# COMEXT
# USER GUIDE


9.1.5

# INTRODUCTION

## Welcome to COMEXT!

## What you will learn from this user manual.

In preparing this guide, we have made every effort to avoid technical terminology and computing jargon. This guide, therefore, explains how to work with COMEXT using easy, non-technical language.

It shows the different steps to follow to prepare requests for data in various formats.

This manual is divided up into five sections:

- INTRODUCTION
- THE PLAN
- THE EXTRACTION
- USERLISTS/AGGREGATES
- ADVANCED FUNCTIONS

## Prerequisites

To use the system efficiently, you must know how to use Microsoft Windows. If you have little or no knowledge of Microsoft Windows, part I of the Microsoft Windows user guide (Basic Principles of Windows) provides a useful reference.

# Requirements

## Architecture

Using COMEXT remotely manipulates the data at the central site. The extraction of data (for example plans) starts a downloading operation, and displays the results on your screen.

## Hardware

To use COMEXT, a PC equipped with a Pentium 1.4Ghz Processor and at least 256MB of memory is recommended. Please check with your informatics support team to make sure you have the recommended equipment.

# Terms and definitions

## Client-Server

These two terms refer to a system architecture where all processing is performed by a central machine (the server), and the results are transmitted to a remote machine (the client). The advantage of this model is that the client machine needs little processing power because its main task is only to display information.

## The Database

The Database is where the (statistical) data is stored and which enables extractions of subsets of data according to certain rules.

## The Dataset

A dataset comprises a large set of related data and is valid for a specific period. For example, the dataset "**Monthly, Since 1988**" under [Trade Domain (EU)/Trade by Product(CN)] includes the EU trade data since 1988.

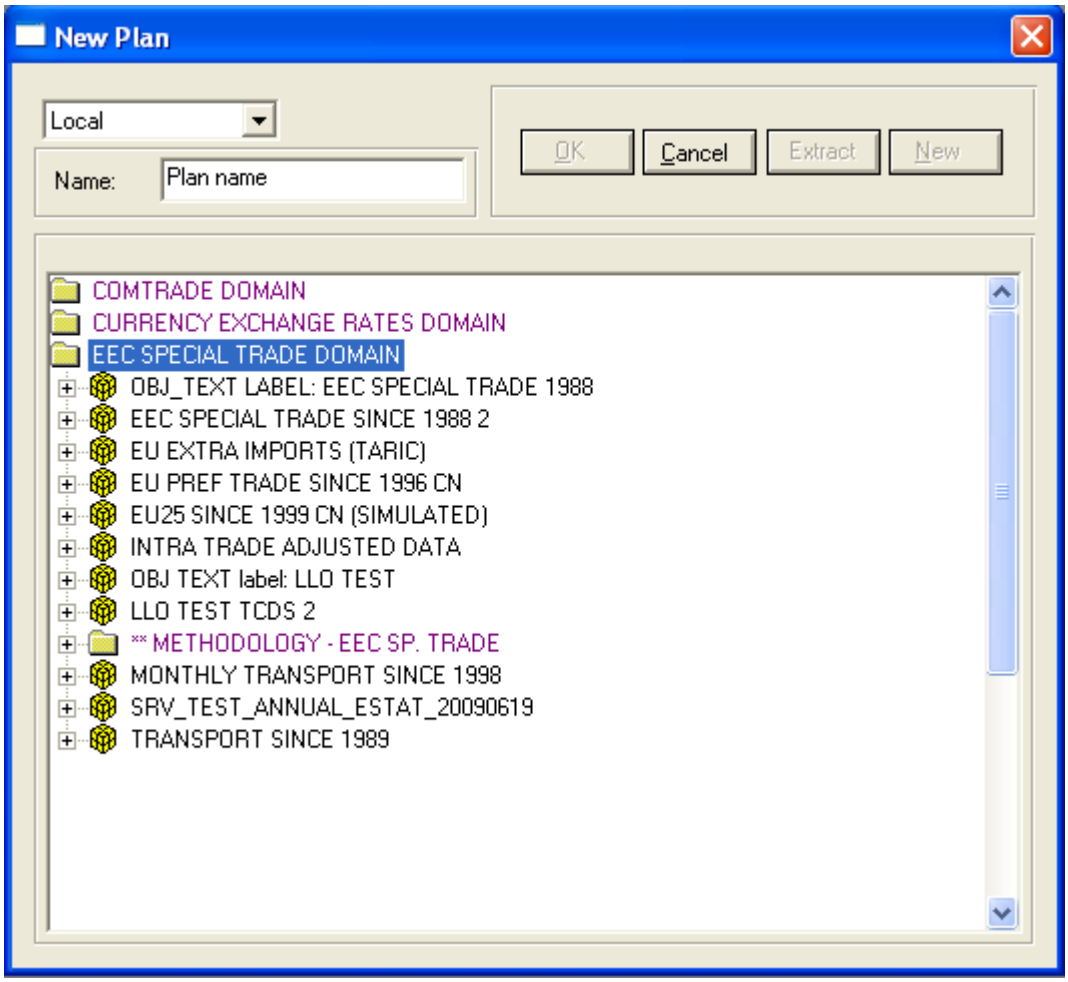



**Figure 1: Examples of datasets.**

Each dataset has different content. The number of parameters (dimensions) and their content will depend on the dataset selected.

# The Dimension

Dimensions are the different parameters for statistical information contained in a dataset. They are directly related to the nature of the information they represent, and for this reason, their number and content vary according to the selected dataset. An example of COMEXT dimensions is shown in Figure 2 on the next page.

# The Plan

The plan is the scheme used to save all the parameters (dimensions) of an extraction. It can be re-used and changed so that the parameters do not need to be specified again.

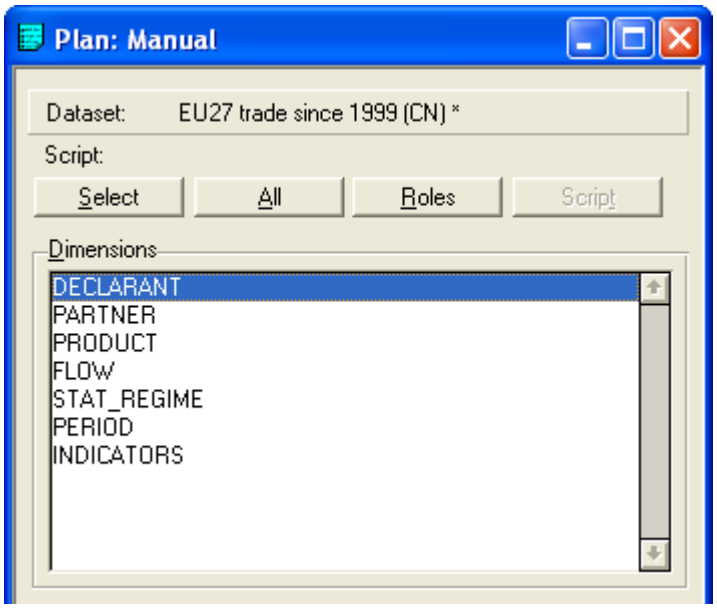



**Figure 2: The dimensions of dataset.**

# The Nomenclatures

A nomenclature represents a statistical entity or parameter (product, country) using a code.

For example, the “Combined Nomenclature” classifies available products by 8-digit codes. Because rules for these classifications are not the same worldwide, COMEXT stores statistical information using several different nomenclatures.

# Starting the COMEXT Client

## Login

Using the mouse, double-click on the COMEXT 

icon to start the system.

## Selecting the Language:

COMEXT allows nomenclatures to be displayed in several languages. The selected language remains until it is changed. The procedure for setting the language is shown below. To change the language, simply follow the same procedure again.

The commands and menus of COMEXT are always in English, regardless of the selected label language.

**To select the language:**

1) From the Main menu, click Option
2) Select Label's Language
3) Click on the selected language
4) Click on OK

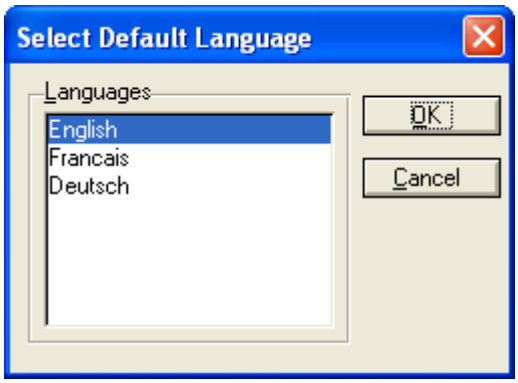



**Figure 3: The "Set Labels's Language" window**

In the above example, English has been selected. All dimension labels will display in French.

# THE PLAN

Through COMEXT, the extraction is defined according to the Plan. The plan is built from a Dataset and composed of Dimensions. Each dimension of a Plan may be filled with one or more codes to perform an extraction. If no code was selected for a dimension, all codes are included by default. At least one indicator must be selected.

# Creating a new plan

**To create a new plan:**

1) From the Main Menu, select Open Extraction File
2) Select the Eurotrace Domain to be open

The "New Plan" window is open

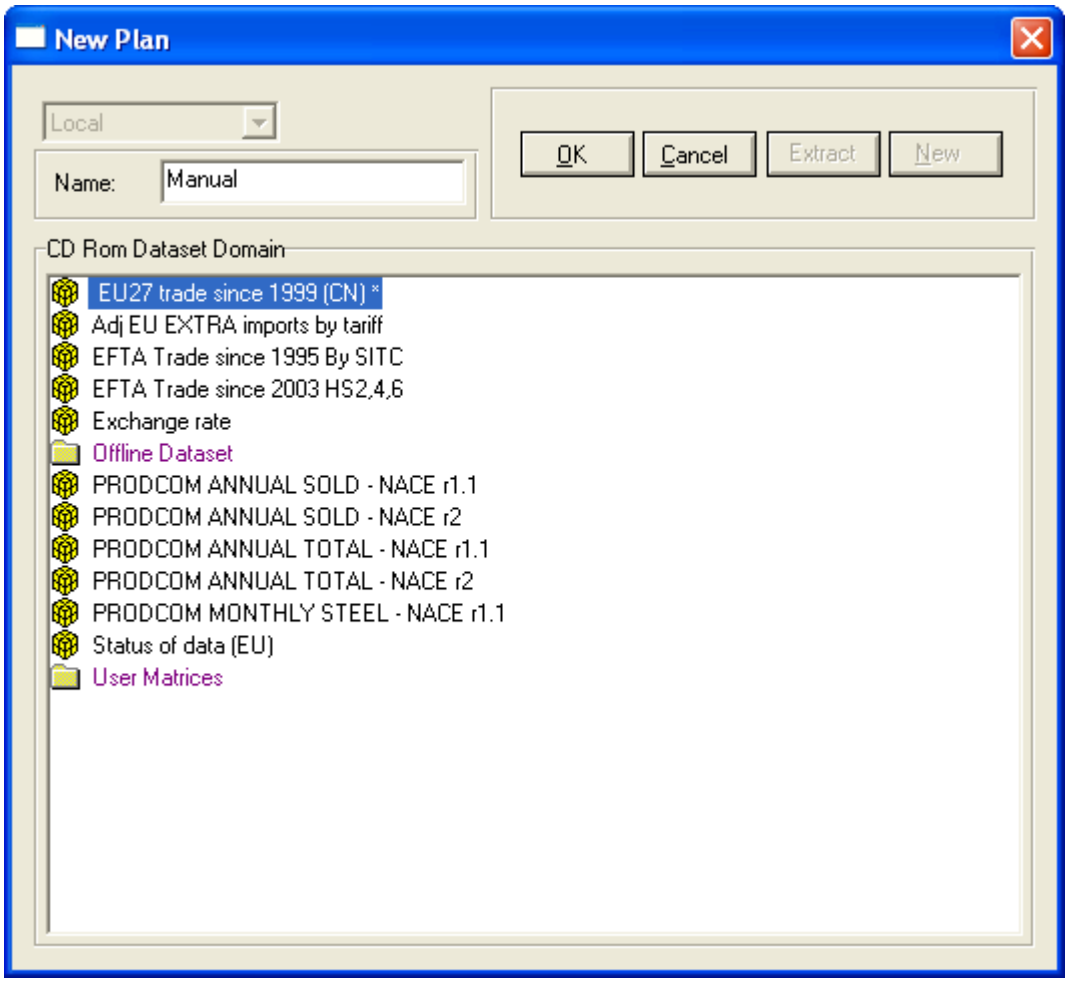


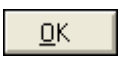

**Figure 4: The New Plan window**

3) From the menu New Plan, give a name to the Plan in the Name Text Box


4) From the menu New Plan, select a dataset

You must then select the appropriate dataset. In the above example, we will select "Trade Domain (EU)", then "Trade by Product (CN)" and finally “Monthly, Since 1988”. This will provide us with monthly data available since the year 1988.

5) Click on OK Button



# Choosing the Dimensions

Once your plan has been defined (when you click "OK" in selection window), the dimensions window will automatically appear. The dimensions (number, order and name) are related to each dataset.

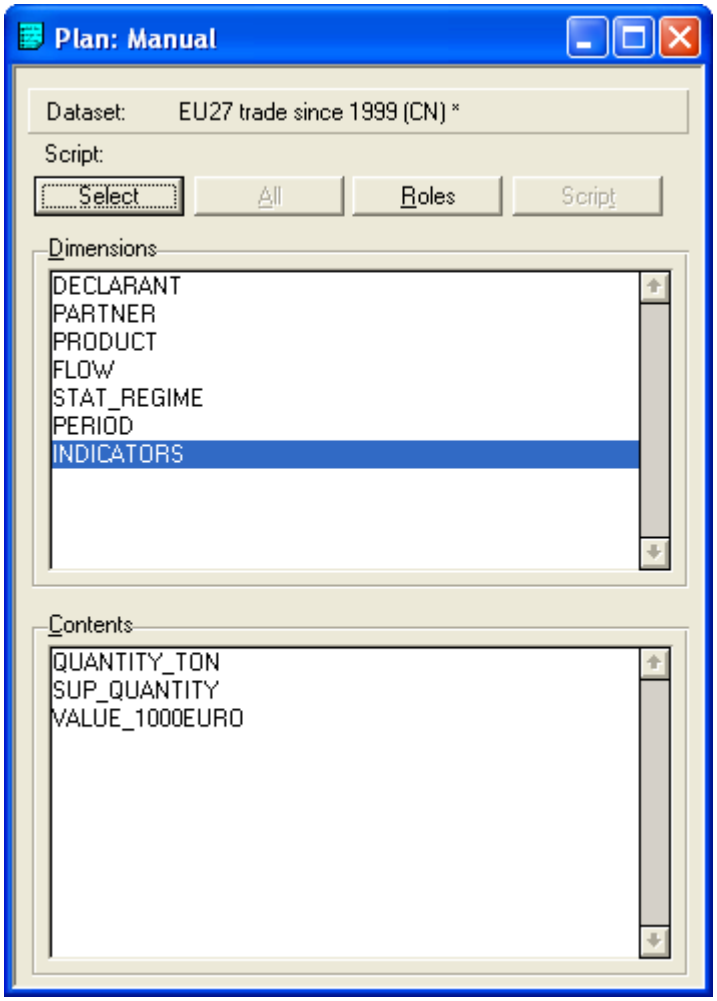



**Figure 5: "The Plan" Window**

In the selected dataset, the dimensions are as follows:

- PERIOD
- YEAR
- TARIFF
- PARTNER
- FLOW
- INDICATORS

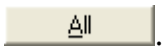
**To edit a dimension (selection of codes):**

1) From the Main menu, click Extract / Edit Dimension. You can also open the Dimension windows by using the Select button on the Top of the dimensions window or double click on the dimension name in the list.

The dimension window will appear.

Note: The window for a Dimension will always be the same for any dimension of any dataset.

If you want to select all the available code for a dimension, you can use the All button on the dimensions window.




# Window of a Dimension

In all the dimensions, the selection window will be the same, with the commands: Filter, Add, Add All, Clear, Remove, Edit, Sort, Hierarchy, Cancel and Close.

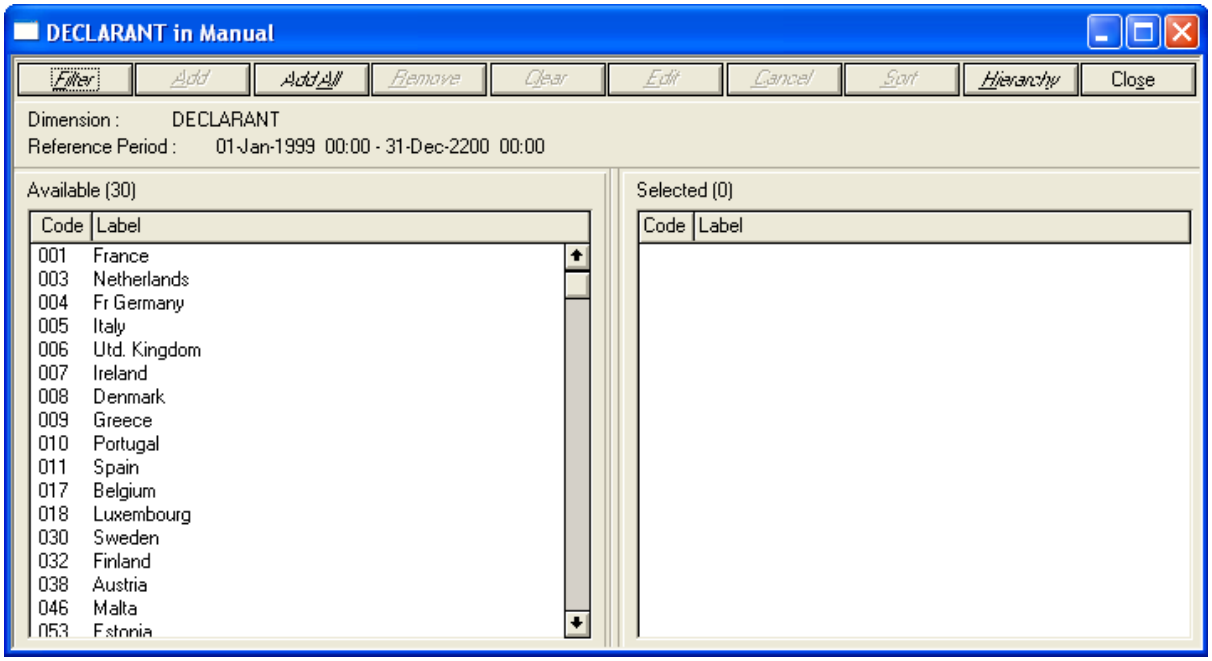



**Figure 6: Window of a dimension.**

The Dimension window is composed of two sub windows:

AVAILABLE:

> This is the window that contains all available codes and labels. This sub window is located on the Left of the main window

SELECTED:

> This is the window that contains all codes and "Available" labels you have selected. This sub window is located on the Right side of the main window

When filling a dimension, select the required code(s) from the “Available” window and transfer them into the “Selected” window.

**To fill a dimension, follow these two steps:**

1) Find the requested code in the “Available” window
2) Transfer the selection to “Selected” window

Note: Two buttons in the Dimension window are not linked to the codes:

CANCEL: if you have made changes in "SELECTED" and you wish to remove them, this function allows you to undo these changes.


CLOSE: Closes the window (identical to clicking on the button) and returns to the dimension definition window.


# To Find the code(s) in a dimension (the Available window)

COMEXT offer several ways of finding the requested code(s). According to the nomenclature, you will need to choose one of the following methods:

- **Selecting Code**

Select a code (or a group of codes) in the Available window by using the mouse

**To select a Code:**

1) In Available window, click on the code.

The selected code is then highlighted.

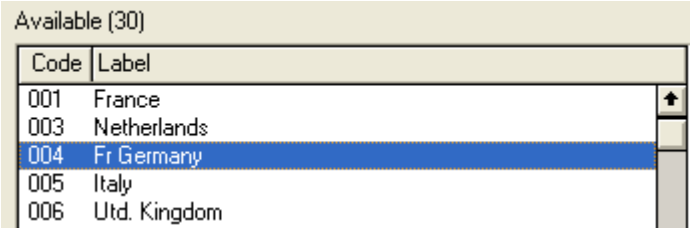



**Figure 7: Germany is selected.**

Note: Use the Shift and Ctrl keys on the keyboard to select several codes at once

- **Quick find Code/Label**

**If the code or label is known use the "Quick Find" Box, to quick find Code /Label:**

1) In Dimension windows, click Code or Label to open the Code or Label text box


2) Type the Code or Label into the text box

The requested code will then be selected.

- **Filter:**

For any code or label search, filters can be introduced to reduce the system search field and speed up the work.

The search for a product may be painstaking when working with nomenclatures comprising more than 20,000 codes, as in the case of the Combined Nomenclature (NC). With COMEXT, it is possible to introduce filters to narrow the search and speed up the work. This

function is accessed via the code selection window by activating the "Filter" button. The filter selection window will then display:

**To use Filter option:**

1) On Dimension window, click Filter to open the Filter window


2) In "Filter On" pane, select either "Code" or "Label".

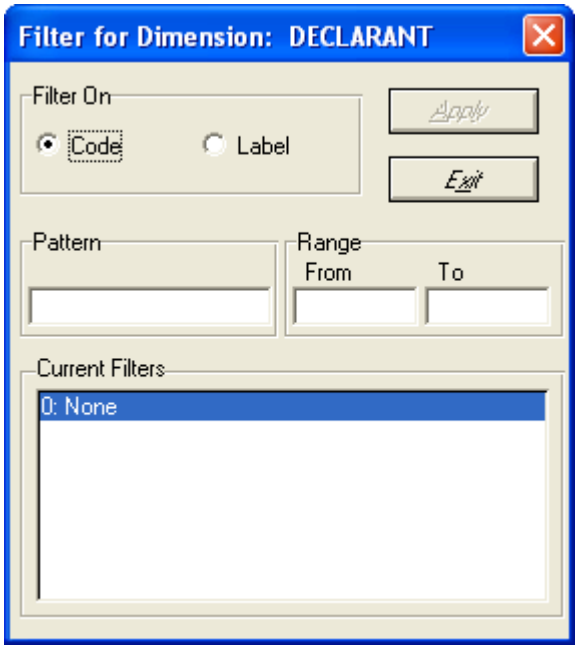



**Figure 8: Filter window with filter activated on Label.**

The search type must now be defined. As indicated in the selection window description, the basic filter functions allow a search to be carried out with the aid of two symbols, which are '?' and '*'.

Note: A '?' (question mark) is used to mask one position of the code or label whereas an '*' (asterisk) is used to mask zero or more positions.

3) Enter filter parameters in the filter window, either in the "PATTERN" or "RANGE" fields.

The "PATTERN" field carries out a search for a particular type of code or label conforming to a specified 'pattern', and the "RANGE" field searches for codes that fall within a specified range.

Here are some examples that will give you a better understanding of the filter function:

**Code filter, in "PATTERN":**

| Input mask | Result requested |
|---|---|
| 2? | Two-position codes starting with the digit '2'. |

| | |
|---|---|
| 2* | All codes starting with the digit ‘2’ |
| *2* | All codes containing the digit ‘2’. |

**Code Filter, in "RANGE":**

| FROM | TO | Result requested |
|---|---|---|
| 2 | 3 | All codes starting with ‘2’ and code 3 |

Note: the parameter encoded under "TO" is included in the selection. Even numerical codes are compared as words, i.e. code ‘15’ is between ‘1’ and ‘2’.

**Label filter**

| Input mask | Result requested |
|---|---|
| CARS | All codes with label ‘CARS’ and only ‘CARS’. |
| *CARS | All codes with label ending in ‘CARS’. |
| *CARS* | All codes with label containing ‘CARS’. |

First choose the type of search to be carried out. You may opt for a search based on digits ("codes") or on characters ("labels").

A ‘?’ replaces a character or digit.

A ‘*’ replaces all the characters or digits.

**The filter: The hierarchies**

When carrying out a search, the filter window displays the type of filter(s) applied and the order of execution. The "CURRENT FILTERS" part of the filter window displays the hierarchical tree of the different filters. This tree will allow you to return, at any time, to the required level. To restart a main search, return to "0:None" level.

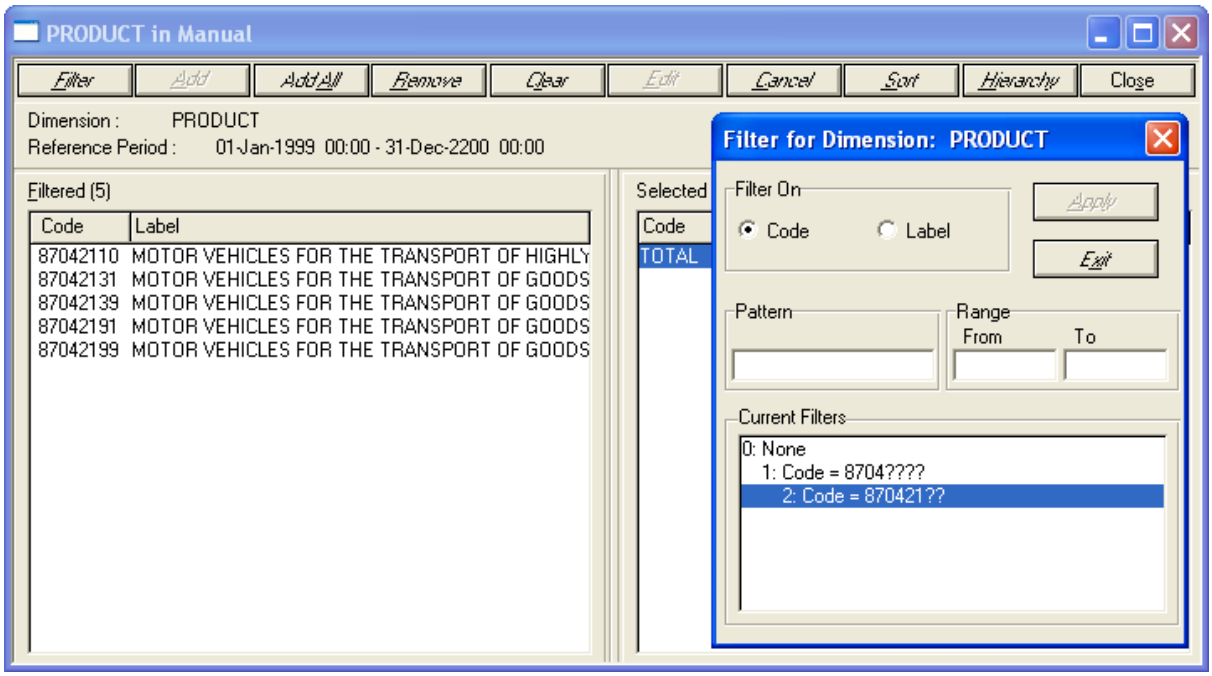



**Figure 9: Example of two successive hierarchical searches.**

### The filter symbols

The filter system also allows other symbols. These features are designed to make your search easier, using either codes or labels. This involves the following characters:

| Symbol | Replaces |
|---|---|
| ? | Any type of character |
| @ | An alphabetical character |
| # | A numeric character |
| * | Any number of characters |
| \| | OR |

Note: In French, COMEXT does not recognize accents.

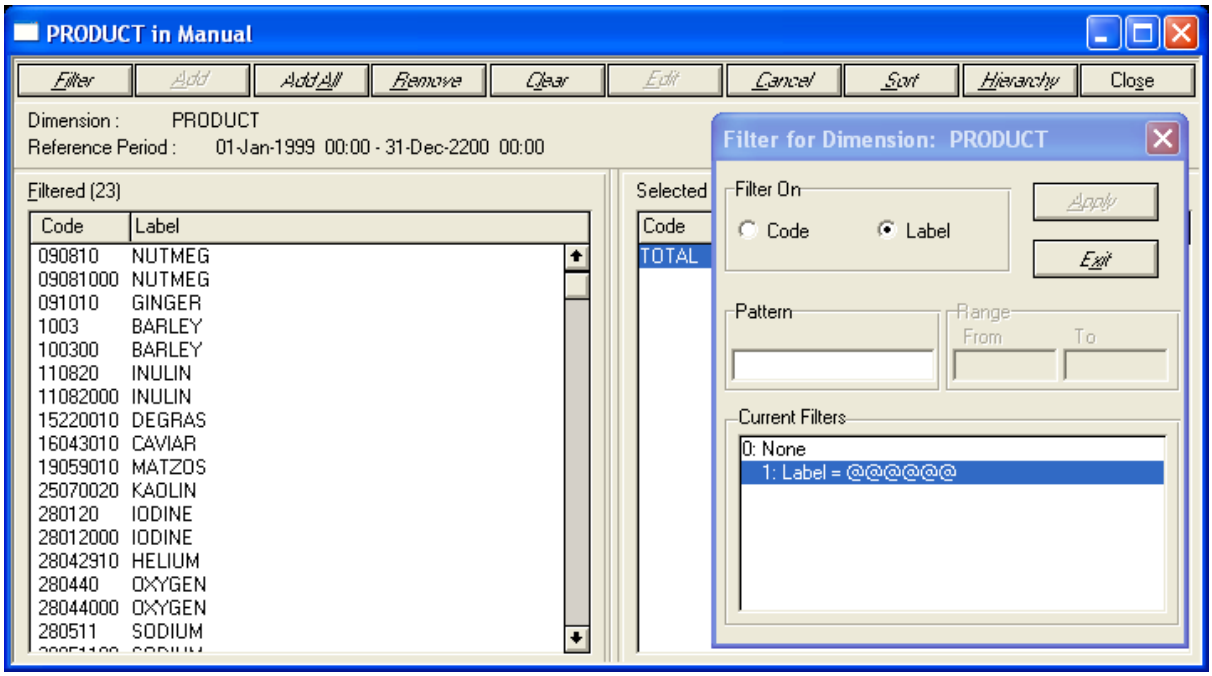



**Figure 10: Under "LABEL". '@@@@@@' produces all 6-letter labels.**

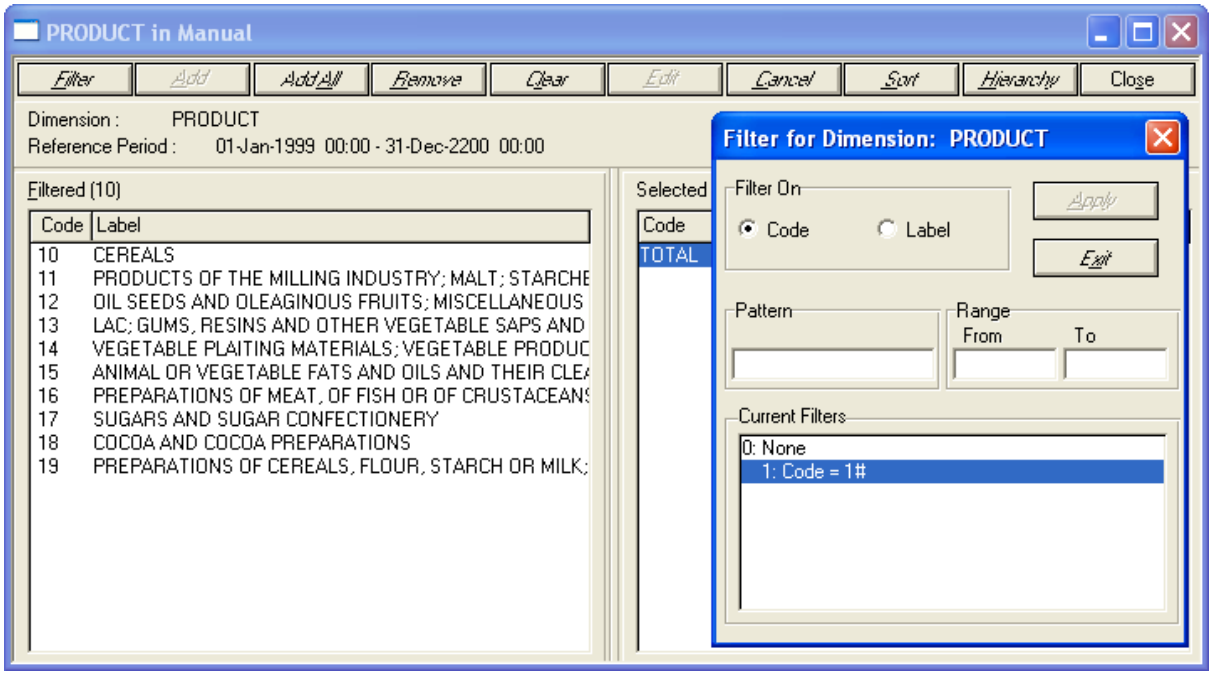



**Figure 11: A 1 followed by # will produce all digits following 1 in a 2-digit code.**

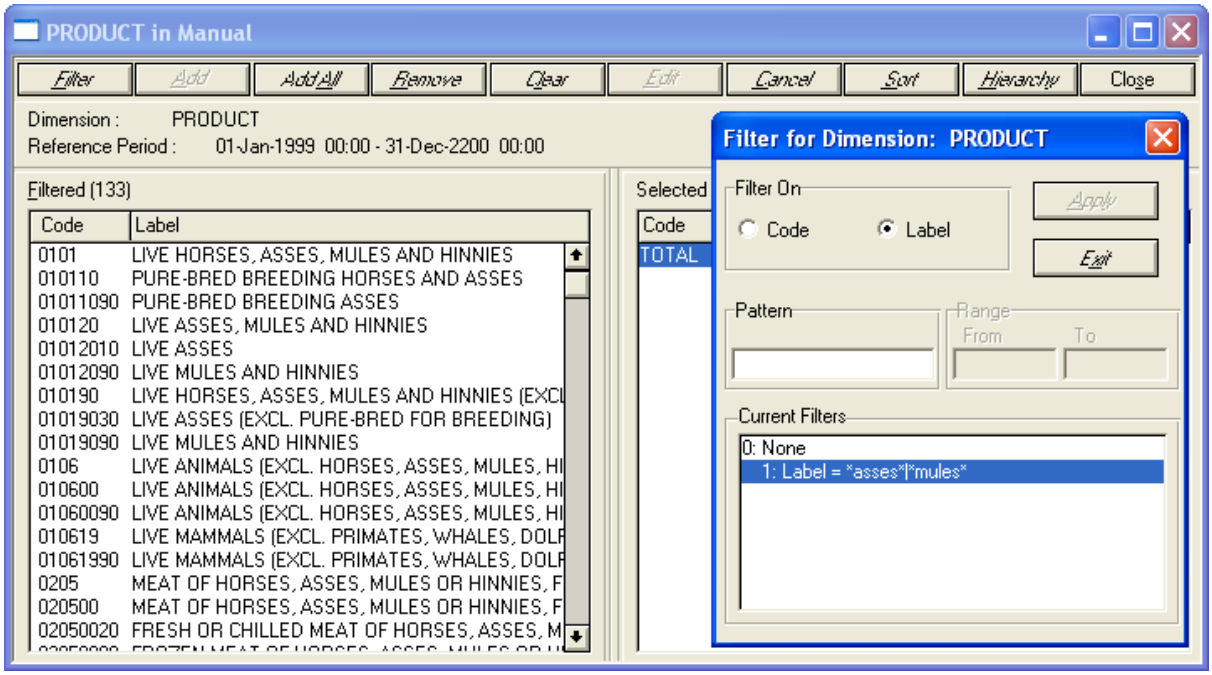



**Figure 12: The string of characters *anes*|*mulets* will produce all codes including strings "anes" or "mulets'.**

### Filtering by model

| Model | Explanation |
|---|---|
| a[bcd]e | *a* followed by the characters *b* or *c* or *d* with *e* at the end of the list, i.e.:<br>*abe*<br>*ace*<br>*ade*<br>but not *aee* |
| a[!xy] | *a* followed by any alphabetical character except *x* and *y* |
| a[e-h]i | *a* followed by the characters from *e* to *h* (inclusive), with *i* at the end of the list, i.e.:<br>*aei*<br>*afi*<br>*agi*<br>*ahi* |

# To select the requested code(s) (the Selected window)

When you have found the required code(s), you must transfer it (them) into the “Selected” window. This can be done with the following options:

ADD: when you have chosen ONE OR MORE codes by highlighting them, this is used to add them to the "SELECTED" window.

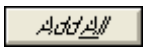
**To add the highlighted codes:**

1) From the Dimension Window, click Add button


ADD ALL: this function is used to transfer ALL codes from the “AVAILABLE” to the "SELECTED" window.

**To add all the code from the Available window:**

1) From the Dimension Window, click Add All button




REMOVE: when you have chosen ONE OR MORE codes by highlighting them in the "SELECTED" window, this is used to remove them.

**To remove the highlighted code from the Selected window:**

1) From the Dimension Window, click Remove button


CLEAR: this function is used to remove ALL the codes in the "SELECTED" window.

**To clear all the code from the Selected window:**

1) From the Dimension Window, click Clear button


EDIT: if you have created or selected a Userlist, you can use this function to change the validity dates and weighting (see Chapter USERLISTS)

**To edit a Userlist:**

1) From the Dimension Window, click Edit button


SORT: this function is used to sort the selection in the "SELECTED" sub window, by codes or labels, in ascending or descending order.

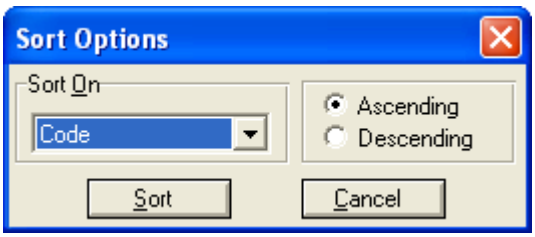
**To sort the code in the Selected sub window:**

1) From the Dimension Window, click Sort button


2) From the Sort Option Window, select the sort parameters:

a) Ascending or Descending

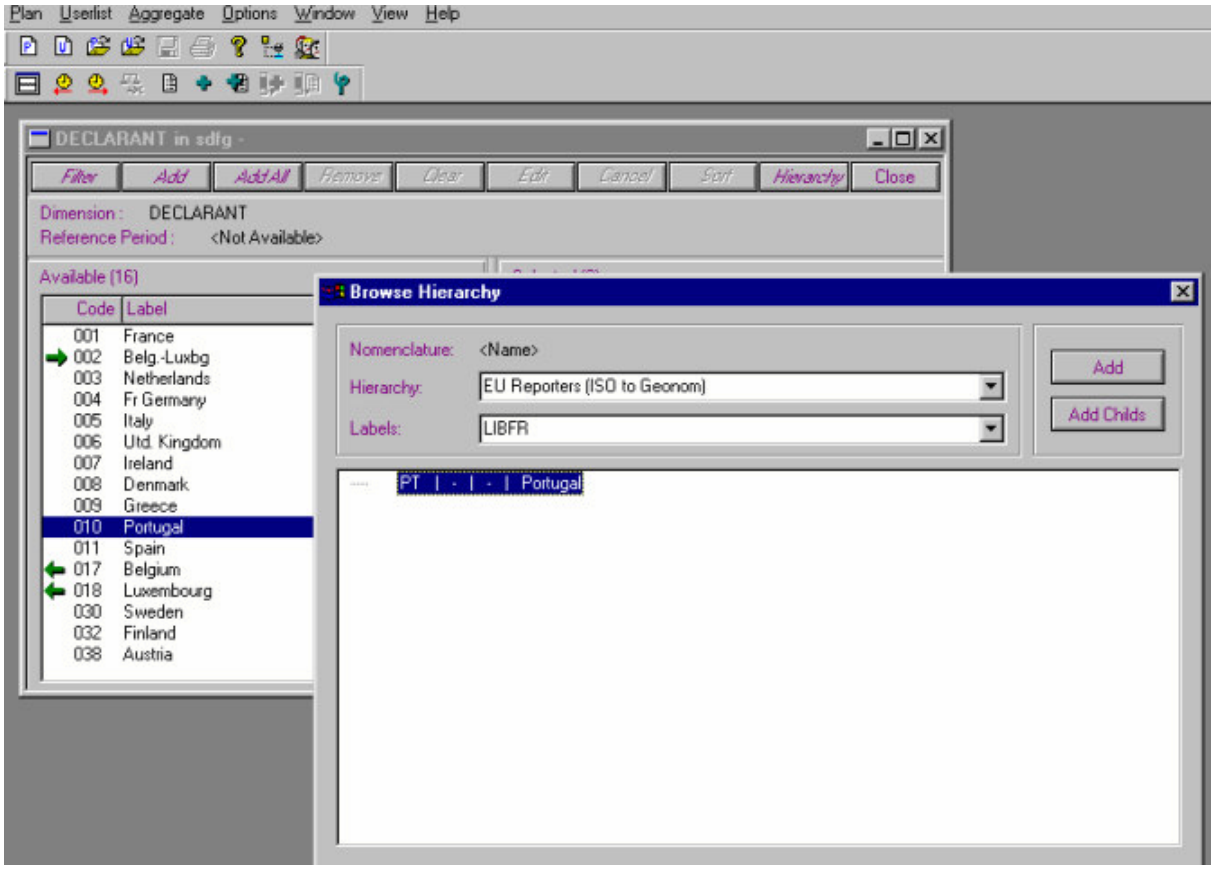
b) Sort on Code or on Labels




3) Click Sort button

HIERARCHY: Displays information in a hierarchical (tree) structure related to the chosen dimension.


This can be the structure of a chosen nomenclature or a table with matching codes from other nomenclatures. In the example shown in the following image, the hierarchy button produces the ISO Geonomenclature code “PT” for the code “010” which was originally selected.




In our example, we wish to have as the reporter (declarant) all the member countries of the European Union. We therefore click on the "ADD ALL" button. All the countries will then appear in the "SELECTED" window.

**To return to the "dimension" window:**

1) Click once on the icon  to access the system commands in the top left-hand corner and select "Close".
2) Double-click on  .
3) Click on  on the top right corner of the "dimension" window.
4) Click on the "Close" button in the "dimension" window.

# Saving the Plan

Save the plan once it is complete. This will ensure it is available for future use.

**To save a plan:**

1) From the Main Menu, click Plan / Save option.

   You can also save the plan by clicking on the icon.

Note: A plan can be used for extraction only when at least one indicator is selected.

If the user has not defined another dimension, all the codes belonging to this dimension will be extracted.

# Information on Plan

The name of the Plan is not always enough to describe its content or purposes. It is possible to supply additional information on your Plan as follows:

**To give additional information to the Plan:**

1) From the Main menu, click Option / Info

The "ADDITIONAL PLAN INFORMATION" window will appear. You can select the language you wish to use by clicking on the down arrow and highlighting the required language from the drop down list. Add notes to your plan in the "DESCRIPTION" text box.

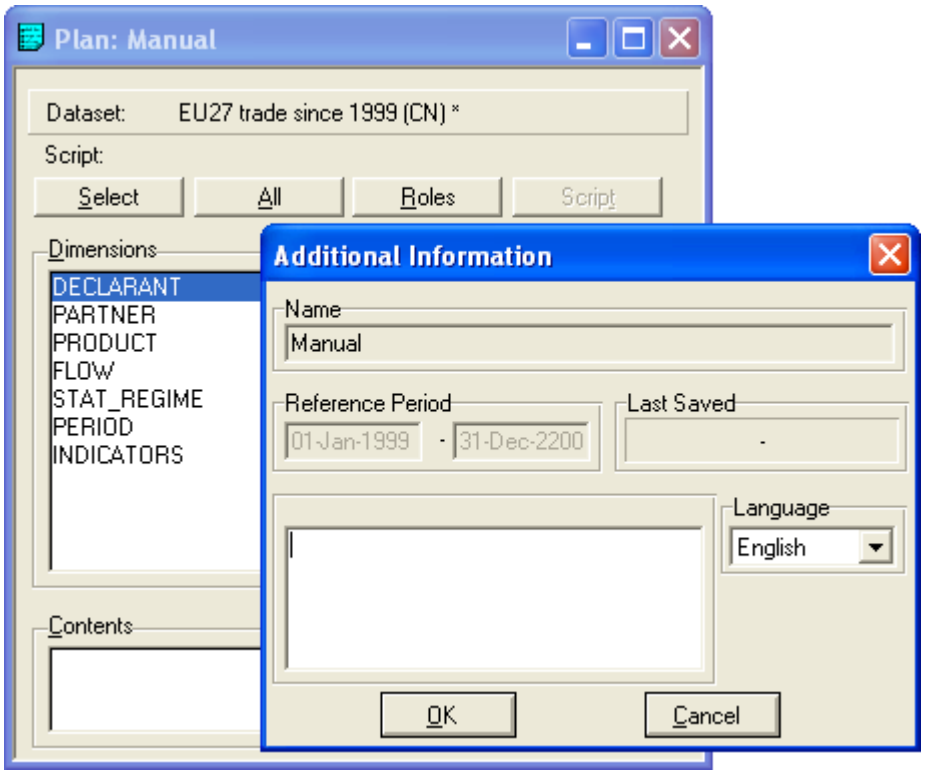
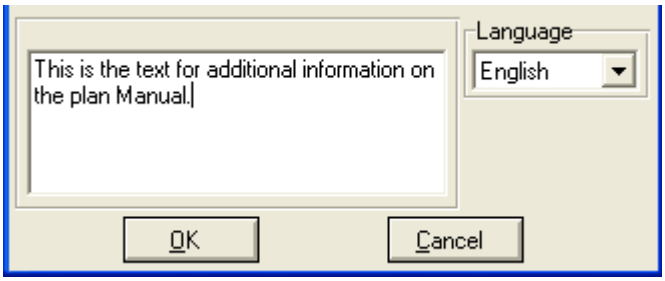



**Figure 13: Plan information window.**

2) Choose the appropriate Language from the Language drop down list.


3) Write the information in the text box




4) Click OK button

# Printing the Plan

You can print the content of your plan. This function produces a text file with a complete description of the plan describing the selected codes dimension by dimension.

**To print the content of a Plan:**

1) From the Toolbar Main, click the printer icon .
2) From the Print Plan windows, select:
   a) Orientation
      i) Portrait, or
      ii) Landscape
   b) Label Option
      i) No Label, or
      ii) Label
   c) Userlist Option

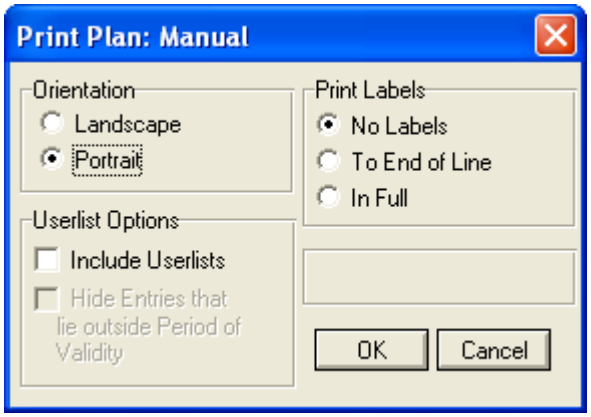



**Figure 14: Plan printout window.**

# Importing/Exporting Plan and Userlists

COMEXT offers the option of saving files containing plans or user lists at defined locations and transferring them back to the system when necessary. This option is very useful for exchanging plans (and user lists) between users. The output format is an XML file.

Additionally, Eurostat includes on-line documentation such as user guide, methodological guide, standard aggregate, etc. You can get this information using the Import function.

## Importing/Exporting plans

This section describes the import and export procedure for the Plan and Userlists.

**To Export Plan(s):**

1) From the Main Menu, select Plan, click Export

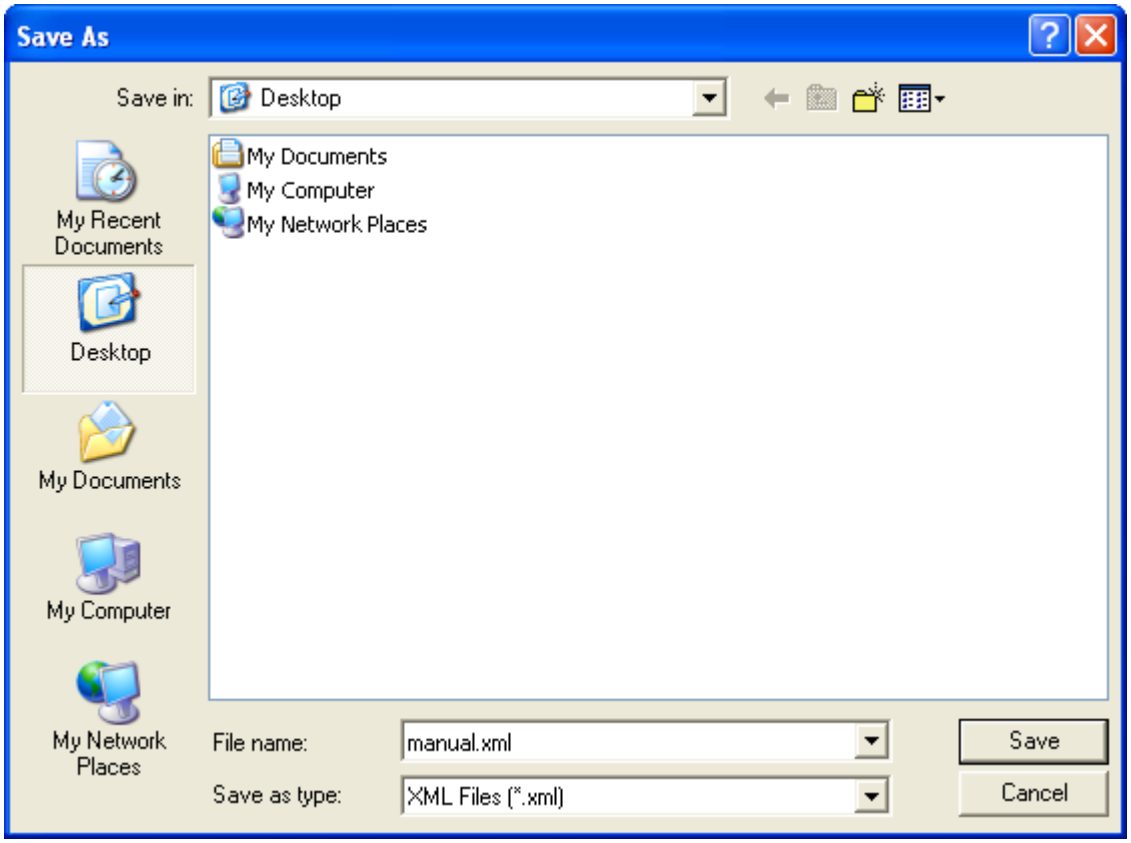


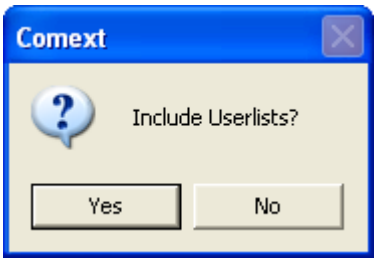

**Figure 15: Save window (for exported plans).**

2) From the Save window, Select the location (path) where the plan will be exported.

3) Give a name to the file (before the 'XML' extension) in the File Name text box.

4) Click Save.

5) Answer Yes or No to question "Include Userlists?" depending on whether or not you wish to export the user list for the plan.




6) Click on OK when the message “Export completed successfully” displays.


**To Import Plan(s):**

1) From the Main Menu, select Plan, click Import.

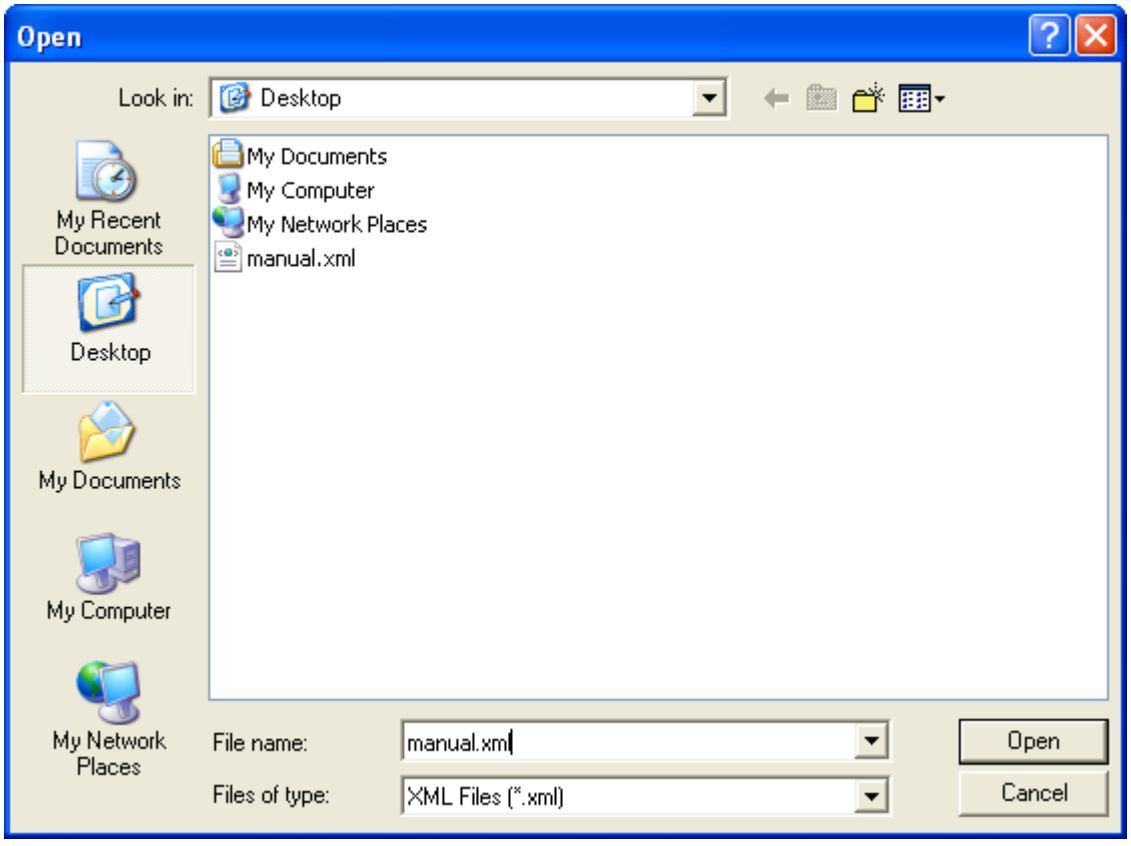
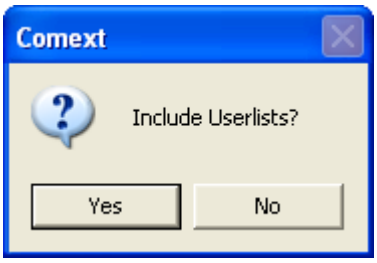



**Figure 16: Open window (Import function).**

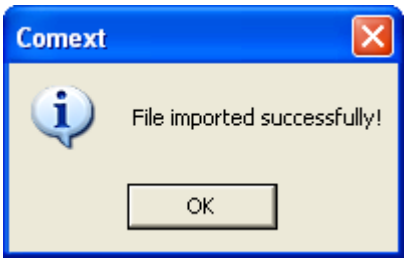
2) From the Open window, Select the location (path) of the file to be imported.

3) Select the file to be imported (XML).

4) Answer Yes or No to question “Include Userlists?” depending on whether or not you wish to import the user list for the plan.




5) Click on OK when the message “Import Completed successfully” displays.



# THE EXTRACTION

**To launch an On line extraction:**

1) From the Extract Menu, click Extract.

This option can also be launched via the "Extract" icon.

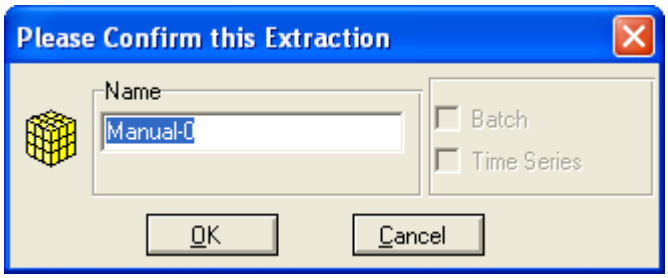
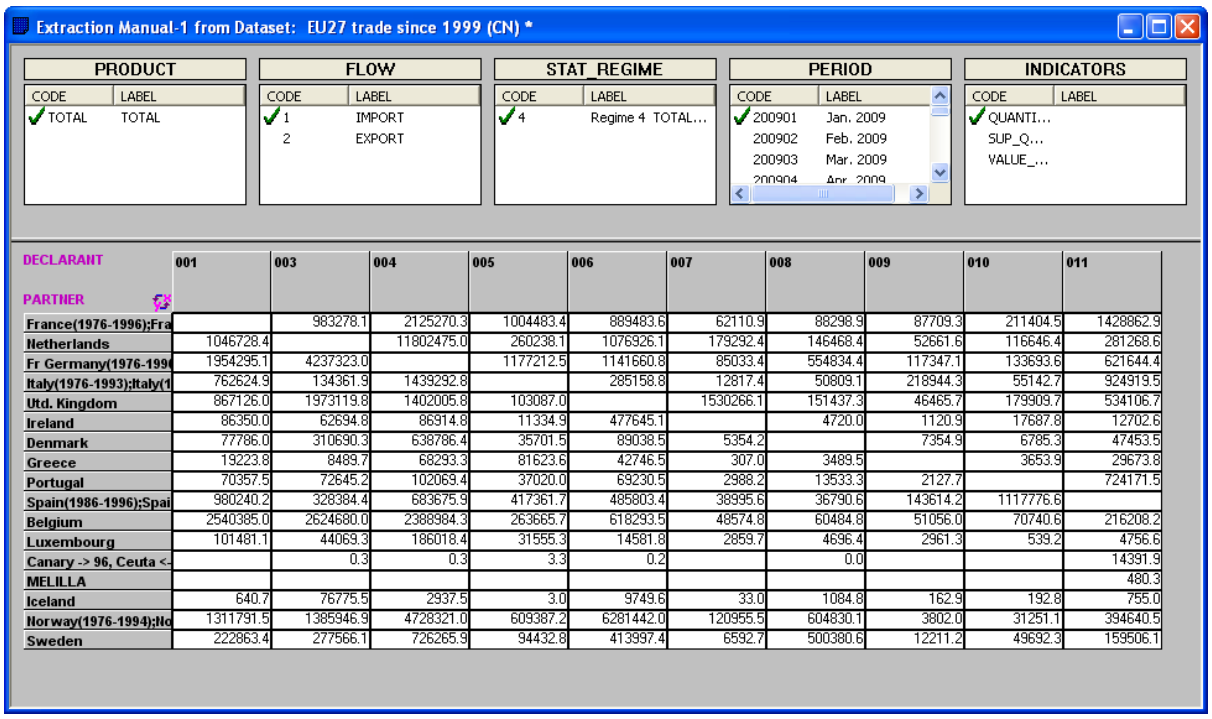



**Figure 17: The On-Line extraction confirmation window**

2) From the Confirm Extraction window, click OK button.

This function blocks use of the COMEXT application until the results are obtained. The results are presented in the form of a spreadsheet.

The following spreadsheet shows the result of an online extraction:




| PARTNER | 001 | 003 | 004 | 005 | 006 | 007 | 008 | 009 | 010 | 011 |
|---|---|---|---|---|---|---|---|---|---|---|
| France(1976-1996);Fra | | 983278.1 | 2125270.3 | 1004483.4 | 889483.6 | 62110.9 | 88298.9 | 87709.3 | 211404.5 | 1428862.9 |
| Netherlands | 1046728.4 | | 11802475.0 | 260238.1 | 1076926.1 | 179292.4 | 146468.4 | 52661.6 | 116646.4 | 281268.6 |
| Fr Germany(1976-199 | 1954295.1 | 4237323.0 | | 1177212.5 | 1141660.8 | 85033.4 | 554834.4 | 117347.1 | 133693.6 | 621644.4 |
| Italy(1976-1993);Italy(1 | 762624.9 | 134361.9 | 1439292.8 | | 285158.8 | 12817.4 | 50809.1 | 218944.3 | 55142.7 | 924919.5 |
| Utd. Kingdom | 867126.0 | 1973119.8 | 1402005.8 | 103087.0 | | 1530266.1 | 151437.3 | 46465.7 | 179909.7 | 534106.7 |
| Ireland | 86350.0 | 62694.8 | 86914.8 | 11334.9 | 477645.1 | | 4720.0 | 1120.9 | 17687.8 | 12702.6 |
| Denmark | 77786.0 | 310690.3 | 638786.4 | 35701.5 | 89038.5 | 5354.2 | | 7354.9 | 6785.3 | 47453.5 |
| Greece | 19223.8 | 8489.7 | 68293.3 | 81623.6 | 42746.5 | 307.0 | 3489.5 | | 3653.9 | 29673.8 |
| Portugal | 70357.5 | 72645.2 | 102069.4 | 37020.0 | 69230.5 | 2988.2 | 13533.3 | 2127.7 | | 724171.5 |
| Spain(1986-1996);Spai | 980240.2 | 328384.4 | 683675.9 | 417361.7 | 485803.4 | 38995.6 | 36790.6 | 143614.2 | 1117776.6 | |
| Belgium | 2540385.0 | 2624680.0 | 2388984.3 | 263665.7 | 618293.5 | 48574.8 | 60484.8 | 51056.0 | 70740.6 | 216208.2 |
| Luxembourg | 101481.1 | 44069.3 | 186018.4 | 31555.3 | 14581.8 | 2859.7 | 4696.4 | 2961.3 | 539.2 | 4756.6 |
| Canary -> 96, Ceuta <- | | 0.3 | 0.3 | 3.3 | 0.2 | | 0.0 | | | 14391.9 |
| MELILLA | | | | | | | | | | 480.3 |
| Iceland | 640.7 | 76775.5 | 2937.5 | 3.0 | 9749.6 | 33.0 | 1084.8 | 162.9 | 192.8 | 755.0 |
| Norway(1976-1994);No | 1311791.5 | 1385946.9 | 4728321.0 | 609387.2 | 6281442.0 | 120955.5 | 604830.1 | 3802.0 | 31251.1 | 394640.5 |
| Sweden | 222863.4 | 277566.1 | 726265.9 | 94432.8 | 413997.4 | 6592.7 | 500380.6 | 12211.2 | 49692.3 | 159506.1 |

**Figure 18: The result of an online extraction (Spreadsheet).**

When the spreadsheet appears, it displays the information according to X and Y axis. The default presentation is:

- The first dimension of the dataset (DECLARANT in our example) on the X axis

- The second dimension of the dataset (PARTNER in our example) on the Y axis.
- The other dimensions remain as headers of the spreadsheet.

Several options are available from the spreadsheet.

# Formatting the Extraction

The result of an extraction can be viewed as a virtual cube stored on the COMEXT server. To obtain the data in a physical file, it may be necessary to format the extraction. COMEXT offers several format options such as Print, Dump, Generate Table, Out data, Map and Graph.

The formatting of an extraction can be defined from the Extract menu when launching the extraction, or can be performed on an existing extraction via the Completed Work menu.

When using an Online extraction, the result is displayed in a Spreadsheet and can be stored on your dedicated space (local or remote).

When using the Batch extraction, the result is stored in your dedicated space and can be accessed from the “Completed Work” menu.

## The Completed Extractions, Prints and Dumps window

The previous section showed that for Batch extractions results are not displayed automatically resulting in a need to check whether the job has completed by viewing the "COMEXT Jobs" window.

When the job is completed, the result can be obtained via the "Completed Work" menu command or by clicking the icon  .

The following window appears:

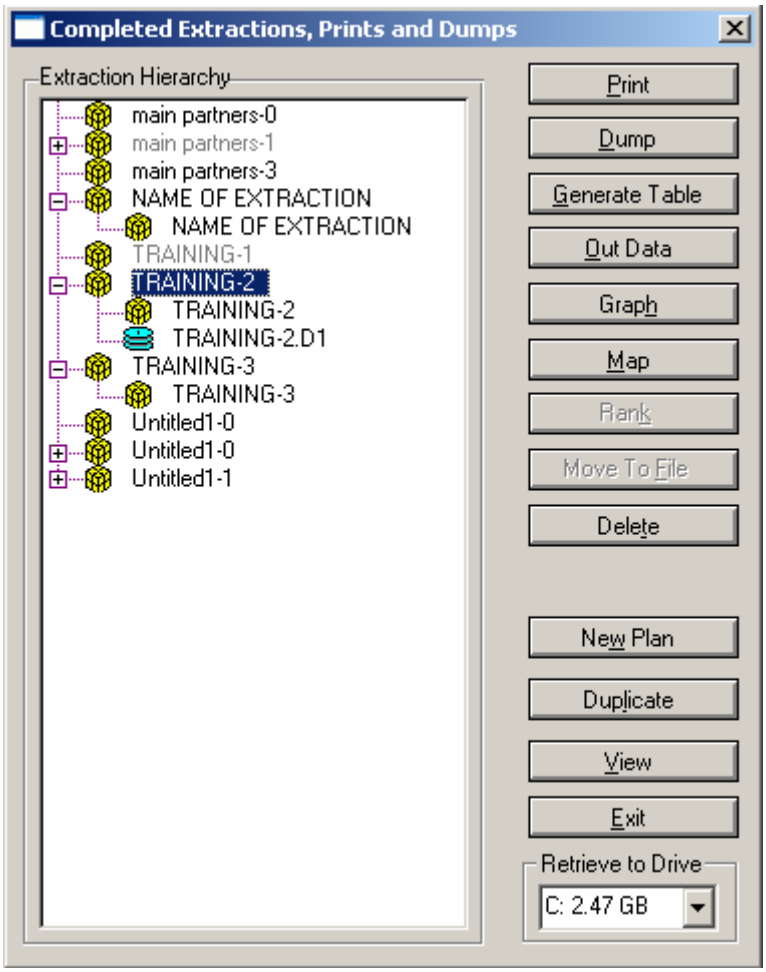



**Figure 19: Extraction, Prints and Dump Window**

In this window all the extractions carried out to your server space will appear in a list. This list shows the name(s) of the extraction(s) preceded by the symbol . By highlighting this symbol we are able:

**To obtain information on the extraction:**

1) Select an extraction from the Completed work window.
2) Right click and select Properties.

Information on the job's creation date, the dataset used and the internal job number will display.

**To view the content of the extraction:**

1) Select an extraction from the Completed work window
2) Click the View Button.


The extraction result will open in a spreadsheet.

**To Delete and free your dedicated server space:**

1) Select one extraction from the Completed work window

2) Click the Delete Button


The extraction will be deleted.

Note: If the extraction has been used to create other files (Print, Dump, Table, etc.), the formatted files will not be removed.

**To Format as a Print, Dump, Generate Table, Out Data, Rank, Graph or Map:**

1) Pressing the respective icons accesses these functions.

For jobs preceded by the ⊞ symbol, the ⊞ indicates that data in this extraction has been formatted in different ways.

2) Clicking on the ⊞ reveals the list of formatting. Six types of icons can appear:

Indicates out Data
Indicates a Dump
Indicates a Print
Indicates a Graph
Indicates a Generate Table
Indicates a Subcube
Indicates a Map

# Print

This option formats the extraction data so that it is suitable for printing on paper.

**To perform a Print:**

1) From the Completed work window, select one extraction.
2) Click on Print.

   This option can also be launched via the Extract menu, Print option.

The following window will appear, allowing configuration of the printout. By default, the system suggests a title, but this can be changed:

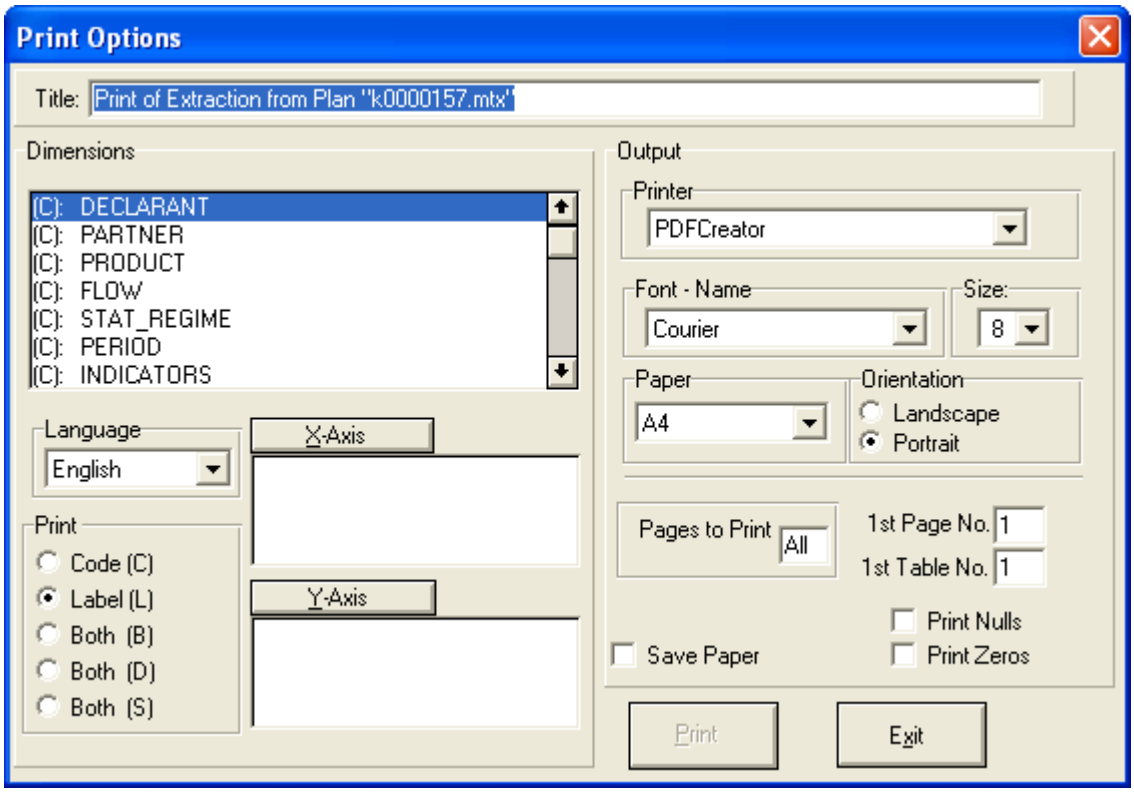



**Figure 20: The printout configuration window.**

**To define the presentation of your print:**

1) Select a dimension from the Dimension text box
2) Click on X or Y-Axis to move the dimension on one of the axis

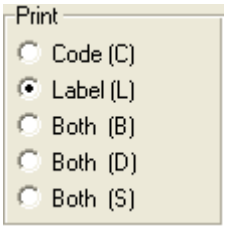
- Click on X-Axis to put the dimension as a Column in the print


- Click on Y-Axis to put the dimension as a Line in the print


Another possibility is to “drag and drop” the highlighted dimensions using the right mouse button.

**To define the printout options:**

1) From the Title text box, give a title to the print
2) From the Print heading, select the appearance of the headings:




The "PRINT" heading gives the option to display:

| Option | Meaning |
|---|---|
| Code (C) | codes only |
| Label (L) | labels |
| Both (B) | abbreviated codes and labels |
| Both (D) | full codes and labels |
| Both (S) | codes and labels |

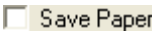
3) Select the orientation of the Print from the Orientation box.


4) Select the font and the size from Font Name and Size Name boxes.


5) Choose whether or not to print the “Nulls” or “Zeros” by checking or unchecking the "Print Zeros" or "Print Nulls" boxes


6) To specify the number of pages to be printed, or to print from one specific to another specific table, use following boxes:


7) To reduce the space between the tables of your print, check the “Save paper” box. This option removes page breaks.




8) Launch the print, click “Print” button.


9) Confirm the print in the confirmation window.

# Dump

This option formats the extraction for import and treatment in another database or spreadsheet. It produces flat files containing data separated by a comma (or other symbol) and is particularly useful for large extractions.

**To launch a Dump: spreadsheet**

1) Select an extraction from the Completed work window.

2) Click on “Dump” button.

This option can also be launched via the Extract menu, Dump option.

The following window appears, allowing configuration of the Dump.

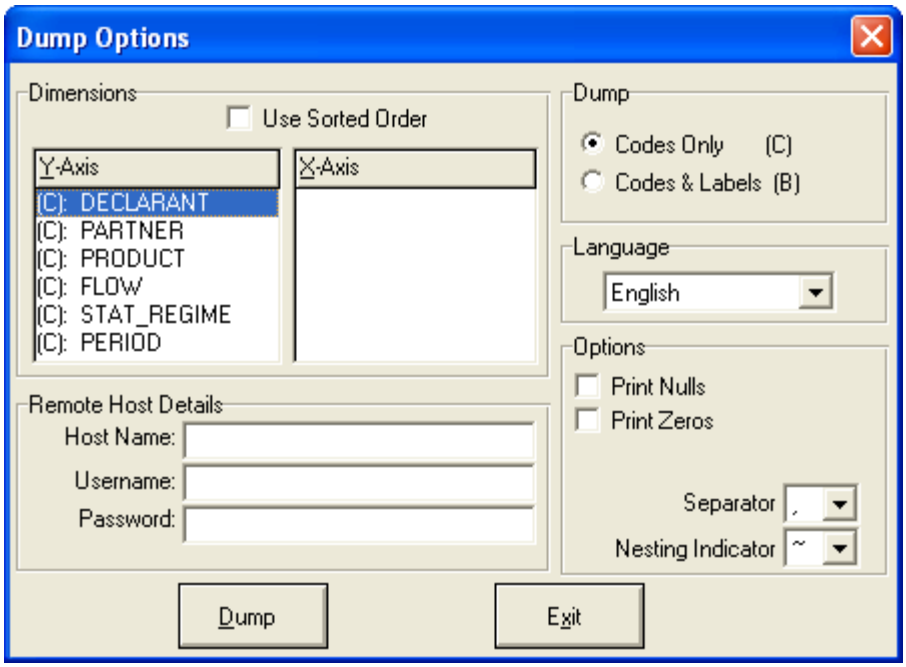



**Figure 21: Setting the parameters of a dump.**

By default, all dimensions are on the Y-axis. As for the Print, some dimensions can be placed on the X-axis (they should not all be placed on the X axis, because the application will no longer run).

**Define Dump options:**

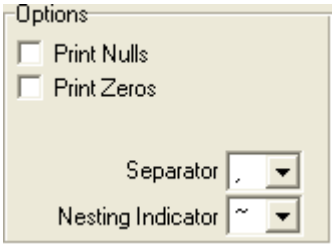
1) To move a dimension from the X to the Y axis, double click on one dimension.

Note: The same procedure will enable you to move one dimension from the X-axis to the Y-axis.

2) To select the label or code display, click C (codes) or B (Both Codes and Label) from the “Dump” option windows


3) To set the language for the output labels, select the requested language under Language option.


4) Choose whether or not to print the “Nulls” or “Zeros”, by checking or unchecking "Print Zeros" or "Print Nulls" boxes.




5) To apply sorting on your result select Use Sorted Order option.

6) Use ‘Separator’ to change the field delimiter (; is the default delimiter)

7) Use the ‘Nesting indicators’ list to change the symbol. The nesting indicators show the line containing “information” in the Dump file. This can be useful if data is loaded onto other database systems.

8) Launch the Dump, click “Dump” button.


9) Confirm the dump in the confirmation window

# Generate Table

This option formats the results for display and treatment in programs like Microsoft Excel.

**To launch a Generate Table:**

1) Select an Extraction from the Completed work window.

2) Click on the ‘Generate Table’ Button.

   This option can also be launched via the Extract menu, Generate Table option.

The following window appears, allowing configuration of the parameters for the generated table. By default, the system suggests a title, but this can be changed.

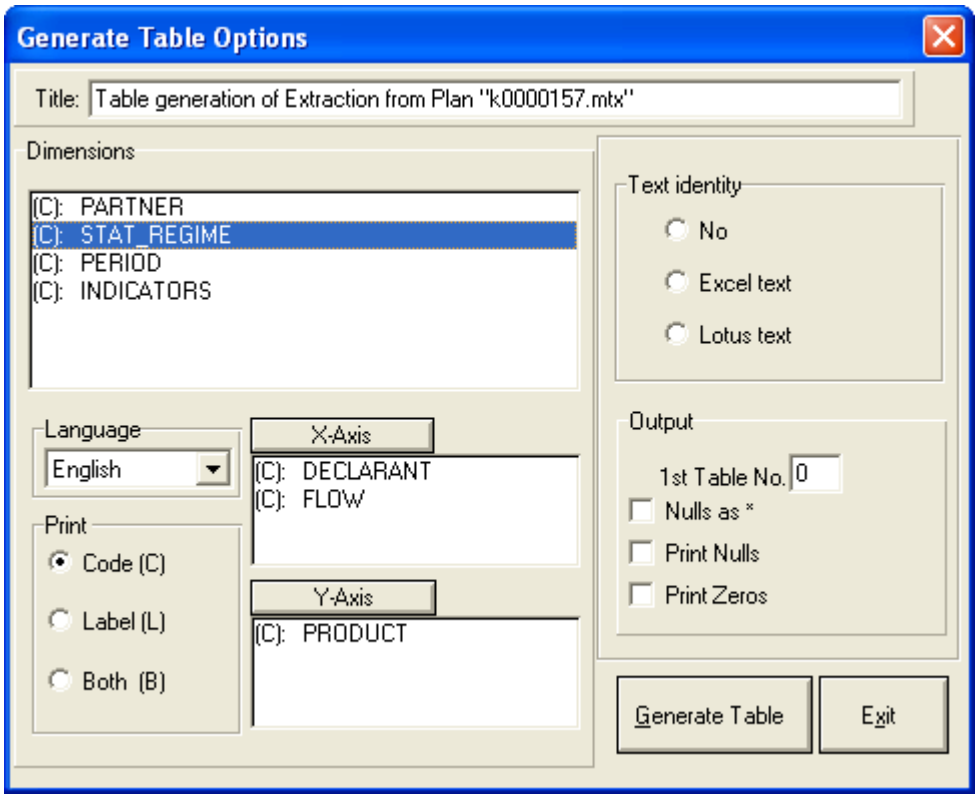



**Figure 22: The "Generate Table" configuration window.**

**To define the presentation of your Tables:**

1) Select a dimension from the Dimension text box.
2) Click on X-Axis or Y-Axis to move the dimension on one of the axis.

- Click on the X-Axis to put the dimension as a column in the Table.

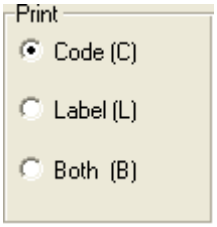

- Click on the Y-Axis to put the dimension as a row in the Table.


- Another possibility is to “drag and drop” using the right mouse button.

**To define the printout options:**

1) Give a title to the Tables in the Title text box.

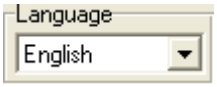

2) Select the appearance of the headings from the Print heading




The "PRINT" heading gives the option to display:

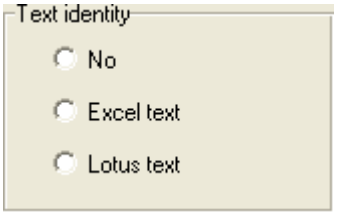
| Option | Meaning |
|---|---|
| Code (C) | codes only |
| Label (L) | labels |
| Both (B) | abbreviated codes and labels |

3) To set language of the output labels, select the required language from the ‘Language’ drop down list.

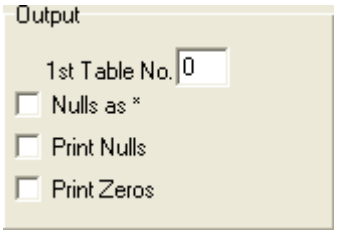



4) Select the Text identity from the ‘Text Identity’.

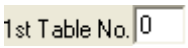
This option will “Format” the textual information of the tables (labels and title) according to the software you will use to edit the tables.




5) Select the output From the ‘Output’ selection box for:

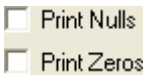



a) To start the generation of the tables at another number other than the first, enter the Table Number in the text box.




b) To print Nulls as *, check the box.


c) Choose whether or not to print “Nulls” or “Zeros”, check or uncheck "Print Zeros" or "Print Nulls" boxes.




6) Click “Generate Table” button to launch the generation of the table.


7) Confirm the Generate Table in the confirmation window

# Out Data

This option formats the results in several ways suitable for display and treatment in a variety of programs like Excel, SAS, SPSS, etc.

**To launch a Out Data:**

1) Select an Extraction from the Completed work window.
2) Click on 'Out Data' Button

   This option can also be launched via the Extract menu, Out Data option.

The following window appears, allowing configuration of the parameters for the generated file.

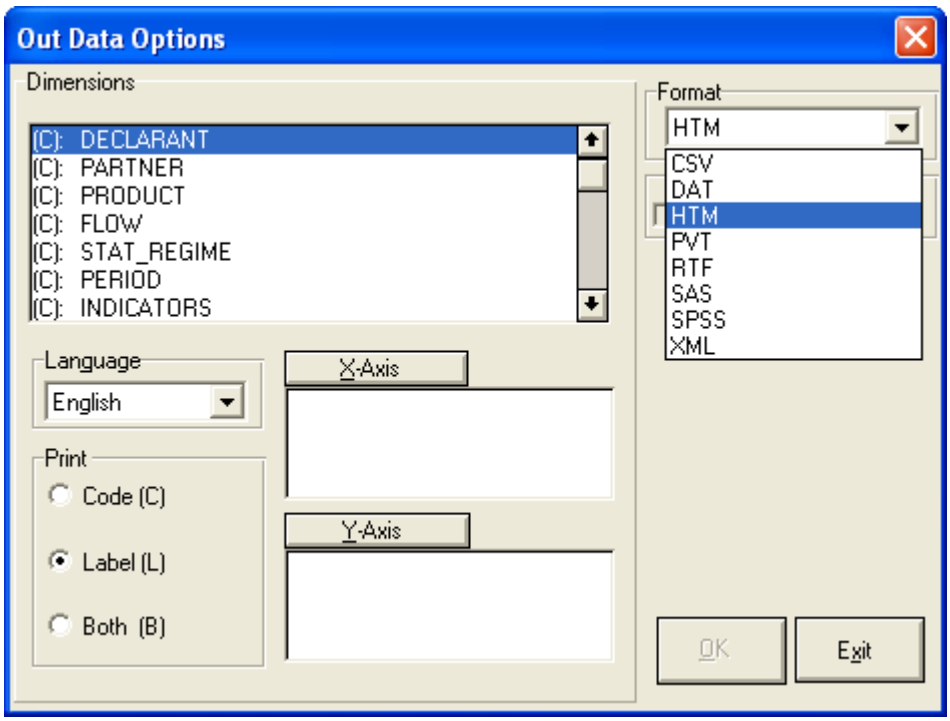



**Figure 23: The "Out Data" configuration window.**

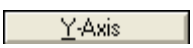
**To define the presentation of your File:**

1) 1) Select a dimension from the Dimension text box
2) 2) Click on X-Axis or Y-Axis to move the dimension on one of the axis

- Click on X-Axis to put the dimension as a Column in the File

- Click on Y-Axis to put the dimension as a Line in the file

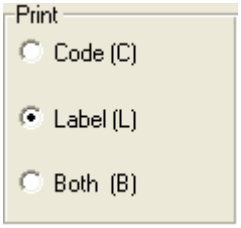
  
  

- Another possibility is to "drag and drop" the highlighted dimensions using the right mouse button.

Note: This option will only be available for some output formats

**To define the printout options:**

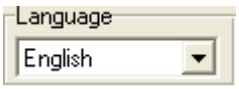
1) Select the appearance of the headings from ‘Print’




The "PRINT" heading gives the option to display:

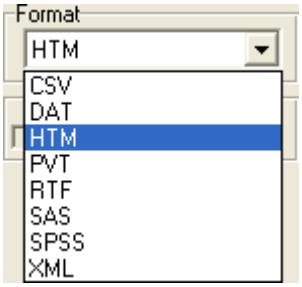
| Option | Meaning |
|---|---|
| Code (C) | codes only |
| Label (L) | labels |
| Both (B) | abbreviated codes and labels |

2) To set the output labels language, select the requested language from the Language drop down list.




3) Select the output format from ‘Format’.

This option defines the output format.




4) Click “OK” button to launch the Out Data formatting.


# Graph

This option formats the results of the extraction into a Graph. COMEXT can generate several types of Graph (Pie, Bar, Line, Scatter etc).

**To launch a Graph:**

1) Select an Extraction from the Completed work window.
2) Click on ‘Graph’ button.

The Graph function can be accessed via the "Graph" button, the icon or the "Extract/Graph" menu command. The following window appears, allowing configuration of the parameters for the Graph.

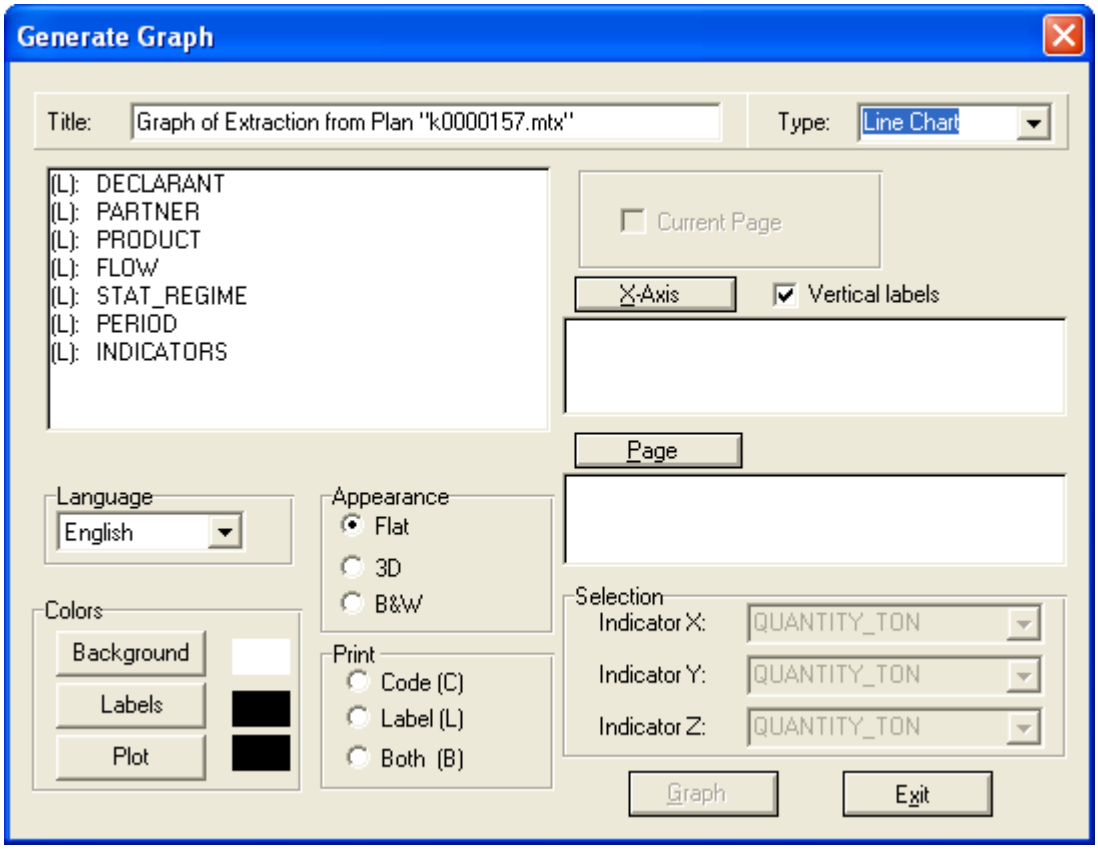



**Figure 24: The "Graph" configuration window.**

Select the type of Graph from the list available n the “Type” box on the top Right of the windows. Several options are available:

- PIE
- LINE
- BAR
- AREA
- BUBBLE
- SCATTER

See below for more details on graph types.

Highlighting the required dimensions and clicking on the “X-Axis”, or "Page" buttons, selects the design of the output for some formats. To restore a dimension, double-click on it to return it to the dimension window (the order of the dimensions on the axes will be the same as the spreadsheet produced).

Another possibility is to “drag and drop” the highlighted dimensions using the right mouse button.

You can also choose:

- The appearance of the headings in the Worksheet, by clicking on any dimension and selecting one of the options of the "print" radio buttons (Code, Label or Both).
- The Indicators of the Axis (X, Y and Z).
- The colors.
- The appearance (3D, Flat or B&W).

By clicking the "Graph" button, the system produces a new entry in the "Completed extractions, prints and dumps" window. Proceed as described in the previous section.

## Graph types

In general many graphs are generated in the single step depending on size of the extraction. Dimensions specified for X axis and Page form data for each single graph. How exactly it depends on graph type. For all codes of the others dimensions graphs are generated and put into a column one by one in order to cover the whole extraction. In other words, for each element of cartesian product of Page dimensions a color is allocated and used to represent the element. For each element of cartesian product of other dimensions (not X-Axix nor Page) one graph is produced.

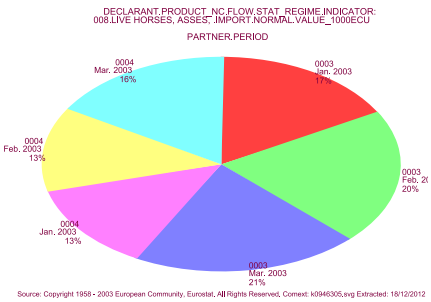



**Figure 25: Pie chart**

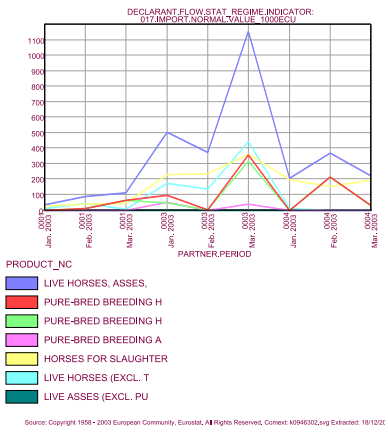



**Figure 26: Line chart**

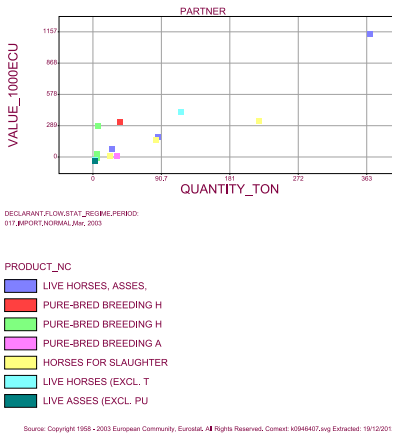



**Figure 27: Scatter graph**

- Pie chart displays ratio of indicator values. There is no notion of the scale. See Figure 26.

  Only X-Axis dimensions are specified. Percentage ratio of values of these dimensions will be displayed in each graph. Other dimensions are covered by drawing many charts into a column as for all graph types.

- Line, bar and area charts are very similar to each other. See Figure 27.

  Dimensions from X-Axis list becomes dimensions of horizontal axis of the graph. Indicator value is displayed as height of point or bar above the code.One dimention may be listed as Page. Codes of this dimension are displyed in various colour in a common chart. As soon as there are too many codes in Page it will be split into more charts.

- Scatter and Bubble graphs are the other group of similar couple. See Figure 28.

Scatter and Bubble graphs on the contrary to previous charts displays more than one indicator in a single graph. They are useful to show dependency of two or three indicators – value of Indicator X determines position on x axis, value of Indicator Y on y axis and for Bubble graph Indicator Z determines size of the point. Variable point size in Bubble graph is the only its difference from Scatter graph.

Codes of Page dimensions are drawn in different colors to a single graph, however, X-Axis dimensions are drawn without any sign of their code.

Note: The order of the dimensions in the lists in Generate Graph window is not significant. The dimensions are processed in their order in Plan window and thus it cannot be changed.

The order of codes is explained in the following example. Let PARTNER and PERIOD dimensions are put to X-Axis and there are codes 0001, 0003 and 0004 in PARTNER dimension and Jan. 2003, Feb. 2003, Mar. 2003 in PERIOD. Then the codes will be listed in the following order:

1. 0001.Jan. 2003
2. 0001.Feb. 2003
3. 0001.Mar. 2003
4. 0002.Jan. 2003
5. 0002.Feb. 2003
6. Etc.

I.e. codes of later dimension changes more frequently. The codes with NULL value are omitted.

# Map

Note: This option formats the results in a Map. This option is not yet available in the actual Stand alone version.

# Move to File: downloading from “server”

The "Move to File" function downloads a file from the COMEXT server to a specific location. This function is used to retrieve a file to use in other software.

**To launch a Move to File:**

1) Select an output file from the Completed Work window
2) Click “Move to File” button

The following screen will appear, to specify where the file will be saved (on your computer or on any network location).

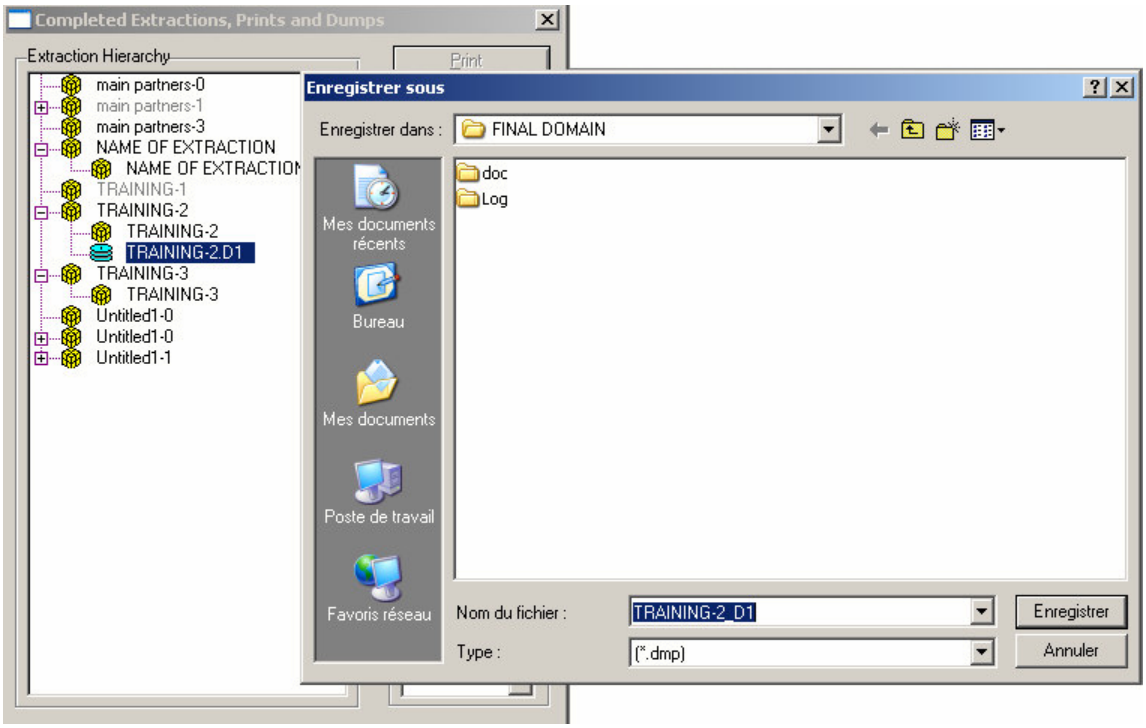



**Figure 28: Copying a format type to a local file using the "Move to File" button.**

3) Give a name to the file in the “File Name” text box of the Save window

4) Click Save button

# Delete

This option deletes Extraction and/or formatted files.

# View

This option will open an existing extraction in “Read only” mode. The data will be displayed into the spreadsheet. Only some function from the spreadsheet will be available.

# Duplicate

This option enables you to open an existing extraction in “Edit” mode. The data will be displayed into the spreadsheet and all the functions of the spreadsheet will be available. From the server side, this function will generate a copy of the previous extraction which will be available for editing, through the spreadsheet functions.

# New Plan

This function will enable you to use a previous extraction as a dataset, and to build a new plan in this “user dataset”. It can be used to extract a part of an existing extraction by reducing the codes selected in the original extraction.

# Exit

This option closes the Completed Work window.

# The spreadsheet

The spreadsheet is the COMEXT data viewer and editor. It can be accessed:

- From the Online extraction (the result of the extraction is displayed in the spreadsheet)
- From the Completed work menu using the functions View or Duplicate.

Several functions are available from the spreadsheet for defining the presentation (multidimensional table), retrieving data from your computer (direct extraction on several formats) or applying calculations on the data (using the Rank functions).

# Formatting the spreadsheet

**To change a dimension in the spreadsheet:**

1) From the spreadsheet, click on one dimension in the header to obtain the following symbol:

2) Hold the left mouse button down (to keep the symbol displayed) and drag to one of the X or Y dimension.
3) Release the left mouse button. The spreadsheet will then display the requested dimension and the previous dimension will appear in the header of the spreadsheet.

Several dimensions can also be displayed on the same axis.

**To display several dimensions on one axis of the spreadsheet:**

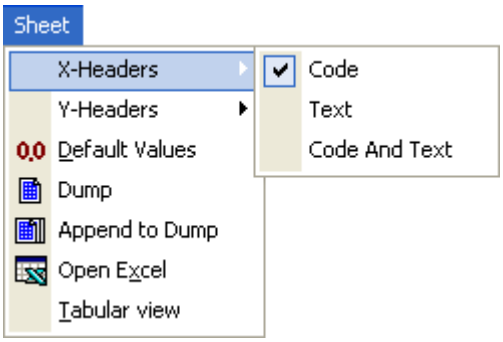
1) From the spreadsheet, click on one dimension in the header to obtain the following symbol:

2) Hold the left mouse button down (to keep the symbol displayed) and drag it on top of, or under, one of the X or Y dimensions.
3) Release the left mouse button. The spreadsheet will then display both the previous and the newly inserted dimension.

**To change the display of the spreadsheet (the Sheet menu):**

1) The Sheet menu becomes available when you are browsing the data with the spreadsheet.

2) This menu allow you to display codes, labels or both, to set default values, to generate a text file and to save the extraction to the repository.




When the required spreadsheet has been generated, the presentation can be defined in more detail by deciding to view either codes or labels, or both, in the header dimensions.

**To display code and/or Labels into the spreadsheet:**

1) From the spreadsheet main menu, click on Sheet menu.
2) From the menu select X-Header or Y-Header.
3) From the Sub menu, select Code, Text or Text and Code.

Note: This function can also be accessible via the Icons for the X- Headers and for the Y- Headers .

# To remove the empty lines/columns of the spreadsheet

When the COMEXT spreadsheet displays the data, all the selected codes are available in the spreadsheet. You can reduce the spreadsheet to keep only the codes for which information is available. This option is called ‘Compress’ and is accessed from the Rank menu.

**To compress the spreadsheet:**

1) From the Rank menu, click on Rank menu.
2) From the menu select Compress.

The spreadsheet will be compacted to remove the code combinations that contain no information (empty lines/columns).

Note: This function can also be accessed via the icon r.

**To save and return to the original spreadsheet:**

1) From the spreadsheet menu, click on Save and Return.

The compressed spreadsheet will be saved in the completed work repository and the original spreadsheet displayed.

Note: This function can also be accessed via the icon .

**To generate an EXCEL file from the spreadsheet:**

1) From the spreadsheet main menu, click on Sheet menu.

2) From the menu select Open Excel.

3) Excel is open with a copy of the spreadsheet.

Note: The data in the Excel file is a copy of the spreadsheet displayed on the screen. It does not automatically contain all the data from the extraction.

This function can also be accessed via the icon u.

**To set default value:**

1) From the spreadsheet main menu, click on Sheet menu.

2) From the sheet menu, select Default values.

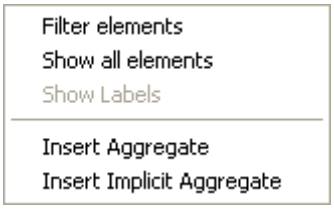
3) In the spreadsheet, the “empty cells” are set to 0.00.

Note: This function can also be accessed via the icon 0.0.

**To Apply filter on a dimension:**

1) From the spreadsheet, right click on one dimension in the table.

2) The following Sub menu becomes available.




3) Select Filter elements to apply filter (or show all after a filter).

4) From the selection window, select the requested codes.

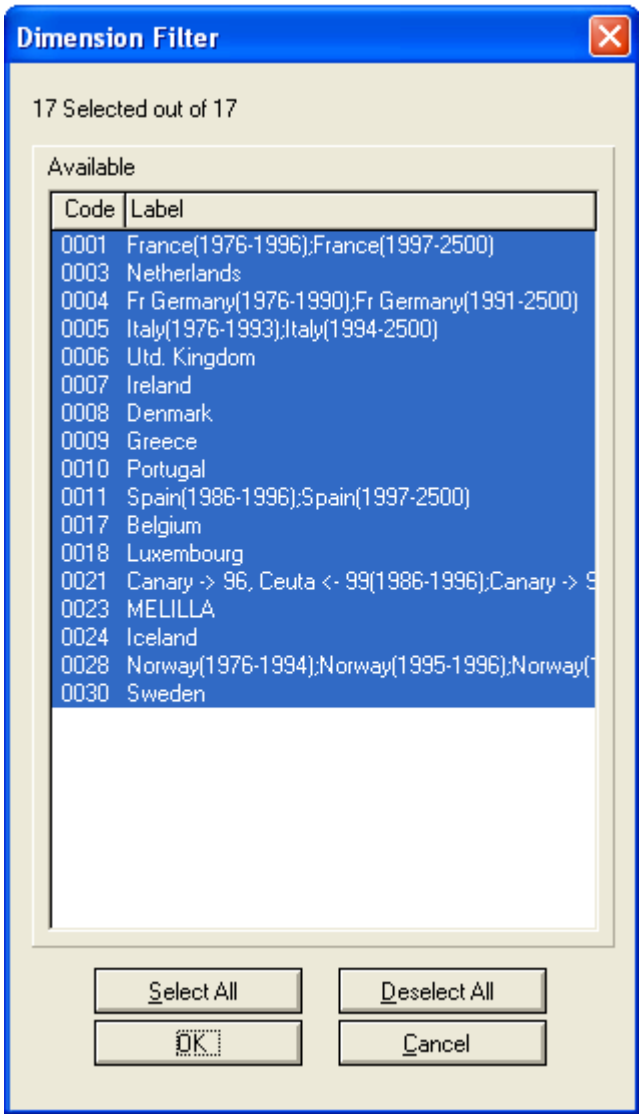



**Figure 29: Filter window.**

5) Click OK button

**To Insert or create an aggregate from the spreadsheet:**

1) From the spreadsheet, right click on one dimension in the table.
2) The following Sub menu becomes available.


3) Select Insert Aggregate or Insert Implicit Aggregate.

Note: For Aggregate or Implicit Aggregate, see chapter USERLISTS

# The Tabular view

The COMEXT spreadsheet allows a tabular view. This option shows the data in a 'flat' format. The Tabular View Parameters allow you to select the information to display in Tabular mode. By default, all the codes of all dimensions are displayed and the 'Asc' parameter selected. The 'Asc' parameter ranks the data in ascending order (according to the selected indicators).

The spreadsheet displays the data in a table that can be formatted. COMEXT gives the option to display the data in Tabular mode.

**To view the data in Tabular mode:**

1) From the Sheet menu, select Tabular View

2) The "Tabular View Parameters" window is displayed.

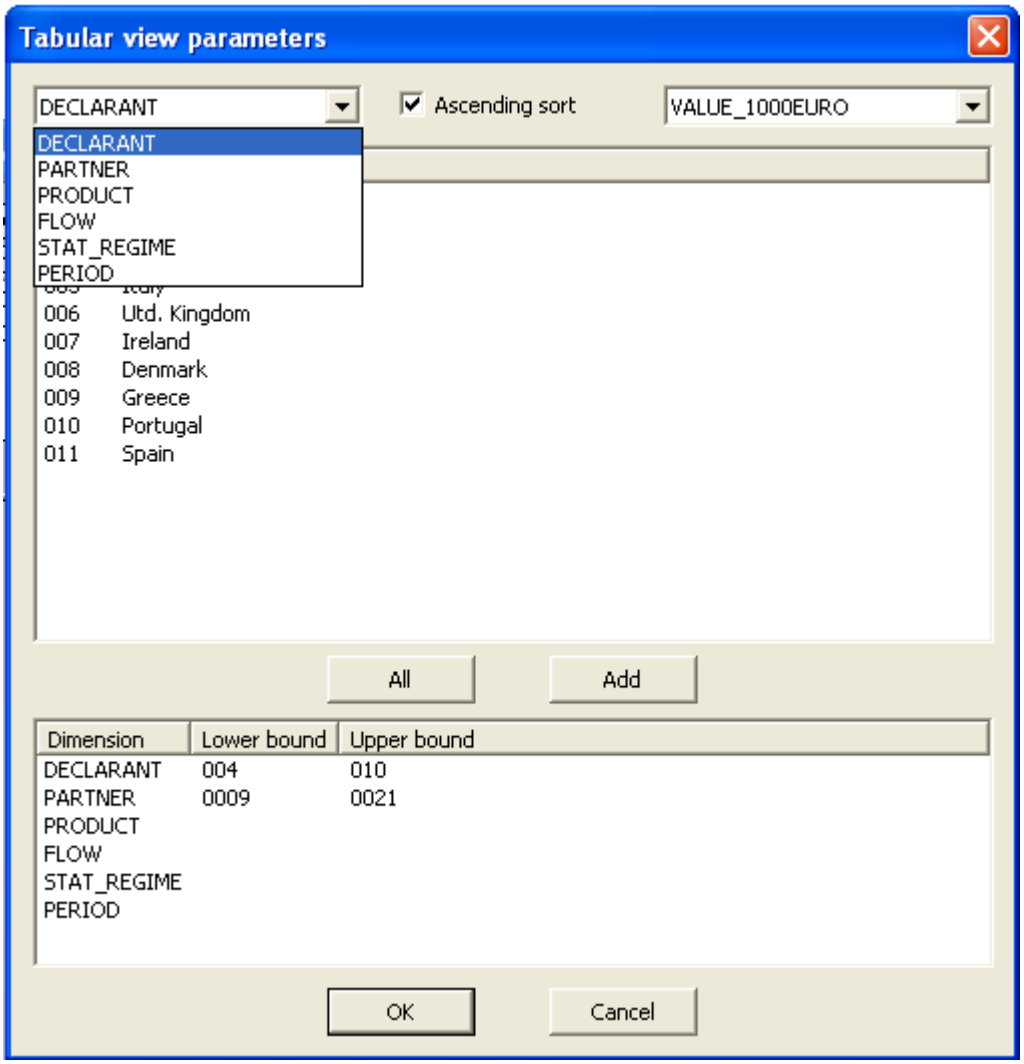



**Figure 30: Tabular View Parameters window.**

3) Select the Indicator (on the **top right** of the window) to order by it. In Figure 31 the selected indicator is 'VALUE_1000EURO'.

4) Click OK to extract all the data in Tabular View

**To apply a selection of codes for a particular dimension to be included in the view from the Tabular View Parameters window:**

1) Click on the drop down list to select the dimension.
2) Select or unselect codes in the upper part of the window.
3) Click on Add button. The selected lower and upper bounds are indicated in the bottom part of the window.
4) Click the OK button to display the Tabular View.

**To reorder columns in the Tabular View:**

Drag and drop the column to change their order.

**To save the Tabular view to a file:**

1) Close the view with clicking on the button and answer 'Yes' the question 'Do you want to save the file?'

# Saving data from the spreadsheet

To retrieve data from your computer (or from a dedicated location), COMEXT spreadsheet offers several options.

Some of the available functions have already been explained in the previous section (Dump to Text, Save from Tabular View), but COMEXT offers additional ways to save data to your computer.

# Dump from the Spreadsheet

To dump data from the spreadsheet, use the "spreadsheet dump" option by clicking on the icon .

The following window appears.

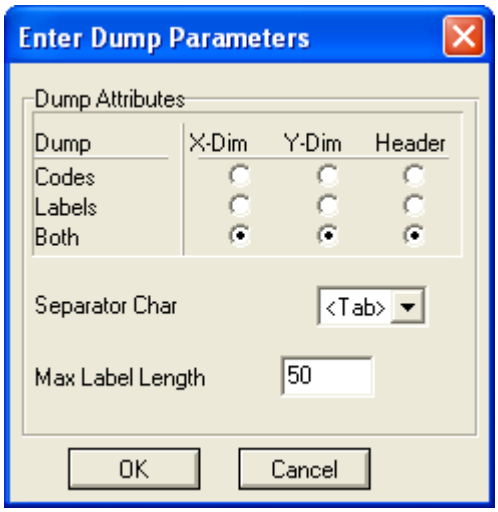



**Figure 31: Enter Dump Parameters window.**

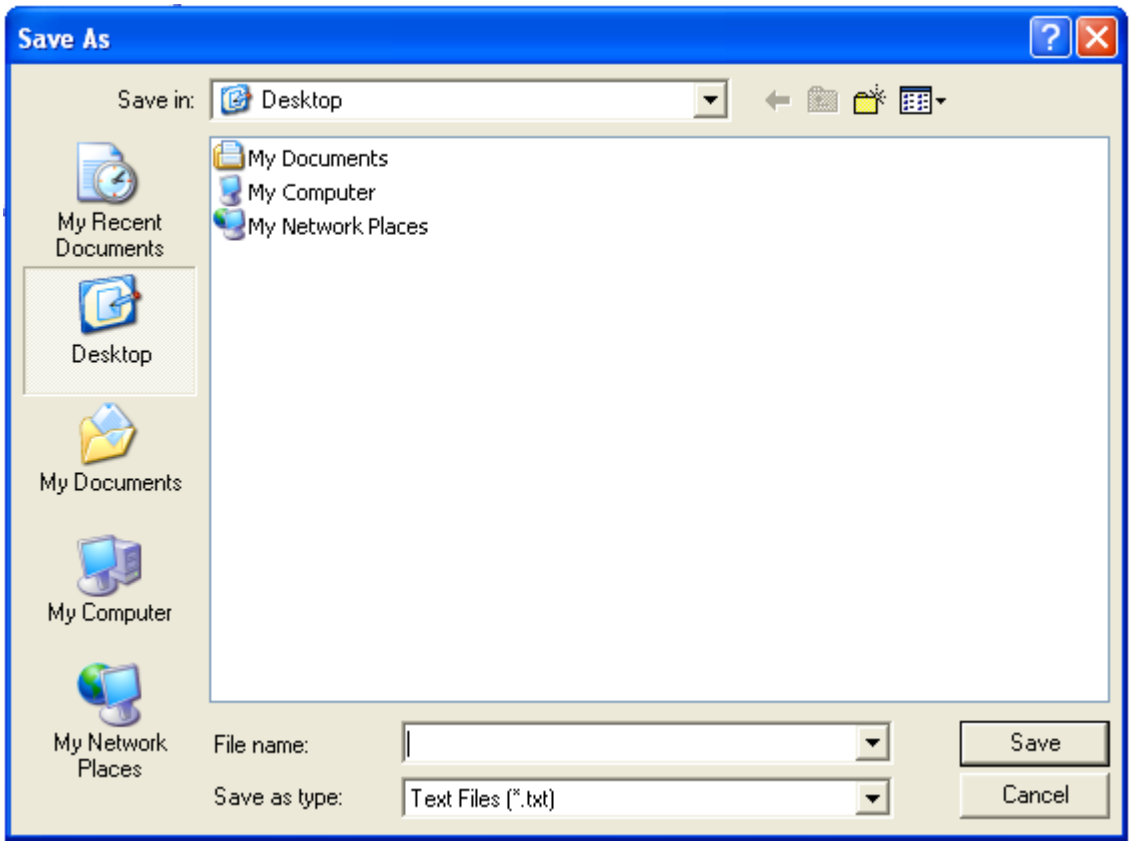
Under "DUMP ATTRIBUTES", choose the required display on the X and Y axes, and their header.

To display:

- codes and labels, click on "Both".
- labels only, click on "Labels.
- codes only, click on "Codes".

These operations must be repeated for the X-axis ("X-Dim"), the Y-axis ("Y-Dim") and the header ("HEADER"). However, only the data for the 'checked' dimensions is saved.

To choose file to save, click to "OK". "Save As" dialog appears:




Other selections from the same extraction can be added to this file. The icon (append to dump) performs this action. Each click on this icon copies the selection displayed on the screen into the previously created file.

# USERLISTS

# Definition

A Userlist is a collection of codes that belong to the same nomenclature. An example of a Userlist would be the group of codes representing the Member States of the European Union, referred to as EUR25. You can create Userlists for all the dimensions in a given dataset.

Userlists are stored on the PC.

COMEXT allows creation of two kinds of Userlists:

- Explicit Userlist: An explicit Userlist is a defined list. It is composed of codes defined during the creation.
- Implicit Userlist: An Implicit Userlist is defined according to an “Expression”. This enables you to create Userlists that follow the Nomenclature evolution and include or remove codes according to revisions that occur over time.

Note: A Userlist can be used in COMEXT as a List or as an Aggregate.

# How to create a Userlist

### Explicit Userlist

### To Create an Explicit Userlist:

1) From the Plan window, click on Select button.
2) Transfer the requested codes to the "Selected" part of the selection window, and then highlight them.

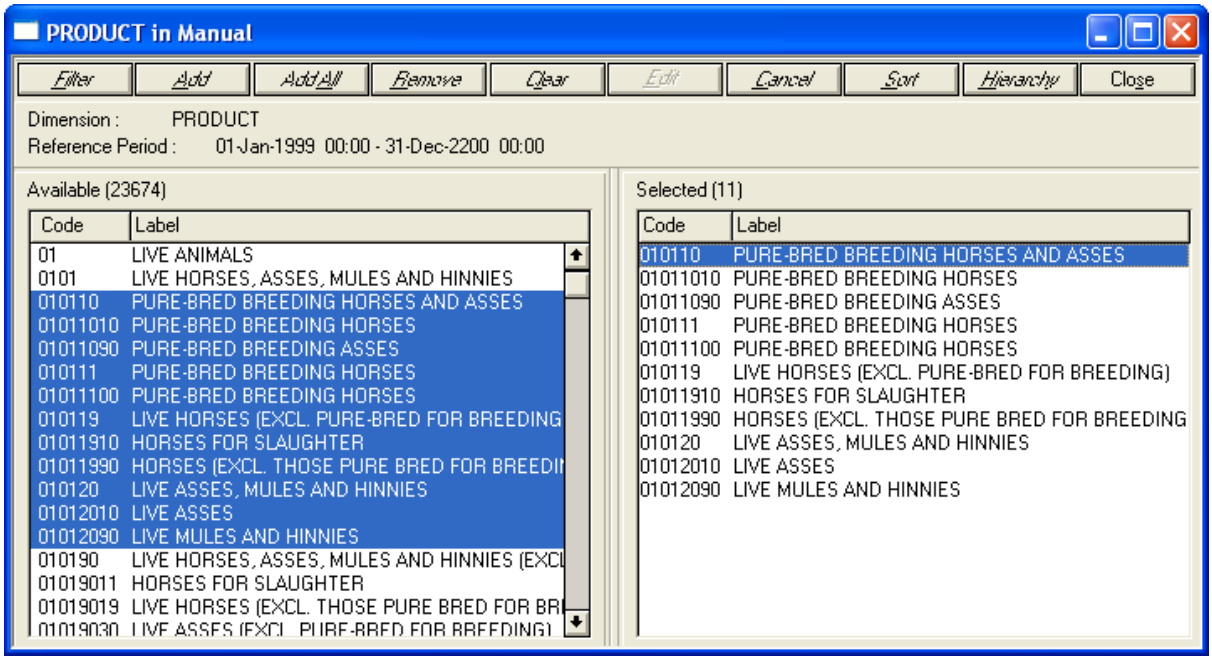
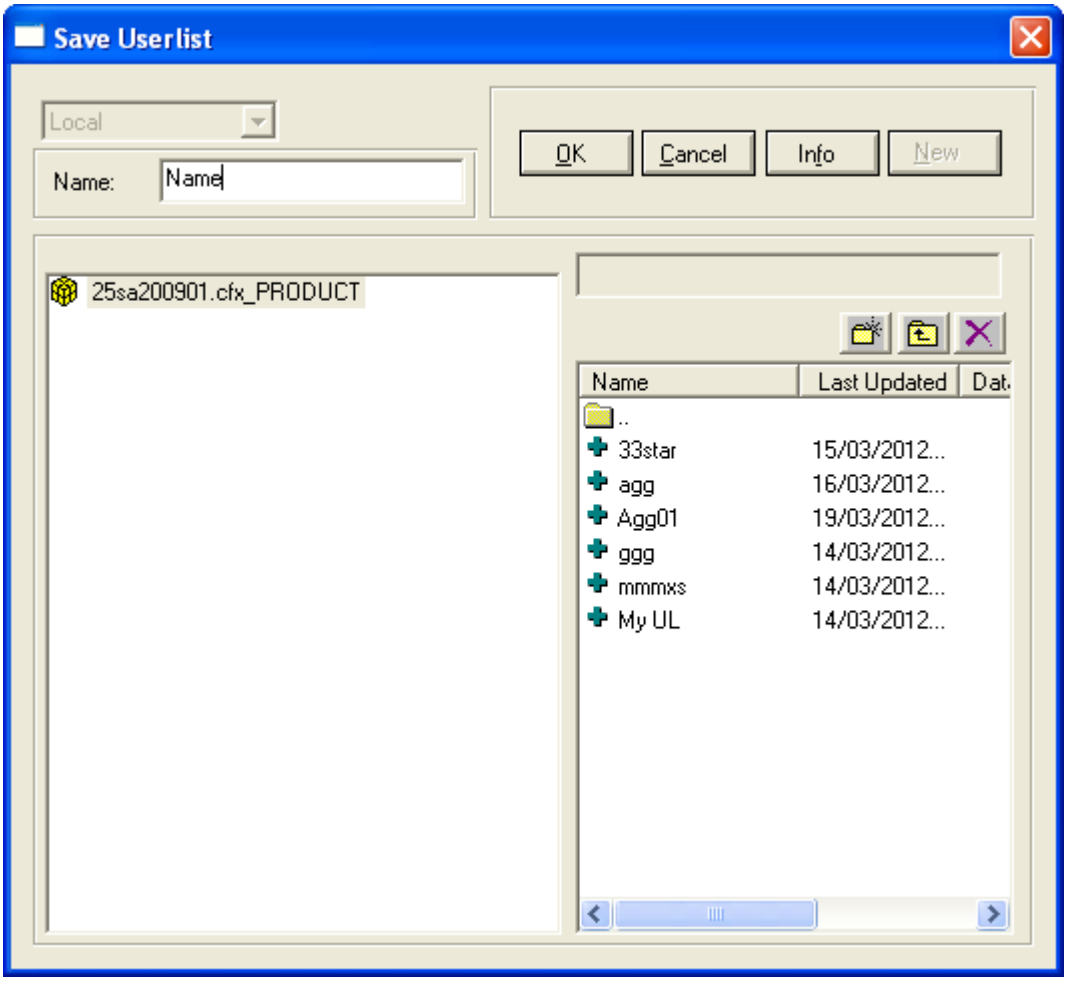



**Figure 32: Product dimension window.**

3) Save the selection and insert it into the plan in the form of a List. This action can be performed via the Aggregate menu, ‘Replace as Userlist’ option or via the icon .

4) Give a name to the Userlist.




5) Select the storage location (Local or Remote).

6) Click OK to save the Userlist.

# Implicit Userlist

**To Create an Implicit Userlist:**

1) From the Plan windows, click on Select button.

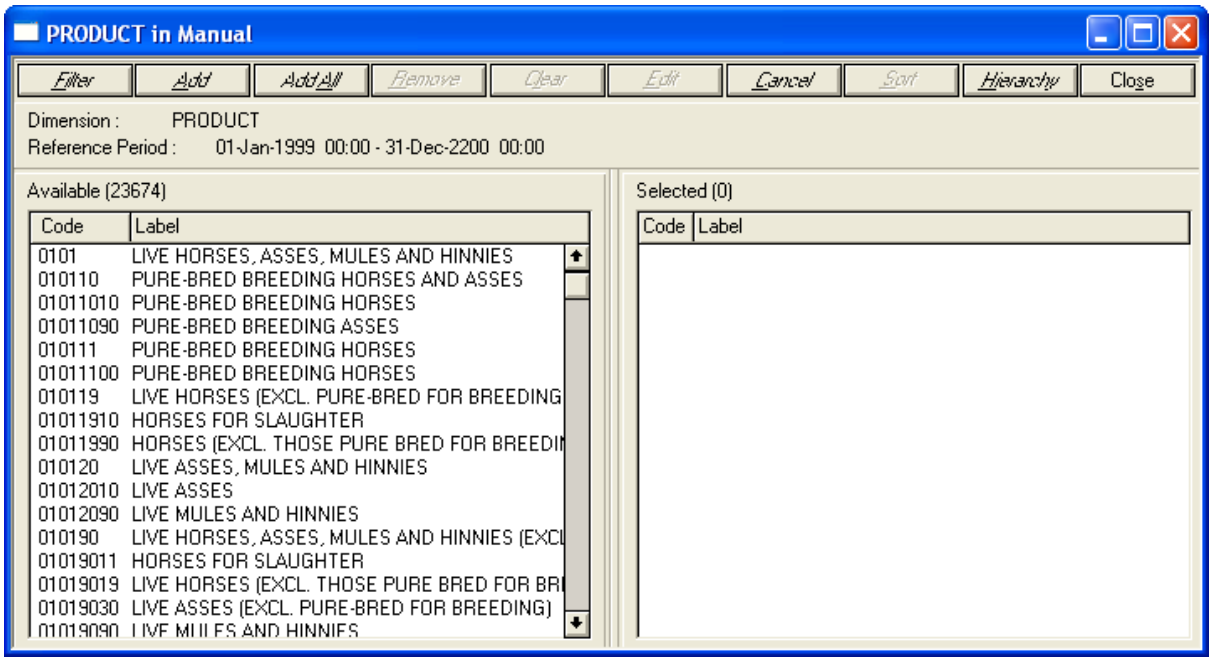



**Figure 33: Product dimension window.**

2) From the Aggregate menu, select Insert Implicit (Aggregate/User List or Aggregate + Userlist).

   This function can also be accessed using the icons.

3) From the Insert Implicit window, give a name to the Userlist in the Selected text box and click on the Insert New button.

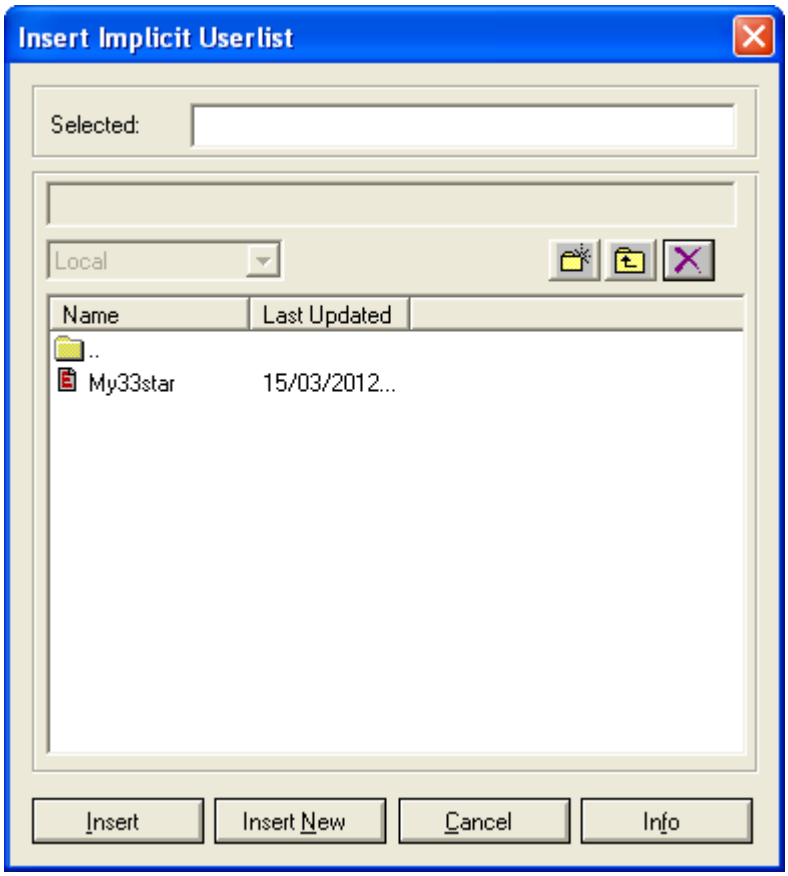
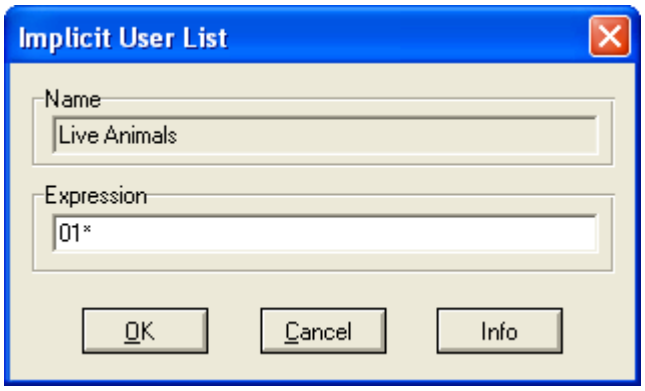



**Figure 34: Create/Open Implicit Userlist window.**

4) From the Implicit Userlist window, write the expression of the Userlist.




Note: To write the expression of the implicit Userlist, you can use the same wild cards described in the 'Filter' section of this manual

# Using the Explicit Userlists

A Userlist can be used in two ways.

- In the form of a list:

A Userlist inserted into a plan in the form of a list will produce a result for each of the codes forming the Userlist. Users can therefore define code groups (countries, products, etc.), permanently save them on their PCs, and insert them into their plans.

- In the form of an aggregate:

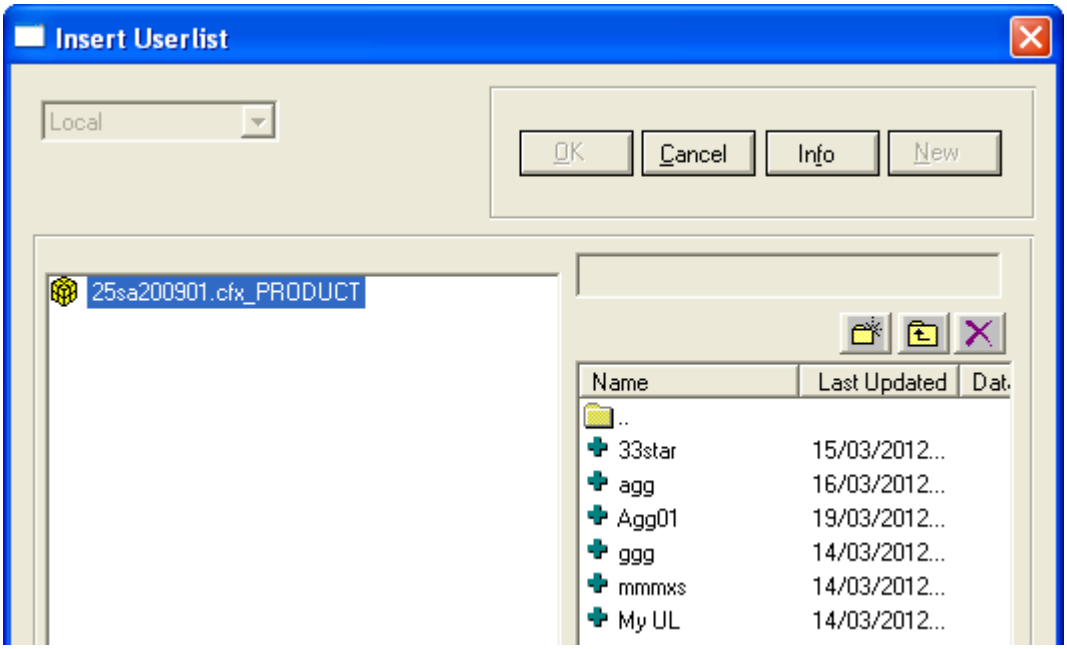
  A Userlist inserted in the form of an aggregate will produce a single result for the entire list. This result will be the sum of the results of each of the codes forming the Userlist.

The combination of the two will display a result for each of the codes forming the Userlist plus a total for the aggregate.

**To Insert a Userlist as a list:**

1) From the Aggregate menu, select Insert Userlist or click the icon .

2) From the Insert Userlist window, select the requested code(s) and click Ok button.




Result: In your extraction, each code of the list will be listed.

**To Insert a Userlist as an Aggregate:**

1) From the Aggregate menu, select Insert Aggregate or click the icon .

2) From the Insert Userlist window, select the requested code(s) and click Ok button.

Result: In your extraction, one code named as the Userlist will be displayed. The resulting value will be the sum of all the individual values of the codes building the Userlist.

**To Insert a Userlist as a List and as Aggregate:**

1) From the Aggregate menu, select Insert Usl +Aggr or click the icon .

2) From the Insert Userlist window, select the requested code(s) and click Ok button.

Result: In your extraction, each code in the list will be listed and the sum will be displayed under the code named as the Userlist.

# Using the Implicit Userlists

As for the Explicit Userlist, Implicit Userlists can be used in two ways.

- In the form of a list:

  An Implicit Userlist inserted into a plan in the form of a list will produce a result for each of the codes forming the Userlist.

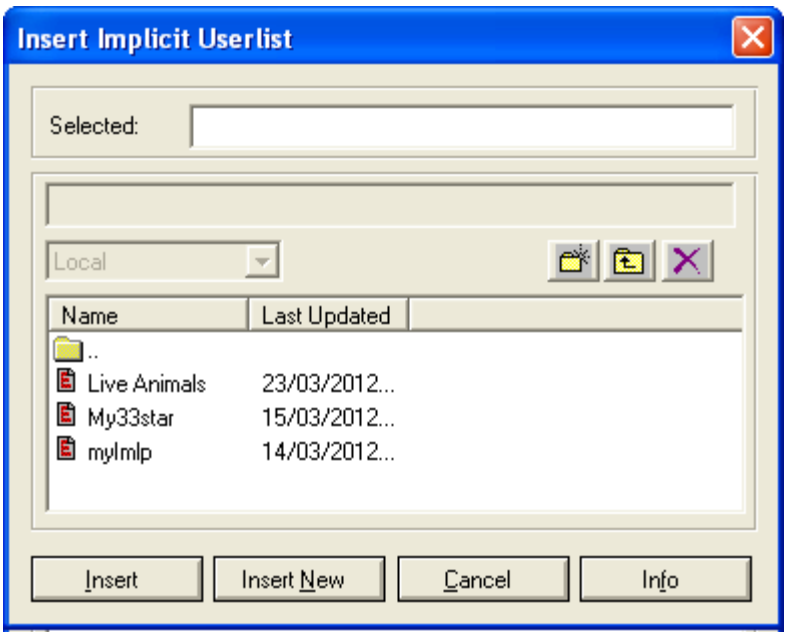
- In the form of an aggregate:

  A Userlist inserted in the form of an aggregate will produce a single result for the entire list. This result will be the sum of the results of each of the codes forming the Userlist.

The combination of the two will display a result for each of the codes forming the Userlist plus a total for the aggregate.

**To Insert an Implicit Userlist as a list:**

1) From the Aggregate menu, select 'Insert Implicit Userlist' or click the icon .




2) From the Insert Implicit Userlist window, select the requested code(s) and click Ok button.

Result: In your extraction, each code in the list will be listed.

**To Insert a Userlist as an Aggregate:**

1) From the Aggregate menu, select Insert Implicit Aggregate or click the icon .

2) From the Insert Implicit window, select the requested code(s) and click Ok button

Result: In your extraction, one code named as the Userlist will be displayed. The resulting value will be the sum of all the individual values of the codes building the Userlist.

**To Insert a Userlist as a List and as Aggregate:**

1) From the Aggregate menu, select Insert Implicit Userlist +Aggregate or click the icon .

2) From the Insert Userlist window, select the requested code(s) and click Ok button.

Result: In your extraction, each code in the list will be listed and the sum will be displayed under the code named as the Userlist.

# Printing the content of a Userlist

The COMEXT system allows printing of Userlist and nomenclature content. A Userlist must be open to use this function.

**To print a Userlist**

1) From Userlist menu, select Print.
2) From the Print Userlist window
   a) select Portrait or Landscape in the Orientation box.
   b) select the Label options.
   c) select the Userlist options (if your Userlist contain Userlists).
   d) click Ok button.

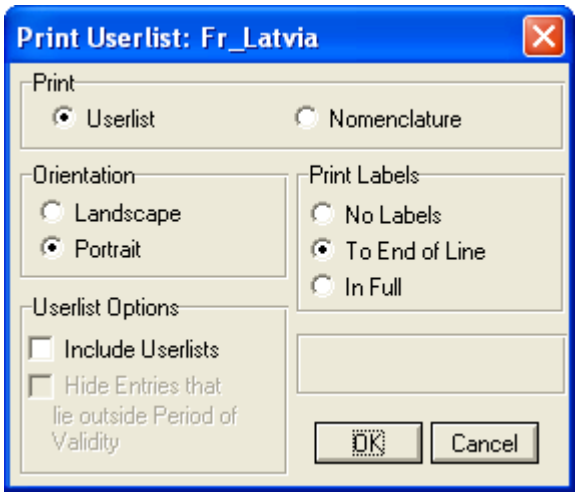



**Figure 35: Printing Userlists.**

# Storing Userlists

Userlists are stored automatically by the system according to the nomenclature to which they belong. For example, the "Olives" Userlist will be stored under the nomenclature NC. You can, however, create your own groups, in which you can store your Userlists. For example, under the nomenclature NC, you may wish to create a textile group, a food group etc.

This can be done in the "Save as Userlist" window by highlighting ‘NC’ and double clicking on it. A window, prompting you to type in the group name, will then appear: Simply type in the name of the group you wish to create.

**To Create a group:**

1) From the Userlist menu, select New Userlist.
2) From the New Userlist window, click the  icon.
3) Give a name to the Group.
4) Click OK.

# Userlist Properties

## Activating/deactivating the Userlist and Aggregate properties.

The  symbol displayed before the name of the Userlist means that the expected result will be composed of the detailed list of code of the Userlist.

The  symbol displayed before the name of the Userlist means that the expected result will be composed of one aggregated. This aggregate will contain the sum of all codes in the Userlist.

In both case, you are using the same definition (list of codes). The use of the Userlist can be changed as follow:

**To activate/deactivate the Aggregate Property:**

1) From the Aggregate menu, select Unset Aggregate Property or select Set Aggregate Property as your wish.
2) In your selection, the aggregate symbol will be removed or added according to your selection.

**To activate/deactivate the Aggregate Property:**

1) From the Aggregate menu, select Unset Userlist Property or select Set Userlist Property as your wish
2) In your selection, the Userlist symbol will be removed or added according to your selection.

# The Reference Period of a Userlist

A reference period can be assigned to each code of a Userlist. This reference period indicates a period of time within which the codes are valid. The user can edit this validity period.

## The Start-Date, End-Date, Weight fields.

In the code selection window, some fields may (or may not) be visible. This particularly concerns fields containing information on the validity dates of the codes and their respective weights.

The validity period and the weight can be displayed, hidden and edited.

Note: code weight is displayed in the Userlist window but not in the Edit dimension window.

## Displaying validity period and weight fields

**To display the validity dates of the codes:**

1) From the Edit dimension window or Userlist window, click the  icon to display the validity period of the codes in the 'Available' part of the window.
2) Click the icon  to display the validity period of the codes in the 'selected' part of the window.

Note: No validation is carried out on the validity dates of a code. If a validity date does not correspond to the actual validity of the code, no trade will appear for this code when the extraction is carried out. For example, choosing 2001 as a end of validity for Latvia will result in no trade appearing for this country (Latvia started to provide data in 2004).

## Editing validity period and weighting

Editing of validity period and weighting is only available in the Userlist window and for aggregates in the Edit dimension window.

**To edit validity period and weight properties:**

1) To edit validity period and weighting of Userlist click the Update icon
2) From the Selected part of the Userlist window, highlight one or several codes.
3) Click the Edit button to open the Edit Weight(s) and Period(s) window.

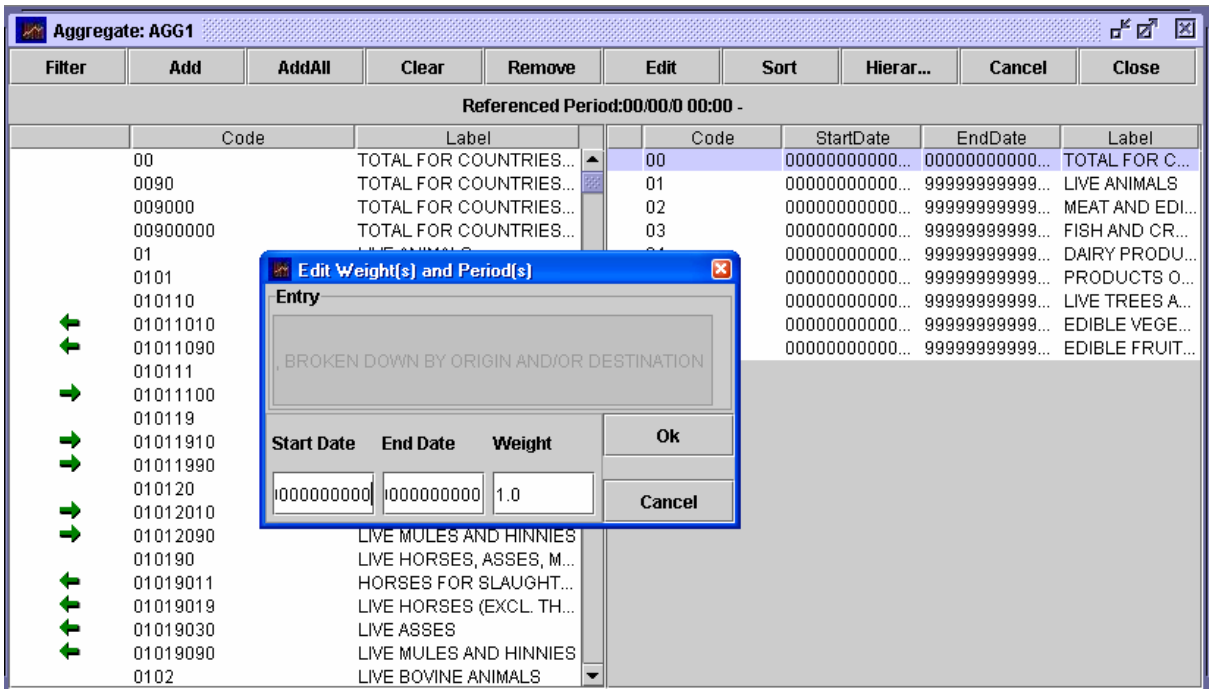



**Figure 36: Example of a code edit.**

4) Enter the start date, the end date and the weight in the respective boxes.

5) Click OK button.

Note: By default, the weight is set to 1. If a weighting other than 1 or -1 is allocated, the unit in which the result will be displayed will be modified. For example, if a code is weighted 0.001 and the result is requested in tons, the actual value of the result will be kilos, since the result will be divided by 1000.

An example of the use of weights is the balance. The Balance Userlist is installed automatically when COMEXT consultation software is installed. This Userlist uses the EUFLUX nomenclature and contains the two codes 1 and 2, for IMPORTS and EXPORTS respectively.

Since the balance formula is EXPORTS - IMPORTS, the means used are as follows: The creation of a Userlist, to be used in the form of an aggregate only (therefore in the form of a sum) is created by applying to code 1 (IMPORTS) a weighting of -1. In this way, the system can carry out a subtraction.

Note: This Userlist should not be used in the form of a list since, having applied a negative weighting to the imports, the result of the imports will be rendered negative.

# ADVANCED FUNCTIONS

This chapter describes the advanced functions of COMEXT. To use these functions, extensive knowledge of both COMEXT and the data used is strongly recommend.

The advanced functions enable advanced data manipulation.

# Role

When using an extraction plan, the ‘Reorganize’ function may be used to change the “roles” of the dimensions and indicators.

This option is accessed via the ‘Roles’ button of the Plan:

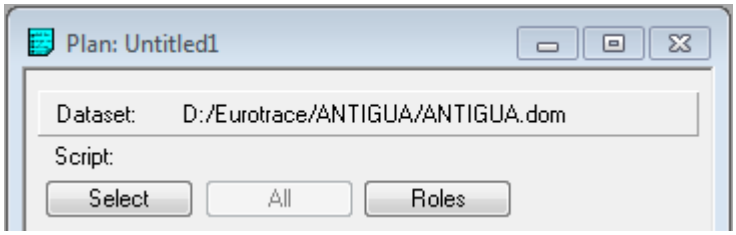



**Figure 37: Roles Button.**

The Roles windows become available with the following options:

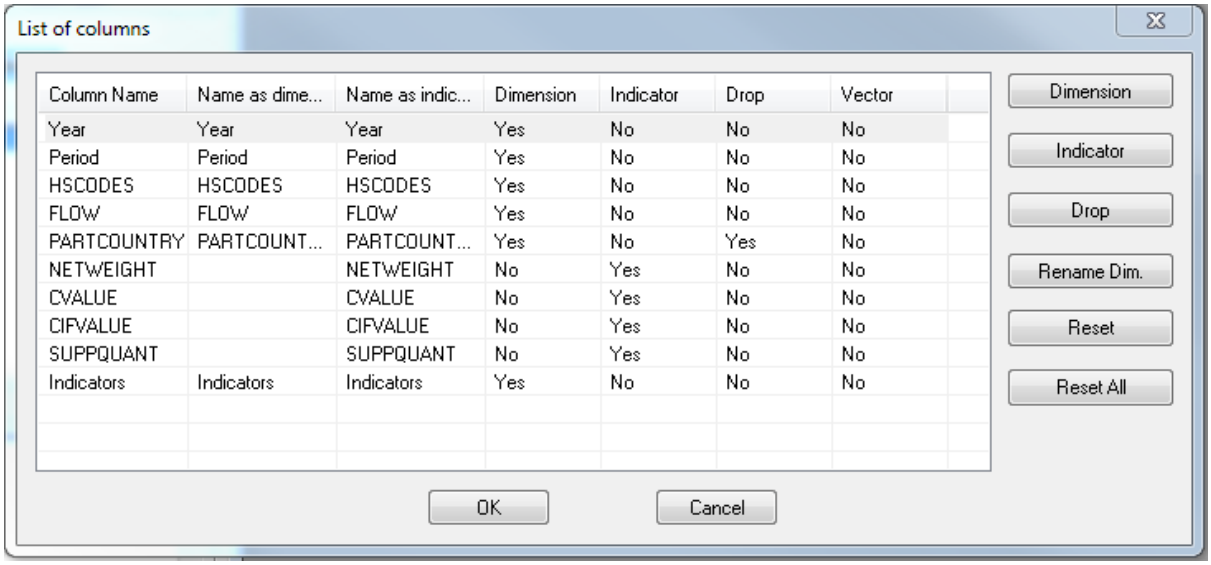



**Figure 38: Role window.**

The ‘Role’ window describes the actual use of the dimension and the indicators. This windows display the dimensions list and the status of each Dimension/Indicator .

## Reorganizing the status of the Dimensions/Indicators

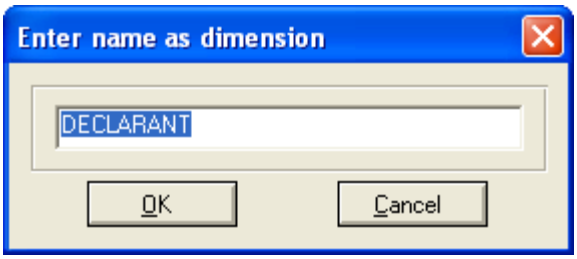
By default, the organization of the Dimension and Indicators are displayed in the Role window.

The Reorganize function enables you to:

- Drop a column
- Change the name of a dimension;
- change the role of a dimension or of an indicator (one dimension can become an indicator and an indicator can become a dimension);

**To change the Dimension name**

1) From the Role window, select a dimension.
2) Click on the "Rename Dim." button to open the following sub window.




3) Give a new name to the dimension.
4) Click OK button.

The given name will be used in your extraction as the dimension name. For example, DECLARANT can be changed to REPORTER.

**To Drop a Dimension:**

1) From the Roles window, select a Dimension.
2) Click on Drop (the option Drop will be turn to Yes).
3) Click OK.

Note: If you drop a column, you won't be able to change the role of any other column. Conversely, if you change the role of a column, the DROP functionality will be disabled.

**To Reset one or all columns to their original state:**

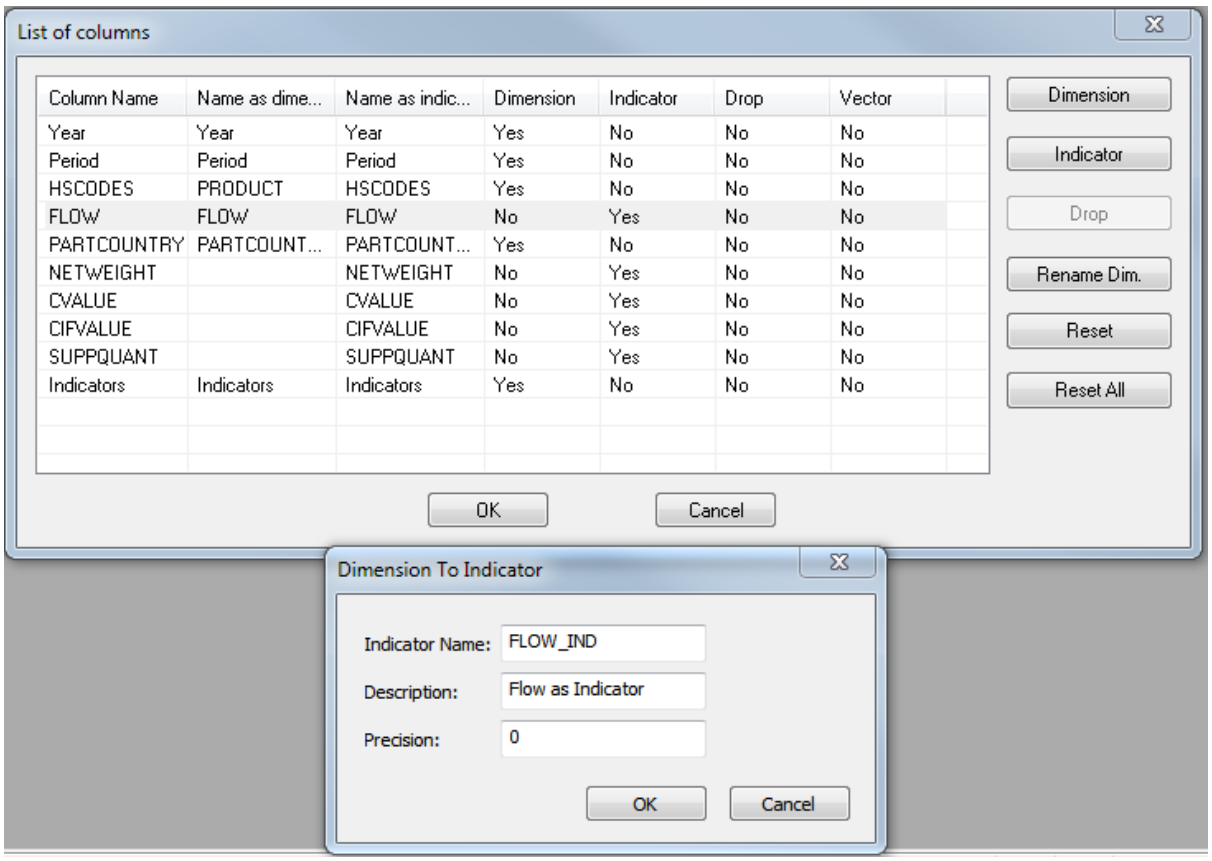
1) Clock on Reset to reset the currently selected column to its original state

OR

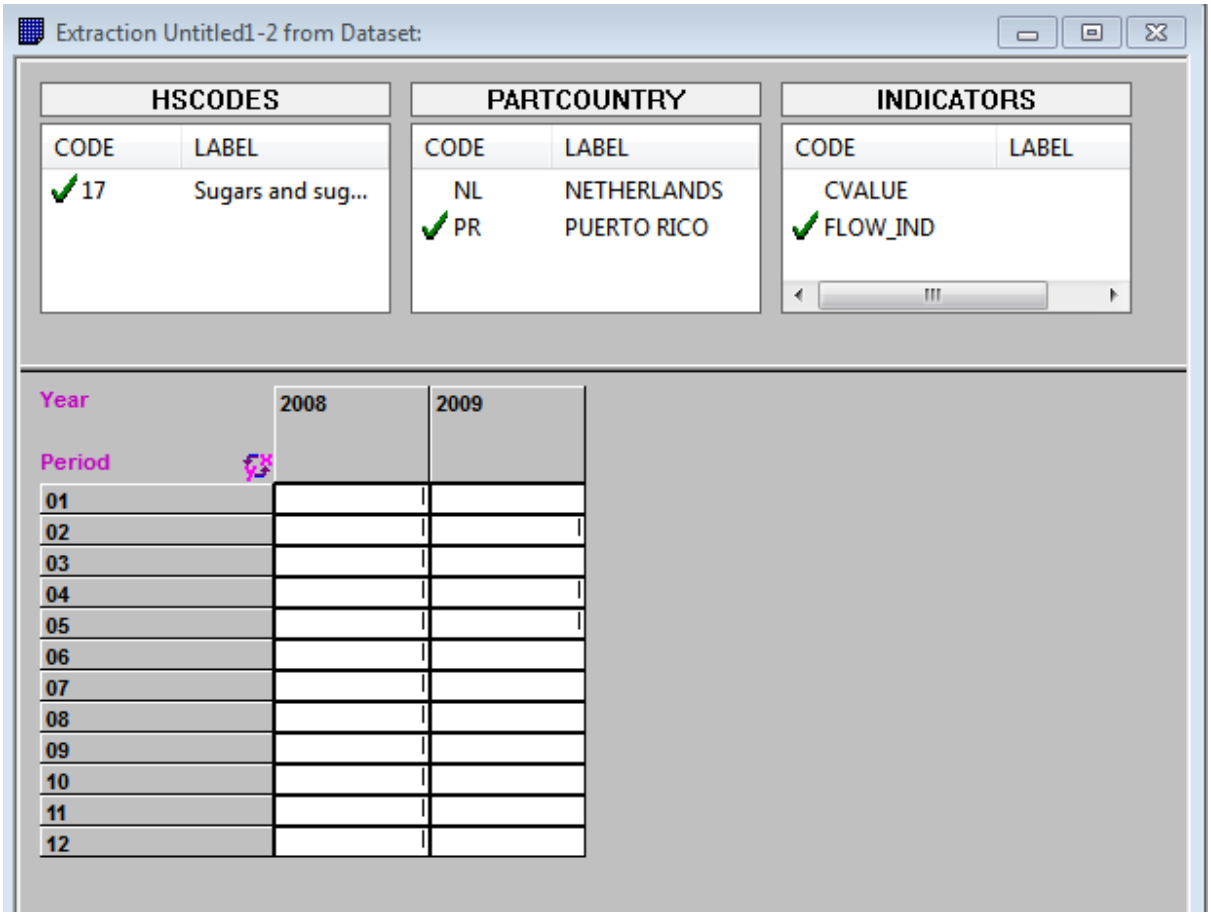
1) Click on Reset All to reset all columns to their original state.

**To set a dimension as Indicator:**

1) From the Roles window, select a Dimension.
2) Click on the "Indicator" Button and enter the new indicator name, its description and the precision (for numbers, you can put 0 for strings)
3) Click OK



The resulting extraction will contain a new indicator named FLOW_IND (the original FLOW dimension is not present anymore)



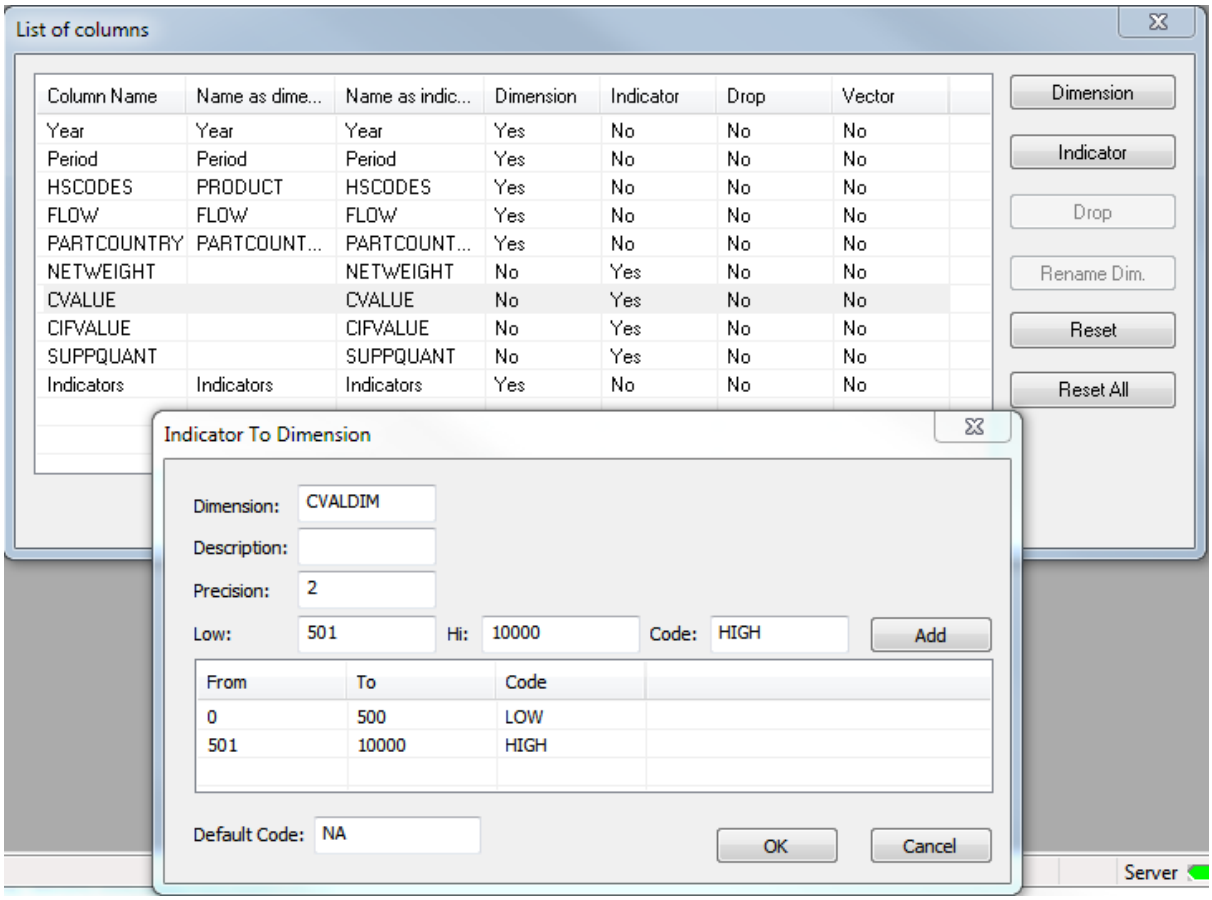
**To set an indicator to dimension:**

1) From the Roles window, select an Indicator.
2) Click on the "Dimension" Button and enter the new dimension name, its description and the precision.

   Transforming an indicator to a dimension requires to define a mapping between a continous measure (the indicator's value) and a set of discrete elements (the codes of the newly created dimensions). There you need to define a set of values ranges and their associated codes by entering the **low** and **high** values of the range and the related **code** (the Add button will add the newly defined range)

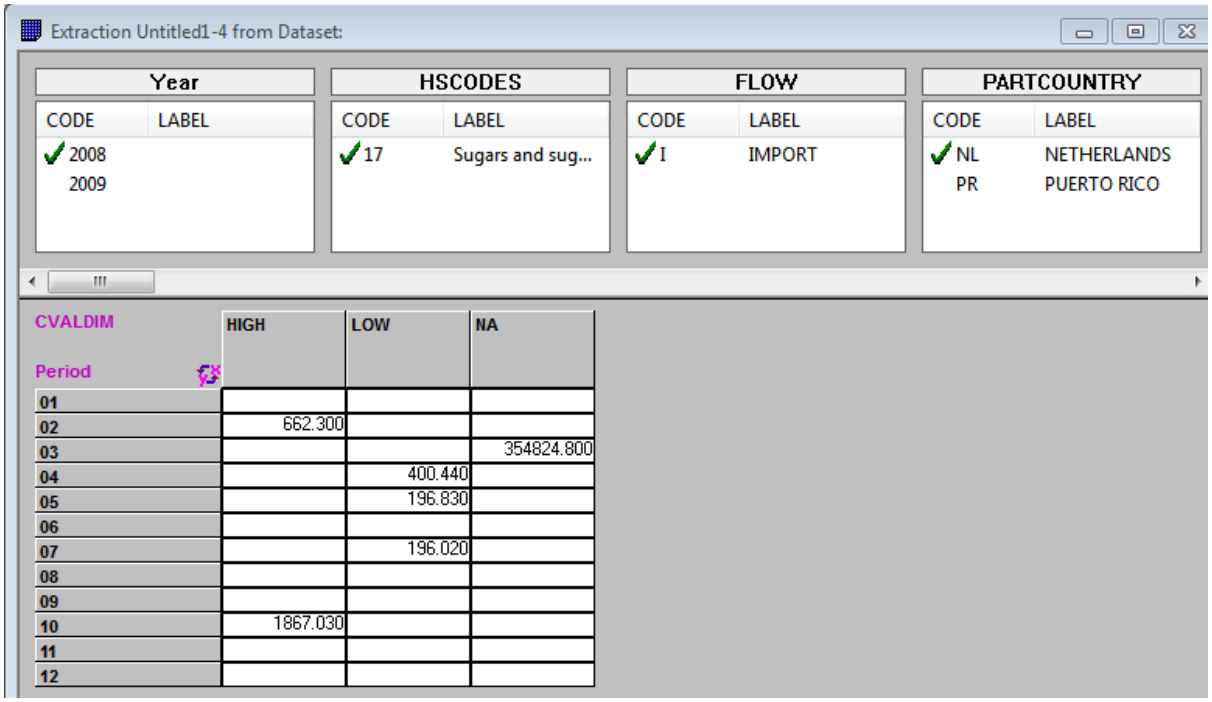
   It is mandatory to define a **default code** as well for all values that will not match any range definition.

3) Click OK




Note: The indicator transformed to a dimension will not appear in the extraction's indicator anymore. Therefore, you need to pay attention that this indicator is not the only defined in the extraction plan.

The resulting extraction will contain a new dimension named CVALDIM with the two defined ranges named LOW & HIGH and the default code NA.



# Ranking and calculation (Subcube functions)

COMEXT enable you to apply several calculations on the extracted data. The Rank menu will enable you to define, via the Subcube window, which calculation you wish to apply on the data.

The Subcube functions window is composed of 4 tabs:

- Aggregate,
- Time (only available with Period as vectorized dimensions),
- Rank,
- Filter.

Each tabs enables specific computation functions.

**To open the Subcube windows:**

1) From the Rank menu, select Rank

   (This function can also be accessed from the Completed Work window, Rank menu)

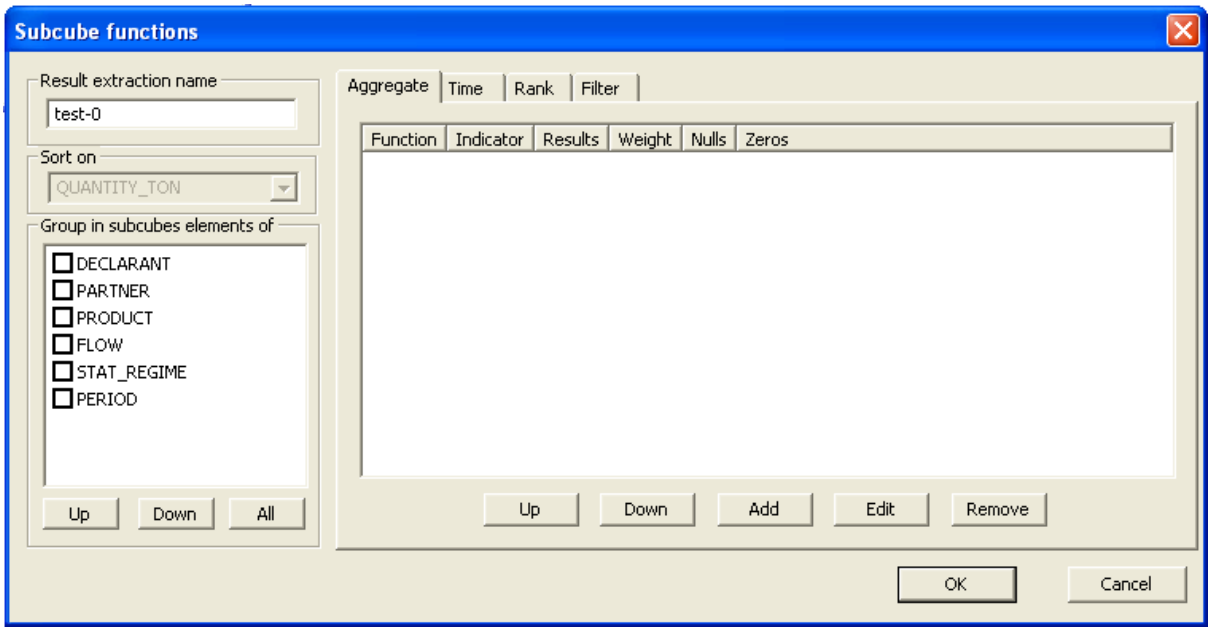



**Figure 39: The Subcube Functions window.**

# Aggregate

This function will allow you to apply calculations on one or several dimension of the extraction. The result of the aggregation will be stored in a new extraction.

**To use aggregate from Subcube windows:**

1) From the Aggregate Tab of the Subcube windows, select the dimension(s) on which the calculation will be applied.

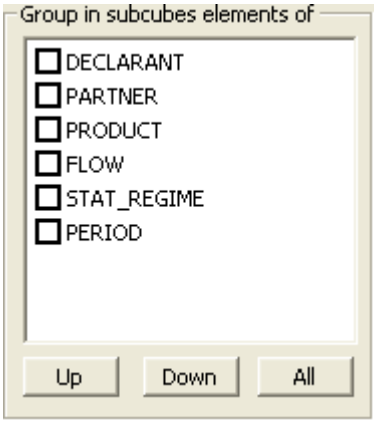



**Figure 40: The Subcube Element window.**

2) You can change the order of the dimension in the list by selecting the dimension and using the Up or down buttons.

3) Give a name to the new extraction to be generated in the "Result of Extraction Name" text box


4) Click on Add button to select the function to apply to the data. Several "Aggregates functions" are available.

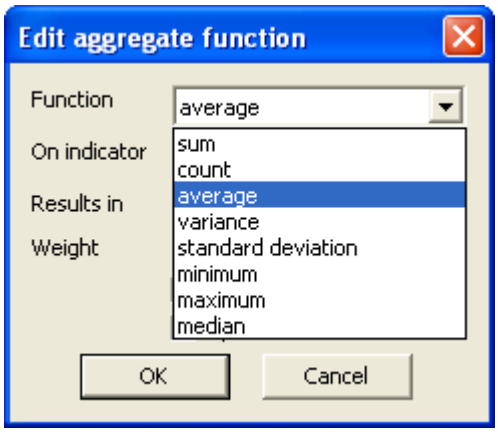


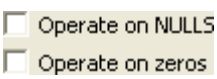

**Figure 41: The Aggregate functions selection Element window**

5) From the "On Indicator" list, select the indicators on which the calculation will be done.

6) From the "Results in" text box, give a name to the new indicators.

7) You will be able to give a weight through the Weight text box. This is only allowed with Median function.

8) Select the "Operate on Nulls" and/or "Operate on Zeros" boxes if needed.

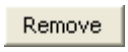



9) Click the OK button to close the "Aggregate functions" windows.

Note: You can apply several calculation functions in the same extraction. The order of the "new calculated indicators" can also be defined by Up and Down buttons.


The button Edit allows you to change the parameters previously set.


The Delete button allows you to delete a function previously set.




10) Click the OK button to start the aggregation.

Note: When launching Subcube functions from the spreadsheet, the result is displayed in a new spreadsheet.

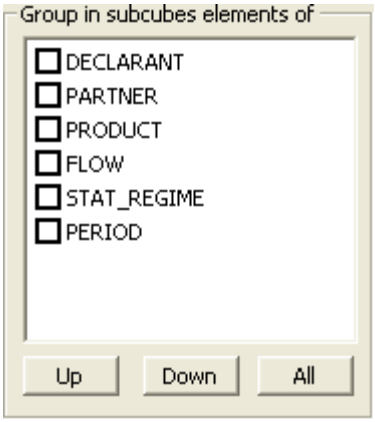
When launching the Subcube functions from the Completed Work windows, the result is stored in a new extraction, as a child of the selected extraction.

# Time

This function allows you to apply calculations based on the time dimension. The Time tab is only available when the Reshape option has veen used to put the Period and only the period as vector (i.e. when the only remaining dimension is the Period).

**To use Time functions from Subcube windows:**

1) From the Time Tab of the Subcube windows, select the Period dimension.




2) Give a name to the new extraction to be generated in the "Result of Extraction Name" text box


3) Click on Add button to select the function to be applied on the data. Several "Time functions" are available.

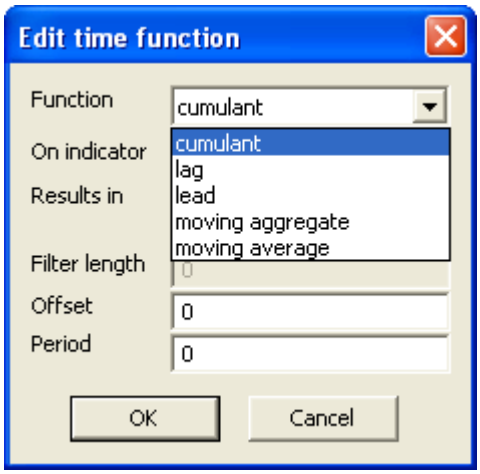



**Figure 42: The Time function selection window.**

4) From the "On Indicator" list, select the indicators on which the calculation will be done.

5) From the "Results in" text box, give a name to the new indicators.

According to the Time function selected, it is mandatory to enter other parameters such as:

- Cumulant

  This function allows you to create cumulated value. To use this function, specify two parameters:

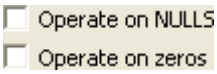
  - Offset: The starting period of the calculation; i.e. the cumulated value will start from the first period of your extraction.
  - Period: The period on which the cumulated value will be done before reset. For example, Period: 6 signifies that the cumulated indicator will increase from period (Month) 1 to 6 and will be reset at period 7.

- Lead, Lag, Moving aggregate and Moving average

  You will need to supply one parameter to run the functions mentioned above.

  - Filter Length: This parameter defines the reference period of your time function.

6) Select the "Nulls" and/or "Zeros" boxes if you wish to use them in the calculation.




7) Click the OK button.

Note: You can apply several calculation functions in the same extraction. The order of the "new calculated indicators" can also be defined by Up and Down buttons.


- Edit button allows you to change the parameters previously set.

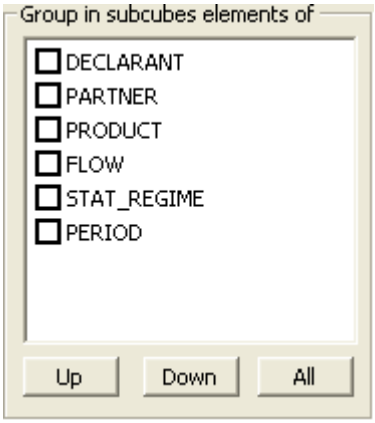

- Delete button allows you to delete a function previously set.

# Rank

This function allows you to apply Ranking calculation on your extraction.

**To use Rank functions from Subcube windows:**

1) From the 'Rank' Tab of the 'Subcube' window, select the dimension(s) on which the calculation will be applied.




2) Give a name to the new extraction to be generated in the "Result of Extraction Name" text box


3) Click on Add button to select the function to be applied on the data. Several functions are available.

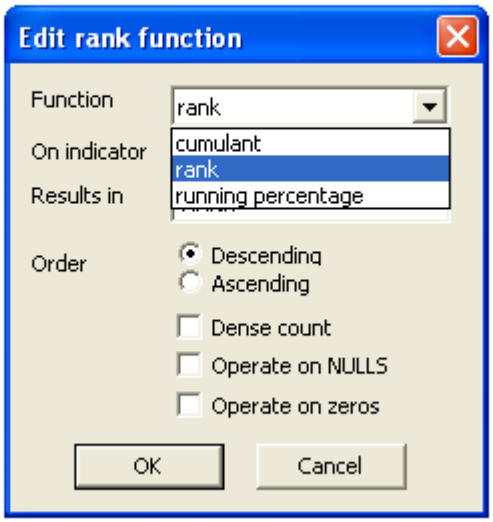



**Figure 43: The Rank function selection window.**

4) From the "On Indicator" list, select the indicators on which the calculation will be done.

5) From the “Results in” text box, give a name to the new indicators.

According to the Time function selected, it is mandatory to enter other parameters such as:

**Rank**

This function allows you to Rank the data. Ranking calculations are performed on the subcube defined by the dimension previously selected.

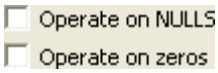
**Cumulant**

This function enables you to add cumulated indicators into your process. A cumulated indicator is linked to the period selected in the extraction. Each new period will display the amount of T + T-1.

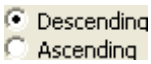
**Running percentage**

This function will allow you to generate the running percentage.

6) Select the order (Ascending or Descending).




7) Select the “Operate on Nulls” and/or “Operate on Zeros” boxes if needed.

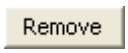



8) click the OK button to close the “Rank functions” windows.

Note: You can apply several calculation functions in the same extraction. The order of the “new calculated indicators” can also be defined by Up and Down buttons.


Edit button allows you to change the parameters previously set.


Delete button allows you to delete a function previously set.




9) Click the OK button to launch the ranking.

# Filter

This function allows you to apply Filters on the data. It will be used to reduce the data in an existing extraction according to defined criteria.

**To use Filter functions from Subcube windows:**

1) Give a name to the new extraction to be generated in the “Result of Extraction Name” text box.


2) From the Filter Tab of the Subcube windows, select the criteria you want to apply on the data. This selection is performed using following sub window:

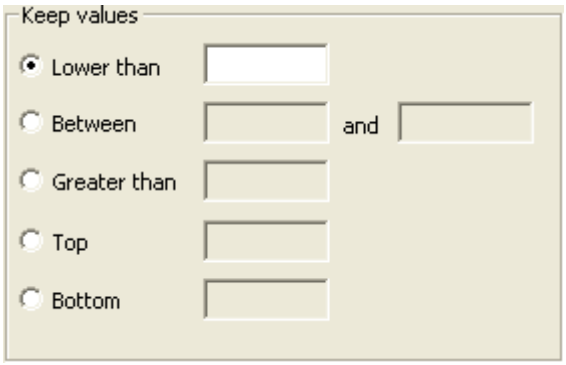
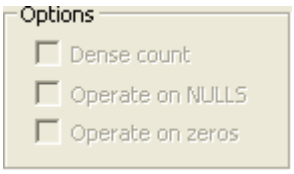



**Figure 44: The Filter "Ranking" window.**

3) Select the indicators on which the filter will be applied.


4) Select the options "Dense Count", "Operate on Nulls" and "Operate on Zeros" boxes if needed.




5) Click on the OK button to launch the Filter.

# Formulas

The use of Formulas permits the inclusion of calculated fields in the extraction, much in the same way as spreadsheet programs, like Excel.

Because of this our dependency on spreadsheets can be reduced, allowing us to easily obtain answers to familiar problems like:

1) What is the growth rate of trade between years 1998 and 1999 for a given country? Using the formula: `rate=(total1999-total1998)/total1998` in the "period" dimension
2) For a given product and a given declarant country, what is the percentage of exports to partner country x in comparison with the total of exports? Using the formula: `XW=x/world` in the "partner" dimension
3) For a given product, what is the relationship between the value and the quantity exported? Using the formula: `indexvq=(val/quantity)*100`.

Note:The system will calculate the aggregates first, and then the formulas. The formulas are post-computed, i.e. if we assume:

`Aggr1=x1+x2 and Aggr2=x3+x4`

and the formula `Z=val/quantity` then

`(aggr1/aggr2) * Z=(x1*val+x2*val)/(x3*quantity+x4*quantity)`

The creation and insertion procedure is similar to that for the Userlists.

**To use formulas:**

1) From the Edit Dimension window, Select Aggregate menu.
2) Select Insert Formula option or click the icon .

The following window appears.

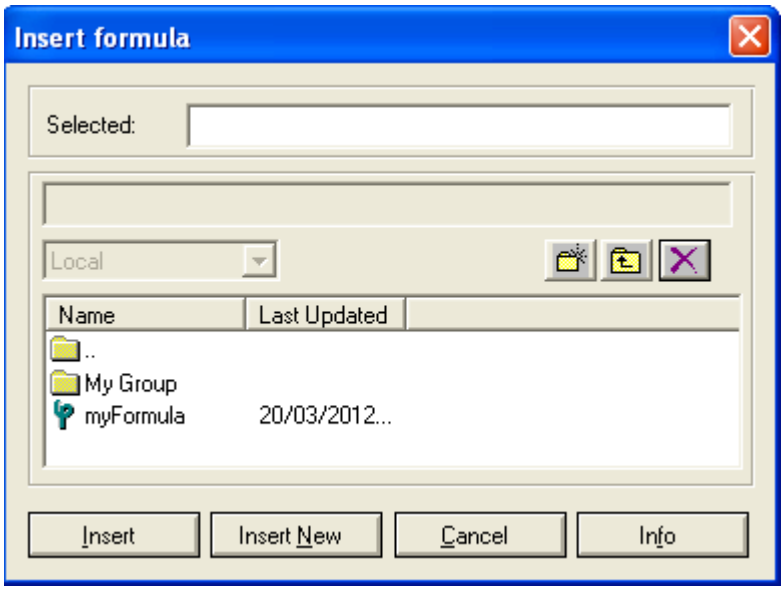



**Figure 45: Inserting a formula.**

3) Select an existing Formula

**To create a formula:**

1) From the Edit Dimension window, Select Aggregate menu.
2) 2) Select ‘Insert Formula’ or click the icon.
3) Give a name to the formula.
4) Click on ‘Insert New’.

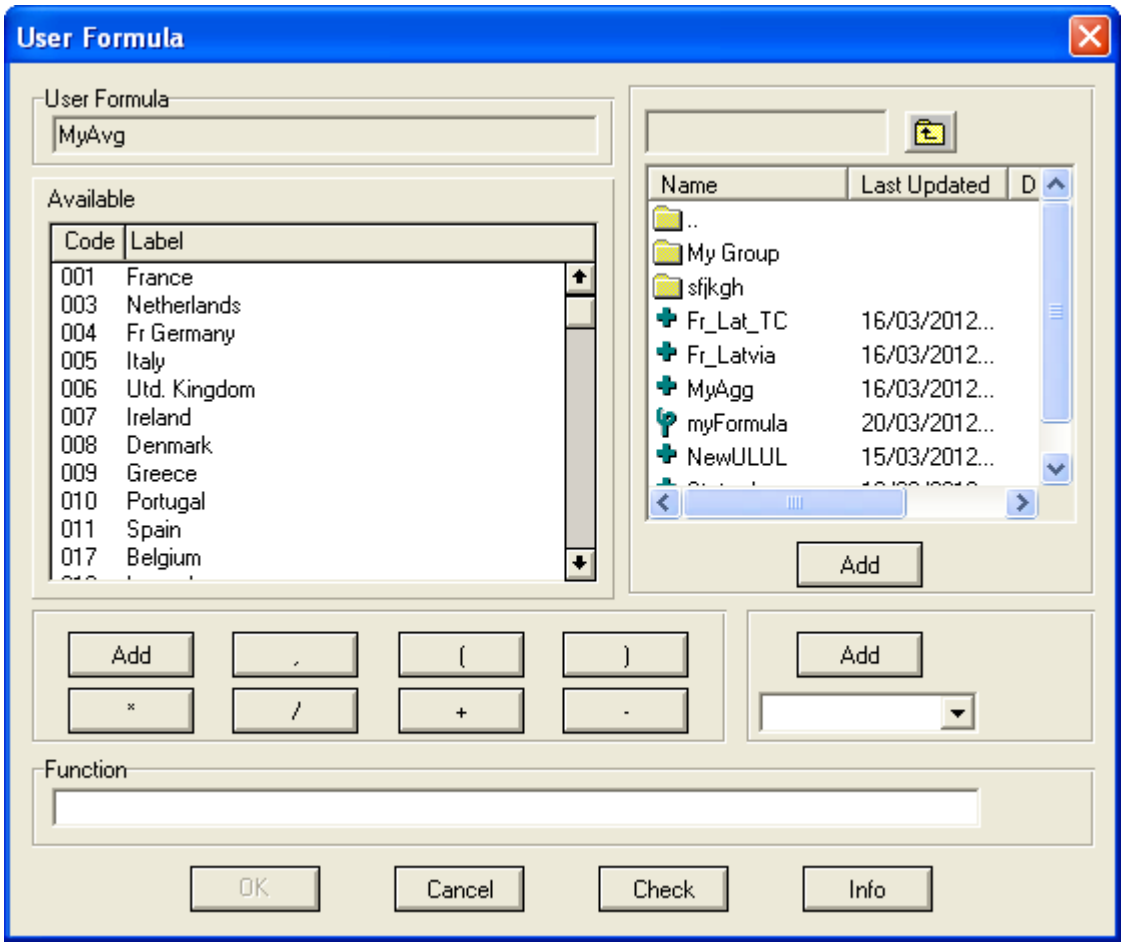



**Figure 46: Creating a formula.**

5) Insert the formula into the "function" box.

   The various options include the usual arithmetic operators as well as the basic mathematical functions.

6) Use the 'Check' button to validate the syntax.

7) Click 'OK'.

Note: Several basic operators are available from list.


You can use the operators selecting and then clicking on them. The Add button will include it in your formula.

The "Info" button is used to attach additional information about the formula.

# Frequently Asked Questions

**What is the difference between a query and an extraction?**

The query is the file containing instructions given to the system to extract the data. It is stored by XTNET on the user's PC and is a way to save the work done for the selection of different dimensions. By contrast, the extraction is the "raw data". A subset of the data stored in the XTNET database, selected by following the instruction in the query. This selection of numbers is stored on the server, in the server space provided for every user.

**Why do some datasets have different dimensions than others?**

The number of dimensions in a dataset depends on the different types of information the dataset stores. The more detailed the statistical information, the more dimensions are required to store the data. For example:

The **"Information Availability"** Dataset (from the Trade Domain EU) requires **four types** of information to give a complete view of the availability of data:

The Reporter country, the period concerned, the type of trade, and the indicators (upload & last update). By contrast, for the **"Monthly, Since 1988**" (from the Trade Domain EU /Trade by Product (CN)) that stores the actual trade information, we see that the partner, product and statistical regime dimensions are absent because this information is not needed.

**I forgot my password, what can I do?**

E-mail the XTNET support team at: Comextsupport@ec.europa.eu

**How do I exchange queries with someone else?**

There are special menu commands that facilitate this procedure. Every query can be saved in the form of a text file with the use of the command: "File/Export Query". Any file saved in this form (.sde) can be imported to the same or other XTNET installation by using the command: "File/Import Query".

**What is the difference between a Null and a Zero?**

In XTNET, as in all databases, a null signifies the absence of data (i.e. no information sent). In the spreadsheet view, this is presented as an empty cell, however, this is totally different to having a cell with zero in it. The latter, means that information was sent and it has the value '0'.

**What is the difference between a Userlist and an aggregate?**

Both Userlists and aggregates are groups (lists) of codes. In reality, XTNET stores them and treats them (in operations like delete, open etc.) in exactly the same way.

It is the way we insert them into a dimension that makes the difference. If we chose “Aggregate/Insert as Userlist” then what we selected will be treated as a collection of separate entities (Userlist).

If we choose “Aggregate/Insert Aggregate”, then what is selected will be treated as a single entity (the total of all codes).