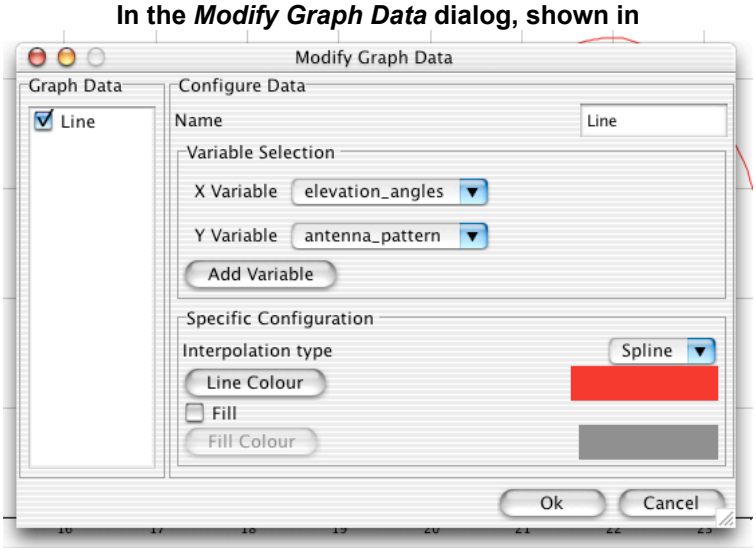
  - The *name* text field allows the marker line to be labelled.

  - The *Orientation Selector* can be set to either horizontal or vertical. Hence, displaying a marker line either parallel to the x-axis or the y-axis.

  - The *Position* option allows the marker line to be offset from the origin.

  - The *Colour* option allows a preferential colour to be selected for the marker line.

In the *Modify Graph Data* dialog, shown in



- Figure 6.13, the configurable options specific to the *2D Line Graph* include:

  - The *Interpolation type* selector can be set to either linear or spline. If set to linear, data points are connected with straight lines. If set to spline, data points are connected with curved lines.

  - The *Line Colour* button allows a preferential line colour to be set.

  - The *Fill Colour* check box can be used to activate or deactivate the fill colour option.

  - The *Fill Colour* button allows a preferential background colour to be selected.

6.3.8.2 The 3D line graph

- The *3D line Graph* allows the plot to be rotated. This can be achieved by clicking and holding the left mouse button and moving the mouse to the angle of rotation required.

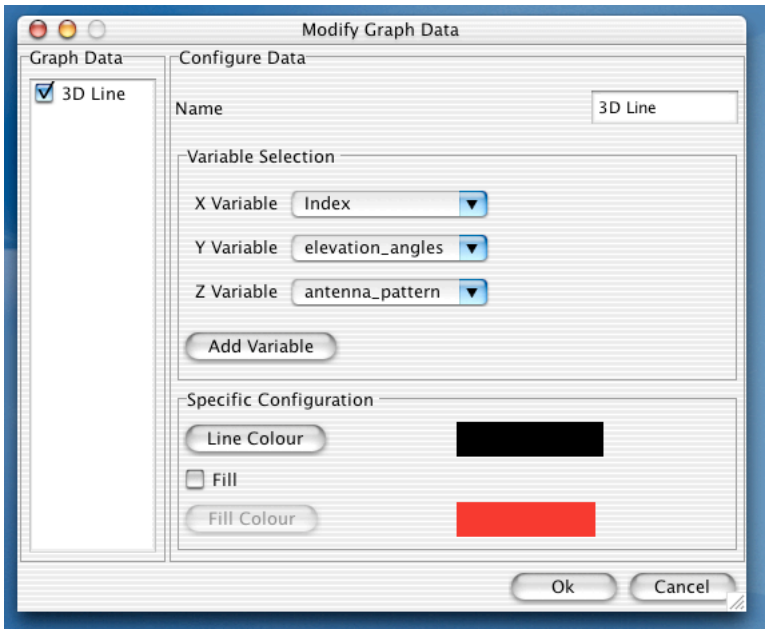
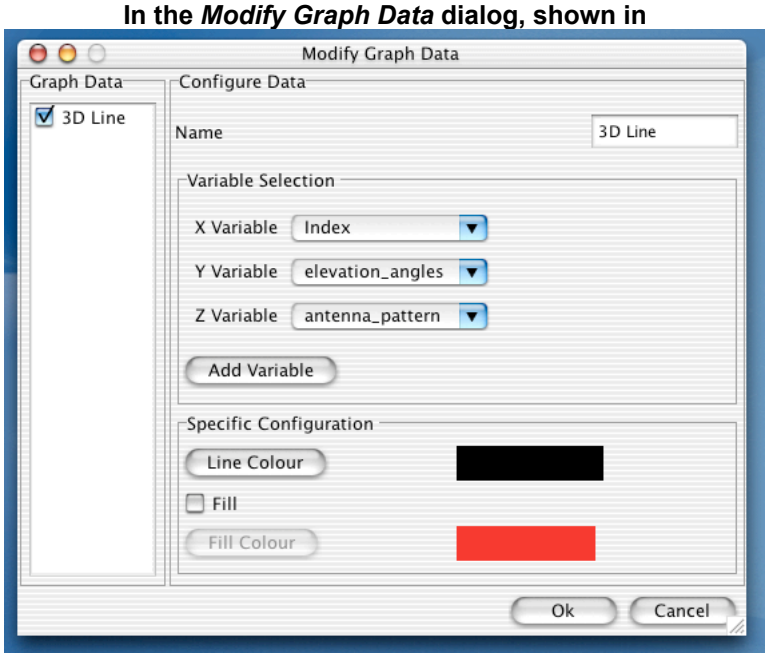


Figure 6.17 Specific panel for 3D Line graph data

In the *Modify Graph Data* dialog, shown in



- Figure 6.17 above, the configurable options specific to the *3D Line Graph* include:

  - The *Line Colour* button allows a preferential line colour to be set.

  - The *Fill Colour* check box can be used to activate or deactivate the fill colour option.

  - The *Fill Colour* button allows a preferential background colour to be selected.

6.3.8.3 The Pie Chart

- The *Pie Chart* has no specific configurable options.

6.3.8.4 The Bar Chart

- The *Bar Chart* has no specific configurable options.

6.3.8.5 The Contour Graph

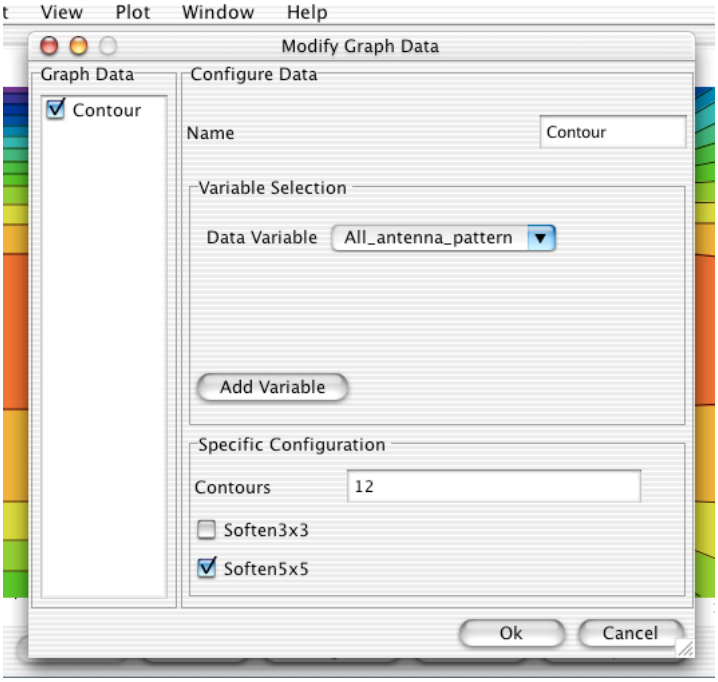
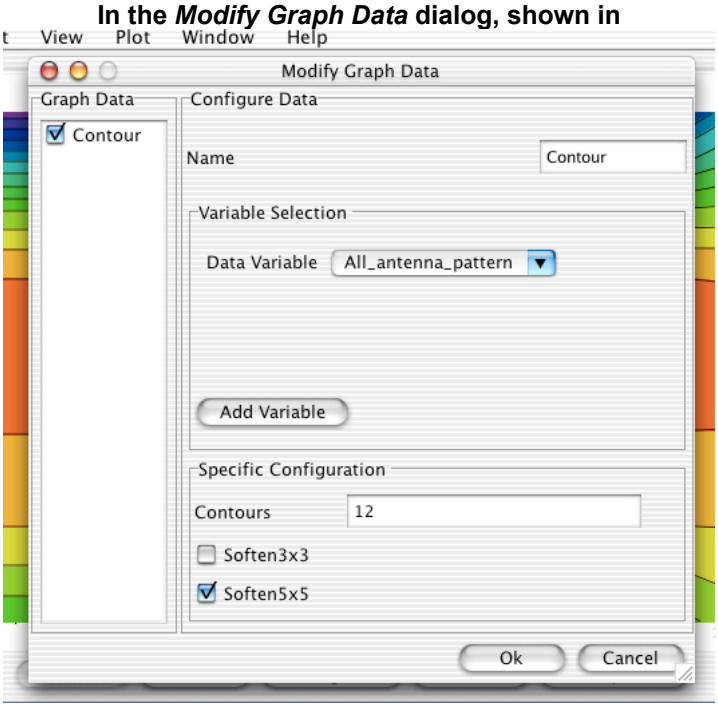


Figure 6.18 Specific panel for contour graph data

In the *Modify Graph Data* dialog, shown in



- Figure 6.18 above, the configurable options specific to the *Contour Graph* include:

  - The *Contour number* option allows the number of contours to be set.

  - The *Palette number* option allows the colour palettes to be set.

- The *Soften 3 x 3* check box can be selected to interpolate the data using a 3 by 3 matrix.
- The *Soften 5 x 5* check box can be selected to interpolate the data using a 5 by 5 matrix.

6.3.8.6 The Density Graph

- The *Bar Chart* has no specific configurable options.

6.3.8.7 The 3D Surface Graph

- The *Bar Chart* has no specific configurable options.

6.4 Saving Graph configurations

To save a graph which has been created in the Plot Window. Make sure that the graph to be saved is top graph in the Plot Window. From the Plot Menu in the Plot Window select the 'Add to Plots' menu item.

A Save Dialog box appears asking for a menu item name to be inputted. This name is the name of the menu item on the Plot menu which the graph will be placed under. If the name is left blank the graph will appear directly under the Plot menu. This allows the user to group together graphs or a similar nature.

If this graph has been saved before the current group menu will be displayed in the dialog box.

The text on the menu item, which represents the graph, will be the title of the graph.

Pressing the 'Save' button will save the graph, and the dialog will close.

If you are saving a graph which has been saved before, when the 'Save' button is pressed a second dialog box appears asking whether to overwrite the existing graph or save it as a new graph.

Once the graph has been saved it will appear on the Plot menus. On the Plot menu for the Plot Window, there is a sub menu for each open product and the graph will appear there. On the Product Window, just the graphs for that product are displayed there.

N.B. graphs that have data on from more than one product can not be saved.

6.4.1 Loading a Saved Graph

To load a saved graph, make sure that the product that contains the data for the graph is open.

To Load from the Product Window

From the Plot menu in the Product Window select the graph to be loaded. If the Plot Window is not open, it will be opened and the loaded graph displayed. If the plot Window is open, it will be brought to the front and the graph displayed.

To Load from the Plot Window

From the Plot Menu, select the product to which the graph belongs. A Sub menu will be displayed containing all the saved graphs for this product. Select the graph to be loaded and the graph will be displayed.

6.4.2 Deleting a Saved Graph

To delete a saved graph, open the graph and make sure this is the top graph in the Plot Window. Go to the Plot window and select 'Remove From Plots'. A confirmation dialog box is displayed. Press Yes to delete the graph.

Only user saved graphs can be deleted. No graphs provided with the EnviView installation may be deleted.

7. VIEWING ENVISAT IMAGE DATA

Envisat has three imaging instruments, AATSR, ASAR and MERIS, each producing a number of different data products containing image data sets, i.e. containing arrays of image pixel scan lines. If an open product contains an image data set, the *New Image View* option from the *View menu* is available (otherwise it is greyed out). Select this menu item to open a new image view.

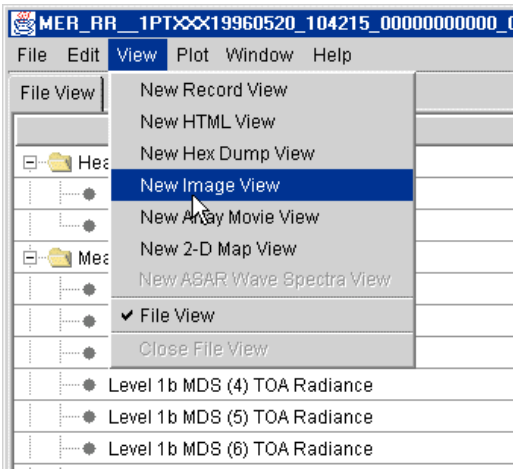


Figure 7.1 Starting a new image view

By default, the first image measurement data set is displayed in the image viewer. Select the data set required from the data set pop up menu as shown in Figure 7.2.

Note that the AATSR level 2 averaged surface temperature product (AST_AR_2P) does not contain image scan lines, and hence can not be displayed in the image viewer. It provides a set of geolocated sample points, which may be displayed using the 2-D map view (see Chapter 9 for details).

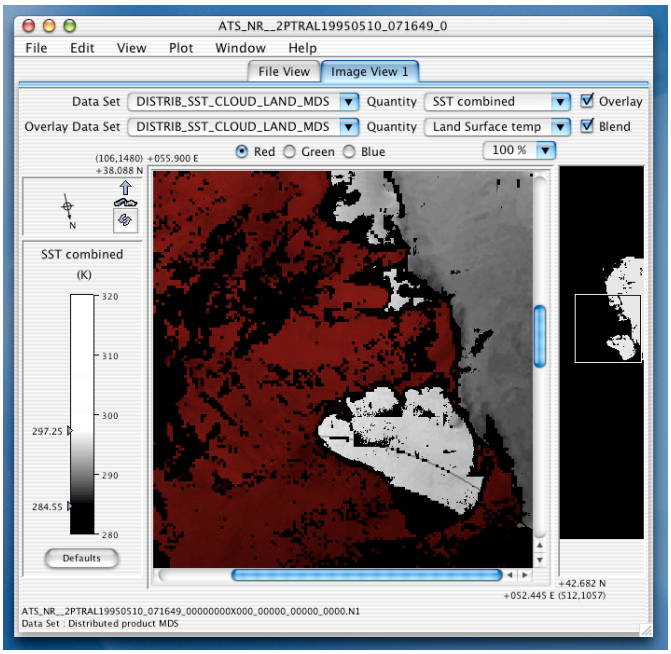


Figure 7.2 Image Viewer Window

7.1 The Image Viewer

An Image Viewer contains a sub-region of the image data set at full resolution. On the right hand side a thumbnail overview of the entire image can be seen. The overview image is built up automatically beginning as soon as the data set as been selected.

For best performance, only use files copied to ones local disc. Accessing image files from a CD-ROM or over the network results in a significantly impaired user experience.

The Image Data Set View consists of the following elements

- Image Viewer at the centre
- Image Overview at the right
  - Initially blank, the overview automatically builds up completely every time a new data set is loaded. Changes to the colour intensity using the colour bar are shown on the overview as well as the main image.
  - Image View Location - a black and white square representing the position of the Image Viewer window in the entire image.
- Image Viewer Co-ordinates - at the top left and bottom right of the Image Viewer
  - The pixel co-ordinates of the top-left and bottom right corner are displayed
  - The approximate geographic co-ordinates of the top-left and bottom right pixels are displayed, If a geographic tie point data set is available in the product.

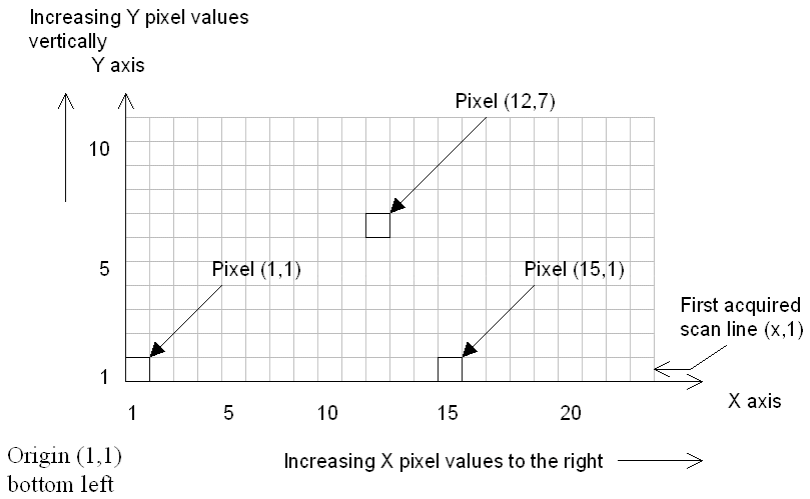


Figure 7.3 Image pixel co-ordinate system

- The image viewer magnification menu. This has several options, the meanings of which are:-
  - 100% - one screen pixel per image pixel (default).
  - 200% - four screen pixels per one image pixel.
  - 400% - sixteen screen pixels for one image pixel.

- A colour bar and scale to the left hand side. This has two arrows on the left, that can be moved to adjust the image contrast. On the right of the colour bar is a numbered range, which can be used to adjust the range of pixel values mapped to pixel intensity.

- A compass above the colour bar shows the direction of North with respect to the image display.

- A rotation toggle button, allowing the image to be rotated by 180º. By default, EnviView rotates an image so that North is always displayed at the top when an image is opened. The button is useful for rotating images that span the North or South pole. Note only the central image view is rotated, the overview remains in product scan line order.

- A data set selector pop up menu allows easy selection of another measurement data set, within the product, to be displayed.

- A quantity selector pop up menu allows one of multiple quantities that may be available in an image data set.

- An overlay button, allowing a second data set or quantity to be overlaid on top of the first. The colour channel of the overlay image may be either Red, Green or Blue, selected through three small buttons below the overlay data set selector.

- A working indicator “barber pole” is displayed below the main image every time a new image area is selected whilst loading of the image takes place. Once the loading and display is completed, the working indicator disappears.

- Specific pixel information and statistics are displayed in a popup-window.

7.2 Moving around the Image

The Image View can be moved around the image using the scrollbars, the keyboard cursor keys or the mouse. To use the keyboard, ensure that either the Image View or Image Overview has the input focus. A simple click of the mouse in either of these views will restore the focus if it has been lost to another component. Movement with either the mouse or the keyboard is very responsive if the image pixels needed to fill the screen have already been loaded. Otherwise, a small delay is incurred as the new pixels are read from the data set. In order to keep the memory requirements to a minimum, pixel data will be discarded when moving to other areas of the image.

There are various ways to move the Image and control the distance moved to the required precision using the keyboard.

- The cursor keys + CONTROL move the Image View by 1 pixel.

Ctrl

↑

← 1 pixel →

↓

- The cursor keys alone move the Image View by 10 pixels.

↑

← 10 pixels →

↓

- The cursor keys + CONTROL + SHIFT move the Image View by 20 pixels.

Ctrl ⇧ Shift

↑

← 20 pixels →

↓

- The cursor keys + SHIFT move the Image View by 200 pixels.

⇧ Shift

↑

← 200 pixels →

↓

The mouse can also be used to move the Image View by clicking in the square marking the position in the Image overview. The mouse button is held down and the square dragged to the new location, see Figure 7.2. After a short delay as the image data is loaded and the Image window is updated.

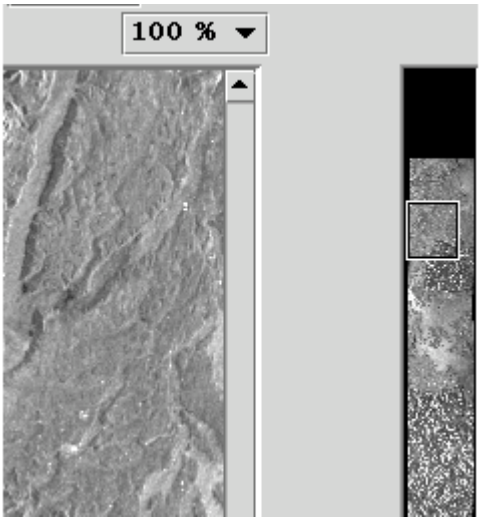


Figure 7.4 Overview Window

Horizontal and vertical scrollbars indicate the position of the image view in the larger image area. In addition to the overview and keyboard, the scrollbars may also be used to navigate around the image area.

While the image data is being read from disk, scaled and prepared for display, a barber pole 'busy' indicator is displayed at the bottom right hand corner of the window.

7.3 The Colour Bar

Images are displayed on screen using a 256 colour look-up table, from black to white (an exception are images stored in RGB format). The Colour Bar indicates and controls the mapping from pixel values to intensity (or shades) on screen.

A pixel value axis determines the range of pixel values that are mapped to the colour table. Markers on the colour bar allow the colour look up-table to be adjusted.

By manipulating the colour bar in the Image Data Set Viewer, the contrast and colour range may be adjusted to a certain degree.

The colour bar consists of the following elements (see Figure 7.5)

- The pixel value axis, on the right.
  This determines the range of the pixel values that correspond to the colour scale.

- Two adjustable colour markers.
  The colours of intermediate pixel values are interpolated between markers.

- The pixel value at each marker on the left.

- A button to reset the default marker positions and axis range.

Clicking on a marker and dragging the mouse up or down moves the colour marker to higher or lower pixel values. Releasing the mouse button updates the colour look-up table, and the image changes immediately.

Clicking on the right of the colour bar brings up a colour scale axis dialog. This allows one to modify the range of pixel values that are mapped onto the colour scale. Changes to the colour scale result in a short pause, as the new image colours are recomputed for the new data range.

The Defaults button returns the markers to their default positions. A triple click on the colour bar itself will also reset the defaults.

NOTE : MERIS and AATSR browse images contain RGB pixel data, so no colour bar is provided.

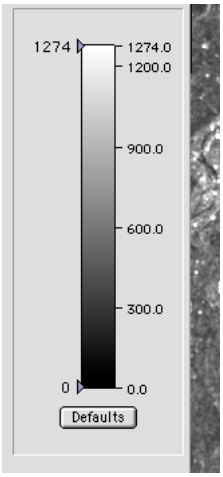


Figure 7.5 Colour Bar

7.4 Magnifying the image display (zoom)

A popup menu at the top right hand corner of the central image allows the display to be magnified by 2x or 4x (200 % or 400 %). Coordinates are adjusted accordingly.

Some high resolution images benefit from a zoom-out. By default EnviView does not provide zoom out options, as this requires significant memory and processor performance to provide an acceptable user experience. Zoom-out may be enabled by selecting a checkbox in the preference panel EnviView -> ImageView.

7.5 Geographical Co-ordinates

Most image data sets are associated with a geographical tie point data set. A tie point data set associates an approximation of a geographical co-ordinate with a given pixel in the image. EnviView is capable of using this information to estimate the geographical location of any pixel on the image using linear interpolation.

7.6 Pixel quantities

Many Image data sets contain multiple quantities, using various different methods of pixel and quantity interleaving. EnviView includes the capability to decode these data set specific quantities, and apply any quantity scaling and offset that might be required.

The quantity is selected from the quantity popup menu on the right side of the image viewer. The values given in the popup menu and the colour scale adjust accordingly. Each quantity has it's own colour scale and units.

7.7 Overlaying Images

Two image quantities, from one or two data sets may be overlaid on top of each other. This may be useful when comparing image data over the same area measured in different frequency bands or polarisations. The overlay may also be useful for displaying a per pixel flag in an associated data set (AATSR and MERIS).

An overlay image may either be blended onto the base image, using a single colour chanel (red, green or blue), or mixed with the base image, replacing one colour channel.

When blending, the base image remains grey scale. When mixing, the overlay image takes one colour channel, and the base image takes the remaining two. For instance, if the overlay image is displayed in Red, the base image would be displayed as cyan (Blue+Green).

The colour scale at the left of the image affects only the base image.

7.8 Image Statistics

Clicking on a pixel and keeping the button pressed in the Image Viewer brings up a popup window detailing the image pixel co-ordinates and approximate geographic location (if a tie point data set is available) and the image pixel value and units. If an overlay image is selected, the values of both the base image and overlay image pixels are provided.

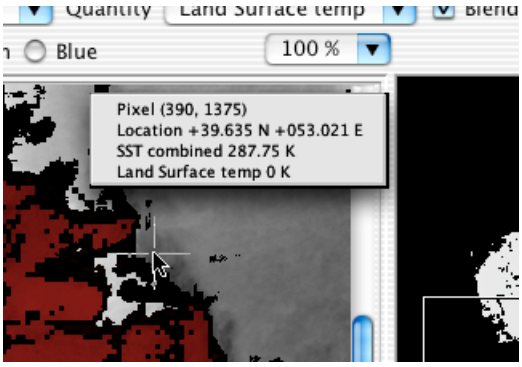


Figure 7.6 Image pixel pop-up window

A large cross for identification marks the pixel. This function is typically used at 200 % or 400% magnification because it is easier to identify individual pixels of interest. Dragging the mouse selects an area of the image.

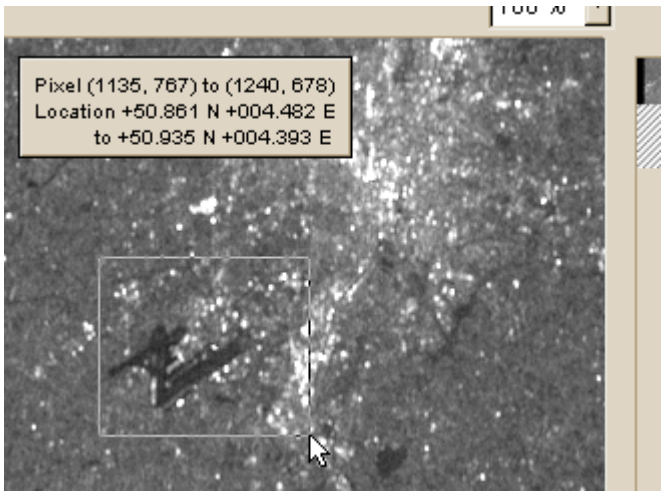


Figure 7.7 Selecting an area of an Image

The pixel and geographic co-ordinates (if a tie point data set is available) of the top left and bottom right corners of the area are updated in the dialog as the mouse is dragged. Releasing the mouse button produces an Image Selection window, Figure 7.8.

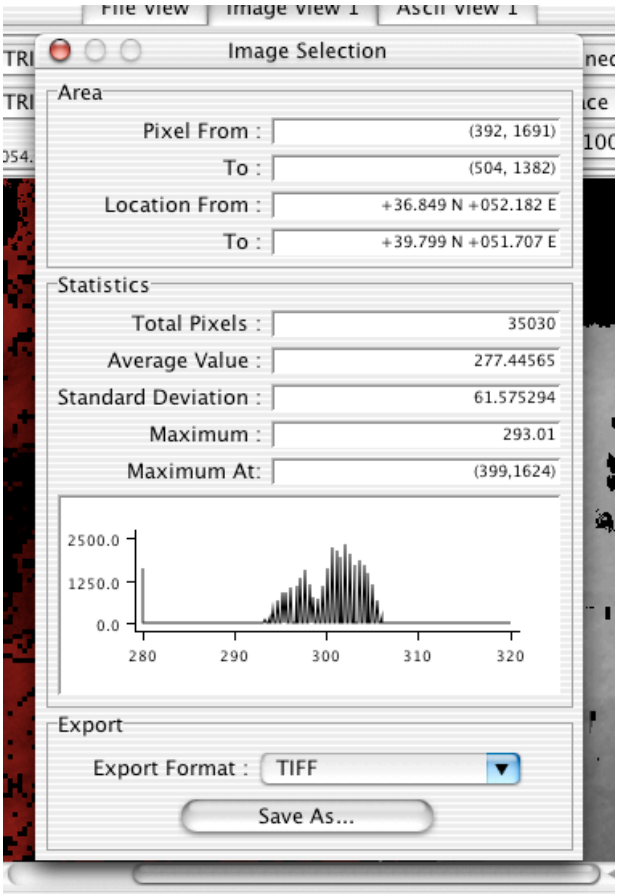


Figure 7.8 Image Selection window

The selection window is split into three parts

- **Area** : which repeats the top-left and bottom right pixel location, and geographic co-ordinates (if a tie point data set is available).
- **Statistics** : which presents a few simple statistics about the image data in the selected area :
  - Total number of pixels in the area.
  - The average pixel value.
  - The standard deviation of the pixel data values.
  - The maximum pixel value and its location.
  - A histogram of the pixels in the selected image area

  When a large area is selected, it may take a short time before the computations are complete. The statistics fields contain a '-' until the computation is complete.
- Export : Save the pixels of the selected area to a file. Available formats include TIFF, GIF, PPM, Tab separated text and raw pixel values. See section 11.1 for more information.

Close the dialog by clicking the window close button in the window frame.

8. VIEWING WAVE SPECTRA

A wave spectrum viewer displays a false colour polar plot showing level 1B or level 2 ASAR wave spectra data. As the Level 1B data values are complex numbers, having both real and imaginary parts, various representations of the spectra are possible.

Data is displayed one record at a time. Buttons are provided for selecting a specific record, or stepping backwards or forward by one record. The evolution of wave measurements may be evaluated by viewing subsequent spectra as a movie, automatically stepping through the product , record by record.

To open an ASAR Wave view, select the *New ASAR Wave Spectra View,* from the *View menu* as shown in Figure 8.1. This option is only enabled if an appropriate data product is open (ASA_WVI_1P, ASA_WVS_1P, ASA_WVW_2P).

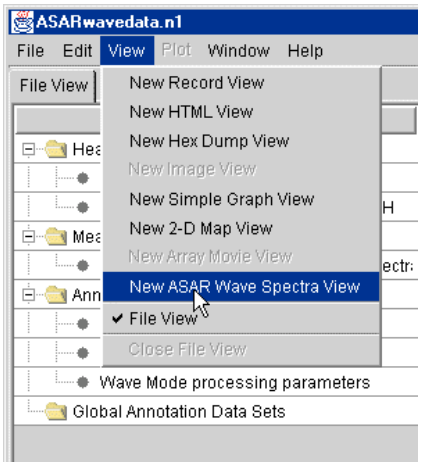


Figure 8.1 Starting an ASAR Wave Spectra View

An ASAR Wave Spectrum viewer will open similar to that shown in Figure 8.2.

8.1 Feature Summary

Figure 8.2 shows an example of an ASAR wave spectrum viewer, displaying a level 1B cross spectrum. It contains a control area at the top, below which is the graph, information and identification area. The spectra data itself are displayed on a polar graph plot in the middle of the image (with the 50(m), 100(m) and 200(m) radii).

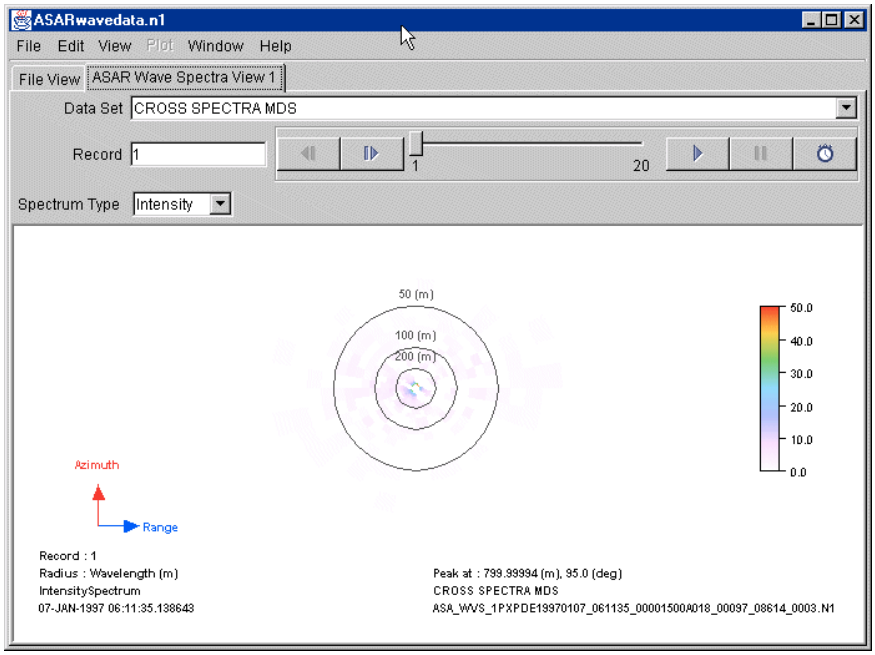


Figure 8.2 ASAR Wave Spectra Viewer

The characteristics and features of the ASAR wave data are summarised below, starting at the top of the window and moving downwards.

- Menu bar and Menus
  These are identical to those on other EnviView windows.

- View tabs
  These select other open views with a single mouse click.

- Data Set selector
  This displays the name of the current wave data set, in the data product, and allows selection of other wave data data-sets, if available. Nominally there is only one wave spectra data set in a product.

- Record Selector Controls and indicators.
  These consist of

  - a Record field, which indicates the currently displayed record number, and allows a particular record to be accessed directly, by typing its index.

  - nudge back and forward buttons allows progress through the data set, one record at a time

  - a record slide control which allows fast access directly to a particular part of the data set.

- Movie Controls.
  The movie controls are to the right of the record slider. They consist of a play button, a pause button and a movie speed control.

- Spectrum type selector (level 1B products only).
  This allows selection of various views of the spectra data. See section 8.7 for more details.

- Graph Area
  The graph area shows a wave spectrum polar plot. See section 8.8 for more details. The Orientation axes show information about the along track and cross track directions (level 1B products) or North and East (level 2 products). The colour scale, to the right of the polar plot shows how values are converted to colours on the plot. The scale can also be entered, see section 8.5 for more information about this.

- Identification and Information Area.
  This area, at the bottom of the window, shows orientation information (see section 8.8.2), identification information for the data product, data set and record number, the type of spectrum displayed and the data on which the data was acquired. Also the magnitude and position of the peak point of the spectra is stated.

8.2 Selecting a data set to view

The data set pop-up menu, towards the top of the window, contains a list of available data sets for the product. It displays the name of the data set that is currently being viewed. To select a different data set (if available), click and open the pop-up menu and select the data set.

8.3 Selecting Records

When an ASAR wave spectrum viewer is opened, data from the first record is displayed by default. There are various ways of navigating to other records. The controls available are shown in Figure 8.3.

Figure 8.3 Record Navigation controls

These controls enable one to go;

- directly to a specific record,

- back or forward one record at a time

- and browse through the records.

The details of how to carry out these operations are below;

The *Record* field at the left indicates which record is currently being displayed. To go directly to a specific numbered record, select the *Record* field and type in the record number. Remember to press enter. EnviView will display data from the record required. Note the maximum number of records is displayed under the right hand edge of the record slider.

To move one record at a time through the data set press the back and forward “nudge” buttons ( ) located between the Record field and the record slider.

To slide through the data set; use the record slider. Click and drag the slider control to the approximate position in the data set required.

8.4 Playing a movie

The EnviView ASAR wave spectrum viewer can display one record at a time in the forward direction, with a controllable interval between each frame, just like a movie. The controls to do this are shown in Figure 8.4.

Figure 8.4 Movie Controls

To start a movie, drag the record slider to the left to the point at which to start the movie, then press the play button ( ).

To stop the movie, press the pause button ( ). The movie will stop automatically at the last record.

The speed control button ( ) activates a menu that enables control of the interval between frames of the movie of between 1/10 second and 2 second. Shorter intervals between frames mean that the movie plays faster, and longer frame intervals mean that the movie will play more slowly. To alter the movie speed, press the speed control button and select the frame interval from the pop-up menu. The speed can be changed interactively whilst a movie is running.

8.4.1 Rescaling behaviour

Note that the default settings mean that the plots are rescaled every movie frame. To make it easier to follow the progression of data through the movie, set the scaling to manual, and fix the colour scale range to appropriate values. See section 8.5 for details about how to do this.

8.5 Changing the spectra display

Level 1B spectra can be displayed in a number of different ways, more details can be found in section 8.7. To change the type of spectra; select the *Spectrum type* pop-up menu and choose the type of spectrum to display from the list displayed. The Intensity spectrum is the default displayed. Resizing the ASAR wave spectrum viewer window changes the size of the plot on the screen.

The colour scale of the spectrum plot can be modified, to emphasise certain parts of the field range, if required. Double click on the colour bar (on the right hand side of the window) to change the mapping of colours to field values (the colour scale). An “Axis parameter” dialogue box will up similar to that shown in Figure 8.5 .

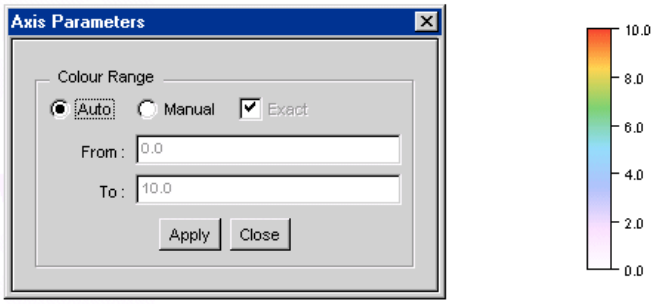


Figure 8.5 Changing the Colour scale

If the **Auto** radio button is checked EnviView will fit the spectrum data to maximise the use of the colour scale.

Select the **Manual** radio button to enter a specific range of values for the colour scale. Then type the lower limit into the **From:** text field and the upper limit into the **To:** text field. Press the **Apply** button to implement the changes. Any data outside the range entered will be displayed as red, if it is greater than the upper limit, or white, if it is less than the lower limit. The manual setting is useful for displaying movies as it prevents the plot values from being rescaled every frame.

Note; if the **Exact** check box is left unchecked EnviView may round the axis limit values entered.

8.6 Getting specific bin information

Detailed information about a specific spectral bin on the image can be obtained simply by clicking on it. A pop-up panel shows the bin index number, the wavelength and direction of the bin, and the plotted value. Figure 8.6 shows an example of detailed spectral bin information by clicking on bin (7,7).

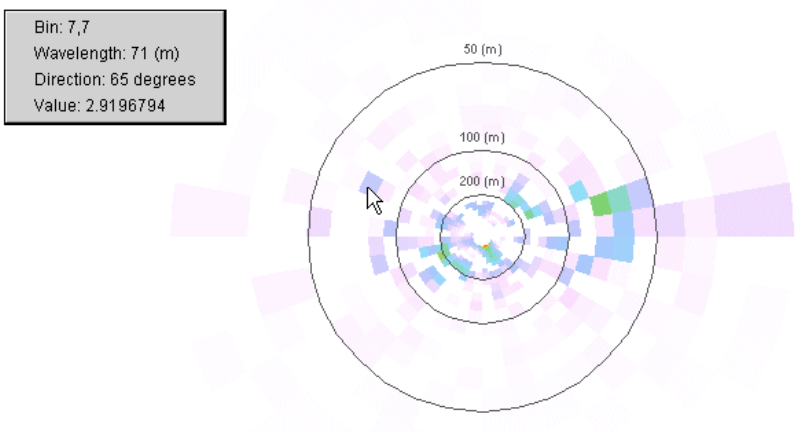


Figure 8.6 Detailed spectral bin information

By clicking and dragging the mouse over the spectra it is possible to interactively see the detailed information for each bin that the mouse pointer passes over.

8.7 ASAR Wave Spectra display types & characteristics

The ASAR level 1B wave spectrum data is complex, containing both real and imaginary parts, as stated in the introduction. EnviView only displays scalar (1 dimensional) values on the ASAR wave spectrum viewer, which means that it can only display either the real parts, the imaginary parts or some scalar combination of the real and imaginary parts.

ASAR Level 2 data is not complex, hence there is no need selection provided for different display types when viewing such a data set. Level 2 data is neither symmetric nor anti-symmetric, and contains information for the entire 360º spectrum.

8.7.1 Level 1B Spectra symmetries

The level 1B wave spectra are 180° (half turn), rotationally symmetric (real part) or anti-symmetric (imaginary part), about an axis through the centre of the plot. This means that the data shown on a complete spectrum (360°) can be redundant i.e. the left hand side of the plot gives the same information as the right hand side of the plot. In fact, only the first 180 degrees of data are available for both the real and imaginary parts of the spectra. The second half (180-360 degrees) is synthesised, by EnviView, using the symmetry properties of the data.

Figure 8.7 and Figure 8.8 show examples of images that are half turn rotationally symmetric and anti-symmetric. Note, in both cases, that image data is redundant. Given any half of either image, the other half can be constructed from it (as is the case for wave spectra real and imaginary spectra).

Figure 8.7: Half turn rotationally symmetric image

Real spectra are half turn rotationally symmetric.

Figure 8.8 Half turn rotationally Anti-symmetric image

Imaginary spectra are half turn rotationally anti-symmetric.

8.7.2 List of level 1B spectra types

The types of Level 1B spectra display available in the ASAR wave spectrum viewer are as follows;

- **Real Spectrum**

  Only the real part of the spectrum is plotted. Note that the real spectrum is symmetric (see Figure 8.7).

- **Imaginary Spectrum**

  Only the imaginary part of the spectrum is plotted. Note that the imaginary spectrum is anti-symmetric (Figure 8.8)

- **Both Spectra**

  Both half the real and half the imaginary spectra are plotted, with the real on the left and the imaginary on the right.

- **Power Spectrum**

  In power spectra, the values are computed as:

  $$real^2 + imaginary^2$$

  NB. If the real and imaginary are out of phase (i.e. real / imaginary < 0) then the corresponding bin value is set to zero. This helps remove the wave direction ambiguity from the plot.

- **Intensity Spectrum**

  This is the default plot. In intensity spectra, values are computed as:

  $$(real^2 + imaginary^2)$$

  NB. If the real and imaginary are out of phase (i.e. real / imaginary < 0) then the corresponding bin value is set to zero. This helps remove the wave direction ambiguity from the plot.

- **Multiplied Spectrum**

  In multiplied spectrum, values are computed as:

  real * imaginary

  NB. If the real and imaginary are out of phase (i.e. real / imaginary < 0) then the corresponding bin value is set to zero. This helps remove the wave direction ambiguity from the plot.

8.8 The ASAR Wave spectra display characteristics

ASAR wave spectra are encoded in a polar grid in the measurement data set. While visually similar, there are a number of differences in the encoding and storage order that must be taken account of when reconstituting the spectral values. This section describes the particular characteristics of Level 1B and Level 2 ASAR spectta data.

8.8.1 Level 1B spectra characteristics

Figure 8.9 shows an example of a Level 1B ASAR wave spectra. The image is a polar plot of ASAR wave spectrum data placed into discrete bins. Each wave spectra image is composed of 864 bins arranged in a polar plot.

- Level 1B data is organised in the ***anti-clockwise*** direction
- Data is stored in the product in sectors, from the longest to the shortest wavelength.
- The first bin (1,1) is at the centre (from –5° to 5°) of image.
- The last bin of the first sector (1,24) is at the top (from –5° to 5°) of image.
- Each bin has a real and an imaginary value

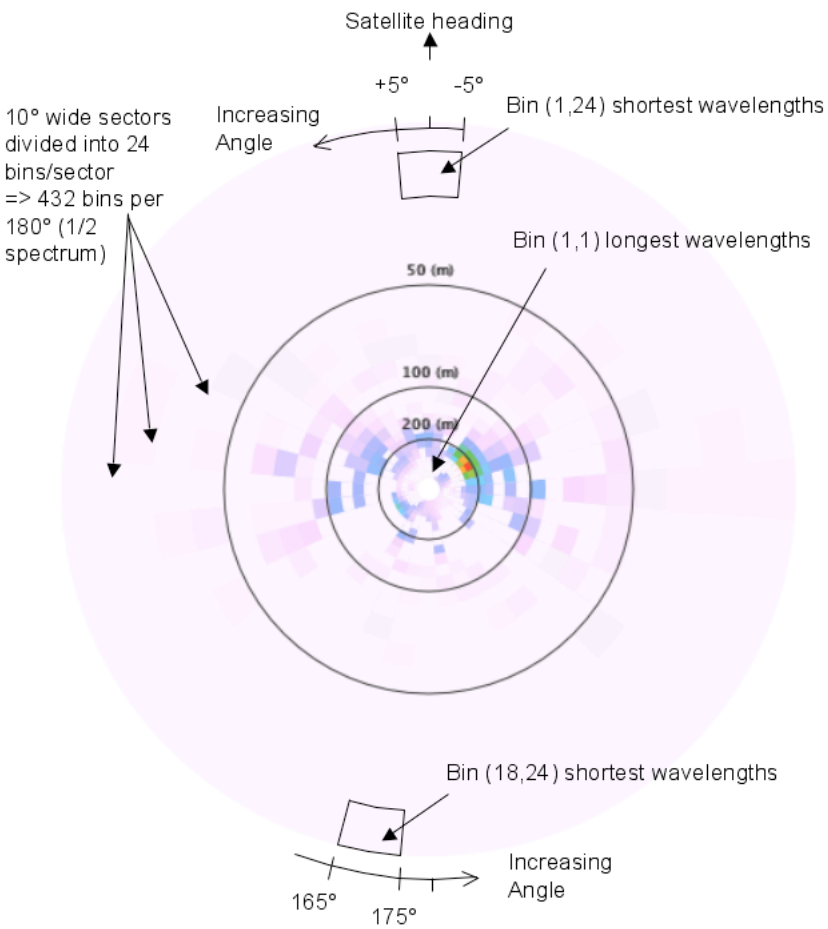


Figure 8.9 Level 1B ASAR Wave spectra image

- The spectrum image is divided up into 36 sectors or “spokes”, each 10° wide. Note that due to the symmetry of the spectra, only data from the first 180° (anti-clockwise) of the spectra are provided i.e. sectors 1 to 18. The other half of the spectrum display (sectors 19-36) can be constructed from the first half as the real part of the spectrum is symmetric and the imaginary part is anti-symmetric.

- Each radial sector or “spoke” contains 24 bins numbered from 1, at the centre of the spectra display to 24 at the outside edge

Each bin is colour coded based on the value of the bin data mapped onto a colour scale shown to the right hand side of the image

8.8.2 Level 1B Spectra Orientation

The direction origin (0°) is at the top of the Level 1B spectrum, and the angle increases ***anti-clockwise***, reaching 180° at the bottom of the display continuing back to 360°. The satellite heading (the along track direction) is at 0°. The orientation axes (see Figure 8.10) show the orientation of the plot.

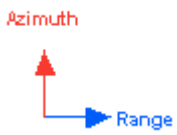


Figure 8.10 Orientation Information

The arrow, labelled “Azimuth”, shows the along track satellite direction which is always parallel to the 0° direction of the spectra image. The arrow, labelled “Range”, shows the satellite cross track direction. This is always orthogonal (i.e. at right angles to) the Azimuth direction.

8.8.3 Level 2 spectra characteristics

Figure 8.11 shows an example of a Level 2 ASAR wave spectra. The image is a polar plot of ASAR wave spectrum data placed into discrete bins. Each wave spectra image is composed of 864 bins arranged in a polar plot.

- Level 2 data is organised in the ***clockwise*** direction

- Data is stored in the product in sectors, from the longest to the shortest wavelength.

- The first bin (1,1) is at the centre (from –5° to 5°) of image.

- The last bin of the first sector (1,24) is at the top (from –5° to 5°) of image.

- The spectrum image is divided up into 36 sectors or “spokes”, each 10° wide. Unlike the level 1B product, all sectors for the full 360° (clockwise) of the spectra are provided i.e. sectors 1 to 36.

- Each radial sector or “spoke” contains 24 bins numbered from 1, at the centre of the spectra display to 24 at the outside edge

- Each bin has a single real value

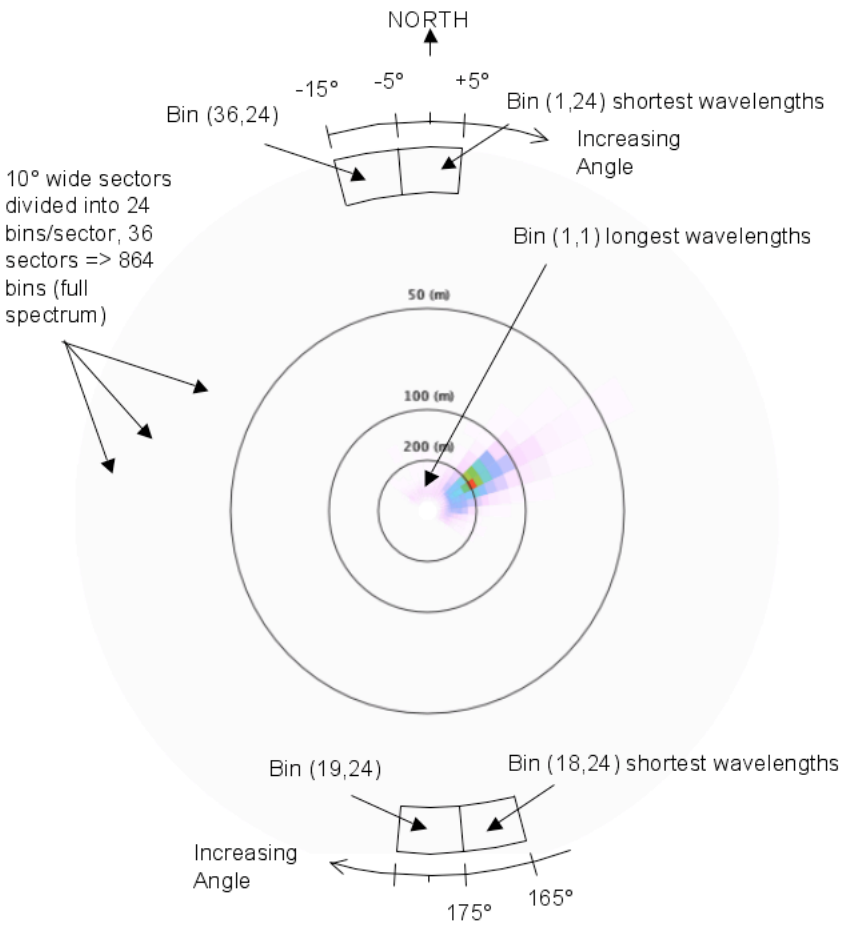


Figure 8.11 Level 2 ASAR Ocean spectra image

Each bin is colour coded based on the value of the bin data mapped onto a colour scale shown to the right hand side of the image

8.8.4 Level 2 Spectra Orientation

The direction origin (0°) is at the top of spectrum image and angle increases ***clockwise***, reaching 180° at the bottom of the display continuing back to 360°. For Level 2 data, North is at 0° and East at 90° The orientation axes indicate the orientation of the image.

9. VIEWING 2-D MAP DATA

A 2-D map view is similar to a simple graph view, except the x-axis is longitude, the y-axis is latitude and a world-map (coastline) is drawn in the background. Points may be plotted on the graph, and the colour of the point may be controlled by a third parameter. Only fields from data sets that have geographical "tie point" information may be plotted in the view.

It is possible to zoom in to view particular regions and select the field values to plot from a list of those available.

9.1 Quick Start

Open a level 1b or level 2 data product. The *File View* lists the available data sets. From the *View* menu, select the *New 2-D Map View* menu item (as shown in Figure 9.1).

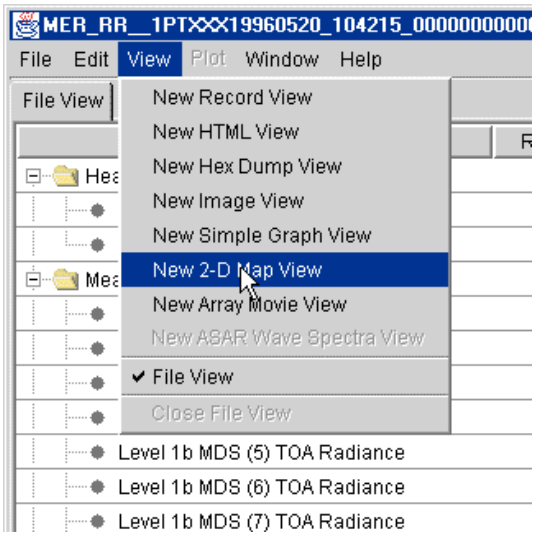


Figure 9.1 Starting a 2D Map view

A 2-D map viewer will open similar to that shown in Figure 9.2 containing a map of the whole world. The first data set containing geographical data will be automatically selected, and a range of data set records will be extracted and plotted immediately.

The latitude/longitude points will be initially all in red. A third field may be selected to vary the colour of each of the points on the graph. This may be performed using the *Value Field* pop-up menu. More details about this process are available further on in this Chapter.

9.1.1 Differences between an image view and a 2-D map view

EnviView Image View (see Chapter 7) and 2-D map View look somewhat similar as they both display two-dimensional false colour images of Envisat data associated with geographical (latitude and longitude) position. However there are important differences between them that are described below.

An EnviView Image is a "flat" two-dimensional pixel array that is simply mapped onto the two-dimensional pixel array of the computer screen. Linear scaling is the only additional type of transformation that EnviView carries out on Image data. Latitude and longitude data is associated with the pixels of an Image by EnviView however this information is

not intrinsic to the image data itself. Zooming and scaling of Images is based on pixels (e.g. screen pixels/image pixel).

Every point displayed on the 2-D map view has an associated geographical tie point reported in the product file. The map data (field values) are plotted, using false colours, onto the computer screen using a rectangular projection oriented with north as the top of the image. Zooming and scaling of a 2-D map view image is based on latitude and longitude (e.g. height and width of the image are scaled as angle).

The measurement data in the AATSR AST_NR__2P image product can be visualised using the Image View. On the other hand the AATSR AST_AR__2P and AST_MET_2P products can be visualised using the 2-D map viewer (Figure 9.3).

9.2 Feature Summary

This section briefly states the features available for viewing 2-D maps. More details are available below.

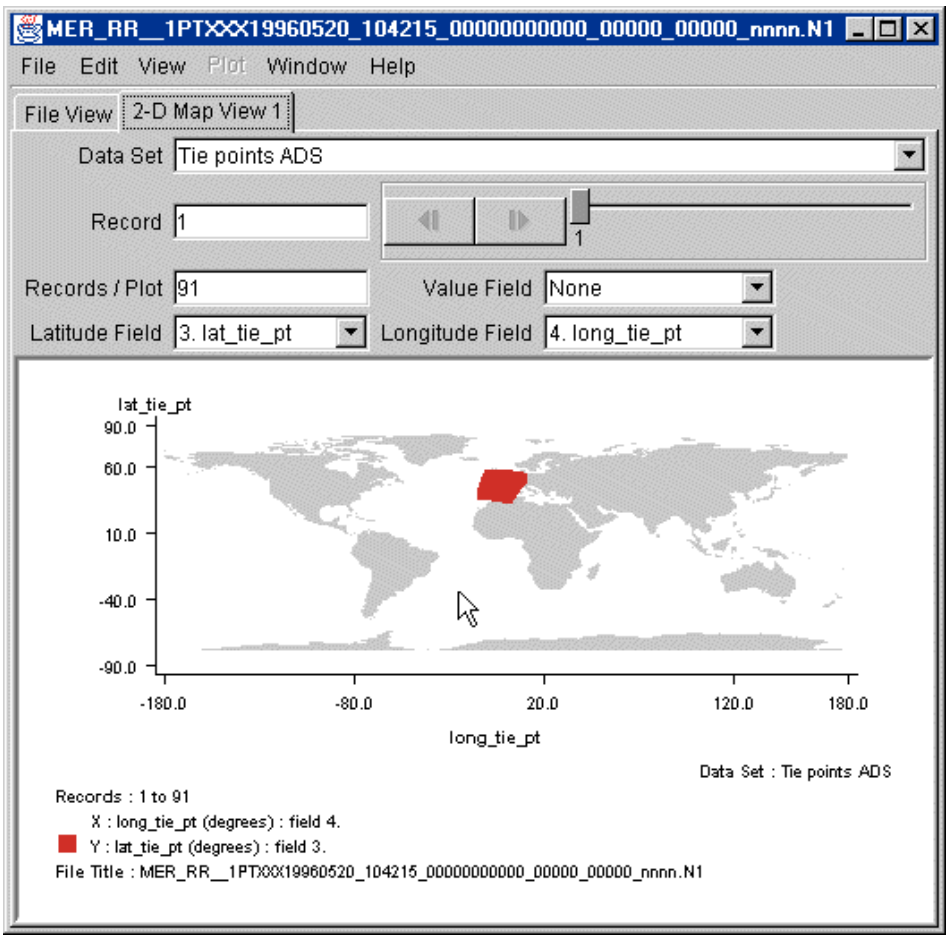


Figure 9.2 2-D Map Viewer

Figure 9.2 shows a 2-D map Viewer window that has just been opened. The features available are;

- Menu bar and menus.
  These are identical to those on other EnviView views.

- View tabs
  These select other open views with a single mouse click

- Data set selector
  This displays the name of the current data set and allows selection of other data sets in the data product that are associated with geographical tie point data.

- Record selector and navigation features.
  The Record field shows the record number of the first record displayed. The records/plot field shows how many records are currently displayed on the plot. The nudge buttons and slider enable navigation through records.

- Value Field
  This is the field that may have its values plotted as false colours on the data area.

- Latitude and longitude field (tie point selectors)
  These are the fields used to define the geographical co-ordinates that are used to plot the data points.

- Data area
  This shows a map, with land areas as grey and the sea as white, over which are plotted the field values (as false colours). The colour to value mapping is shown on the colour bar to the right of the map. The x and Y axes show the latitude and longitude scales that can be altered. Figure 9.3 shows an example of these features.

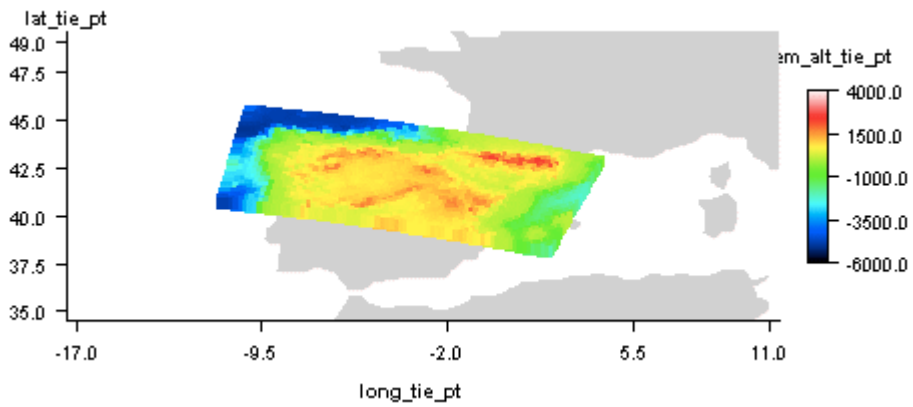


Figure 9.3 Geolocated AATSR Image Data

- Information and identification area.
  At the bottom of the window, the data set and file names are shown together with the range of records displayed. Finally the X, Y (grid co-ordinates) and field value (y*) names, units and field indexes are specified.

9.3 How EnviView selects Map View Data

Only data sets containing geographical tie point data can be displayed by the 2-D map viewer. EnviView handles this selection automatically; only listing data sets that are available for map viewing. The algorithm EnviView uses to decide looks through the data sets for a field with a Geocoordinate data type, and a name containing "**lat**", or "**lon**". If EnviView finds matching fields, it makes it available for display using the 2-D map viewer. It then displays the first matching fields it finds.

9.4 Selecting a data set to view

The Data Set pop-up menu, towards the top of the window, contains a list of available data sets for the product.

Figure 9.4 Selecting the data set

To select a different data set (if one is available) click and open the pop-up menu and select the data required.

9.5 Selecting records to view

When a 2-D map view opens it displays the selected field value of every record in the data set. This means that the record navigation features, the nudge buttons and slider, are initially not available.

To display a subset of the records, first decrease the number of records per plot. Do this by selecting the **Records/Plot** and entering the number of records to be displayed. This number must be less than the initial value in the field (which is the maximum value). Make sure *Enter* is pressed after entering the value.

The nudge buttons (to go back and forward one record at a time) and the record slider will now be available. Figure 9.5 shows an example with 10 records displayed per plot. The nudge buttons move one record at a time through the data set and the slider can be used to move many records at a time.

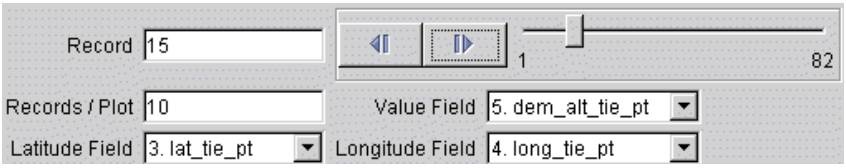


Figure 9.5 Record navigation & field selector features

Note that the false colour to field value mapping is rescaled, every time the record selection is changed, to use the full colour range with particular range of field values for the selected records. This means that points on the map may change colour when the record selection is altered although they keep the same numerical field value. Use a manual axis scale to fix the range between plots (click on the colour bar to select the axis range).

9.6 Choosing a value to plot

The field selected in the **Value Field**, below the record slider, controls the colour of the each point. This pop-up menu contains a list of all the field values available to be displayed. To select a different field to plot, select the **Value Field** pop-up menu and choose a different field from the list.

EnviView automatically selects the first latitude and longitude fields it finds in a data set. Normally, there should be no need to alter the selection.

9.7 Changing the scale of the map

A map of the whole world is displayed by default (see Figure 9.2). Generally, a smaller area but in more detail will be used, usually because the data area covers only a small region of the globe. The ways to do this are;

- Click and drag over the area to view in more detail
- Explicitly set the limits of the latitude and longitude axes
- Let EnviView maximise the view of the data area.

To select an area, move the pointer to the top left corner of the area to be viewed in more detail. Click and drag towards the bottom right as shown in Figure 9.6.

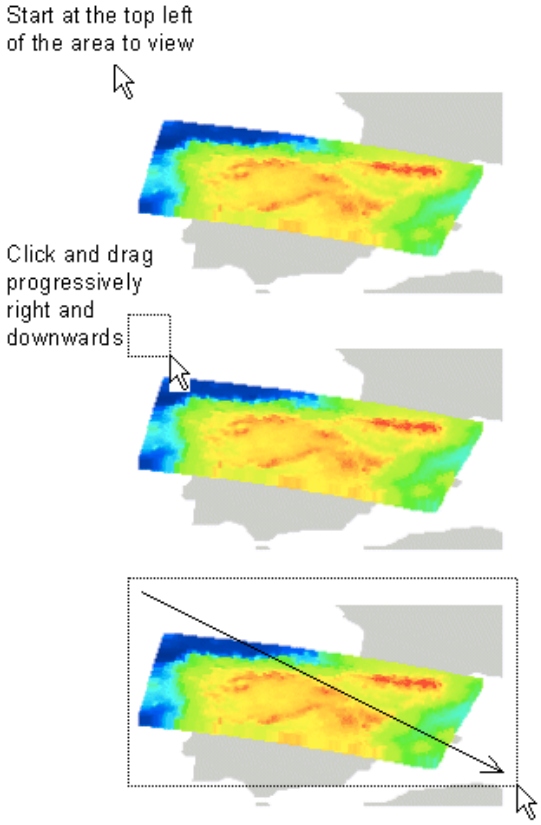


Figure 9.6 Drag selecting on a 2-D map area

Do not drag select in any other direction to select an area. This will reset the map, i.e. display a map of the whole world.

The X and or Y scales (latitude and longitude limits) can be set to specific values. e.g. for the X –axis, double click the X-axis on of the display.

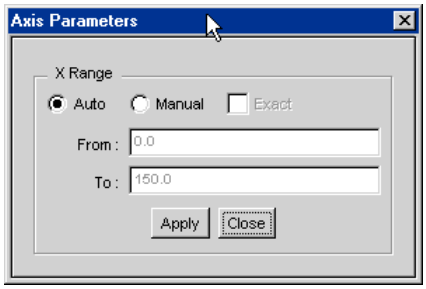


Figure 9.7 X-axis scaling

An “Axis Parameters” dialogue appears similar to that shown in Figure 9.7. Select the **Manual** radio button and enter the desired limits for the longitude scale in the **From:** and **To:** fields. Then press the **Apply** button to see the changes. If the **Exact** check box is not crossed, EnviView may round the values entered.

Exactly the same process is used to adjust the longitude scale. The only difference being that the Y-axis must be double clicked to activate its “Axis parameters” dialogue box.

To let EnviView maximise the area of the plotted data, double click the X-axis to activate the Axis parameters dialogue box and select the **Auto** radio button. Apply and close the dialogue box. Then do the same for the Y-axis. EnviView will maximise the area displaying the plotted data for the set of records chosen.

To reset the map, i.e. return to the world view; click and drag, on the map area in any direction except towards the bottom right.

10. THE ARRAY MOVIE VIEWER

The array movie viewer shows an animated display of one-dimensional array data sets, with each subsequent frame of the movie advancing one record through the data set. Each line on the movie display corresponds to a set of field array values of a record. Several records can be displayed at a time to help show the progression of the data, with the records stacked one on top of another.

It has been designed to display Radar Altimeter waveforms, but may be used to display a sequence of records for any fixed length array field.

To start an array movie, select the *New Array Movie View* option from the *View* menu, (Note: the data set must be an array, with certain characteristics, to be viewable as an array movie. See section 10.1, for more information). An array movie viewer will open similar to that shown in Figure 10.1. To start playing the array movie press the play button ( ) towards the top right of the window.

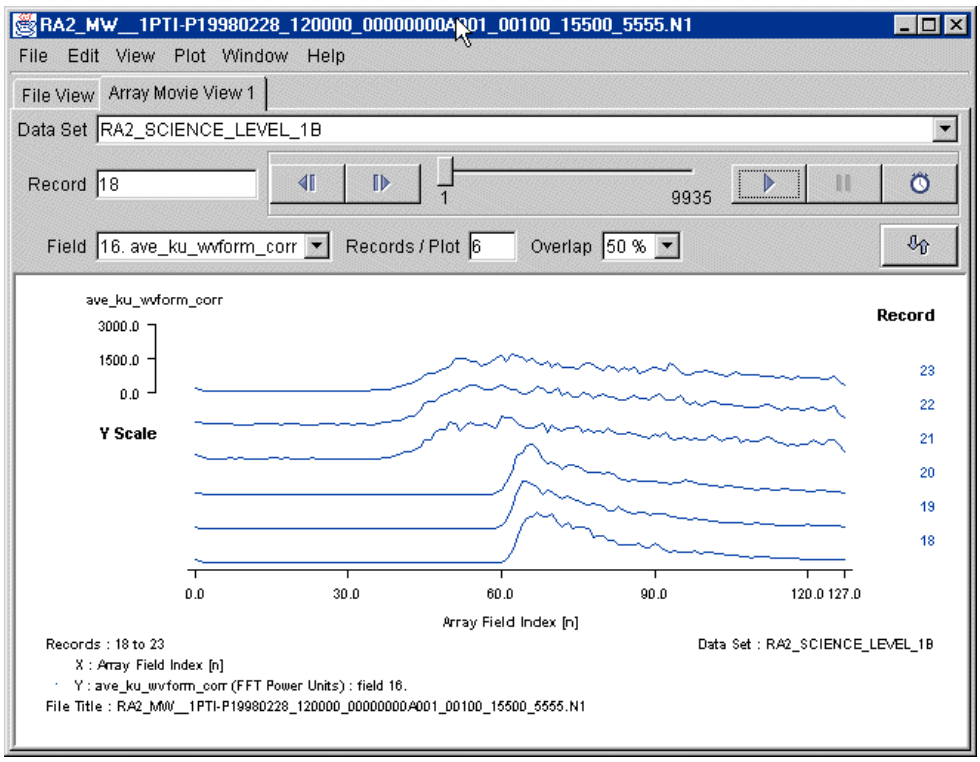


Figure 10.1 Array Movie Viewer

The array viewer window features are briefly summarised in the list below. More detailed information about array movie controls and features follows after the summary. Starting at the top of the window, referring to Figure 10.1, the features are;

- Menu bar and menus.
  These are identical to those in other EnviView windows.

- View tabs.
  Other open views can be selected with a single mouse click.

- Data Set selector
  This displays the name of the current data set and allows selection of other data sets within the data product that can be displayed as array movies.

- Record Number.
  The field shows the lowest record number of the displayed records.

- Record navigation. These consist of

  - nudge buttons to move 1 record at a time through the movie,

  - a record slider control which also displays the number of records in the data set,

- movie controls consisting of

  - a play button to start the movie,

  - a pause button to stop the movie ,

  - and a speed control button ( ) which controls the time interval between movie frames.

- Field selector

  To choose which array field in the record to plot.

- Number of records (lines) per plot and how they are displayed
  The overlap pop-up selector closes up the space between the plots and the order button changes the display order of the records.

- Data area
  This shows the field name and (Y) axis scale, a set of plotted lines each of which corresponds to the field array values of a record to the right of which are the record numbers. Below the plots is the array index (X) axis.

- Identification and statistics area.
  At the bottom of the display the range of records displayed is stated together with the X and Y-axis data names. The data set and file (corresponding to the data product) identifiers are also displayed.

10.1 Rules for viewing Arrays

Only certain fields from a data set record can be viewed as an array movie. EnviView handles this automatically, only allowing valid fields to be selected. This section explains the rules that EnviView uses to decide whether a field is valid for the array movie viewer. These rules are as follows;

- The data set must contain 10 or more records.

- The records must contain at least one field that is an array field.

- Only one-dimensional array fields are viewable.

- Each viewable array field must contain 5 or more elements.

The easiest way to check these characteristics of a data set is to open an HTML view of it and then check the record and field properties.

10.2 Array Movie Features

This section explains in more detail, how to use specific features of the Array Movie view.

10.2.1 Selecting a Data set and starting an Array Movie View

To start an array movie;

- Select the *New Array Movie View* from the *View* menu,
  (Note: the data set must be an array, with certain characteristics, to be viewable as an array movie. See section 10.1 for more information).
  An array movie view will open similar to that shown in Figure 10.1.

- Select the data set to view using the Data Set pop up menu below the tab view selectors.

- To start playing the array movie press the play button ( ) towards the top right of the window.

10.2.2 Selecting the fields & records to be viewed

The record slider control can be used to move to the approximate position in the data set. If the data set contains a large number of records use the nudge backward and forward buttons to move to the exact record number required once in approximately the correct location. The record number field, to the left of the nudge buttons, shows the exact record number. Note that this field is not editable and the desired record number cannot be typed into this field.

To select the field array to be plotted, use the pop-up **field** menu on the left-hand side of the Array movie viewer, just below the record field. This pop up displays all the fields, of the data set records, that can be displayed as array movies.

10.2.3 Changing the speed of a Movie

When the array movie window opens, the movie is stopped at the first record. The movie can be forwards at varying speeds, stopped, or manually moved backwards and forwards one or more frames at a time.

- Press the play button ( ) to start the movie.

- To change the speed of the movie, vary the time interval between frames using the speed control button ( ). This can be done before or whilst the movie is playing. Press the speed control button to bring up a menu of time intervals between movie frames and select the interval required. Smaller time intervals (1/10 sec minimum) give faster play speeds whereas longer intervals (2 seconds maximum) give slower speeds

- Stop the movie by pressing the pause button ( ). The movie will stop automatically when it reaches the last record.

- Single record step; Stop the movie and use the nudge backward and forward buttons ( ) to move through the movie a single record at a time.

- Moving to a different part of the movie can be done whilst the movie is playing. Drag the slider control to the part of the data set to view and the movie will play forwards from this point if it was already playing. If the movie was stopped press the play button.

10.2.4 Changing the look of the data displayed

The number of graphs plotted and their look can be controlled.

- The number of graphs/records displayed on the movie viewer can by varied between 1 and 50. Enter the value required into the records/plot field and press Enter. Note that the default value is 6 records per plot.

- The spacing between graphs can be varied using the overlap feature. This enables the plot areas of each line to overlap by up to 50% thus compressing the amount of screen space needed to display the data. Select the amount of overlap by clicking on the **Overlap** pop-up menu and selecting either 0% (no overlap), 25% or 50%. Resizing the EnviView array movie viewer window changes the spacing and size of graphs plotted as the data area, containing the plots, expands or shrinks. To increase the size of the plots, expand the window (the details of this vary according to the user interface being used). Conversely shrink the window to make the plots smaller

The scale of graphs can be varied. Both the X axis (array indexes) and the Y axis (field values) scales be varied independently. To alter the X-axis scale, double click on the X-axis. A dialogue box appears as shown in Figure 10.2.

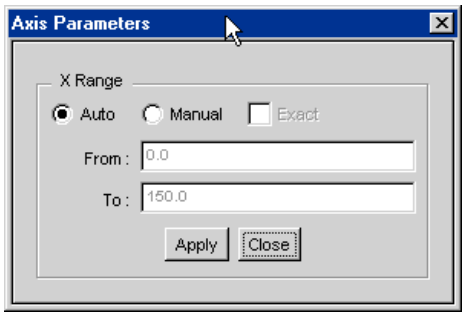


Figure 10.2 X-axis parameters

The X-axis scale is set to Auto by default. When Auto range is set the array movie viewer chooses the scale to optimally fit all the record field array data onto the display. However, to concentrate on a certain part of the array click the **Manual** radio button and enter the start and end array elements into the **From:** and **To:** text fields. Click the **Apply** button to see the changes. The **Exact** check box is only available when a **Manual** range is selected. This constrains the axes to use exactly the values entered into the **From:** and **To:** fields as the start and end points of the X-axis. If **Exact** is left unchecked, the axis range displayed may be larger than the values entered manually as EnviView rounds up the range.

Note that the Axis parameter dialogue box can be open whilst the movie is running and the graph scale interactively changed.

Altering the Y-axis (field values) scale is done in the same way as that for the X-axis scaling. Double click the Y-axis and a dialogue box will open as shown in Figure 10.3. The fields, check boxes and button behave in the same way as those of the X-axis parameters dialogue box.

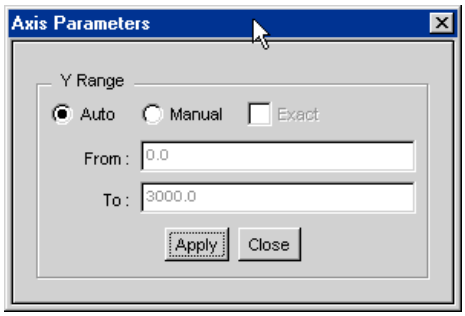


Figure 10.3 Y-axis parameters

Note that all lines plotted on an array movie viewer window use the same X and Y-axis scales.

10.2.5 Y axis automatic rescaling behaviour

The default behaviour of the movie viewer, whilst playing an array movie is to rescale the Y-axis every frame to optimise the fit of the latest record. As all the other lines plotted use the same scale this usually means that every line on the plot is rescaled every frame.

Figure 10.4 and Figure 10.5 illustrate this effect. In Figure 10.4 the plots are scaled to fit the plot of record 9, giving a Y-axis range from 0 to 150. Figure 10.5 shows the next frame. In this case the Y-axis has been automatically rescaled to fit the plot of record 10 giving a Y-axis range of 0 to 600. Note however that the record 9 plot is displayed on a scale of 0 to 600 meaning that the height of it has shrunk to a quarter of the size it was displayed as in Figure 10.4.

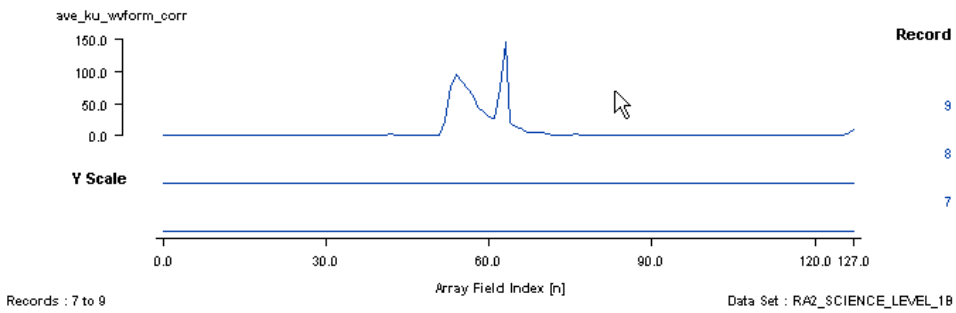


Figure 10.4 Y-axis scaling, record 9 before rescale

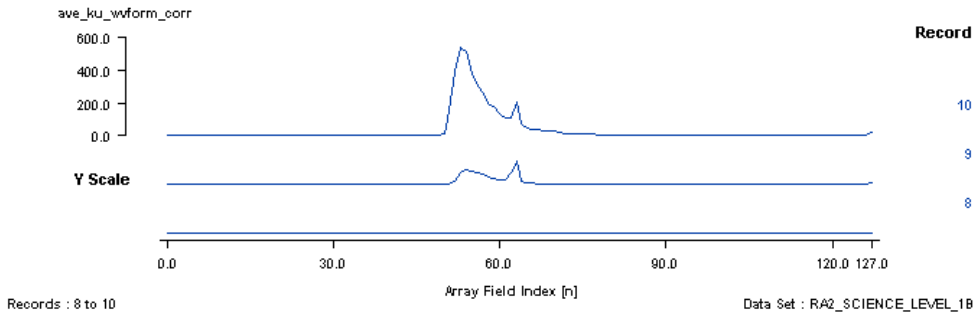


Figure 10.5 Y-axis scaling, record 9 and 10

Set the Y-axis range to **Manual** to avoid the plots continually being rescaled. This makes it easier to follow the progression of the data through the movie. However care must be taken to choose a y-scale that can meaningfully plot the set of data of interested.

11. SAVING AND EXPORTING ENVISAT DATA

EnviView provides a number of different facilities for extracting and exporting Envisat data to other file formats. Many of these methods depend on the type of data being extracted. A number of command line tools are also provided for converting and extracting Envisat data, see Chapter 14 for more details.

11.1 Saving a Selected Region of an Image

The pixels of a selected image area can be exported to an image file, tab-separated text or binary data, from the *Image Selection* dialog in the *Image View*, Figure 7.8. Refer to Section 7.8 for an explanation of how to select an area of an image.

At the bottom of the Image Selection dialog, in the *Export* section of the window, choose an output format in which to save the data, from the pop-up list, see Figure 11.1. Click on the *Save as* button, whereupon a file selection window will open where the location and name of the specified file can be chosen.

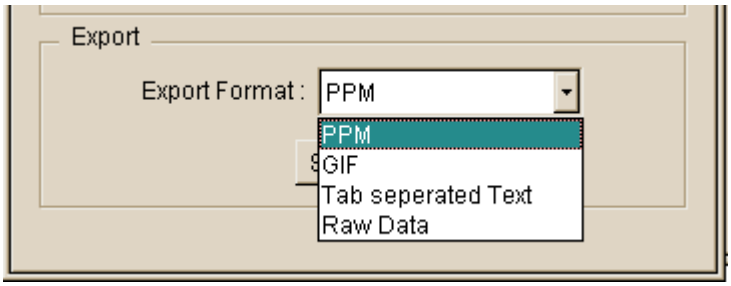


Figure 11.1 The Export selection for image data

Three image formats are available:

- TIFF: A 24-bit image file.
- PPM: A 24-bit Portable Pixel Format common on Unix platforms.
- GIF: An 8-bit image file. GIF only supports up to 256 colours. If the selected area contains more than 256 different colour values (e.g in RGB browse images), PPM should be used.

Exporting data as TIFF, PPM or GIF should be used for presentation purposes only. The pixel values contained in the data set are converted to colour values depending on the settings of the Image Viewer colour bar.

To export the actual Envisat image data of the selected region, save as either: -

- Tab separated Text: A line of pixels is represented by a line of text, terminated by the platform specific line terminator. Each pixel value is separated by a tab character.
- Raw Data: Each pixel value is written to the file as an 8-byte IEEE double.

To export an entire image data set, or a subset of scan lines see section 11.2 on exporting to HDF.

Command line tools for extracting image data are also provided. See `PdsPixels` and `PdsPixelTrack` in Chapter 14 for more details.

11.2 Exporting Data in HDF

The NCSA Hierarchical Data Format is supported by most commercial scientific data visualisation packages, including PV-Wave and IDL, and a library for reading HDF data is freely available for C, Fortran and Java. HDF is a self-describing format, so once data is written into an HDF file, it's content can be automatically determined, without the need for an external format description. There are two versions of HDF in common use, HDF 4 and HDF 5. EnviView supports only HDF 4 today.

EnviView can be used to export all or part of an Envisat product in PDS format to a new HDF 4 file. In addition, specific data sets and records can be chosen rather than exporting the whole file.

A command line tool is also provided for extracting and exporting to HDF format. See `pds2hdf` in Chapter 14 for more details.

Appendix B provides information and examples for reading Envisat data exported to HDF in Research Systems IDL data visualisation package.

NOTE: HDF support requires the addition of a native library compiled for the platform that EnviView is being executed on. If the library is not available, or EnviView cannot locate it, the menu options described below are not accessible. Choose *About EnviView* in the Help menu to determine whether EnviView can locate the correct library, also refer to the README file provided with the software distribution for installation instructions.

11.2.1 Organisation of Envisat data in HDF format

EnviView writes Envisat data into HDF files, conforming as closely as possible to the format used in the ESA PDS products.

The following approach is used to convert Envisat data to HDF format:-

- EnviView uses the NCSA HDF 4.1r3 libraries.
- No data values are lost
- All data values are stored in the original format, with the exception of:
  - ASCII records are converted to their binary equivalents
  - Time values are written as a 64-bit double precision floating point value, as Julian days since January the 1st 2000.
  - 64-bit integers are converted to 64-bit double precision floating point values. No support for 64-bit integers is available in HDF 4.
- An HDF Vgroup is created for each data set, named after the DS_NAME found in the corresponding PDS DSD. Additional groups called MPH and SPH are also created.
- To distinguish Envisat data set Vgroups from any other, they are assigned the HDF class "`PDS Data Set Record`".
- Each data set Vgroup contains
  - Scalar fields in a Vdata table, with one row for each exported record from the PDS data set, and named the same as the enclosing Vgroup. i.e. the DS_NAME found in the corresponding PDS DSD.The table contains
    - An integer entry for the original record number
    - An entry (column) for each scalar field, the field name coming from the Envisat format database.

- An entry for each element of arrays with five or fewer elements. The field name is appended by the field index in round brackets. e.g. '`mw(2)`'
- An entry for each scalar field of non-repeating structures. The name of the sub-field is the name of the parent field, followed by a full stop and the name of the sub-field. e.g. `image_parameters.swst_value` in the ASAR Main Processing Parameters data set. Note that Envisat products do not contain two or more levels of structures.

- Array fields larger than five elements (or with a variable number of elements) are written to individual HDF SDS objects (Scientific Data Sets).. Each SDS contains the values for one field. The first dimension of the data set is the Envisat record offset. e.g. All image pixels from a single ASAR data set are written into one SDS are written to an SDS. Each SDS is named after the field name in the Envisat format database (refer to Product Formats in the EnviView *Help* menu for more details).
- Variable length array fields are also written to SDSs, however these are sized to the largest dimensions of the variable length field, over the exported record range. Unused elements are written as zeros for integer fields, and the special IEEE NaN value for floating point data. Each SDS is named after the field name in the Envisat format database (refer to Product Formats in the EnviView *Help* menu for more details).
- The dimensions of each variable length array field, for each exported record, is written into an optional extra VData named "`dimensions`". The entry in the dimensions VData table matches the name of the SDS (derived from the field name in the Envisat file format database). This Vdata is not written if there are no variable length arrays in the data set.

11.2.2 Selecting data for export

If a product is selected in the Product View window and the HDF library is correctly installed, the *Export Product As HDF* option in the *File* menu is enabled. This menu item leads to a dialog that allows the details of the data sub set for export to be specified.

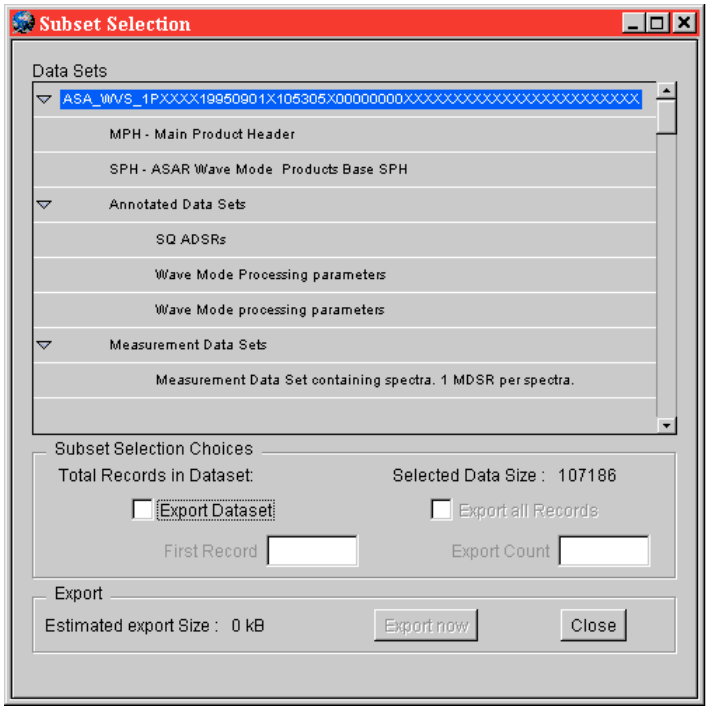


Figure 11.2 The Subset Selection Window

The HDF subset selection dialog (see Figure 11.2) is split into three sections.

- The top part of the new dialog ,*Data Sets,* displays the product name and all the data sets in this product. The data sets are organised into annotated data sets and measurement data sets but are all treated in exactly the same way. Note that by default the ADS list is fully expanded, unlike in the *Product View*. This area is not editable. Data sets including the MPH and SPH are selected from here.

- The second section, *Subset Selection Choices,* contains details about the currently selected data set. It shows the number of records in the data set and, if appropriate, the size of the data set in bytes. Below these are two checkboxes.

- *Export Dataset* is selected if this data set is to be exported to HDF

- *Export all Records* can be used to select all records in a single click.

  The fields labelled *First Record* and *Export Count* contain the index of the first record and the number of subsequent records to be exported.

- The lowest section, *Export*, contains the estimated total size of the exported data selected, the value of which is updated as data sets are selected and deselected, a button labelled *Export Now*, and a button to *Close* the dialog. Note that the *Export Now* button is not enabled until at least one data set has been selected for export.

NOTE: The total export size can only be estimated because the HDF format includes structural and formatting information in addition to the data itself, resulting in a somewhat unpredictable overhead. In general, the overhead is of a fixed amount per data set, so will not vary with the number of records written. Furthermore, the export data size of data sets with variable length records is given as a rough approximation.

11.2.3 Selecting a Subset of Data for Export

To select a data set for export, click on its name in the Data Sets list. In the *Subset Selection Choices* box, the number of records and size of the data set will be displayed automatically. The checkbox labelled *Export Dataset* will be enabled but not ticked (selected). At this point, no other fields can be edited. The *size* field reports the size of the raw data that will be exported.

Clicking the *Export Dataset* checkbox adds a tick next to the data set name in the list, to indicate that this data set is marked the export. By default, the *Export all Records* checkbox is ticked and the fields *First Record* and the *Export Count* are filled in with the values corresponding to the data set contents. The estimated export size in the lower window is updated accordingly, and the *Export Now* button is enabled if it was not done so already.

Other data sets can be added in the same way. As they are added, the *Estimated Export Size* field is updated accordingly.

To select a subset of the records for a particular data set, the *Export all Records* checkbox can be deselected. The *First Record* and *Record Count* fields become editable and the desired values can be entered. Negative or non-integer entries are rejected. The record count is also checked so that the value entered is not too large for the data set, taking into account the *First Record* value. The *Estimated Export Size* is updated accordingly.

To export the selected data , click the *Export Now* button and enter an output filename and destination directory using the platform specific save dialog that appears.

See Figure 11.3 for an example. NOTE :The SPH size is smaller than the format documentation specifies, as the DSD's are not exported (they are replaced by the directory and catalogue information in the HDF file).

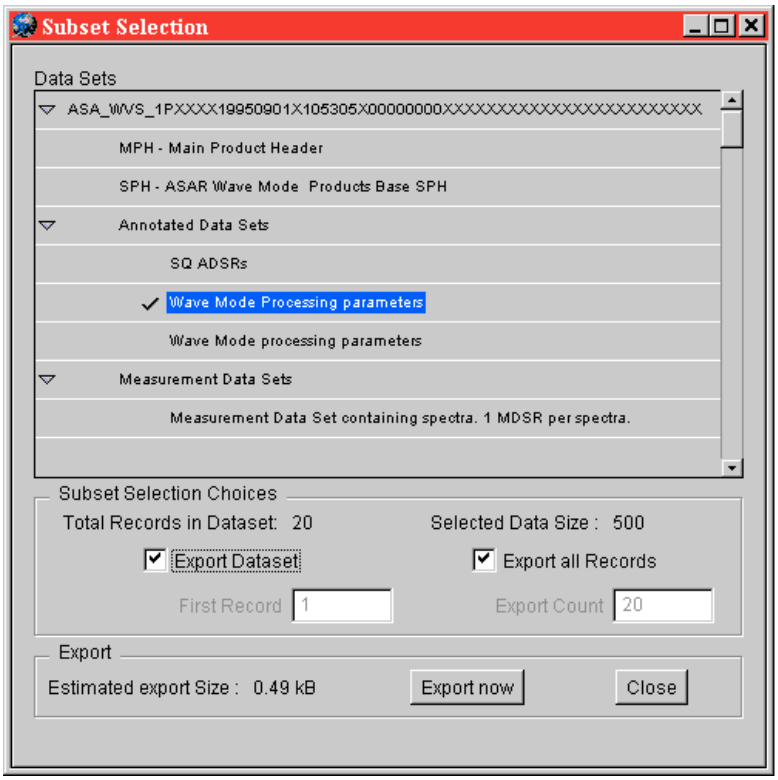


Figure 11.3 An Example of Subset Selection

11.2.4 Shortcuts for Exporting the Data to HDF

Rather than explicitly specifying each data set for export, one by one, there are some shortcuts to select multiple data sets with a single click.

- To select the whole product, select the product name in the *Data Sets* menu and then *Export Dataset*.

- To select all annotated data sets, select the item *Annotated Data Sets* in the *Data Sets* menu and then *Export Dataset*. Note, that it is not possible to select a first record and a record count and have this value applied to all data sets when this item has been chosen. By default all records are exported. It is, however, possible to then select individual data sets and set the values separately.

- To select all measurement data sets, select the item *Measurement Data Sets* in the *Data Sets* menu and then press *Export Dataset*. NOTE: it is not possible to select a first record and a record count and have this value applied to all data sets when the this item has been chosen. By default all records are exported. It is, however, possible to then select individual data sets and set the values separately.

TIP: In certain circumstances, it may be quicker to select all data sets or all MDS or ADS for export, then deselect ones that are not actually required. See Figure 11.4 for an example.

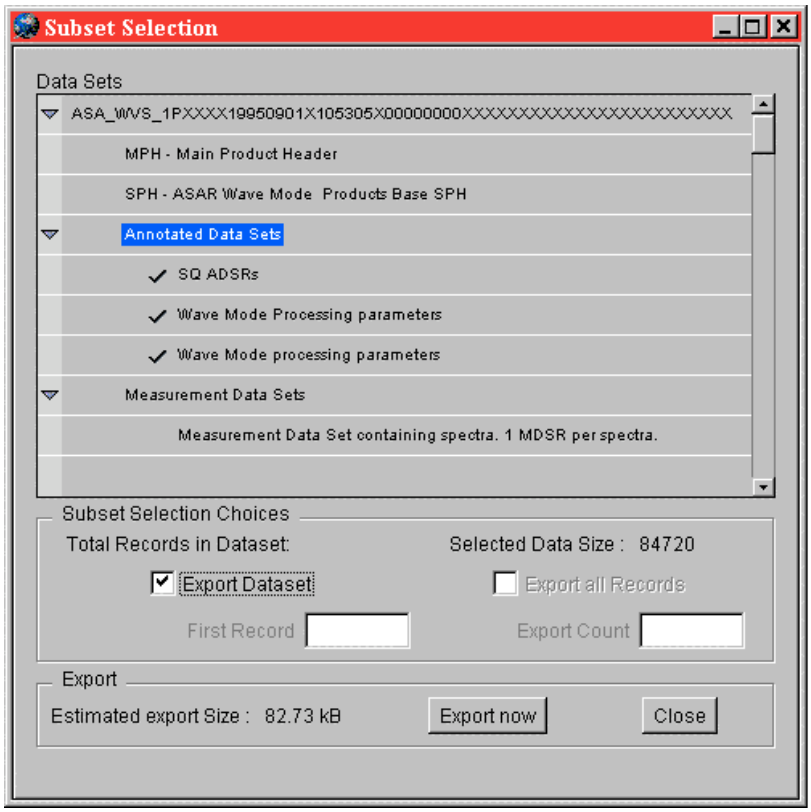


Figure 11.4 Example of a Shortcut being used to Export Data

11.3 Creating a Sciamachy Calibrated (Level 1C) Product

This section deals with creating a Sciamachy Level 1C (SCI_NLC_1P) file from EnviView. A command line tool to create this file is also available, see section 14.9. A description of the calibration process and the file format is given in accompanying documents. Detailed file format information is also available in the EnviView File Format window.

11.3.1 Creating a Level 1C file

A Level 1C file is derived from a Sciamachy Level 1B file (SCI_NL__1P). If a level 1B file is open in EnviView, the *File* menu item *Extract Level 1 C* is enabled. Selecting this menu item opens a Wizard type dialog to guide you through the process of creating a calibrated file. Arrow buttons at the bottom of the window are for navigation between each of the wizard panels. The first panel provides some introductory information.

11.3.1.1 Selecting the extraction Criteria

On the second panel there are five tabbed panes for specifying the extraction criteria. The first criteria type is the Category Filter (Figure 11.5).

Figure 11.5 Category Filter

Select this filter with the *Apply this filter* checkbox. Choose one or more Sciamachy Categories for extraction by clicking on items in the list. Selected items are marked with a tick icon appearing beside them. As a short cut, buttons are provided for selecting all or none.

The second criterion is the Cluster filter (Figure 11.6).

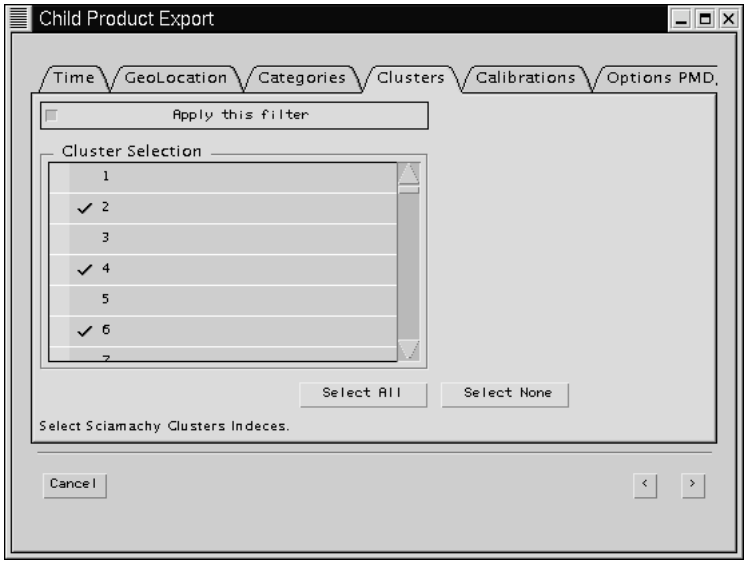


Figure 11.6 Cluster Filter

Select this filter with the *Apply this filter* checkbox. Choose one or more measurement clusters for extraction by clicking on items in the list. Selected items are marked with a tick icon appearing beside them. As a short cut, buttons are provided for selecting all or none. The types appear as check boxes beside the list. The default is "All" clusters.

The third criterion is the Calibrations filter (Figure 11.7).

Figure 11.7 Calibration Selection

To apply calibrations, select the *Apply this filter* checkbox. Choose one or more calibrations for application by clicking on items in the list. Selected items are marked with a tick icon appearing beside them. As a short cut, buttons are provided for selecting all or none. The default is that no Calibrations are applied.

The fourth criteria selection is the Options filter (Figure 11.8).

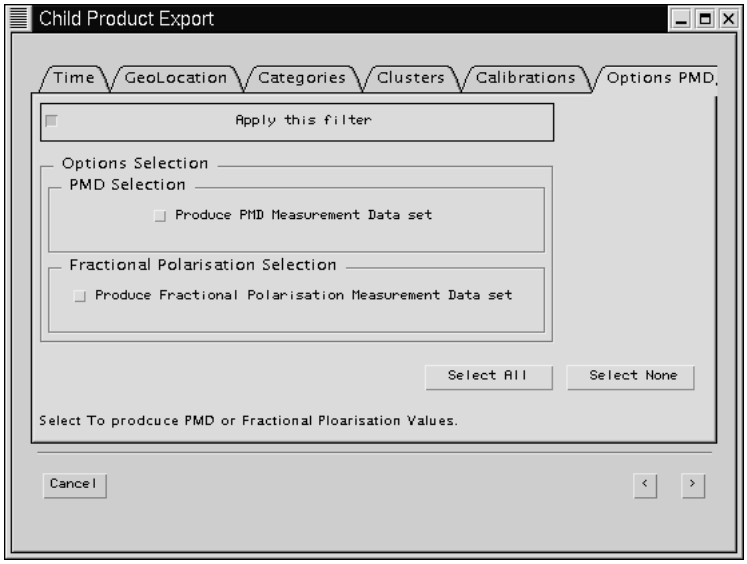


Figure 11.8 Options Filter

To export PMD and/or fractional polarisation data, select the *Apply this filter* checkbox. Select which data sets should be exported to the L1C file. As a short cut, buttons are provided for selecting all or none.

The fifth criterion is the Type filter. Select this filter with the *Apply this filter* checkbox. Choose one or more measurement types for extraction by clicking on items in the list. Selected items are marked with a tick icon appearing beside them. As a short cut,

buttons are provided for selecting all or none. The types appear as check boxes beside the list. The default is "All" clusters. The default is Nadir type only.

Warning: if the Time, Geolocation or Category filters are applied, ensure that there is a data of the selected type passed through by the other filters.

Click on the *Next* button to perform the extraction and calibration. A subset of datasets may be selected for extraction, and the location of the output file is set through a standard file chooser.

Depending on the performance of the machine, the IO speed, and the range and calibration types selected, the operation may take a significant time to complete.

When processing large data sets, use the command line tool, SciaL1C, in batch mode.

11.4 Saving a Data Set Record in HTML

Choose *Save* from the *File* menu, when using an *HTML View*.

11.5 Saving Individual Data Fields

Data extracted for a graph may be viewed in tabular form, and saved to a text file. See Extended Plotting Facilities in section 6.3 for more details.

For more advanced export, see the command line tools Pds2Ascii, PdsPixels and Pds2Hdf (Chapter 14).

11.6 Exporting to Printable Files.

This feature is platform dependent. Choose print from a view of choice. Your printer drivers may allow you to save the print to a data file.

12. DECODING LEVEL 0 DATA

This chapter explains how to read level 0 data using EnviView. The principle is identical to that used for any other data set, however some configuration is necessary before a particular level 0 field can be read.

12.1 Introduction

The exact contents of a Level 0 MDS is instrument and mode specific. This means for a specific Level 0 product, the fields in a MDS can be in different locations within the source packet for each mode of the instrument. The format of level 0 source packets is not available in the Envisat product format database. However, EnviView is able to decode and display Level 0 data, if it is provided with a description of which parameters are contained in a given source packet, and how to retrieve them.

If the configuration for a given Level 0 product is provided, EnviView will display, plot and extract the information in exactly the same way as for fields in any other product, using the record View, Graph View, etc.

To enable EnviView to be able to open a Level 0 MDS, a script language has been developed to enable users to describe the locations and sizes of fields in the source packet, which EnviView can then decode and display in the normal manner.

User written Level 0 script files contain descriptions of fields to extract from certain source packets.

If no user script exists for a given Level 0 product, EnviView has a default script which it can use to open a Level 0 MDS. This default header decodes the Packet Header and the first 2 fields of the Data Field Header. These fields are the only fields which are constant in every Level 0 MDS.

The detailed description of the format of each source packet may be found in the Envisat PGICD [RD.3].

12.2 Introduction to the scripting language

The scripting language is very simple. All it does is define a series of variables, and for each variable set the properties for that variable. The basic structure of the language is in the following format.

```
variableName = Field( variableProperties );
```

Where:-

- variableName is the name of the variable.
- variableProperties is a list of statements which define the properties of the variable.

Syntax:- There must be a space before and after the = between variableName and Field.
Syntax:- There must **not** be a space between the end bracket and the semi colon at the end of the line.
Syntax:- variableName must be a string without any spaces in it.

Figure 12.1 is an example script file. It is the Default Level 0 script file, defining the first fields of every source packet. If a user writes their own Level 0 script file, it is recommended that they include these definitions at the start of the file.

```
packver = Field( bitoffset = 0, size = 3, type = "IntegerU1", description =
"Packet Version Number");
packtype = Field( bitoffset = 3, size = 1, type = "IntegerU1", description =
"Packet Type");
datafieldflagheader = Field( bitoffset = 4, size = 1, type = "IntegerU1",
description = "Data Field flag header");
AppId = Field( bitoffset = 5, size = 11, type = "BinaryU2", description =
"Application id");

segflag = Field( bitoffset = 16, size = 2, type = "IntegerU1", description =
"Segmentation Flag");
segcounter = Field( bitoffset = 18, size = 14, type = "IntegerU2",
description = "Source Segmentation Counter");

packlen = Field( bitoffset = 32, size = 16, type = "IntegerU2", description
= "packet length");

dataFieldHeaderLength = Field( bitoffset = 48, size = 16, type =
"IntegerU2", description = "data field header length");
modeId = Field( bitoffset = 64, size = 16, type = "BinaryU2", description =
"mode Id");
```

Figure 12.1 Example Script File

12.3 Variable Properties

The variable properties are a list of properties, which define or initialise the variable. The syntax of the list is as follows.

(propertyName1 = statement1, propertyName2 = statement2);

Where:-

- PropertyName is a name of the property.
- Statement is a constant or expression which initialises the property.

Syntax:- There must be a space before and after the = between PropertyName and the statement.

Syntax:- There must be a comma between each property.

12.3.1 Name Property

Property name = name

The name property defines the name of the variable. If the property is not present, it will use the variable name as the field name.

12.3.2 Bitoffset Property

Property name = bitoffset

The bitoffset property defines the offset in bits from the start of the Instrument Source Packet. Bitoffset is expecting an integer.

12.3.3 Size Property

Property name = size

The size property defines the size of the field in bits. The size property expects an integer.

12.3.4 Type Property

Property name = type

The type property defines the type of the field, i.e. is it a string, integer, binary etc. The type property expects one of the following strings to tell it what is the field type.

Type	*Description*	*Type*	*Description*
AsciiCharacter	A character represented externally as a single ASCII character	IntegerS2	2 byte signed integer
AsciiFloatNumber	A Float represented externally as a set of ASCII digits	IntegerS4	4 byte signed integer
AsciiNumber	A Number represented externally as a set of ASCII digits	IntegerS8	8 byte signed integer
AsciiString	A String represented externally as a set of ASCII Bytes	IntegerU1	1 byte un-signed integer
BinaryU2	2 byte un-signed binary	IntegerU2	2 byte un-signed integer
BinaryU4	4 byte un-signed binary	IntegerU4	4 byte un-signed integer
BooleanFlag	Boolean represented by a single byte	IntegerU8	8 byte un-signed integer
FloatDo	8 byte IEEE float	Geolocation	geographical coordinate stored in 8 bytes
FloatFl	4 byte IEEE float	Utc	Coordinated Universal Time in processing format
IntegerS1	1 byte signed integer	UtcExternal	UTC represented externally as a 27 character ASCII string

Table 12.1 Field Types

12.3.5 ElementCount Property

Property name = elementCount

The element count property defines the number of elements the field has. ElementCount is expecting an integer.

12.3.6 Units Property

Property name = units

The units property defines the units of the field. units is expecting a String.

12.3.7 Description Property

Property name = description

The description property defines a description of the field. description is expecting a String. If the description includes spaces, use double quotes to delimit the contents.

12.4 Statements

Statements are used to initialise the field properties. There are several statement types.

12.4.1 Basic Statement

Basic Statement is the most simple statement type. It simple initialise the property with a string. If spaces are required within the string surround it with quotes. Below are a few examples of basic statements.

```
propertyName = some_value;

propertyName = "initialise with this string";

propertyName = 54;

propertyName = 23.67;
```

The last two example properties have been given a number constant. If the property is expecting to be initialised with a string, the property value will consist of the digits of the specified number.

Syntax:- There must be a semi colon at the end of the basic statement.

12.4.2 If Statement

The If Statement is a simple conditional expression. If the result is true the code (statements) inside the if-statement are executed, if the result is false then the statements inside are not executed. Below is the syntax of an if statement.

```
if ( lhsValue condition rhsValue ){
   statements;
}
```

Where:-

- lhsValue is a left hand side value.
- condition is the condition part of the statement. There are 6 conditions which are excepted, they are:-

- == (equals)
- != (not equals)
- < (less than)
- <= (less than or equal to)
- > (greater than)
- >= (greater than or equal to)

- rhsValue is the right hand value.

- statements the inner statements which are executed if the if statement is true.

Syntax:- There must be a space between the if and the (

Syntax:- There must be a space between the lhsValue and the condition.

Syntax:- There must be a space between the condition and the rhsValue.

Syntax:- if rhsValue has spaces in it, it must be surrounded by double quotes.

The lhsValue is a variable (field) already defined within the script. When EnviView is executing the script, and it comes to an if-statement, it takes the value of the field defined as a lhsValue and evaluates it against the rhsValue. So if the field is an integer the rhsValue must be an integer.

A reason to do this might be that the bitoffset of field 'A' in the product is a different value for each mode the instrument is in. Below is an example script to do this.

```
AppId = Field( bitoffset = 5, size = 11, type = "BinaryU2", description = "Application id");
A = Field( bitoffset = if (AppId == "0000 0000 1100 0100"){
                        500
                    },
                    if (AppId == "0000 0000 1100 1110"){
                        550
                    },
                    size = 10;,
                    type = "IntegerU2";);
```

12.4.3 If Else Statement

The if-else-statement is a extension of the if-statement. The Syntax is as follows.

```
if ( lhsValue condition rhsValue ){
  statements;
}else{
  elseStatments;
}
```

Syntax:- There must not be any spaces between the curly brackets and the 'else'.

It works in the same way as the if-statement. i.e. the lhsValue is a variable name already defined in the script. If the if-statement is true then the code inside is executed, if the condition is false, the code inside the else loop is executed.

12.4.4 Math Statement

The math statement is provide for the case where the value of a property of a field is dependent on the value of another field. E.g. the bitoffset of field B is the value of field A + a constant. The script for this would look like.

A = Field(bitoffset = 234, size = 16, type = "IntegerU2", description = "field a");
B = Field(bitoffset = A + 25, size = 10;, type = "IntegerU2";);

The math statement can handle the basic math operators, + - * / and can handle parenthesis. Each term in the expression must be separated by spaces, this includes parenthesis. Below is an example of a complex math statement.

Bitoffset = (fieldA + 25) * (fieldB – 55) ;

12.5 Using Level 0 Scripts

12.5.1 File naming conventions

The file naming convention used by EnviView to find if a Level 0 MDS has a script is as follows.

ProductType + '_' + DataSetName + '.txt'

e.g. for the MIPAS_SOURCE_PACKETS dataset in a MIP_NL__0P product the script file name would be:- MIP_NL__0P_MIPAS_SOURCE_PACKETS.txt

12.5.2 Where to save script file

All scripts must be stored in a directory called level0scripts

The location of this directory is relative to the current working directory, the first time a level 0 data set is opened.

12.5.3 Modifying a script

EnviView reads the script file the first time a dataset is opened. Changes to a script are only registered by restarting the application.

12.6 Example

The following script is an example of a MIPAS file, decoding a number of items in the data field header :-

```
#
# Source Packet Header
#
packver = Field( bitoffset = 0,size = 3,type = "IntegerU1",description = "Packet Version Number");
packtype = Field( bitoffset = 3, size = 1, type = "IntegerU1", description = "Packet Type");
datafieldflagheader = Field( bitoffset = 4, size = 1, type = "IntegerU1",
                      description = "Data Field flag header");
AppId = Field( bitoffset = 5, size = 11, type = "BinaryU2", description = "Application id");

segflag = Field( bitoffset = 16, size = 2, type = "IntegerU1", description = "Segmentation Flag");
segcounter = Field( bitoffset = 18, size = 14, type = "IntegerU2",
             description = "Source Segmentation Counter");

packlen = Field( bitoffset = 32, size = 16, type = "IntegerU2",
          description = "packgfntenehsgbet length");
#
# Data Field Header
#
dataFieldHeaderLength = Field( bitoffset = 48, size = 16, type = "IntegerU2",
                        description = "data field header length");
modeId = Field( bitoffset = 64, size = 16, type = "BinaryU2", description = "mode Id");
icu_code = Field( bitoffset = 80, size = 32, type = "IntegerU4",
           description = "ICU On Board time code");
packet_type_id = Field( bitoffset = 112, size = 4, type = "IntegerU1",
                 description = "Packet Type ID");
igm_id = Field( bitoffset = 116, size = 16, type = "IntegerU2", description = "IGM ID");
num_blocks = Field( bitoffset = 132, size = 3, type = "IntegerU1",
             description = "Number of blocks in Packet");
b1_src_id  = Field( bitoffset = 144, size = 5, type = "IntegerU1",
             description = "Source ID Block#1");
b1_seq_num = Field( bitoffset = 149, size = 11, type = "IntegerU2",
             description = "Block Sequence Number Block#1");
b1_num_samp = Field( bitoffset = 160, size = 11, type = "IntegerU2",
              description = "Number of Samples Block#1");
b1_num_samp_bits = Field( bitoffset = 171, size = 5, type = "IntegerU1",
                   description = "Number Bits Per Sample Block#2");
b2_src_id  = Field( bitoffset = 176, size = 5, type = "IntegerU1",
             description = "Source ID Block#2");
b2_seq_num = Field( bitoffset = 181, size = 11, type = "IntegerU2",
             description = "Block Sequence Number Block#2");
b2_num_samp = Field( bitoffset = 192, size = 11, type = "IntegerU2",
              description = "Number of Samples Block#2");
b2_num_samp_bits = Field( bitoffset = 203, size = 5, type = "IntegerU1",
                   description = "Number Bits Per Sample Block#2");
b3_src_id  = Field( bitoffset = 208, size = 5, type = "IntegerU1",
             description = "Source ID Block#3");
b3_seq_num = Field( bitoffset = 213, size = 11, type = "IntegerU2",
             description = "Block Sequence Number Block#3");
b3_num_samp = Field( bitoffset = 224, size = 11, type = "IntegerU2",
              description = "Number of Samples Block#3");
b3_num_samp_bits = Field( bitoffset = 235, size = 5, type = "IntegerU1",
                   description = "Number Bits Per Sample Block#3");
b4_src_id  = Field( bitoffset = 240, size = 5, type = "IntegerU1",
             description = "Source ID Block#4");
b4_seq_num = Field( bitoffset = 245, size = 11, type = "IntegerU2",
             description = "Block Sequence Number Block#4");
b4_num_samp = Field( bitoffset = 256, size = 11, type = "IntegerU2",
              description = "Number of Samples Block#4");
b4_num_samp_bits = Field( bitoffset = 267, size = 5, type = "IntegerU1",
                   description = "Number Bits Per Sample Block#4");
```

Copy the contents of the above script into the file `MIP_NL__0P_MIPAS_SOURCE_PACKETS.txt`, and place the file in a directory called level0scripts.

13. GETTING HELP IN ENVIVIEW

13.1 Documentation

This document is intended to provide a complete description of the features of the EnviView application. If there are any mistakes or omissions, please send the details and any requests for more information by e:mail to enviview@jw.estec.esa.nl

In addition to this document, a README file is provided with the distribution, containing installation instructions.

13.2 On-line help

This manual is available as on-line help from within EnviView. Choose *EnviView Help* from the *Help* menu. The manual is broken up into sections. Each section title is displayed on the left and the contents of the section is displayed on the right hand side. The window may be resized, open and closed as often as one desires.

Some components also provide *tooltips*. These are small windows that appear when a cursor is held above a component (such as a button or a text field) that contain information or instructions pertinent to that component and its use.

14. COMMAND LINE TOOLS AND ENVIVIEW

The EnviView package includes not only the GUI based EnviView application, but also a growing number of tools that may be used directly from the command line. This section describes how to use each of those tools.

The following tools are provided in the standard EnviView package (version 2.0.1).

- Pds2Ascii – extract field values to an ASCII text file
- Pds2Hdf – extract datasets (and subsets) to an HDF 4 file
- PdsSubset – extract datasets to a smaller PDS child product (i.e. Pds2Pds)
- PdsPixels – extract portions of image datasets to various image file formats
- PdsPixelTrack – extract individual pixels from an image data set, along a given track
- SciaL1C – extract and calibrate Sciamachy measurement data

14.1 Maximising performance

EnviView and all it's family of command line tools are written in java. Using an efficient and optimised java runtime is essential for maximising performance of EnviView and it's tools. We currently recommend a Java 1.3.1 runtime, using the hotspot VM. The latest release for Windows, Solaris and linux x86 may be downloaded from

http://java.sun.com/j2se/1.3/jre/

Mac OS X includes a built in Java 1.3.1 runtime. Updates are included in the standard operating system updates. Mac OS 9 only provides a Java 1.1 runtime. We strongly recommend upgrading to Mac OS X for improved performance. Runtimes for other operating systems are available from the OS vendor.

Furthermore, if running the same command multiple times, improved overall execution times will be had from using multiple input files on the same command line, or by using the batch facility, see below.

14.2 Batch files

When starting one of the EnviView command lines tools, the first task that must be performed is to start the java runtime, and when the command is finished, the runtime must be closed down. If one is running a tool on multiple files, it may be useful to use the batch file facility.

Essentially the batch file provides a series of command line arguments to the tool. It's advantage is that the java runtime does not have to be restarted for every operation, thereby reducing the overhead and improving performance.

Note that if the same options are being used for every input file, most commands allow a list of input files to be given on the command line, alleviating the need to turn to a batch file.

For the tool <PdsTool> to make use of a batch file use only the arguments `–b <batchfile>` where <batchfile> is a text file containing a list of command line arguments, one line per command. For example

```
Pds2Ascii –b p2a.txt
```

Where the file p2a.txt contains:-

```
-ds 2 –field dsr_time –file time.txt GOM_NL__2PXXXX20011221X103525X0
-ds NL_GEOLOCATION –field lat –file latitude.txt GOM_NL__2PXXXX20011221X103525X0
```

Each line contains a list of command line arguments for one invocation of the command. Each of these lines is executed sequentially, from the beginning of the file to the end.

14.3 Tips for using the command line options

14.3.1 Arrow keys on command line under DOS and Windows.

One some versions of windows, the DOS terminal does not provide the ability to edit the command line with the arrow keys, or re-use previously typed commands. If this is the case, type `DOSKEY`, to add the functionality.

14.3.2 Arrow keys on command line under Unix.

One some unix shells, previously typed commands cannot be accessed with the arrow keys. If this is the case, use a shell such a `tcsh`, to add the functionality.

14.3.3 Quoting arguments containing spaces.

Some data set names have spaces in them, making it difficult to use as command line arguments. While various command line shells deal with this issue in different ways, the most widely accepted method is to place double quotes (“) either side of the argument. This is also supported in our own batch files.

14.4 Pds2Ascii

NAME

```
Pds2Ascii – extracts field values from a dataset in an Envisat PDS format file
            and exports to an ASCII text file.
```

SYNOPSIS

```
Pds2Ascii [-....] pdsfile1 [pdsfile2...]
```

DESCRIPTION

Envisat data products are stored in a non-standard file format (PDS). This tool enables field values from a specified dataset to be output to an ASCII text file.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Additional help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the data sets in the product file can be displayed. This list gives the type of the data set, either annotated (ADS) or measurement data set (MDS), the name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used in the export command to specify the data set rather than by using the full name.

-listfield "data-set-name"
-listfield n
Using this option, followed by a data set name or index, a list of the fields in the given dataset are displayed. This list gives the index of the field, the name of the field, and a description of the field. To list the fields from the MPH or SPH, use "MPH" or "SPH" respectively, as the datasets have no index.

EXTRACTING DATA

-batch "file-name"

Option for grouping multiple command lines. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For a more detailed description, go to the Batch section.

-ds "data-set-name", …
-ds n1, n2, …
This option defines which data set will be used for extracting field values.

To define the dataset either use the data set name or by a unique index (as declared by the –list option). Data set names with spaces must be enclosed in quotes.

-field "field-name", …
-f This option defines which fields from the datasets specified by the –ds option are exported. Multiple field names are comma delimited.

-file outfile.txt
This option defines the output file. If the option is not used the results are outputted to screen.

-batch batch.txt
-b Using this option, the tool will run a batch file.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
Pds2Ascii –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 2 –field dsr_time –file time.txt GOM_NL__2PXXXX20011221X103525X0
-ds NL_GEOLOCATION –f lat –file lat.txt GOM_NL__2PXXXX20011221X103525X0
```

EXAMPLES

Listing the contents of a PDS product

```
Pds2Ascii -list GOM_NL__2PXXXX20011221X103525X0
```

The output would look like this

```
Dataset details :
Index  : type :      Data set  name
1      : ads  : NL_GEOLOCATION  (contains  70  records)
2      : ads  : NL_ACCURACY_ESTIMATION (contains 70 records)
3      : gads : NL_SUMMARY_QUALITY (contains 1 records)
```

Listing the fields from a dataset

```
Pds2Ascii -listfield 2 GOM_NL__2PXXXX20011221X103525X0
```

The output would look like

```
Index    field name       description
1.     dsr_time     Start time of the Data Set Record
2.     attach_flag  Attachment Flag
...
```

To Extract fields dsr_time and attach_flag from dataset 2 and output the values into a file called results.txt

```
Pds2Ascii –ds 2 -field dsr_time, attach_flag –file results.txt
GOM_NL__2PXXXX20011221X103525X0
```

To use a batch file.

```
Pds2Ascii –b batchfile.bat
```

The batch file will contain a list of commands will be executed sequentially. Below is an example of a batchfile

```
-ds 2 –field dsr_time –file time.txt GOM_NL__2PXXXX20011221X103525X0
-ds NL_GEOLOCATION –f lat –file lat.txt GOM_NL__2PXXXX20011221X103525X0
```

14.5 Pds2Hdf

NAME

```
Pds2Hdf – convert Envisat PDS format files into HDF 4.1
```

SYNOPSIS

```
Pds2Hdf [-options] pdsfile1 [pdsfile2...]
```

DESCRIPTION

Envisat data products are stored in a non-standard file format (PDS). These files can be converted to the HDF 4.1 specification which can then be read by many common off-the shelf tools (COTS) for further analysis.

The list of files at the end of the command line will be executed sequentially with the set of options.

The output file name will be the input file name with ".hdf" appended onto the end.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the data sets in the product file can be displayed. This list gives the type of the data set, either annotated or measurement data set, name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used in the export command to specify the data set rather than by using the full name.

EXTRACTING DATA

-ds "data-set-name", …
-ds n1, n2, … This option defines which data sets and their record ranges are used in creation of the child product. To define the data set either use the data set name or by a unique index (as declared by the –list option). Data set names with spaces must be enclosed in quotes. Multiple datasets are comma delimited. If the option is not used all data sets are used in creating the child product. To define a record range for these datasets use the following firstRecord:NumberOfRecords, e.g. –ds 1,2 10:15 this will select datasets 1 and 2 with records 10 through to 24. If no record range is provided, it will use all records.

-batch "file-name"

Option for grouping multiple command line. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For more information, please go to the Batch section.

-out "directory-path"
This option specifies the output directory where the program is to place the produced files. By default, the program places the files in the directory where the program was executed.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
Pds2Hdf –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 1,2,3 –out C:\OutputDirectory GOM_NL__2PXXXX20011221X103525X0
–ds 1, 2 1:50 GOM_NL__2PXXXX20011221X103525X0
```

EXAMPLES

Listing the contents of a PDS product

```
Pds2Hdf –list ASA_IM__BPXXXX20011221X103525X0
```

The output would look like this

```
Product  file is : ASA_IM__BPXXXX20011221X103525X0
      pds2hdf v1.1b6
      Data Formats DDT v2.2 EvD 1.1 (22/10/1999)

      Index   : type :      Data set  name
       0      : mds  : MDS1  (contains  405  records)
       1      : ads  : MDS1 SQ ADS  (contains  1  records)
       2      : ads  : GEOLOCATION GRID ADS  (contains  9  records)
```

Extracting a data set

```
pds2hdf –ds MDS1 ASA_IM__BPXXXX20011221X103525X0
```

In this case, the data sets MDS1 are to be exported. Alternatively, the indices for the data sets could be found first using the -list option and the same command could have been entered as

```
pds2hdf -ds 0  ASA_IM__BPXXXX20011221X103525X0
```

Exporting a data set record set

```
pds2hdf -ds MDS1 100:50 ASA_IM__BPXXXX20011221X103525X0
```

In this case, the product is **ASA_IM__BPXXXX20011221X103525X0** and the data set **MDS1** is chosen, but only 50 records starting from the 100th record are to be exported .

14.6 PdsPixels

NAME

`PdsPixels – takes a Envisat PDS format file and extracts an area of pixels from an Image dataSet.`

SYNOPSIS

```
PdsPixels [-....] pdsfile1 [pdsfile2...]
```

DESCRIPTION

This tool extracts an area of pixels from an Image dataset. The area can be given in either pixel co-ordinates or as a geo-location. The area can also be described by a topleft and bottomright corners of a centre and radius. The output of the pixels for the area can be one of 5 formats, saved as raw pixel values, binary floating point, ascii text file delimited by either commas or tabs, or as hdf output.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the Image data sets in the product file can be displayed. For each image dataset, it will also list the quantities for the image. This list gives the type of the data set, either annotated (ADS) or measurement data set (MDS), name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used with the –dataset option to specify the data set rather than by using the full name. The list of quantities display the quantity name with an index.

EXTRACTING DATA

-batch "file-name"

Option for grouping multiple command lines. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For a more detailed description, go to the Batch section.

-dataset "data-set-name", …
-dataset n1, n2, …
-ds This option defines which image data sets are to be used. To define the data set either use the data set name or its unique index (as declared by

the –list option). Data set names with spaces must be enclosed in quotes. Multiple datasets are comma delimited.

-quantity "quantity-name", …
-quantity n1, n2, …
-q This option defines which quantities are used to extract the pixel values. To define a quantity use either the name or index (as declared by the –list option). Multiple quantities are comma delimited. If option is not used the default quantity is used.

-out "directory-path"
This option defines the output directory. If the option is not used the output directory is the current working directory.

-pixelcoord
-p This option defines that the pixel area is declared in pixel co-ordinates. If this option is not used then by default, the area is assumed to be declared in geo-location.

-topleft n1 n2
The top left corner of the area. The co-ordinates which follow the option are separated by a space. Pixel co-ordinates are integers and geo-locations are floats. Must be used with the –bottomright option.

-bottomright n1 n2
The bottom right corner of the area. The co-ordinates which follow the option are separated by a space. Pixel co-ordinates are integers and geo-locations are floats. Must be used with the –topleft option.

-centre n1 n2
The centre point of the area. The co-ordinates which follow the option are separated by a space. Pixel co-ordinates are integers and geo-locations are floats. Must be used with the –radius option.

-radius n1 n2
The radius of the area (in a vertical and horizontal directions). The co-ordinates which follow the option are separated by a space. Pixel co-ordinates are integers and geo-locations are floats. Must be used with the –centre option.

-floatingPoint
-f Use this option to set the output to be binary floating point. Cannot be used with options -asciiTab, -asciiComma or –hdf. If none of these options are set the output will be in raw pixels values.

-asciiTab
-at Use this option to set the output to be ascii text file, tab delimited. Cannot be used with options -floatingPoint, -asciiComma or –hdf. If none of these options are set the output will be in raw pixels values.

-asciiComma
-ac Use this option to set the output to be ascii text file, comma delimited. Cannot be used with options -floatingPoint, -asciiTab or –hdf. If none of these options are set the output will be in raw pixels values.

-hdf Use this option to set the output to be hdf. Cannot be used with options -floatingPoint, -asciiComma or -asciiTab. If none of these options are set the output will be in raw pixels values.

-littleEndian
-le Use this option to save the binary files in Little Endian format. If this option is not used it will save binary files in Big Endian Format. This option is only used when the output type is raw pixels or floating point.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
PdsPixels –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 3 –q 0 –f –centre 25.657 –45.685 –radius 5 10 -le
ASA_IM__BPXXXX20011221X103525X0
-ds 3 –q 0 –hdf –p –topleft 50 100 –bottomright 200 300
ASA_IM__BPXXXX20011221X103525X0
```

EXAMPLES

Listing the image data sets of a PDS product

```
PdsPixels -list ASA_IM__BPXXXX20011221X103525X0
```

The output would look like this

```
Dataset details :
Index  : type :     Data set  name
3      : mds  :     MDS1  (contains  5355  records)
  quantity index 0      : name pixel
```

To extract pixels 50, 100 to 200, 300 from data set MDS1 quantity 0 and save the file in hdf format.

```
PdsPixels -ds 3 –q 0 –hdf –p –topleft 50 100 –bottomright 200 300
ASA_IM__BPXXXX20011221X103525X0
```

To extract from a centre point 25.657N –45.685E with radius +5 N/S +10 E/W from data set MDS1 quantity 0 and save the file in floating point format in Little Endian.

```
PdsPixels -ds 3 –q 0 –f –centre 25.657 –45.685 –radius 5 10 -le
ASA_IM__BPXXXX20011221X103525X0
```

To use a batch file.

```
PdsPixels –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 3 –q 0 –f –centre 25.657 –45.685 –radius 5 10 -le
ASA_IM__BPXXXX20011221X103525X0
-ds 3 –q 0 –hdf –p –topleft 50 100 –bottomright 200 300
ASA_IM__BPXXXX20011221X103525X0
```

14.7 PdsPixelTrack

NAME

`PdsPixelTrack – takes a Envisat PDS format file and extracts individual pixels from an Image dataSet.`

SYNOPSIS

```
PdsPixelTrack [-....] pdsfile1 [pdsfile2...]
```

DESCRIPTION

This tool extracts individual pixels from an Image dataset. The pixels to extract are listed in a text file and can be stored as either a pixel co-ordinate or as a geo-location. The output is a text file which list the value, the pixel co-ordinate and the geo-location.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the Image data sets in the product file can be displayed. For each image data set, it will also list the quantities for the image. This list gives the type of the data set, either annotated (ADS) or measurement data set (MDS), name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used with the –dataset option to specify the data set rather than by using the full name. In the Quantity list, for each quantity, it will list the quantities name with an index.

EXTRACTING DATA

-batch "file-name"

Option for grouping multiple command lines. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For a more detailed description, go to the Batch section.

-dataset "data-set-name", …
-dataset n1, n2, …
-ds This option defines which image data sets are to be used. To define the data set either use the data set name or its unique index (as declared by the –list option). Data set names with spaces must be enclosed in quotes. Multiple datasets are comma delimited.

-quantity "quantity-name", …
-quantity n1, n2, …

-q This option defines which quantities are used to extract the pixel values. To define a quantity use either the name or index (as declared by the –list option). Multiple quantities are comma delimited. If option is not used the default quantity is used.

-out "directory-path"
This option defines the output directory. If the option is not used the output directory is the current working directory.

-input "file-name"
This option defines the input file. The input file contains a list of pixels to extract.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
PdsPixelTrack –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 3 –input pixels1.txt ASA_IM__BPXXXX20011221X103525X0
-ds 3 –input pixels2.txt ASA_IM__BPXXXX20011221X103525X0
-ds 3 –input pixels3.txt ASA_IM__BPXXXX20011221X103525X0
```

EXAMPLES

Listing the image data sets of a PDS product

```
PdsPixelTrack -list ASA_IM__BPXXXX20011221X103525X0
```

The output would look like this

```
Dataset details :
Index  : type :     Data set  name
3      : mds  :      MDS1  (contains  5355  records)
  quantity index 0      : name pixel
```

To extract some pixels from data set MDS1 quantity 0

```
PdsPixelTrack -ds 3 –q 0 –input pixels.txt
ASA_IM__BPXXXX20011221X103525X0
```

Where pixels.txt contains the list of pixel co-ordinates and/or geolocations. The format of the pixels.txt would be.

```
125 256
569 6
25.678 –45.689
28.159 -42.753
```

The numbers which are integers represent pixel co-ordinates and the numbers which are floats represent the latitude and longitudes.

To use a batch file.

```
PdsPixelTrack –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 3 –input pixels1.txt ASA_IM__BPXXXX20011221X103525X0
-ds 3 –input pixels2.txt ASA_IM__BPXXXX20011221X103525X0
-ds 3 –input pixels3.txt ASA_IM__BPXXXX20011221X103525X0
```

14.8 PdsSubset

NAME

PdsSubset – takes a Envisat PDS format file and creates a child product which contains a subset of the data.

SYNOPSIS

PdsSubset [-....] pdsfile1 [pdsfile2...]

DESCRIPTION

This tool creates child product files which contain a subset of the data from the parent product. The subset of data is created by defining data sets to use, record ranges within those data sets and a time range.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the data sets in the product file can be displayed. This list gives the type of the data set, either annotated (ADS) or measurement data set (MDS), name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used with the –dataset option to specify the data set rather than by using the full name.

EXTRACTING DATA

-batch "file-name"

Option for grouping multiple command lines. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For a more detailed description, go to the Batch section.

-ds "data-set-name", …
-ds n1, n2, … This option defines which data sets and their record ranges are to be used in creating the child product. To define the data set either use the data set name or its unique index (as declared by the –list option). Data set names with spaces must be enclosed in quotes. Multiple data sets are comma delimited. If the option is not used all data sets are used in creating the child product. To define a record range for these datasets use the following firstRecord:NumberOfRecords. E.g. –ds 1,2 10:15 this

will select datasets 1 and 2 with records 10 through to 24. If no record range is provide it will use all records.

-out "directory-path"

This option defines the output directory. If the option is not used the output directory is the current working directory.

-starttime t

This option defines the start time in a time range filter. It must be used with the –stoptime option. The format used for the time is DD-MMM-YYYY HH:MM:SS e.g. 15-JAN-2001 17:53:49

-stoptime t

This option defines the stop time in a time range filter. It must be used with the –starttime option. The format used is the same as starttime.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
PdsSubset –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 1,2,3 –out C:\OutputDirectory GOM_NL__2PXXXX20011221X103525X0
–ds 1, 2 1:50 –starttime 15-JAN-2001 17:53:20 –stoptime 15-JAN-2001
17:53:50 GOM_NL__2PXXXX20011221X103525X0
```

EXAMPLES

Listing the contents of a PDS product

```
PdsSubset -list GOM_NL__2PXXXX20011221X103525X0
```

The output would look like this

```
Dataset details :
Index  : type :      Data set  name
1      : ads  : NL_GEOLOCATION  (contains  70  records)
2      : ads  : NL_ACCURACY_ESTIMATION (contains 70 records)
3      : gads : NL_SUMMARY_QUALITY (contains 1 records)
```

To create a child product containing all the records in data set 1 and record 10 to 25 in dataset 2

```
PdsSubset -ds 1 –ds 2 10:15 GOM_NL__2PXXXX20011221X103525X0
```

To create a child product containing records 1 to 50 in dataset 1 and 2 and those within the time period 17:53:20 and 17:53:50 on 15-JAN-2001

```
PdsSubset –ds 1, 2 1:50 –starttime 15-JAN-2001 17:53:20 –stoptime 15-JAN-
2001 17:53:50 GOM_NL__2PXXXX20011221X103525X0
```

To use a batch file.

```
PdsSubset –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 1,2,3 –out C:\OutputDirectory GOM_NL__2PXXXX20011221X103525X0
–ds 1, 2 1:50 –starttime 15-JAN-2001 17:53:20 –stoptime 15-JAN-2001
17:53:50 GOM_NL__2PXXXX20011221X103525X0
```

14.9 SciaL1C

NAME

```
SciaL1C – convert Envisat Sciamachy Level 1B file to a Sciamachy Level 1C file
```

SYNOPSIS

```
SciaL1C [-options] sciafile1 [ sciafile2... ]
```

DESCRIPTION

The Sciamachy Level 1B files have non-calibrated measurement data. These files can be exported to the intermediate Level 1C file specification, which are calibrated to the user specification and then can be used for further analysis. The description of the Level 1C MDS can be found in the EnviView help section.

The defaults for this command line are only type Nadir is exported; no calibrations are applied; no PMD data is exported; no Fractional Polarisation data is exported. The type default changes to "all types exported" if the category, time (-starttime, -stoptime) or geolocation (-topleft, -topright) filters are applied.

The list of files at the end of the command line will be executed sequentially with the set of options.

The output file name will be the input file name with ".child" appended onto the end.

The options are all pre-fixed with a '-' and may be followed by one or more parameters. Some options have shortened forms that can be used instead of the long option name. The available options are:

RETRIEVING INFORMATION

-help
-h A description of the tool with some examples is printed out. Help on specific options can also be obtained by adding the option, including the prefix - at the end.

-list
-l Using this option, a list of the data sets in the product file can be displayed. This list gives the type of the data set, either annotated or measurement data set, name of the data set and the number of records it contains. In addition, each data set is prefixed with a unique index that can then be used in the export command to specify the data set rather than by using the full name.

EXTRACTING DATA

-ds "data-set-name", …
-ds n1, n2, …
This option defines which datasets are used in creation of the child product. To define the data set either use the data set name or its unique index (as declared by the –list option). Data set names with spaces must be enclosed in quotes. Multiple datasets are comma delimited. If the option is not used, by default, all data sets are used in creating the child product.

-out "directory-path"

This option specifies the output directory where the program is to place the produced files. By default, the program places the files in the directory where the program was executed.

-batch "file-name"

Option for grouping multiple command lines. The –batch option is followed by a file. In this file, multiple command lines are stored. The program will execute each command line in turn. For a more detailed description, go to the Batch section.

CALIBRATION OPTIONS

-allcalibrations
-allcal Option for specifying that all calibrations are applied to the data.

-calibration n1, n2, …
-cal n1, n2, … Option used to define the calibrations applied to the exported data. The values 0 to 7 are entered, comma delimited. The default is no calibrations applied. If wavelength calibration is not specified then Polarisation or Radiance calibration cannot be completed. These values correspond to
0 :: Memory Effect Correction
1 :: Leakage Correction
2 :: StrayLight Correction
3 :: PPG Correction
4 :: Etalon Correction
5 :: WaveLength Calibration
6 :: Polarisation Sensitivity Calibration
7 :: Radiance Sensitivity Calibration

-category n1, n2, …
-cat n1, n2, … Option used to export specific measurement categories. The categories indexes 0-N are entered comma delimited. For a list of these categories and their indexes, use the SciaL1C –help –cat command. This list is liable to change. Hence, it is dependant on the version of EnviView currently used.

-cluster n1, n2, …
-clus n1, n2, … Option used to define a list of clusters to be exported. A list of cluster indexes (1 to 64); comma delimited is supplied after the option.

-limbcluster n1, n2, …
-limbclus n1, n2, … Option used to define a list of Limb clusters to be exported. A list of cluster indexes (1 to 64); comma delimited is supplied after the option.

-moncluster n1, n2, …
-monclus n1, n2, … Option used to define a list of Monitoring clusters to be exported. A list of cluster indexes (1 to 64); comma delimited is supplied after the option.

-nadircluster n1, n2, …
-nadirclus n1, n2, … Option used to define a list of Nadir clusters to be exported. A list of cluster indexes (1 to 64); comma delimited is supplied after the option.

-occcluster n1, n2, …
-occclus n1, n2, … Option used to define a list of Occultation clusters to be exported. A list of cluster indexes (1 to 64); comma delimited is supplied after the option.

-noclus
Option used to export no cluster. No Measurement datasets will be output.

-fracpol
Option used to produce the Fraction Polarisation Level 1C MDS. The default is that is not used.

-pmd
Option used to produce a PMD Level 1C MDS. The default is that this is not produced.

-bottomright n1 n2
This option defines the bottom right co-ordinate in a geolocation filter. It must be used with the –topleft option. The format used for the co-ordinate is longitude latitude e.g. 23 45.5

-topleft n1 n2
This option defines the top left co-ordinate in a geolocation filter. It must be used with the –bottomright option. The format used for the location is longitude latitude e.g. 23 45.5

-starttime t
This option defines the start time in a time range filter. It may be used with the –stoptime option if the –stoptime is not used the stopime is taken from the stop time of the product. The format used for the time is DD-MMM-YYYY HH:MM:SS e.g. 15-JAN-2001 17:53:49

-stoptime t
This option defines the stop time in a time range filter. It may be used with the –starttime option, if the –starttime is not used the startime is taken from the start time of the product. The format used is the same as starttime.

-type "option", …
Option to set the type of measurement data exported. The options are 'limb ' ' nadir ' ' occultation ' ' monitoring ' ' all '. These options are listed comma delimited after –type. Default is type nadir except if a category, time or geolocation filter is applied. In this case, all types are exported.

BATCH

This option is followed by a batch filename. In the batch file, there are multiple command lines each on a separate line. Each of these separate command lines will be executed sequentially.

To use a batch file.

```
SciaL1C –b batchfile.bat
```

The batch file will contain a list of commands that will be executed sequentially. Below is an example of a batchfile

```
-ds 1,2,3 –out C:\OutputDirectory SCI_NL__1PTD-
P19960620_103652_00004200A001_00179_00176_0000.N1
–ds 1, 2 -allcal SCI_NL__1PTDP19980620_A.N1
```

EXAMPLES

Listing the contents of a Level1B product

```
SciaL1C –list SCI_NL__1PTD-
P19960620_103652_00004200A001_00179_00176_0000.N1
```

Extracting a list of clusters and specifying the calibrations

```
SciaL1C –cal 0,1 –nadirclus 1,2,3 –limbclus 6,7 SCI_NL__1PTD-
P19960620_103652_00004200A001_00179_00176_0000.N1
```

This will extract Nadir clusters 1, 2, and 3, and limb clusters 6 and 7 and export them applying memory effect and leakage correction.

Extracting a type and all calibrations:

```
SciaL1C –allcal –type nadir,occultation SCI_NL__1PTD-
P19960620_103652_00004200A001_00179_00176_0000.N1
```

This will extract all nadir and occultation clusters and export them applying all calibrations.

Extracting a data set and Fractional Polarisation values.

```
SciaL1C –ds "NEW_LEAKAGE" –noclus -fracpol SCI_NL__1PTD-
P19960620_103652_00004200A001_00179_00176_0000.N1
```

This extracts the fractional polarisation values for all types with no cluster data and copies the NEW_LEAKAGE dataset to the level 1C product.

Appendix A Installing EnviView

In short, double click on the installer for your platform, provided on the CD-ROM.

Full details of installing the EnviView package is provided in a separate document. Please refer to the **README.txt** (or **README.pdf**, or **README.htm**) on the CD-ROM distribution for detailed instructions on installation.

Appendix B .Processing EnviView Data with IDL

EnviView can export Envisat data products, and data sets to HDF files. The HDF format is supported by many off the shelf data analysis tools, including Noesys, IDL, ENVI from Research Systems Inc, and PV-Wave from Visual Numerics.

This chapter includes instructions and examples of reading HDF data exported from EnviView in IDL. NOTE: HDF support in EnviView requires the addition of a native library compiled for your platform. If the library is not available, or EnviView cannot locate it, the menu options referred to below are not accessible. Choose 'About EnviView in the Help menu to determine whether EnviView can locate the correct library. In case of difficulty, please refer to the installation instructions.

A detailed description of the organisation of Envisat data in HDF4 files is given in Section 11.2.1.

For further information about HDF, please refer to the NCSA HDF 4 User Guide and Reference guide provided on the EnviView CD.

B.1 Introduction to examples

A set of utility functions is used here for reading this data into your own IDL programs. For your convenience, all these routines have been collected together in one file EnvisatHdf.pro. The contents of this file are listed at the end of this appendix.

To prepare these examples, the simulated L1B and L2 AATSR data files were converted as follows: -

```
pds2hdf -p ATS_TOA_1PTRAL19950510_071649_00000000X000_00000_00000_0000.N1
-out L1.hdf

pds2hdf -p ATS_NR__2PTRAL19950510_071649_00000000X000_00000_00000_0000.N1
-out L2.hdf
```

NOTE These examples are not specific to AATSR. The same principles are used to read any data element from an Envisat HDF file. Only the Data Set name and variable names need to change. The List utility functions allow a user to determine the names of any items in the file.

The important concepts are that all fields for a given data set in the Envisat file are stored together in a group in the HDF file. Scalar fields are collected in a table (VData), with one row (record) per Envisat data set record. Array fields are stored as individual array objects (SDS).

The following examples demonstrate how to read items from a VData or SDS into IDL variables. For further information, read Chapter 3 of the IDL manual "Scientific Data Formats", and feel free to inspect the code if the utilities in EnvisatHdf.pro

B.2 Load the utilities into IDL

Copy the EnvisatHdf.pro file from the EnviView CD to the local directory. Then type the following at the IDL prompt.

```
IDL> .run EnvisatHdf.pro
```

This file provides a set of convenience routines to make reading Envisat data from HDF files much easier.

B.3 Listing the datasets in an Envisat HDF file

Using the utilities, to List the Envisat data sets in the HDF file :-

```
IDL> listPdsDataSets, "L1.hdf"
MPH
SPH
SUMMARY_QUALITY_ADS
GEOLOCATION_ADS
SCAN_PIXEL_X_AND_Y_ADS
NADIR_VIEW_SOLAR_ANGLES_ADS
FWARD_VIEW_SOLAR_ANGLES_ADS
VISIBLE_CALIB_COEFS_GADS
NADIR_VIEW_SCAN_PIX_NUM_ADS
FWARD_VIEW_SCAN_PIX_NUM_ADS
11500_12500_NM_NADIR_TOA_MDS
10400_11300_NM_NADIR_TOA_MDS
03505_03895_NM_NADIR_TOA_MDS
01580_01640_NM_NADIR_TOA_MDS
00855_00875_NM_NADIR_TOA_MDS
00649_00669_NM_NADIR_TOA_MDS
00545_00565_NM_NADIR_TOA_MDS
11500_12500_NM_FWARD_TOA_MDS
10400_11300_NM_FWARD_TOA_MDS
03505_03895_NM_FWARD_TOA_MDS
01580_01640_NM_FWARD_TOA_MDS
00855_00875_NM_FWARD_TOA_MDS
00649_00669_NM_FWARD_TOA_MDS
00545_00565_NM_FWARD_TOA_MDS
NADIR_VIEW_CONFIDENCE_MDS
FWARD_VIEW_CONFIDENCE_MDS
NADIR_VIEW_CLOUD_MDS
FWARD_VIEW_CLOUD_MDS

IDL> listPdsDatasets, "L2.hdf"
MPH
SPH
SUMMARY_QUALITY_ADS
GEOLOCATION_ADS
SCAN_PIXEL_X_AND_Y_ADS
NADIR_VIEW_SOLAR_ANGLES_ADS
FWARD_VIEW_SOLAR_ANGLES_ADS
NADIR_VIEW_SCAN_PIX_NUM_ADS
FWARD_VIEW_SCAN_PIX_NUM_ADS
DISTRIB_SST_CLOUD_LAND_MDS
```

B.4 Listing the contents of an Envisat HDF data set

Each data set is a HDF VGroup (a kind of inner directory), named after the DS_NAME of the data set (in the DSD at the beginning of the Envisat file). Each one contains a VData table (also named after the DS_NAME) which contains all the scalar variables in the data set. Array variables are stored in individual SDS within the same VGroup.

To list the variables in a data set :-

```
IDL> listpdsdataset, "L1.hdf", '00649_00669_NM_FWARD_TOA_MDS'
00649_00669_NM_FWARD_TOA_MDS
     00649_00669_NM_FWARD_TOA_MDS        2272 records
        .Record Index  ULONG
        .dsr_time  DOUBLE
        .quality_flag  BYTE
        .img_scan_y  LONG
     bt_rad_pix (SDS) : INT [512,2272]
```

This first lists the scalar variables in the Vdata table, which has the same name as the Vgroup (the data set DS_NAME). There is one value per record (2272 records). These variables are :-

```
record index, dsr time, quality flag and image scan line y
```

It then lists the array variables (SDS), in this case the image pixels, a 2d array, 512 pixels by 2272 records, named `bt_rad_pix`.

B.5 Reading data from an SDS

This example function will return an entire image into an IDL variable :-

```
FUNCTION LoadEnvisatImage, filename, datasetname, pixelfieldname

    ; Open The HDF file

    Hid = HDF_OPEN(filename, /READ)

    ; Start the SD interface

    SDid = HDF_SD_START(filename, /READ)

    ; Open the VGroup table containing the image field

    SDSid = DATASET_SDS_ATTACH(Hid, SDid, datasetname, pixelfieldname)

    IF (SDSid EQ -1) THEN BEGIN
        PRINT, "Data Set '"+datasetname+"."+pixelfieldname+"' not found."
        HDF_SD_END, SDid
        HDF_CLOSE, Hid
        RETURN, -1
    ENDIF

    ; Get information about the size of the data

    HDF_SD_GETINFO, SDSid, DIMS = dimensions

    ; Read the data

    HDF_SD_GETDATA, SDSid, pixels

    ; Close the SDS

    HDF_SD_ENDACCESS, SDSid

    ; Close the HDF file

    HDF_SD_END, SDid
    HDF_CLOSE, Hid

    return, pixels
END
```

This function might be used as follows :-

```
IDL> img=loadEnvisatImage("L1.hdf", '00649_00669_NM_FWARD_TOA_MDS',
'bt_rad_pix')
```

The size of the resulting image :-

```
IDL> help, img
IMG             INT       = Array[512, 2272]
```

Displaying part of the image :-

```
IDL> tvscl, img(*,1500:2000)
```

B.6 Reading data from a Vdata

Here is an example function reading the time and image scan parameters from the VData table in the same data set.

```
PRO PlotScanVsTime, filename

    ; Open The HDF file

    Hid = HDF_OPEN(filename, /READ)

    ; Open the VData table containing the scalar values

    datasetname = '00649_00669_NM_FWARD_TOA_MDS'

    VDname = datasetname

    VDid = DATASET_VD_ATTACH(Hid, datasetname, VDname)

    IF (VDid EQ -1) THEN BEGIN
        PRINT, "Data Set '"+datasetname+"."+VDname+"' not found."
        HDF_CLOSE, Hid
        RETURN
    ENDIF

    ; Determine how many records are in the table

    HDF_VD_GET, VDid, count=totalrecs

    ; Read the time values from all records

    Nrec = HDF_VD_READ(VDid, time, FIELDS="dsr_time", NRECORDS=totalrecs)

    ; Rewind the record counter

     HDF_VD_SEEK, VDid, 0

    ; Read the image scan y field

     Nrec = HDF_VD_READ(VDid, scany, FIELDS="img_scan_y",NRECORDS=totalrecs)

    ; Close the VData table

    HDF_VD_DETACH, VDid

    HDF_CLOSE, Hid

      ; Plot the data

      plot, time, scany
END
```

B.7 EnvisatHdf.pro

The contents of the EnvisatHdf.pro is listed for reference below. It includes the following functions :-

Three routines that list the data in an Envisat HDF file.

- List all the PDS data sets in the named HDF file

  PRO ListPdsDataSets, filename

- List all the fields in the named PDS data set in the named HDF file

  PRO ListPdsDataSet, filename, datasetname

- List all the fields in all PDS data sets in the named HDF file

  PRO ListPdsDataSetFields, filename

HINT: These routines are very helpful in determining the names used for specific data sets or fields. These names can also be consulted in EnviView under the product formats menu, or while plotting data.

Two functions that attach to Envisat data structures in an HDF file:

- Attach to an SDS given a dataset name and the SDS (field) name. Returns SDSid.

  FUNCTION DATASET_SDS_ATTACH, Hid, SDid, DATASET_NAME, SDS_NAME

  Hid is provided by the IDL function HDF_OPEN (Close with HDF_CLOSE)
  SDid is provided by HDF_SD_START (End with HDF_SD_END)

  Use the IDL routines

  HDF_SD_GETINFO to obtain information about the SDS

  HDF_SD_GETDATA to read from the SDS

  HDF_SD_ENDACCESS to close the SDS

- Attach to an Vdata given a dataset name and the Vdata name. Returns VDid

  FUNCTION DATASET_VD_ATTACH, Hid, DATASET_NAME, VDATA_NAME

  Hid is provided by the IDL function HDF_OPEN (Close with HDF_CLOSE)

  Use the IDL routines

  HDF_VD_GET to obtain information about the Vdata

  HDF_VD_READ to read records from the Vdata

  HDF_VD_SEEK to skip records in the Vdata

  HDF_VD_DETATCH to close the Vdata

```
;----------------------------- Utility functions -----------------------------;

;
; Search an open VGroup for a named VData
;
; return the HDF REF to the VData
;
FUNCTION GET_VDATA_REF, Hid, VGid, VDname

       N = HDF_VG_NUMBER(VGid)

       FOR i = 0, N-1 DO BEGIN
              HDF_VG_GETTR, VGid, i, TAG, REF
              IF (HDF_VG_ISVD(VGid, REF)) THEN BEGIN

                     VDid = HDF_VD_ATTACH(Hid, REF)

                     HDF_VD_GET, VDid, NAME=VD_NAME
                     HDF_VD_DETACH, VDid

                     IF VD_NAME EQ VDname THEN return, REF
              ENDIF
       ENDFOR

       return, -1
END

;
; Search an open VGroup for a named SDS
;
; return the HDF REF to the SDS
;
FUNCTION GET_SDS_REF, SDid, VGid, SDSname

       N = HDF_VG_NUMBER(VGid)

       FOR i = 0, N-1 DO BEGIN
              HDF_VG_GETTR, VGid, i, TAG, REF

              ; Check to see if it is an SDS

              IF (TAG EQ 720) THEN BEGIN

                     SDSidx = HDF_SD_REFTOINDEX(SDid, REF)

                     SDSid = HDF_SD_SELECT(SDid, SDSidx)

                     HDF_SD_GETINFO, SDSid, NAME=SDS_NAME
                     HDF_SD_ENDACCESS, SDSid

                     IF SDS_NAME EQ SDSname THEN return, REF
              ENDIF
       ENDFOR

       return, -1
END
```

```
;
; Search an open VGroup for a named SDS
;
; return the SDS index
;
FUNCTION GET_SDS_INDEX, SDid, VGid, SDSname

       N = HDF_VG_NUMBER(VGid)

       FOR i = 0, N-1 DO BEGIN
              HDF_VG_GETTR, VGid, i, TAG, REF

              ; Check to see if it is an SDS

              IF (TAG EQ 720) THEN BEGIN

                     SDSidx = HDF_SD_REFTOINDEX(SDid, REF)

                     SDSid = HDF_SD_SELECT(SDid, SDSidx)

                     HDF_SD_GETINFO, SDSid, NAME=SDS_NAME
                     HDF_SD_ENDACCESS, SDSid

                     IF SDS_NAME EQ SDSname THEN return, SDSidx
              ENDIF
       ENDFOR

       return, -1
END

;
; Search the top level directory for a named VGroup
;
; return the HDF REF to the VGroup
;
FUNCTION GET_VGROUP_REF, Hid, VGname

  VGs = HDF_VG_LONE(Hid)

  IF VGs(0) EQ -1 THEN RETURN, -1

       N = N_ELEMENTS(VGs)

       FOR i = 0, N-1 DO BEGIN
              VGid = HDF_VG_ATTACH(Hid, VGs(i))
              HDF_VG_GETINFO, VGid, NAME=DS_NAME
              HDF_VG_DETACH, VGid

              IF (DS_NAME EQ VGname) THEN RETURN, VGs(i)
       ENDFOR

       return, -1
END
```

```
;
; Attach to a named Vdata in a named Vgroup
;
; Hid
;          Identifier of the HDF file (fromHDF_OPEN)
;
; DATASET_NAME
;          name of the data set (VGroup)
; VDATA_NAME
;          name of the VData in the VGroup
;
FUNCTION DATASET_VD_ATTACH, Hid, DATASET_NAME, VDATA_NAME

       VG_REF = GET_VGROUP_REF(Hid, DATASET_NAME)

       IF (VG_REF EQ -1) THEN RETURN, -1

       VGid = HDF_VG_ATTACH(Hid, VG_REF)

       ; Find the VData with the given name

       VD_REF = GET_VDATA_REF(Hid, VGid, VDATA_NAME)

       HDF_VG_DETACH, VGid

       IF (VD_REF EQ -1) THEN RETURN, -1

       VDid = HDF_VD_ATTACH(Hid, VD_REF)

       RETURN, VDid
END

;
; Attach (Select) to a named SDS in a named Vgroup
;
; Hid
;          Identifier of the HDF file (from HDF_OPEN)
;
; SDid
;          Identifier of the SD i/f to the file (from HDF_SD_START)
;
; DATASET_NAME
;          name of the data set (VGroup)
; SDS_NAME
;          name of the SDS in the VGroup
;
FUNCTION DATASET_SDS_ATTACH, Hid, SDid, DATASET_NAME, SDS_NAME

       VG_REF = GET_VGROUP_REF(Hid, DATASET_NAME)

       IF (VG_REF EQ -1) THEN RETURN, -1

       VGid = HDF_VG_ATTACH(Hid, VG_REF)

       ; Find the SDS with the given name

       SDS_INDEX = GET_SDS_INDEX(SDid, VGid, SDS_NAME)

       HDF_VG_DETACH, VGid

       IF (SDS_INDEX EQ -1) THEN RETURN, -1

       SDSid = HDF_SD_SELECT(SDid, SDS_INDEX)

       RETURN, SDSid
END
```

```
;------------------------------- List Functions -------------------------------;

;
; List the fields in a given VGroup
;
; Hid
;        HDF file ID returned by HDF_OPEN
; REF
;        VGroup REF
;
PRO ListPdsVGroupFields, Hid, SDid, REF
       VGid = HDF_VG_ATTACH(Hid, REF)
       HDF_VG_GETINFO, VGid, NAME=DS_NAME, CLASS=DS_CLASS, NENTRIES=DS_ENTRIES
       IF (DS_CLASS EQ "PDS Data Set Record") THEN BEGIN
              Print, DS_NAME

              ; Find the VData with the same name as the VGroup

              MAIN_REF=GET_VDATA_REF(Hid, VGid, DS_NAME)

              ; List it's fields

              ListVData, Hid, MAIN_REF

              ; List the fields of any other groups

              N = HDF_VG_NUMBER(VGid)

              FOR j = 0, N-1 DO BEGIN

                     HDF_VG_GETTR, VGid, j, TAG, REF
                     IF (HDF_VG_ISVD(VGid, REF) AND (REF NE MAIN_REF)) THEN BEGIN
                            ListVData, Hid, REF
                     ENDIF ELSE IF (TAG EQ 720) THEN BEGIN ; is SDS
                            ListSDS, SDid, REF
                     ENDIF
              ENDFOR

       ENDIF
       HDF_VG_DETACH, VGid

END
```

```
;
; List the fields in a given VData
;
; Hid
;        HDF file ID returned by HDF_OPEN
; REF
;        VData REF returned by HDF_VG_GETTR
;
PRO ListVData, Hid, REF

       VDid = HDF_VD_ATTACH(Hid, REF)

       HDF_VD_GET, VDid, NAME=VD_NAME, NFIELDS=VD_FIELD_COUNT, COUNT=VD_RECORD_COUNT

       PRINT, "     ",VD_NAME, VD_RECORD_COUNT, " records"

       FOR i=0, VD_FIELD_COUNT-1 DO BEGIN
              HDF_VD_GETINFO,VDid, i, NAME=FIELD_NAME, TYPE=FIELD_TYPE,
                                   ORDER=FIELD_ELEMENTS
              IF (FIELD_ELEMENTS GT 1) THEN BEGIN
                     PRINT, "        .",FIELD_NAME + $
                            "["+STRING(FIELD_ELEMENTS,FORMAT="(I0)") + "]  " + FIELD_TYPE
              ENDIF ELSE BEGIN
                     PRINT, "        .",FIELD_NAME + "  "+FIELD_TYPE
              ENDELSE
       ENDFOR


       HDF_VD_DETACH, VDid
END

;
; List the fields in a given SDS
;
; SDid
;        SD file ID returned by HDF_SD_START
; REF
;        SDS REF returned by HDF_VG_GETTR
;
PRO ListSDS, SDid, REF


       SDSidx = HDF_SD_REFTOINDEX(SDid, REF)

       SDSid = HDF_SD_SELECT(SDid, SDSidx)

       HDF_SD_GETINFO, SDSid, NAME=SDS_NAME
       HDF_SD_ENDACCESS, SDSid


       HDF_SD_GETINFO, SDSid, NAME=SDS_NAME, DIMS=dimensions, TYPE=field_type, NDIMS=Nd

       dim_str = STRING(dimensions,FORMAT='('+STRING(Nd)+'(I0,","))')
       dim_str = STRMID(dim_str, 0, STRLEN(dim_str)-1)

       PRINT, "     ",SDS_NAME+" (SDS) : "+field_type+" ["+dim_str+"]"

       HDF_SD_ENDACCESS, SDSid
END
```

```
;
; List all the PDS data sets in the named HDF file
;
PRO ListPdsDataSets, filename
  Hid = HDF_OPEN(filename, /READ)

  VGs = HDF_VG_LONE(Hid)

  IF VGs(0) EQ-1 THEN BEGIN
       PRINT, "No Data Sets (VGroups)"
  ENDIF ELSE BEGIN
       FOR i=0,N_ELEMENTS(VGs)-1 DO BEGIN
              VGid = HDF_VG_ATTACH(Hid, VGs(i))
              HDF_VG_GETINFO, VGid, NAME=DS_NAME, CLASS=DS_CLASS, NENTRIES=DS_ENTRIES
              IF (DS_CLASS EQ "PDS Data Set Record") THEN BEGIN
                     Print, DS_NAME
              ENDIF
              HDF_VG_DETACH, VGid
       ENDFOR
  ENDELSE

  HDF_CLOSE, Hid
END

;
; List all the fields in the named PDS data set in the named HDF file
;
PRO ListPdsDataSet, filename, datasetname

  Hid = HDF_OPEN(filename, /READ)
  SDid = HDF_SD_START(filename, /READ)

  VG_REF = GET_VGROUP_REF(Hid, datasetname)

  IF (VG_REF EQ -1) THEN BEGIN

       PRINT, "Data Set '"+datasetname+"' not found."
       HDF_CLOSE, Hid
       RETURN
  ENDIF

  ListPdsVGroupFields, Hid, SDid, VG_REF

  HDF_SD_END, SDid
  HDF_CLOSE, Hid
END

;
; List all the fields in all PDS data sets in the named HDF file
;
PRO ListPdsDataSetFields, filename
  Hid = HDF_OPEN(filename, /READ)
  SDid = HDF_SD_START(filename, /READ)

  VGs = HDF_VG_LONE(Hid)

  IF VGs(0) EQ-1 THEN BEGIN
       PRINT, "No Data Sets (VGroups)"
  ENDIF ELSE BEGIN
  FOR i=0,N_ELEMENTS(VGs)-1 DO BEGIN
              ListPdsVGroupFields, Hid, SDid, VGs(i)
       ENDFOR
  ENDELSE

  HDF_SD_END, SDid
  HDF_CLOSE, Hid
END
```