User Manual WEB COMEXT version 17
INTRODUCTION

The User Manual ............................................................................................................. 6

Welcome to XTNET ........................................................................................................ 6

What you will learn from this book .............................................................................. 6

Prerequisites .................................................................................................................. 6

Architecture .................................................................................................................. 7

Connection .................................................................................................................... 7

Training ......................................................................................................................... 7

Hardware ...................................................................................................................... 7

Terms and definitions ................................................................................................... 8

Client-Server ................................................................................................................ 8

The Database ............................................................................................................... 8

The Domain ................................................................................................................ 8

The Dataset ............................................................................................................... 8

The Dimension ......................................................................................................... 9

The Query .................................................................................................................. 9

The Nomenclatures ................................................................................................... 10

Web Client .................................................................................................................. 10

Java Plug-in Installation ............................................................................................. 10

Work with a local database ......................................................................................... 11

Working with the remote database ........................................................................... 11

Error Reporting ......................................................................................................... 11

Storage of Plans and Userlists: .................................................................................. 12

Starting the XTNET Client ......................................................................................... 13

Login to the application through ECAS (Single Sign On) ................................... 13

New Users of ECAS that want access to COMEXT ............................................. 13

Existing ECAS users that want access to COMEXT ............................................ 14

Setup Default Query and Default Userlist Domains ......................................... 15

Main interface .......................................................................................................... 16

Selecting the Language: .......................................................................................... 23

Detailed information ............................................................................................... 23

Managing your server dedicated space ................................................................. 26

Protecting your installation .................................................................................. 26

THE QUERY

Contextual menu for the Query ................................................................................. 27

Creating a new query: ............................................................................................. 27

Defining the content of the Dimensions .............................................................. 30

Contextual menu for the Query when a dimension is in Edit mode ................ 32

Window of a Dimension ......................................................................................... 32

Find code(s) in a dimension (the ‘Available’ window) ...................................... 33

FILTER ..................................................................................................................... 34

Saving the query: .................................................................................................... 51

Information on query: .......................................................................................... 52

Printing the Query ................................................................................................ 52

Importing/Exporting query, Userlists and documentations ........................ 54

Importing/Exporting queries: ............................................................................... 54
### FORMULAS

- 137

### THE PREDECESSORS AND SUCCESSORS:

- 142

## EXTRACTION OPERATIONS

- 147

### Filter Data
- 147

### Transpose
- 149

### Interpolate Time Series
- 153

### Values Types
- 154

### Drop
- 154

### Reshape
- 157
  - Concatenate dimensions
    - 157
  - Split Dimensions
    - 159
  - Swap a dimension to Indicator
    - 161

### Subcube Functions
- 163
  - Aggregate
    - 163
  - Advanced aggregate functions
    - 165
  - Time
    - 167
  - Rank
    - 170
  - Filter
    - 172

## SERVICES

- 173

### Services functions
- 173

### Create New service
- 174

### Edit an existing service
- 174

### Subscribe to a service:
- 175

### Outputs
- 176

## OUTPUTS

- 177

### Print
- 177

### Dump
- 178

### Generate Table
- 180

### Out Data
- 183

### SDF
- 186
  - SDF Without Transposition
    - 187
  - SDF with Pure Transposition
    - 188
  - SDF with Tuple Transposition
    - 189
  - SDF With EUROBASE
    - 190
Graph ................................................................. 182
Map ....................................................................... 195
XTTable ................................................................... 196
EUROBASE ............................................................ 197
CFX ........................................................................ 199

Macros .................................................................. 201
Macros general user options .................................... 201
Viewing new macros ................................................ 202
Scheduling macros execution .................................... 203
List of already defined user macros ......................... 204
Examples of using macros ....................................... 205
  Example 1: How to record a macro (simple) ............ 205
  Example 2: Defining the Drop operation (advanced macro recording) ........................................... 205
Important Macros information - Troubleshooting ...... 206
  My macro show no result in the application ............ 206
  My macro recording session is expired ................. 206
  I cannot use function X in my macro recording ...... 206
  Can I do other operations while recording a macro? 207
  Error X during recording process ........................ 207
  Execution of macro terminated with an error .......... 207
  Editing a macro .................................................... 207

Frequently Asked Questions .................................. 208
  What is the difference between a query and an extraction? .... 208
  Why do some datasets have different dimensions than others? .... 208
  I forgot my password, I have an issue, what can I do? .... 208
  How do I exchange queries with someone else? ....... 208
  What is the difference between a Null and a Zero? .... 208
  Green arrows pointing to the left or to the right. What do they mean? .... 209
  What is the difference between a Userlist and an aggregate? .... 210
  Why is the software disconnected every now and then? ... 210

TABLE OF FIGURES .................................................... 211
INTRODUCTION

The User Manual

Welcome to XTNET

XTNET is the database system for Foreign Trade between the Member States of the European Union and the rest of the world. As with the preceding version of XTNET, this database can produce a whole list of statistical spreadsheets.

What you will learn from this book

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

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

This manual is divided up into six sections:

- INTRODUCTION
- THE QUERY
- THE EXTRACTION
- USERLISTS/AGGREGATES
- EXTRACTION OPERATIONS
- SERVICES
- OUTPUTS
- FREQUENTLY ASKED QUESTIONS

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 manual (Basic Principles of Windows) provides a useful reference.
Architecture

Data stored in the XTNET database resides at a central site that can be remotely accessed by your PC via Eurostat machines.

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

Connection

Using XTNET, requires authorization to access the database. The form used to apply for this authorization is included in Appendix I of this manual.

Users of the European Institutions:
Complete the first part of this form, put “New Comext Connection” as the subject, and e-mail it to the XTNET support helpdesk mailbox (comextsupport@ec.europa.eu).
You will receive an acknowledgement and, a few days later, you will receive the necessary software and tools to install XTNET on-line. Contact your CIS to perform the actual installation.

External Users:
Send your signed request, accompanied by a letter of request stating your need for XTNET, to the XTNET support helpdesk by e-mail (comextsupport@ec.europa.eu). Put “New Comext External Connection” as the subject. Your request will be considered.

Training

You are strongly advised to attend one of the XTNET training courses, held in Brussels and Luxembourg throughout the year. The timetable for these courses is shown in the document entitled "Training courses in the consultation of statistical databases organized by Eurostat". If you have not received this document, or if you require further information, please e-mail the XTNET support team at comextsupport@ec.europa.eu

Hardware

To use XTNET, a PC equipped with a Pentium Processor and at least 256MB of memory is strongly recommended. Please check with your I.T. support 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 Domain**

The domain is a collection of related datasets stored under a single name. The accesses to the Domains are regulated and depend on user profiles.

![Figure 1: The domains accessed by the XTNET client](image_url)

**The Dataset**

A dataset comprises a large set of related data and is valid for a specific period. For example, the dataset "EU27, Since 1988" under [EU27 PUBLIC TRADE DOMAIN] includes the EU trade data since 1988, following the CN nomenclature.
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 XTNET dimensions is shown in figure 3 on the next page.

**The Query**

The query 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.
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, XTNET stores statistical information using several different nomenclatures.

Web Client

Java Plug-in Installation

The Web XTNET Client is an applet written in Java and it works with:

- Microsoft Internet Explorer.
- Mozilla Firefox.
Google Chrome.

It uses the latest functionalities of the Sun’s Java runtime environment (JRE) and, for that reason; the Java Plug-in must be installed on your computer. If it is not, download and install the latest version from the Java web site (http://www.java.com/).

The Web XTNET Client is working with Java version 1.7 and Java version 1.8. We strongly recommend using the Java version 1.7 if possible with last update version.

Work with a local database

Local database is deprecated

From the WEB Analytical Client 14 and later versions the system is only able to read existing user queries and associated object. We strongly advice the user of the system to migrate to the Remote database using WEB Analytical Client 13: Tools/Transfer local database to server. The latest version of the WEB Analytical Client can be reached directly through this link:

http://comext.eurostat.ec.europa.eu/Analytical.html

NB: The migration does not prevent usage of any previous versions of the WEB Analytical Client or the usage of the WEB Analytical Client C++ for any version that supports fully the remote database storage.

Working with the remote database

This is now the default storage used for user queries and any user objects. This prevents any migration issues, and it allows accessing directly all user objects from any computer.

You can use the application without a local access database, storing plans and objects (Userlists, aggregates, formulas, etc.) remotely on the server. This allows you to access your stored objects, using the Web Client, from any other system.

Error Reporting

After you login, the application creates a log file ‘cxtlog.txt’ in your ‘temp’ directory. The log file tracks all messages exchanged with the XTNET Server. (NB: It does not contain any sensitive information).

In case of any error, the WEB Analytical Client will ask you to save a ZIP archive that contains fully such log file. You can give the name you want to this zip file (applying Windows limitations). In order to speed up and to simplify to solve any issue, it is important to attach such ZIP file to the Comext support: comextsupport@ec.europa.eu.
Storage of Plans and Userlists:

XTNET is actually available in two versions:
- **Windows**
- **Web**

The differences are linked to the connection procedure and to the database that can be used. The connection will be done through the Internet for the web client and through a dedicated line for the windows client.

The database that will be accessed by the web client will be the main database, located to the EU Commission computer center.

The windows client will have the capability to access the main database (the same as for the web client), but also a local database (XTNET DVD or dedicated database for the standalone version).

According to the version you are using, you will have the possibility to select the location of storage for the several files generated by XTNET (Plan, Userlists, etc.).

**Selection of storage folder:**

As describe here above, this version can be use with a local database or with the main database.

Using the standalone version (local database), all the queries and the userlists that will be created will be stored locally (on your computer or on your local network)

Using the windows client with the main database will enable you to select the storing location for your queries and your userlists (Remote or Local).

This selection can be done from the Query/Open menu or from the Query/ Save As menu.

![Figure 4: Save as (Remote or Local)](image)

**Note:** The **Remote** selection will enable you to make available all your queries and userlists from any computer having access to XTNET. As stated previously, the WEB Analytical Client 14 does not allow any more storage in the local database, refer to the Section “Local Database is deprecated” for the migration procedure using the WEB Analytical Client 13.
Starting the XTNET Client

Login to the application through ECAS (Single Sign On)

New Users of ECAS that want access to COMEXT

In order to use COMEXT you will firstly need to have a user account in ECAS. Please follow the instructions below to create an EXTERNAL user, Commission users already have an ECAS account. If you are an existing ECAS user you do not need to follow steps 1 and 2. Please continue directly to step 3.

1. Choose a login name. You are strongly advised to choose your individual professional email address (or alternatively, your personal email address), which will be easy to remember next time you try to login to Comext. Please also provide the additional information requested in the form.

2. Set your ECAS password. Once you fill in the personal details requested, an email will automatically be sent to you with a link to set your password. You have 90 min to click on the link (if you do not receive this automatic email, please notify the ECAS Helpdesk).
Existing ECAS users that want access to COMEXT

3. Please contact the Comext support (comextsupport@ec.europa.eu) in order to request to create you a user within the COMEXT. This step is mandatory to grant authorisation for you to fully access the system.

**Note:** Following the creation of the user attention is needed during the login, to select your type of user in the EC’s network. European Commission Users should select “European Commission User”, while all other users “External”.

![ECAS login window](image)

**Figure 5:** The login/password input window. (example for External user)

After a successful login, the application will be available.

(i) **Inactive Account**

The XTNET administrator can deactivate your account due to different reasons like:

- User space at server side is full
- User is submitting too big extractions
- Other…

In this case, you will receive an email notifying you about the de-activation of your account. While your account is not active and you try to login you receive the following message
(ii) Active Account

When the XTNET administrator re-activates your account, you receive again an email notification.

In this case, you can provide your username and password and login to XTNET.

In case of any problem please contact the XTNET administrator and provide detailed description of your problem. Sometimes error can happen. In this case, press Report Error to save the detailed information about the problem on your local computer. If it is possible please attach this error to your email. After reporting the error press Close button and error will be closed.

---

**Setup Default Query and Default Userlist Domains**

This section only applies if you have created a new account on XTNET, or you have to switch to another domain: you have to set a default query domain and set the default userlist.

From the Menu “Options/Set Default Query Domain”, select “REF DATASET DOMAIN (Expanded)” (this is for our example; in true life you can choose any other domain you want to work in and you have access to) as shown in the next figure, then click on Ok.

From the Menu “Options/Set Default Userlist Domain”, select “REFERENCE NOMENCLATURE DOMAIN” as shown in the next figure, then click on Ok.
Main interface

When you are logged into XTNET, the main Interface is displayed. Along with the Comext logo and any Comext news, you can view some details about your available space at the server side and the notifications.

![Main User Interface](image)

**Figure 7**: Main User Interface

Server Disk Space Usage

The dialog “Server Disk Space Usage” works like a warning of the space used by the user (e.g. to store extractions). If this space exceeds 1GB then the message appears, otherwise it is not shown. Still, it is possible to see the disk usage explicitly in going in the Main menu/Server/Connection/Disk Usage information.

Notifications

These notifications are generated by administrators of the Comext system through the Metadata Editor. The notifications are sent to specific users, thus not all can see them or they cannot see the same. Once the user opens a given notification and clicked on close, it is assumed to be ‘Seen’. Then at the next login, such notification will not be shown anymore. Similarly to the Server Disk Space Usage, the user can view already seeing notifications in going in the Main Menu/Server/Connection/Notifications.

Notifications can be filtered by creation date, last modification date, subject, and status (‘seen’ only or all). They may check one or more check boxes.
Menu

Four menus are available:
- Server
- Query
- Options
- Tools

In addition, a click on the top left Icon, a global menu can be displayed as follows:

**Figure 8:** The main Ribbon Options
This menu will enable user to access directly to subsets of functions.

The server:

Under this section, several actions are available, grouped in subsections:

**Connection:** Connect / Start or Disconnect / Stop (according to the current status)

**Disk usage information**

Shows details about the available user space and user files at the server side

**Notifications**

Shows the notifications depending to the filter chosen by the user.

**Jobs**

- **Jobs**: Show the list of jobs and their status
- **Completed Works**: Display the list of completed extractions in batch or online.

**Services**

- **Services**: Access to the services dialog
  - **Publication Services**: Access to the Publication Services Dialog
  - **Publication Instances**: Access to the Publication Instances Dialog

The Query:

Under this section, the following actions are made available
Figure 9: Query options from main Ribbon

- **New**: Create a new query
- **Open**: Open an existing and saved query

**The Options:**

Under the Options section, the following actions are made available:

- Set query language
- Set default nomenclature domain
- Set default Query Domain
- Set interface language
- Set Favorite dataset
- Set Startup Dialogs

Figure 10: The option from main Ribbon
Set query language : To define the default label’s language (this will be applied on all dimensions by default)
Set Default Nomenclature domain : To define the “default Userlist” domain
Set Default Query Domain : To define the default domain
Set Working language : To define the interface language
Set Favorite dataset : To set a list of favorite datasets (filter)
Set startup dialogs : To open at the startup of the WEB Analytical Client for instance Completed Works, Jobs and/or Open query dialogs.

![Preferences: startup dialogs](image)

**Figure 11**: Tools options from main Ribbon

File download
To open the ‘File Download’ dialog. The downloaded file is under **zip** format.
The download window looks as follows:
The user can also delete his files one by one, using the “Delete” button, and close the window with the “Close” button.

**NB:** For big files, the ZIP file format is automatically changed to use the more recent ZIP file format that is able to handle files and archives bigger than 4 GB. In such case, the Windows 7 native unzip will generate and error that the ZIP archive is not correct. Actually this is wrong, the ZIP file is correct. We strongly advise the user to install for instance the freely available 7zip that handles it.

For small files, the previous ZIP file format is retained.

**Unified Interface**

To launch the ‘Unified Interface’. The module offers the same possibilities than the standard Comext interfaces but it has the Easy Comext look.
Help:

Under the section « Help », the following actions are available:

**Figure 13: Help options from main Ribbon**

- **About**: To display the Comext version
- **Help Topics**: To open the help files
- **Helpdesk**: To open the User helpdesk home page (http://ec.europa.eu/eurostat/help/support)
- **User Manual**: To download the User Manual written in the current working language (English, French or German).

**Status bar**

The status bar at the bottom of the main interface presents various information messages regarding a number of user operations like: logging in, save/open a plan, extracting a plan, retrieving the list of plans, etc.

**Figure 14: Status bar**
**Selecting the Language:**

XTNET application is available in 3 languages (DE, EN, and FR). To choose the language of the interface, select the option in the **Option** menu. The following dialog will be display:

![Set Interface Language](image)

*Figure 15: Interface Language selection*

In addition, the system allows nomenclatures to be displayed in several languages. The language can be defined for all the dimensions of the plan (default language) but can also be edited for one specific dimension. To define a “main” language for the labels, select the option in the Option menu. The following dialog will be display:

![Set Query Language](image)

*Figure 16: Set Query Language*

In the above example, English has been selected. The interface and the Labels will be displayed in English.

---

**Detailed information**

In XTNET, there are several ways to obtain system and user information:
System status

On application start up, two windows will open:

- The first window contains current information activated by the system manager. Please refer to this from time to time to find out the latest system information.
- The second window shows information from the Computer Centre on the availability of the machine that hosts the database.

Descriptions, Comments, Attachments

Several sources of information are available, useful to the XTNET user (user manual, standard aggregates, methodological notes, etc.).

This information can be accessed from the “New Query” window as shown below:

► To Access Descriptions and Comments:

1) From the Query Context menu, select the option New
2) Select the Domain/Dataset
3) Right click to obtain the following screen

![Contextual menu “Dataset”](image)

**Figure 17:** Contextual menu “Dataset”

The Comments window displays the list of available files. The files are described according to the Title, the format, the language available, the description and the size.
Select a file and click on the View button:

For the Attachments, the selected file will be downloaded to your computer, under zip format, and available for use.
Managing your server dedicated space

New XTNET users are allocated a user-id. The user-id is associated with an amount of space that may be used on the server. Once this space is full, it is no longer possible to work on jobs. The Computer Centre regularly frees space by deleting user files.

The golden rule is not to leave anything on the server for more than one month. With the help of your CIS, you can arrange to have access to a local space (local server, for example) to which you can transfer your jobs.

Protecting your installation

If your files are not backed up on a regular basis, it is essential that you protect the files in the directory of your XTNET installation by regularly making a backup. This folder stores all information about your work with XTNET (queries, Userlists etc). Making sure the information is backed up ensures restoration of a proper working environment in case of a hardware failure.

**NB:** If you are using remote storage, the system backup is handled directly by Eurostat. We strongly advise you to use the remote storage.
THE QUERY

Through XTNET, the extraction is defined according to the Query. The query is built from a dataset and is composed of Dimensions. The Query defines what you want to extract/to compute from a specific dataset.

Each dimension of a Query must be filled with at least one code to perform an extraction.

Contextual menu for the Query

The main ribbon of the contextual Query menu will be display as follows:

![Query contextual menu]

Creating a new query:

- To create a new query

1) From the Query Context Menu, Click on the "New" button. The "New Query" window is opened.
2) From the menu New query, give a name to the query in the ‘Name’ text box. The length of the query’s name must be less than 88 characters

When defining a new query, several options are available on the top of the dialog:
First option (Selecting a dataset and create a new query):

The default option will enable user to select a dataset within a domain and to create a new plan.

1) From the menu New query, select a dataset

You must then select the appropriate dataset. In the above example, we will select "EEC SPECIAL TRADE DOMAIN", and finally “EU25 SINCE 1999 CN (SIMULATED)”. This will provide us with monthly data available since the year 1999.

2) Click on OK button

Second option (Selecting a dataset and using the associated default query):

This option will open the default query associated to the dataset.

A default query will be defined by the user when saving the query. The option:”Save As Default” will set the property “default query” to the open query.

Third option (Selecting a dataset and use a prepared query):

This option will display the list of the existing queries using done selected dataset.

After the selection, XTNET will open a dialog containing the list of the dimension of the dataset. According to the option selected previously, the dimensions will be filled (second and third option) or empty (first option).
Defining the content of the Dimensions

Once your query 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.

In the "EU28 SINCE 1999 CN(SIMULATED)" dataset, the dimensions are as follows:

- DECLARANT
- PARTNER
- PRODUCT
- FLOW
- STAT_REGIME
- PERIOD
- INDICATORS

To edit a dimension (selection of codes)

1) From the Query Context menu, click on “Edit Dimension” button. You can also open the Dimension windows by using the Edit 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.

The button Outputs will enable user to define output which will be generated at the query extraction.

The other options from the Plan dialog: Reorganize, Formula order, and Operations will enable to perform advanced configuration.

These options are described later on in this user manual.
Contextual menu for the Query when a dimension is in Edit mode

When a dimension has been selected (via the Edit Button), the contextual Query menu will be modified in order to enable the selection of user objects and other options. The contextual menu will be displayed as follows:

![Figure 23: The “Edit” dimension menu](image)

The user objects use is described in the section “User Objects” of this User Manual

Window of a Dimension

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

![Figure 24: Dimension dialog](image)

The Dimension window is composed of two sub windows:

1) AVAILABLE ELEMENTS:

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

2) SELECTED ELEMENTS:
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 and select it
2) Transfer the selection to “Selected” window, by clicking “Add”

**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.

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

XTNET offers 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 using the mouse.

- **To select a Code**
  
  In “Available elements” window, click on code.
  
  The selected code is then highlighted.
  
  Germany is selected

<table>
<thead>
<tr>
<th>002</th>
<th>Belg.-Luxbg</th>
</tr>
</thead>
<tbody>
<tr>
<td>003</td>
<td>Netherlands</td>
</tr>
<tr>
<td>004</td>
<td>Fr Germany</td>
</tr>
</tbody>
</table>

**Figure 25: Selecting Code(s)**

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

**Quick find Code/Label**

If you know the code or label, then you may use the "Quick Find" Box.
To quickly find Code / Label

1) In Dimension windows, click on **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 scope 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 XTNET, 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: **NB**: It is also possible to filter on an existing filter.

To use Filter option

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

![Filter dialog with filter activated on Code](image)

Figure 26: Filter dialog with filter activated on Code

In "Filter on", select either "Pattern" or "Range" or “Date”.

The search type must now be defined. As indicated, in the selection window description, the basic filter functions for a search are ‘?’ and ‘*’

**Note:** A ‘?’ (Question mark) is used to mask **one position** of the code or label whereas a ‘*’ (asterisk) is used to mask **any position**.
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":

<table>
<thead>
<tr>
<th>Input mask</th>
<th>Result requested</th>
</tr>
</thead>
<tbody>
<tr>
<td>2?</td>
<td>Two-position codes starting with the digit ‘2’.</td>
</tr>
<tr>
<td>2*</td>
<td>All codes starting with the digit ‘2’</td>
</tr>
<tr>
<td><em>2</em></td>
<td>All codes containing the digit ‘2’.</td>
</tr>
</tbody>
</table>
In "RANGE":

<table>
<thead>
<tr>
<th>FROM</th>
<th>TO</th>
<th>Result requested</th>
</tr>
</thead>
<tbody>
<tr>
<td>2*</td>
<td>3*</td>
<td>All codes starting with ‘2’ (means from 200000 to 299999 codes)</td>
</tr>
</tbody>
</table>

Note: the parameter encoded under "TO" is included in the selection.

Label filter:

**Input mask**

- LIVE ANIMALS
- *ANIMALS
- *ANIMALS*  

**Result requested**

- All codes with label ‘LIVE ANIMALS’ and only ‘LIVE ANIMALS’.
- All codes with label ending in ‘ANIMALS’.
- All codes with label containing ‘ANIMALS’.

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 any character or digit.
The user can filter codes according to the date of validity of codes. By default, if no filter is specified in “From” and “To” all codes will be taken.
Combining filters

The WEB Analytical Client allows the user to combine successive filters in a hierarchy through a tree.

NB: The tree is stored in memory, any time the dialog is closed the hierarchy of filters will be lost. The user selects in the tree what list has to be filtered and setup a new filter as described above. 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, take the root node.

Additional 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:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Replaces</th>
</tr>
</thead>
<tbody>
<tr>
<td>?</td>
<td>Any type of character</td>
</tr>
<tr>
<td>@</td>
<td>An alphabetical character (without accents)</td>
</tr>
<tr>
<td>#</td>
<td>A numeric character</td>
</tr>
<tr>
<td>*</td>
<td>Any number of characters</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: For French/German: XTNET WEB Analytical Client recognizes accents.
Figure 28: Under "CODE". ‘##’ produces all length 2 numeric codes

Figure 29: 010 followed by ###### will produce all digits following 010 in a 8-digit code
Figure 30: The string of characters *anes*!*mulets* will produce all codes containing strings "anes" or "mulets"

Filtering by model

a[bcd]e

"a" followed by the characters b or c or d with e at the end of the list, i.e.:  

a b e
a c e
a d e
but not a e e

a[!xy]

"a" followed by all the alphabetical characters 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  

a e i
a f i
a g i
**SELECTION OF CODES**

When the requested code(s) has been founded in the “available” pane, it (them) must be transferred into the “Selected” pane. This can be done with the following options:

1) **ADD**

When ONE OR SEVERAL 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

Multiple selections can be performed either using:
- Ctrl + Left Click to select code by code.
- Shift + Left Click to select a contiguous list of codes.
- Ctrl + A select all codes in the left panel.

It is also possible to add codes without clicking on the Add button: first perform a selection on codes in the left panel, then click on the left button mouse and move the mouse cursor in the right panel.

2) **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

3) **REMOVE**

When ONE OR SEVERAL codes by highlighting them in the "SELECTED" window, this is used to remove them.

- It is also possible to remove elements without clicking on the Remove button: first perform a selection on elements in the right panel, then click on the left button mouse and move the mouse cursor in the left panel.

- **To Remove highlighted the code from the Selected window:**
  From the Dimension Window, click **Remove** button

4) **CLEAR**

This function is used to remove ALL the codes in the "SELECTED" window.
To Clear all the code from the Selected window:

From the Dimension Window, click **Clear** button

---

5) **EDIT**

When a Userlist has been created or selected, it is possible to use this function to change the validity dates and weighting (see Chapter **USERLISTS**)

- **To Edit a Userlist:**

  From the Dimension Window, click **Edit** button

---

6) **LABEL**

When a Userlist has been created or selected, it is possible to use this function to give a label (see Chapter **USERLISTS**)

- **To give a Label to a Userlist:**

  From the Dimension Window, click **Label** button

---

7) **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. On available or on selected items

     c. Sort on code or labels or start date or end date.
3) Click **Sort** button

### RELATIONS

This option enables the display of codes from a Nomenclature into another Nomenclature, according to an existing relation. When a relation exists for a selected code, this option will display the correspondence.

In the example, the Relation option will enable the display of a Successor / Predecessor relation.

The Relation displays the history of the code 01059300, having a Successor (01059100) and a Predecessor (01059400).

From this dialog, it is possible to add:

1. A single code:
   - Select the code and press **Add**
2. The entire list of code (as aggregate)
   
   - Select the code in Bold (source code) and press As Aggregate

9) SEARCH

Enable code research using wildcard.

NB: This feature relies on an external index, not all datasets are indexed in general: so performing a search may not return any result due to such datasets not being indexed.

To use the Search:

From the Dimension Window, click Search button

The following dialog will enable the search option:

Note: the Help button will display the available syntax.
10) **CANCEL**

This will cancel all the modifications done after the opening of the dimension dialog.

![Cancel Button]

11) **CLOSE**

Will close the Dimension dialog

![Close Button]

12) **Hide Elements**

In the right panel, select an entry and perform a right click. The following content menu will be shown:

At the extraction stage, the given element will be extracted and it will be removed at the end of the extraction. A common usage of this feature is to compute for instance a formula that is using other formula: the user does not want dependencies of the formula in the resulting extraction; still such formula dependencies as to be provided and set as hidden.

When an element is hidden, it is shown in red in the right panel as shown after:
Display options:

As mentioned in the beginning of this section, the Contextual menu offers several options when the dimension is in Edit mode. The Display options are available at the top left of the Contextual menu:

Display options include:

- '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.

Displaying validity period and weight fields

To display the validity dates of the codes:

1) From the Edit dimension 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.

Figure 32: Display of codes validity periods
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 start 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**

The weighting of a user object (by default, the weighting of a code is set to 1 and cannot be changed) will be displayed in the “Selected pane” by pressing the icon.

To edit validity period and weight properties:

1) From the ‘Selected’ part of the edit dimension window, highlight one or several codes
2) For the user objects, click the ‘Edit’ button to open the ‘Edit Weight(s)’ and ‘Period(s)’ window
3) Enter the start date, the end date and the weight in the respective boxes
4) Click on the 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 XTNET consultation software is installed. This Userlist uses the EU-FLUX 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: this 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.

**Label type:**

This option, available from the **Label Type** button will enable user to select a specific language for the selected dimension. This option will, for example, enable a user to display labels for the PRODUCT dimension in GERMAN for the query editing and for the extraction, when the default language (available by default for all dimension) has been set to English.

A click on the Label Type button will open the following dialog box:

![Figure 34: The label type dialog](image)

The selection of the language for the query edition will be done on the Left side when the selections of the extraction’s language will be done on the Right pane.

**Dump:**

The Dump option, available from the **Dump** button will enable user to download the nomenclature in use for the dimension being edited.
When selecting this option, the following dialog is open:

![Figure 35: Dump Nomenclature dialog](image)

This dialog will allow the selection of the Language from the drop down list and of the attributes (additional information's link to codes) as well as the extraction format. Two formats are available for download, CSV (with the choice of the separator) or XML. The name of the extracted file and the destination folder will also be selected by the user.

**Saving the query:**

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

1) From the **Save Context** click either on
   a. **Save button**
      If you have set a correct name or modifying an existing query.
   b. **Save as button**
      If you want to save as a new query name or inside another folder.
   c. **Save As Default**
      If you want to save a default query for this given dataset.

**Note:** A query can be used for extraction only when in each dimension it contains at least one element.
Information on query:

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

1) From the Query Context menu, select the “Info” option from the Query sub menu. The "ADDITIONAL query 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 query in the "DESCRIPTION" text box.

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

3) Write the information in the text box

4) Click on the OK button
Printing the Query

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

► To print the content of a query:

1) From the Query Context menu, select the “Print” option from the Query sub menu

1) From the Print query window, select:
   a. Orientation
      i. Landscape
      ii. Portrait
   b. Label Options:
      i. No Label
      ii. Labels
   c. Userlist Options.
      i. Include Userlists

Figure 37: Print query dialog
Importing/Exporting query, Userlists and documentations

XTNET offers the option of saving files containing queries or user lists at defined locations and transferring them back to the system when necessary. This option is very useful for exchanging queries (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 queries:

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

To Export Queries:

1. From the Query Context Menu, select Export

   ![Save dialog](image)

   **Figure 38: Save dialog (for exported query).**

   1. From the Save window, Select the location (path) where the query will be exported
   2. Check or uncheck the Export UserObjects option depending on whether or not you wish to export the user list for the plan
   3. Give a name to the file (before the ‘XML’ extension) in the File Name text box
   4. Click Save
   5. Click on OK when the message “Export completed successfully” displays.
To Import Query(ies):

1) From the Query Context Menu, select Import

![Open dialog (to import query)](image)

Figure 39: Open dialog (to import query)

1. From the Open window, Select the location (path) of the file to be imported
2. Check or uncheck the Import Userlist option
3. Select the file to be imported (XML)
4. Click on OK when the message “Import Completed successfully” displays
Outputs

XTNET offers the ability to define directly on a given query, one or several outputs. This feature avoids to go in Completed Works and to perform manually the operation.

For further details about the expected behavior, we defer the reader to the Outputs section.

Grouping queries functions

XTNET offers the option of creating groups of queries in order to reduce the burden on the users when several extractions need to be done. Grouping queries will enable the users to proceed to one action to launch the multiples extractions and to generate the associated output files. In addition, this feature will allow outputs definitions (at the group or at the query level) as well as overrides possibilities of extraction contents. Also through the use of “Overrides” the user will be able to reuse as a “template” one or several queries to override some selection of codes/aggregates (for instance on the period dimension).

The group sub menu is available from the Query contextual menu:

![Groups](image)

Note: A group is composed of one or several query (ies). Each query can have its own predefined outputs. The group extraction will proceed to the extraction of each queries participating to the group and will produce the associated outputs (defined at the query or at the group level).

The dialog of the group displays the existing group and is composed of the following icons:

- **Create a new group**
- **Add queries to a group**
- **Extract Group/Query (according to the selected item)**
- **Define output to Group/Query (according to the selected item)**
To Create a group:

1) From the **Group Dialog**, **click on**
2) A new Group is created with the default name "new group"
3) Enable the renaming of the group (optional) by a double click on the name and type the new name of the group

The group is created and can be populated by query(ies)

To add query to a group:

1) Select an existing Group from the **Group Dialog**
2) Click on the add Query button
3) Select the query in the list of the existing queries

To define outputs (Groups or query):

1) Select an existing Group/Query from the **Group Dialog**
2) Click on the Outputs button
3) Define the output from the output dialog box:
Note: If the group is composed of queries belonging to different datasets, the definition of the outputs at the group level will not be possible. Please note that all the existing outputs formats from XTNET analytical are available.

To copy and paste a Group:

1) Select an existing Group from the Group Dialog
2) Click on the Copy button
3) Click on the Paste button

A copy of the group will be created with the exact content. The name will be: "Copy of (name of the group)" and can be renamed by the user.

To launch an extraction of a group/query:

1) Select an existing Group/query from the Group Dialog
2) Click on the Extract button
3) The following confirmation dialog will be displayed
As you can see at the top of the dialog, the system does not allow you to proceed to the extraction of the group/query as the max number of batch extractions you are entitled to perform is less than the requested ones.

Either you need to wait for any running job to finish or extract a group with less queries.

If the condition for the max allowed batch extractions is met, the confirmation dialog is as following.

4) Click on OK to launch the extraction

The progress dialog appears displaying the different steps of the group extraction
Note: According to the outputs defined at the query or at the group level, the summary of the extraction will display the list of the queries to be extracted and their associated outputs.

Important: If an output cannot be generated (for example if the output definition does not correspond to the status of the query, which can occur when some dimensions have been dropped for example), users will receive a message in Red explaining why the output will not be generated:

If the user has the production role he can select Destination queue (Small, Medium, Big or Production) in the Confirmation dialog. The Small is selected by default.
In addition to the above mentioned actions, a contextual menu is accessible via a right click on the group or on the query. The contextual menu will be display as follows:

From the menu, the extract option is enhanced by two additional features:

Define Query Override and Define Cartesian Override

These option will enable to override query(ies) belonging to the group (Query override) and the split of the extraction results according to codes of selected dimension (Cartesian Override).

To define a query override:

1) Select the option Define Query Override from the contextual menu in order to open the dimension selection dialog:

2) Edit the dimension(s) for the override
3) Close the override dialog and save the definition
4) Extract the group using the extraction process above mentioned
**Note:** when a query override has been defined, the extraction can be done with or without the override. The selection will be done at the extraction confirmation dialog. The override definition will also be mentioned in the header of the confirmation dialog:

<table>
<thead>
<tr>
<th>Extraction with override</th>
<th>Extraction without override</th>
</tr>
</thead>
</table>

**USG**

Number of extractions: 1
Query override on:
- ESTAT_FLOW(10, 108101, 10, 108102, 10, 106104, 11, 140101, 11, 140102, 11, 140103, 11, 140104, 11, 140105)

**To define a Cartesian override:**

1) Select the option Define Cartesian Override from the contextual menu in order to open the dimension selection dialog:

<table>
<thead>
<tr>
<th>Code</th>
<th>Label</th>
</tr>
</thead>
<tbody>
<tr>
<td>04</td>
<td>C</td>
</tr>
<tr>
<td>05</td>
<td>C</td>
</tr>
<tr>
<td>06</td>
<td>C</td>
</tr>
<tr>
<td>07</td>
<td>C</td>
</tr>
<tr>
<td>08</td>
<td>C</td>
</tr>
<tr>
<td>09</td>
<td>C</td>
</tr>
<tr>
<td>10</td>
<td>C</td>
</tr>
</tbody>
</table>

2) Edit the dimension(s) for the Cartesian override
3) Close the override dialog and save the definition
4) Extract the group, using the extraction process above mentioned

**Note:** when a Cartesian override has been defined, the extraction can be done with or without the override. The selection will be done at the extraction confirmation dialog. The override definition will also be mentioned in the header of the confirmation dialog:
As for normal extractions, when you extract a group with extraction overrides (Cartesian / Query), the condition of the max allowed batch extractions will also be checked.

**Reorganize**

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

This option is accessed via the 'Reorganize' button of the Query:

*Figure 40: Reorganize button*

The Reorganize windows become available with the following options:

*Figure 41: Reorganize dialog*
The ‘Reorganize‘ window describes the actual use of the dimension and the indicators. It is composed of a 3 sub-windows displaying the dimensions list and the status of each Dimension/Indicator.

Changing the name of the Dimensions/Indicators:

The Rename button enables users to change the name of the Dimension/Indicator using the following test box:

![Figure 42: Renaming a dimension](image)

Reorganizing the status of the Dimensions/Indicators

By default, the organization of the Dimension and Indicators is displayed in the Reorganize window.

The Reorganize function enables you to:

- change the name of the dimension or of an indicator;
- change the status of a dimension or of an indicator (one dimension can become an indicator and an indicator can become a dimension);

To remove Indicator from its default status:

To remove an Indicator from its actual location (Indicator list), user must double click on the name of the object.

**Note:** You can use a dimension as Indicator and Indicator as Dimension using the properties above.

To change the Dimension name

1) From the Reorganize window, select a dimension
2) Click on Rename button to open the following sub window
3) Give a **new name** to the dimension  
4) Click **OK** button

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

**To change an Indicator name**

1) From the Reorganize window, select an **Indicator**  
2) Click on **Rename** button to open the following sub window  
3) Give a **new name** to the indicator  
4) Click **OK** button

![Enter Indicator Name](image)

Figure 43: Renaming an Indicator

The name given will be used in your extraction as the indicator name.  
For example, **VALUE_1000ECU** can be changed to **1000 EURO**.

**Formula Order**

This function will enable user to select the order of the dimension for the formulas executions.  
The Formula order function is available at the query level, via the **Formula order** button.
When selecting the option, the Formula execution order is displayed:

Using the two buttons (Move Up and Move Down) user can change the execution order. The confirmation of the new order will be done when clicking on the button. Ok

**Behaviour:**

The formula order is used at the extraction stage: if the query is having formulas, there will be evaluated according to the order specified on dimensions except for the INDICATORS dimension that will be always evaluated at the latest stage. The formula order is important according to what you want to compute if there are “intersections” between formulas across different dimensions.

**Example:** Assume that we have a formula on DECLARANT called SUM_DECL=|01| + |02| and we have two formulas on REPORTER called SUM_REPO = |003| + |004| and SUM_REPO_DECL=IF DECLARANT['SUM_DECL'] THEN |005|ENDIF. If the order of
evaluation is DECLARANT, REPORTER then the formula SUM_DECL will be first evaluated, then SUM_REPO. We have the following data:

\textit{DECLARANT, REPORTER, VALUE\_100\textsc{ecu}, QUANTITY\_TONS}

\begin{align*}
01, & 003, 100, 10 \\
01, & 004, 200, 20 \\
02, & 003, 300, 30 \\
02, & 004, 400, 30 \\
01, & 005, 500, 50 \\
\end{align*}

First, we evaluate on DECLARANT, we will obtain this new set of data:

\begin{align*}
\text{SUM}\_\text{DECL}, & 003, (100+300)=400, (10+30)=40 \\
\text{SUM}\_\text{DECL}, & 004, (200+400)=600, (20+40)=60 \\
\text{SUM}\_\text{DECL}, & 005, (500+0)=500, (50+0)=50 \textbf{NB:} \text{ The default behavior in case of missing value is to replace by 0 if at least one operand exists, if both operands do not exist it will be a missing value.}
\end{align*}

Then, we evaluate on PARTNER, we will obtain from the original set of data

\begin{align*}
01, & \text{SUM}\_\text{REPO}, (100+200)=300, (10+20)=30 \\
02, & \text{SUM}\_\text{REPO}, (300+400)=700, (30+40)=70 \\
\end{align*}

Then, we will evaluate also on the previously computed DECLARANT formula(s):

\begin{align*}
\text{SUM}\_\text{DECL}, & \text{SUM}\_\text{REPO}, (400+600)=1000, (40+60)=100 \\
\text{SUM}\_\text{DECL}, & \text{SUM}\_\text{REPO}\_\text{DECL}, 500, 50
\end{align*}

Now, let us assume that we change the order of evaluation to REPORTER, DECLARANT.

SUM\_REPO will be evaluated similarly, but SUM\_REPO\_DECL will not be evaluated at all because its evaluation depends on SUM\_DECL that will be evaluated later.
Operations

The user can define operations on a query that will be performed after the extraction. An operation has a parent operation or a source extraction, and it can have one or several outputs.

Operations can be used to avoid many manual operations.

The user must click on the extraction object. The available operation buttons are activated.

The order of the operations will be the order defined in the hierarchy.

The user has also the possibility to define outputs on an operation. For example, after an “Interpolate” operation, the user can define outputs. For this, he must go to the desired level (on the object under which he wants outputs to be defined), then he must use the mouse right click to make appear the “Outputs” entry, as shown below. The creation of these outputs does not prevent the user to create other outputs at the extraction level of the “Completed Works” window.
For example, after the “Interpolate” action, the user selects to create a graph. The creation of a Graph is shown in another chapter below. After the successful definition of the graph, the “Operations” will look as follows:

The user can define more outputs.

Please note that adding outputs at this stage is possible after the definition of an operation. So, operation may contain (when allowed by the application) other operations and/or outputs.

Once all operations and associated outputs are defined and the user has clicked on the Close button, then inside the Query dialog will updated and “Operations” will be in blue as shown in the next figure:

Then, the user can save the Query as previously described.

When, the extraction is launched on such a query, the user will be notified on the different operations performed using provided names by the user, for instance:
Limitation: Please note that some operations cannot be done under any order. For example, after a “Compress” operation, no child operations can be defined as shown below. The main reason is that some operations can change the number of dimensions, or the list of codes in a given dimension according to data found in the source extraction: in this context this means that the WEB Analytical Client cannot check the validity of subsequent operations or outputs on such kind of operation.

Output map can’t be defined under the Operations menu, the rest of outputs can be defined.
Split

The Split feature enables users to split huge extractions in sub-extractions that can be extracted concurrently and merged if requested. It is a complement approach to the previously introduced Extraction Groups with Cartesian overrides on the Query selection.

First Open a Query or Create a new Query:

Then, **select the dimension** on which you want to define a Split; Then click on the **Split** button in the Query window.

In order to define a Split on a given dimension, it must have at least two elements otherwise the client will print this error message:
Limitations:

It is not possible to split dimensions that contains:

- Formulas
- Aggregates computed as formulas:
  - ‘Missing’ semantics aggregates
  - ‘With Status’ aggregates.

It is not possible to split the indicator dimension.

After, the split definition dialog will be shown:

- The top of the window is to simplify the generation of groups:
  - Group by Code can be used to associate one group to each element of the dimension. Each group will have the name of the associated element. This feature can be used typically on datasets with PERIOD and FLOW.
  - Group by Size can be used to split a dimension according to a given cardinality. The client generates a list of groups that contains at most the given number of elements in each group. This feature can be used typically on a query that contains in a given dimension a massive amount of elements, for instance the PRODUCT dimension is a good candidate if there are thousands of elements.
  - Clear is used to reset the definition of the split on the given dimension.
  - If you click on any of those buttons the client will ask for a confirmation.

- Groups panel contains the list of defined groups. It is initially empty. If you select one of them it will update the bottom window. When a right mouse click is performed in the window, you will access to the contextual menu as shown after:
Using this context menu, you can create a new group, renaming existing group and delete a group.

- The bottom of the window contains two panels:
  - The left one contains the list of elements that are not affected to any group.
  - The right one contains the definition of the selected group.
To move an element or a list of elements in a given group:

1. First select the group
2. Select the list of elements to be inserted in the left list
3. Then click on the button “>>”

To remove an element or a list of elements in a given group:

1. First select the group
2. Select the list of elements to be removed in the right list
3. Then click on the button “<<”

Once the split edition is finished, the user can click on Ok, after the client will update the Query window with a Split icon in front of the splitted dimension:

Note: The client will check if there is at least an element that is not affected to the group and it will show the following popup in this case:

Group by codes:

We detail an example of use to create a split based on group by codes.

1. First, create a query or open a new query.
2. Select the given dimension in the Query window
a. Ensure, there is at least 2 elements in the dimension you want to split

3. Click on the **Split** button in the Query window

4. Then, in the Split window click on the button “**Group by codes**”

5. A confirmation dialog will be shown, click on **Yes**.

6. The client will update the Split window this a list of groups that are named according to each element you have in the selected dimension.

7. After the user is still able to customize fully the split.

**Group by size:**

We detail an example of use to create a split based on group by size.

1. First, create a query or open a new query.

2. Select the given dimension in the Query window

   a. Ensure, there is at least 2 elements in the dimension you want to split

3. Click on the **Split** button in the Query window

4. Then, in the Split window click on the button “**Group by size**”

5. A window will pop-up that will ask the size you want for the number of elements
6. Enter a number greater or equal to 1, if you put a number bigger than the number of elements or a bad number the client will reask you to enter a number. (The case where the size is equal to 1 is the “Group by code” behaviour).

7. Once click on OK (and if the size is valid), the client will generate a list of groups, as shown in the next example:

8. After the user is still able to customize fully the split.
Mirrored Matrix

The external Trade statistics are defined according to several dimensions (Reporters, Partners, Flow, Products, etc…)

The mirror statistics will enable to compare trade between Reporters and Partners according to the direction of trade (imports or Exports).

**Ex:** The import of a country A to a Country B will be compared with the Export of Country B to Country A.

In order to facilitate such comparison, Comext is having a new function called Mirrored Matrix.

**NB:** In order to perform a mirror matrix at least 2 dimensions must have the same nomenclature inside the given dataset.

**Defining a Mirrored Matrix:**

The Mirrored Matrix function can be launched from the Extract menu of the Query Ribbon:

![Image of the Mirrored Matrix function in Comext]

The Mirrored Matrix definition dialog is displayed:

![Image of the Mirrored Matrix definition dialog]

Figure 45: Mirrored Matrix - step 1
The definition will be done in two steps:

Step 1: Mirrored Dimensions selection

Step 2: Symmetric Dimensions

(iii) Step 1: Selection of the Mirrored Dimensions:

In the example, we will compare the Reporters and the Partners as (see above) the Imports of a Reporter from a Partner need to be compared with the Exports of the Partner to the Reporter.

In this first step, user will have the list of all the query dimensions on the Left side of the dialog.

The Right side (Mirrored Dimension) will have to be filled with two dimensions.

**Note:** The two dimensions **MUST** have the same nomenclature in order to be compared.

(iv) Step 2: Selection of the Symmetric Dimensions:

The symmetric dimension will be the dimension on which will be based the reverse information (e.g. Flow). The step 2 dialog is as follows:

User will have to select the appropriate dimension (Flow) and define the Symmetric elements:
Figure 47: Mirrored Matrix - selection of Codes

Selection of the First Element (ex: 1 is Import)

Selection of the Second Element (ex: 2 is Export)

Import data will be compared to Export data for each association Reporter / Partner.

Click on the Finish button to launch the extraction.
Reading the information in the mirrored matrix:

The spreadsheet will display the extraction in which, each original indicators will have its “double” (mirrored).

For ex: Value_1000EUROS will display the original data and the indicator Value_1000EUROS_MIRROR will display the mirror figure

Note: if the extraction query contains several indicators each indicator will have its associated mirror.
THE EXTRACTION

Methods of extracting data

When all the dimensions of the query have been filled, XTNET offers two options for extracting the data:

- Batch extraction
- Online extraction.

Batch Extraction

**Batch Extraction:**

‘Batch extraction’ will extract the data without blocking use of the XTNET application. The information required for the extraction is sent to, and saved on, the server. This is the default extraction mode.

A name should be given to the extraction. The default name is the query name followed by the number of the extraction (automatically allocated number).

To launch a Batch extraction:

From the Extract option (available when selecting the Query context Menu), click Extract

![Extract](image)

Figure 49: Batch Extraction confirmation dialog
1) From the Confirmation window, check the Batch option
2) From the Confirmation window, click OK
3) If the query contains outputs the user can select option Send extraction by email
   The user can unselect the outputs. By default all the outputs are selected.

**Note:** Some outputs may not be compatible with the query definition. In such case such outputs cannot be selected.

When the user selects “Send extraction by email” the following window will appear prompting for a valid email address.

**Notes:** The current maximum size limit for attachments is set to 5mb.

To add more than one email recipients please add the email addresses separated with a comma (,). Attention, the link provided in the email is public, and can be shared with other users.

**Production Users:**

If the user has the production role he can select Destination queue (Small, Medium, Big or Production) in the Confirmation dialog. The Small is selected by default.
The user makes his selection and confirms the extraction by Ok button. Job is sent to the selected queue. When the Batch extraction box is unchecked the Destination queue selection is disabled.

Following a Batch extraction, XTNET will open the Jobs windows.

![Jobs Window](image)

**Batch Extractions number**

It is possible you try to submit in batch more than one and/or big plans. Once you press **OK**, the System will check if you are allowed to proceed with the given batch extraction. If not, you will receive the following message:

![Error Message](image)

In this case, you need to wait for your running jobs (see XTNET Jobs) to finish and try again to submit in batch your plan, or if you need more jobs you can ask to the XTNET Support.

**XTNET Jobs Window**

Once "OK" is clicked, the "XTNET Jobs" window opens.

- **To open the XTNET Jobs window:**
  1. From the **Server Context** menu, click **Jobs**
Figure 52: The list of batch jobs executed by the central site.

This window is a log of all the jobs sent to the server and is purely for information.

The XTNET Jobs window displays the Job name (extraction name), the source dataset, the status of the extraction, and the date of extraction:

The status may be:
- queued: The job is waiting for other jobs to finish
- running: The job is currently running
- completed: The job is finished
- aborted: The job has been aborted, possibly because:
  - no data resulted from the extraction, or errors in the definition of the query.
- error: The job could not be executed (probably due to a technical fault)

► To delete an entry:
1) From the XTNET Jobs window, select the entry (one or several lines)
2) Press Delete

► To clean all the entries:
1) From the XTNET Jobs window, press Clean

► To close the XTNET Jobs menu:
The XTNET Jobs window can also be closed by clicking on the cross icon, located on the top right of the window.

To modify the Jobs window:

User can change the columns order (name column excepted) by dragging the Column headers left or right. Right-clicking in the jobs list will display a Columns dialog to show/hide column. It is not allowed to hide the name column.

The user unselects the columns which he wants to hide. Press button to apply the changes or to cancel it. Once the jobs dialog is closed, the settings will be saved in the user preferences and the same jobs dialog will be opened for the next time.
Online extraction

An online extraction is an extraction performed directly on the server interactively.

NB: The WEB Analytical Client is subject to a timeout of few minutes, it means that for instance any medium to big extractions, you should not use an Online extraction but Batch Extraction which is selected by default.

To launch an Online extraction:

From the Query Context Menu, click Extract

![Extract button](image)

Figure 53: The On-Line extraction confirmation window

1) From the Confirm Extraction window, deselect the Batch extraction, then click OK button

This function blocks use of the XTNET 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:
When the spreadsheet appears, three tabs are available:

1) The **Spreadsheet** tab 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.
2) The Chart tab displays the same information as the one displayed in the data tab under graphic representation (See description below):

3) The Treemap tab displays the same information as the one displayed in the data tab under a tree map representation (See description below):

Several options are available from the spreadsheet. All these options are describes in the section Spreadsheet of this user manual.
4) The **Map** tab displays geographical maps according to the user’s selection. The user’s selection must have only one row of data and one of the dimensions selected must be of type country (e.g., Reporter or Partner dimension). If the user has selected more than one row in the spreadsheet, only the first row will be taken into account. If the selection is not suitable for the map, the system informs the user about it and map is not shown.

**Note:** The user can change map type (Mercator, Equi Rectangular, Azimuthal Equidistant) and request to show legend, Arcs, Grid, zero and change the colors.
Formatting the Extraction

The result of an extraction can be viewed as a virtual cube stored on the XTNET server. To obtain the data in a physical file, it may be necessary to format the extraction. XTNET 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 Works” 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 Works” menu.

The Completed Works 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 "XTNET Jobs" window. When the job is completed, the result can be obtained via the "Completed Work" Menu command. The following window appears:

![Completed extractions, prints and Dump window](image)

Figure 55: Completed extractions, prints and Dump window
In this window, all the extractions carried out on 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 jobs 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. ![View Button](image)

The extraction result will open in a spreadsheet.

**Note:** When extracting data via the “On Line” option, XTNET displays the results 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 ![Delete Button](image)

The extraction will be deleted after clicking on Yes in the confirmation dialog.

**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:**

Pressing the respective icons accesses these functions.
For jobs preceded by the ![formatting_icon] symbol, the ![formatting_icon] indicates that data in this extraction has been formatted in different ways.

Clicking on the ![formatting_icon], reveals the list of formatting. Different types of icons can appear:

- ![icon] Indicates Out Data
- ![icon] Indicates a Dump
- ![icon] Indicates a Print
- ![icon] Indicates a Graph
- ![icon] Indicates a Generate Table
- ![icon] Indicates an XTTTable
- ![icon] Indicates a Subcube/Extraction
- ![icon] Indicates a Map
- ![icon] Indicates a Split Report/Eurobase Loading Report

**Note:** For the extraction formats where a title is shown (e.g. in the Generate Table), the length of the title field is of maximum 255 characters.

**Delete**

This option deletes extraction and/or formatted files.

**Merge**

**To Merge extractions:**

1) Select a minimum of two extractions having the same dimensions from the ‘Completed Work’ window

2) Click “Merge” button

3) Confirm the merge (in Batch or interactive)

This option will generate a new extraction, containing the merged extractions. A new dimension will be added to the new extraction. This new dimension will be called “Cubes” and will contain the number of the original extraction.
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.
Query
This function will enable you to use a previous extraction as a dataset, and to build a new query 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.

► To create a query from an extraction:

1) Select an output file from the Completed Work window
2) Click “Query” button
3) Build your query as it is describe in the Query section

Subcube
From the extraction context menu, click on Subcube button.

The Subcube function, which is also available at the Query definition step, will allow you to apply calculations on one or several dimension of the specific extraction. (See: The spreadsheet).

Close
This option closes the ‘Completed Extractions’ window.
The spreadsheet is the XTNET 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**.

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 Reorder and change the data in the spreadsheet:

This set of functions, enabling user to reorder and to change the content and data display is composed of four "sub functions":

- Changing the dimension of the spreadsheet.
- Sorting the elements according to the values in the spreadsheet.
- Filtering the content of a dimension.
- Inserting an aggregate in a dimension.

To use the reordering function, the spreadsheet must be open:

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 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.

► 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:

\[ \rightarrow \rightarrow \]
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.
4) Sorting the elements according to the values in the spreadsheet

**Sorting the elements according to the values in the spreadsheet:**

The second sub function of the reordering enables a sorting of the element according to the values.

This function is available via the contextual menu obtained with a right click on the mouse.

<table>
<thead>
<tr>
<th>Filter Elements</th>
<th>Show All</th>
</tr>
</thead>
<tbody>
<tr>
<td>Insert Aggregate</td>
<td>Insert Implicit Aggregate</td>
</tr>
<tr>
<td>Insert Formula</td>
<td></td>
</tr>
<tr>
<td>Sort data ascending</td>
<td>Sort data descending</td>
</tr>
<tr>
<td>See Code Attachments</td>
<td></td>
</tr>
</tbody>
</table>

The “Sort options” will enable to reorder the elements (in lines or in columns). To distinguish the selected Element, an arrow will be set on the spreadsheet:

1. Ascending sort: ↓
2. Descending sort:
To Apply filter on a dimension:

The second sub function enables a sorting of the element according to the values. This function is available via the contextual menu obtained with a right click on the mouse. From the spreadsheet, right click on one dimension in the table.

The following submenu becomes available:

1. Select “Filter Elements” to apply a filter (or show all after a filter)
2. From the selection window, select the requested codes
3. Click on the 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 submenu becomes available:
3) Select “Insert Aggregate” or “Insert Implicit Aggregate”.

**Note:** For Aggregate or Implicit Aggregate, see chapter USERLISTS

---

**Extraction Context menu**

When the Spreadsheet is displayed, the extraction context menu is available. This menu will give access to the possible actions:

![Extraction Context Menu](image)

**Figure 58: The Extraction context menu**

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

- To display code and/or Labels into the spreadsheet:
  1. From the extraction context menu, click on X-Header or Y-Header.
  2. From the Sub menu, select Code, Text or Text and Code.

![X-Headers Menu](image)

**To remove the empty lines/columns of the spreadsheet:**

When the XTNET 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.

The compress option will eliminate all the “unused” codes from the extraction.

- **To compress the spreadsheet:**
  1. From the extraction context menu, select “Compress”.

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

- **To eliminate zeroes in the spreadsheet:**

The spreadsheet will be compacted to remove all the zeroes (empty lines/columns).
1) From the extraction context menu, select “Filter Empty Values”

a) To save and return to the original spreadsheet (after a compress):

1) From the extraction context menu, click on Save and Return

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

To set the indicators format:

1) From the extraction context menu, click on Set indicator format

2) From the display dialog, define the format (number of decimals to be displayed:

![Figure 59: The indicators format definition](image)

To generate an EXCEL file from the spreadsheet:

1) From the extraction context menu, click on Excel Icon

To generate an Dump (text file) from the spreadsheet:

1) From the extraction context menu, click on Excel Icon
Under "DUMP", 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.

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

To set default value:
1) From the extraction context menu, click on “Default Values Icon” at 0.0.

To launch Subcube functions:
1) From the extraction context menu, click on "Subcube functions".

The Subcube function, which is also available at the Query definition step, will allow you to apply calculations on one or several dimension of the extraction. When using the Subcube function, a dialog will be display in order to specify the type of function you will use on your extraction.
To launch Edit mode:

1) From the extraction context menu, click on [Set Edit Mode] to launch the data Editing functions:

**Data Editing**

**Definition:**
Under Comext, it is now possible to edit the data. However in order to keep the original data, the data editing process will be available via a dedicated procedure.

This procedure will create a ‘virtual dimension’ in the existing extraction. The virtual dimension will contain a minimum of two codes, one keeping the original data and the other ones for the new “edited” data.

The standard edit procedure will be done in 4 steps:

1) Step1 : Set Edit Mode
2) Step2 :Saving the “Edited extraction”
3) Step3 :Reshaping the plan in order to use the new entered data instead of the original ones
4) Step4 :Production of a file for the Comext Data Loading system

![Virtual Dimension definition](image)

**Figure61: Data Editing - Virtual dimension definition**

When opening the “Set Edit Mode” dialog, the following information’s must be provided:
- Virtual dimension Name

- System data Element Name
  - This will be the “code” of the virtual dimension containing the original data

- User data Elements
  - User can define several additional codes of the virtual dimension for editing purposes

Then click on the **Ok** button to create the dimension in the extraction.

The spreadsheet is displayed with its new structure:

![Spreadsheet in Edit mode](image)

**Figure 62: The spreadsheet in Edit mode - 1**

As shown in the above picture, a dimension having the properties defined previously by the user is part of the extraction (a new dimension called “NEW_DIMENSION” and two codes called “OrigValue” and “NewValue”)

**Note:** The System data Element **cannot** be edited; it will contain the original value

Selecting the “NewVal1”, which is a User data Element, the data grid (spreadsheet), is set to “Edit” mode.
Figure 63: The spreadsheet in Edit mode - 2

By default, the User data element will contain the original value. When selecting a cell, user will have the possibility to replace the value. The new value is taken in account after pressing <ENTER>.

Once the data have been modified by the user, the extraction will have to be saved in order to keep the “new data”.

To Display the data in Tabular mode:
The XTNET spreadsheet allows a tabular view. This option shows the data in a ‘flat’ format.

1) From the Extraction context menu, select Tabular View
2) The “Tabular View Parameters” window is displayed.
3) Select the Indicators (on the top Right of the window)
The Tabular View parameters allow you to select the information to display in Tabular mode. By default, all the extractions are displayed, and the ‘Asc’ parameter selected. The ‘Asc’ parameter ranks the data in ascending order (according to the selected indicators).

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

**To apply a selection before the Tabular View:**

From the Tabular View Parameters windows:
1) Click on the drop down list to select the dimension
2) Select or unselect codes
3) Click on “**Add**” Button
4) Click on the **OK** button to display the Tabular View

**To modify the column order in a tabular view:**

1) Select the column you want to move.
2) Keep the left mouse button held on the column’s header and move the column at the desired place.
3) Release the mouse.
To save or print the tabular view:

From the Tabular View:
1) Click on ‘Print’ to print the file or ‘Save’ to save it to a dedicated location. The saved files has a “.DAT” extension and is readable through Excel for example.

Post computations

Post computations are available in Advanced functions when spreadsheet is opened.

The user can define the following functions: average, count, sum, percentage, maximum, minimum, median, standard deviation and variance.

The user can select if the function should operate on Nulls and Zeros. The post computations can be applied to all dimensions. When the user presses Ok button the post computations are performed and added to the spreadsheet.

Note: The new post computation will not be saved with the plan or extraction; it will be just a temporary post computation for display purposes. The user can repeat the operation for a new post computation. In this case a new column will be added in the same manner.
USER OBJECTS

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.

Userlists are stored by default on the server.

A Userlist can be either attached to a nomenclature that can be further reused across different datasets sharing the same nomenclature (typically PRODUCT dimension) or it can be attached explicitly to a specific dataset.

XTNET allows creation of two kinds of Userlists:

1) Explicit Userlist: An explicit Userlist is a defined list. It is composed of a list of codes defined during the creation. An Explicit Userlist is hierarchic it can contains codes, and also UserLists.

2) 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 XTNET as a Userlist or as an Aggregate.
How to create a Userlist:

**Explicit Userlist**

- To Create an Explicit Userlist:

1) From the Query windows, click on Select button

![Product dimension dialog](figure65.png)

Figure 65: Product dimension dialog

2) Transfer the requested codes to the "Selected" part of the selection window, and then highlight them

3) Save the selection and insert it into the query, in the form of a list. This action can be performed via the Userlist menu, 'Replace' option.

4) Give a name to the Userlist and specify the type:
   a. **Regular**
      i. This is the default kind of a userlist. The word “Regular” refers to the semantic used when computing the aggregate of such userlist: if it exists at least a valid element in the userlist, the aggregate will be computed and missing elements will be replaced by the value 0.0.
   b. **Missing**
      i. The word “Missing” refers to the semantic used when computing the aggregate: if at least an element of the userlist does not exist, the aggregate will not be computed. The user can also specify for each missing element an associated default value.
c. With status

i. In this context, “With status” refers to a computation of the aggregate that will generate additional indicators that will describe the status of the computation for each record. When using such userlist as an aggregate, the user will be able to choose either the “Regular” or the “Missing” semantics.

![Figure 66: Save Userlist / Aggregate](image)

2) Select the storage location (Local or Remote)
3) Click **OK** to save the Userlist

Please note that from the calculation point of view the status has no impact on the pure userlists that are taken as a least of codes. **NB.** If a code is repeated several times due to the use of the hierarchy, it will be show only once in the final extraction.
Using the Explicit userlists

A Userlist can be used in two ways:

In the form of a list:

A Userlist inserted into a query 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 remotely, and insert them into their queries.

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 existing Userlist as a list

1) From the menu, select Userlist and then Insert

2) From the Insert Object window, select that attachment either NOMENCLATURE or DATASET.

3) From the Insert Object window, select either User or System, by default User is selected. System is Eurostat provided common userlist/aggregates.

4) From the Insert Object window, select the type of the userlist either Regular, Missing or w/Status.

5) From the Insert Object window, select the requested code(s) and click Ok button

Figure 67: Insert Userlist
Insert a user defined userlist. By default you see all the available userlists you have already defined.

Insert a system defined userlist: Selecting ‘System’ at the “3rd” drop down list, the available system userlists will be retrieved for you to select.

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

▶ To Insert an existing Userlist as Aggregate

1) From the Aggregate menu, press **Aggregate** and then select **Insert**

2) From the **Insert Object** window, select that attachment either **NOMENCLATURE** or **DATASET**.

3) From the **Insert Object** window, select either **User** or **System**, by default User is selected. System is Eurostat provided common userlist/aggregates.

4) From the **Insert Object** window, select the type of the userlist either **Regular**, **Missing** or w/Status.

5) From the **Insert Object** window, select the requested code(s) and click **Ok** button

Figure 68: Insert Aggregate

As inserting a userlist as a list, you can choose to insert a **User** defined or a **System** defined userlist as aggregate.
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 give or change a label of a Userlist

From the Aggregate menu, select Insert a USL (or Aggregate)

1) From the Insert Userlist window i.e., select the requested code(s) and click Ok button
2) Once the selected list or aggregate is the “Selected elements” area, click on it to select it and then click on LABEL.
3) A new sub-window will appear. Choose the label's language and insert the text, as shown:

Figure 69: Defining a label of a Userlist
Press the OK button

**Result:** The list will have a new label. Keep in mind that the label is not the identifier of the object. It means that you do not change the “1000”.
Using the Implicit functions:

The implicit function option will enable the use of objects are built by expressions (in opposition to explicit objects which are based on a defined list of codes). The implicit objects can be built from formulas (code or Label based) or using a range of codes.

The Implicit function dialog is available from the main selection ribbon
This option opens the dialog “Insert Implicit function”:

![Image of Insert Implicit dialog]

Figure 70: The Insert Implicit dialog

This dialog enables users to define the implicit object that will be defined and inserted. The user can define the type of the implicit object:

- Userlist
- Aggregate
- User list + Aggregate
- Function
- Label
To Create an Implicit Object using the Pattern:

1) From the Query windows, click on Select button

2) From the Query context menu, select Insert Implicit Function:

3) Select 'Implicit Pattern' and Pattern Type (Userlist, Aggregate or both):

4) From the Insert Implicit Userlist window, give a name to the Userlist in the Selected text box and click on the Insert New button
Figure 71: Insert / Insert New Implicit object dialog using the Pattern

1) From the Implicit Userlist window, write the expression of the Userlist

![Implicit Function window](image)

Figure 72: Definition of the implicit object

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

► To Create an Implicit Object, using the Selection Function:

1) From the Insert Implicit function, select the

![Selection Function](image)

1) From the **Insert Selection Function** dialog, Give a **name** to the Userlist in the **Selected** text box and click on the **Insert New** button
2) Figure 73: Selection of an implicit object using Implicit function. From the Edit Range dialog, define the Range Start and End.

Figure 74: Implicit function definition

The dialog will enable the definition of the Range Start and of the Range end. The Range definition can be defined on the Relative order of the codes in the nomenclature (from the beginning or the end) or defined on the code itself. A radio button enables the selection.
The button will proceed to a preview of the content of the implicit object.

To Create an Implicit Object, using the Selection Label:

1) From the Insert Implicit function, select the Selection Label.

2) From the Insert Selection Label dialog, give a name to the Userlist in the Selected text box and click on the Insert New button.

3) From the Implicit Label dialog, define the Selection.
**Note:** The button will display a dialog with the syntax:

**Figure 76: Implicit Label definition**
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 menu, select 'Insert Implicit Userlist' or click the icon.

![Figure 77: Insert Implicit Userlist](image)

1) 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 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 on the 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 on the 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.
Replace a Product group by an aggregate

Some datasets provide product codes by groups as shown below:

When a group of codes is inserted as-is in the ‘Selected’ area, then at the extraction all the elements contained in this group will appear.

If the user wants, he can treat this group as an aggregate (the results will be grouped at the aggregate’s level). To do this, the user must...
• select the group in the ‘Selected’ area
• go to the ‘Aggregate’ button (as shown below) and choose ‘Replace’

The selected element’s icon will change. It will become as shown below:

The result of this change will be the following:

Figure 81: Group replaced as Aggregate.

The new element is an aggregate.
Advanced aggregates

The most common and generic type of aggregate is the 'Regular' one.

The semantics for the computation during the extraction is the following: if AT LEAST one value of an element found in the aggregate exists, then the aggregate will be computed and all missing elements will be replaced by 0.0.

Two other types have been defined: 'Missing' and 'With status'

- An aggregate of ‘Missing’ type is an aggregate that will NOT be computed IF at least a value is missing on an element defining the aggregate.
- An aggregate ‘With status’, is an aggregate where the user can specify the semantics, either to behave as Regular, either as Missing. Additional indicators will be created to denote what is the status of the computation (missing, partial or complete for instance).

If you want to define aggregates on one dimension, then you can define one or more.

For any type of Aggregate or UserList/Aggregate object, the creations and insertion actions, shown below are identical. For our example, we will use the REGULAR Aggregate creation and insert.

**How to create an Aggregate**

When you have chosen the dimension elements you want to calculate the aggregate on, click on ‘Aggregate’ -> Replace (of ‘UserList/Aggregate’ -> Replace)

![Figure 82: Create a Regular Aggregate](image)

You will see then the place where all aggregates exist (if any)

---

1 For the aggregates ‘With Status’, you need also to specify this status.
Note: If you want to create a specific type of Aggregate, then you must indicate this specific type in which the object will be created, and thus select it in the drop-down list:

Click on ‘Insert new’ and give an eloquent name to the new aggregate. You may also attach a Label on it to give more specific information.

► Add Aggregate with successors, predecessors or with MSA

The user can add the new aggregate directly from the Available elements panel when the element has successors or predecessor. Select the element, press the right mouse button and select Add as aggregate With Successors, With Predecessors, With Succ/Pred or With MSA. After that the user selects the storage in the User objects panel and presses Insert new button to insert the new aggregate.
The name will be suggested by the system. The user can change the name and press OK and Aggregate will be added to Selected elements.

Note: The option Add as aggregate is disabled for the elements without successors or predecessors.

► Aggregate ‘With Status’: how to set the status.

By default, an aggregate with status is created to behave as a Regular one. The user can choose to leave it or to make it behaving like a ‘Missing’ one. In order to do this, the user must:

1. Select the aggregate ‘With Status’ he wants.
2. Click on ‘Edit’

A new sub-window will appear as shown below:
The user can change the type (from ‘Regular’ to ‘Missing’) and eventually define validity period and weight for this aggregate. The ‘Indicator prefix’ and ‘Complete value’, ‘Missing value’ and ‘Partial value’ default entries can be changed. Just keep in mind that if you define other aggregates of the same type and for the same dimension, these should be IDENTICAL as for this type of aggregate (‘with status’) NEW INDICATORS are created (shown in the spreadsheet) and calculated depending on the number of the previously selected indicators.

**Note:** Only the aggregates ‘With Status’ have this possibility. The ‘Regular’ and ‘Missing’ aggregates cannot be modified.

---

**How to insert an existing aggregate**

In order to retrieve an existing Aggregate, proceed as show below:

1. **Go to Aggregate -> Insert**
2. Chose the type of Aggregate you want to retrieve and thus, choose the appropriate type.

3. In the chosen repository, select the appropriate aggregate and click on Insert.

**Differences between aggregate types**

We will show the differences between aggregates through an example.

We have defined a query with the following data from the **EEC SPECIAL TRADE SINCE 1988** dataset:

- Dimensions:
• DECLARANT: 002 (Belg-Luxbg), 004 Fr Germany, 010 Portugal, 032 Finland
• PARTNER: 0001 France, 0002 (Belg-Luxbg), 0003 Netherlands, 0004 Germany, 005 Italy
• PRODUCT: 8703
• FLOW: 1, 2
• STAT REGIME: 1 (Normal)
• PERIOD: October 2003 and December 2003

Indicators:
• VALUE_1000ECU and QUANTITY_100KG.

On the PARTNER dimension, we have defined two aggregates:
• ‘partner_status’: aggregate with status set to REGULAR and containing the same Partner values
• ‘new_agg_status’: aggregate with status set to MISSING and containing other values (e.g. 0010 Portugal, 0011 Spain, 0017 Belgium, 0018 Luxembourg, 0021 Ceuta, 0022 Ceuta & Melilla.)

At the extraction we have the following:

The ‘partner_status’ Regular aggregate could be calculated wherever values existed. The ‘new_agg_status’ aggregate of ‘With status’ type set to ‘Missing’ could not be calculated as (some or all) values were missing. This can be shown under the new indicators (that were NOT selected in the query) named PARTNERVALUE_100ECU and PARTNERQUANTITY_100ECU.
When the calculation was possible (because some values existed), the value of the new ‘PARTNERVALUE_100ECU’ indicator was set to ‘partial’.

For the aggregates that are of type ‘Missing’ or ‘With status’ set to ‘Missing’, when the calculation was not complete because of missing values, then the value indicated in the sheet is ‘missing’.

When the calculation is impossible for ALL INDICATORS (whatever the definition of the aggregate might be) as for the DECLARANT 002 (declarant ‘002 has no data), then the cell is empty/missing (nothing is indicated). It means that in case all components of the aggregate have NO data for all indicators = the status is not computed.

BUT if values exist for one indicator and not for the other(s), then the value will be set to ‘missing’

In general, for the following types of aggregates, we can have the following values for the new indicators when computation is possible:

<table>
<thead>
<tr>
<th>Computing semantics</th>
<th>All components of the aggregate are having data</th>
<th>At least one component of the aggregate has no data</th>
<th>Component has values for indicator et not for the other(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>REGULAR – W/S Regular</td>
<td>Complete</td>
<td>Partial</td>
<td>Missing</td>
</tr>
<tr>
<td>MISSING – W/S Missing</td>
<td>Complete</td>
<td>Missing</td>
<td>Missing</td>
</tr>
</tbody>
</table>
Printing the content of a Userlist

The XTNET system allows printing of Userlist and its nomenclature content.

A Userlist must be open to use this function: double click on the Userlist you want to print.

To print a Userlist

1) Once an aggregate or a userlist has been opened
2) From Userlist menu, select Print
3) 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. Press the Ok button, a PDF file will be generated.

Figure 84: Printing Userlists
Dump of Userlist and Nomenclatures

COMEXT enables user to download Userlists and Nomenclatures into text files.

► To download a nomenclature

1) From an open dimension dialog, select the `Dump` button
2) Provide the parameters (fields to include, format and storage folder):

![Dump Nomenclature](image)

Figure 85: Dump of a nomenclature

3) Click on `Dump` button to launch the download.

► To download a Userlist

1) From an open userlist, select the `Export` button from the Userlist menu
2) Provide the storage folder.
3) Click on Save to proceed to the download.

Note: The generated file will be an XML.
Storing Userlists

By default, 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 folders, in which you can store your Userlists. For example, under the nomenclature NC, you may wish to create a textile folder, a food folder 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:

After choosing the values that will construct your Userlist, click on “Replace”

1) Click the following icon:
2) Give a name to the group
3) Click on the OK button.
Userlist properties

Activating/de-activating 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 codes of the Userlist.

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

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

To activate the Aggregate /Userlist property:

1) From the User objects menu, select Set/Remove (User list or Aggregate) property depending on the selected user object in ‘Selected’ window.
2) In your selection, the aggregate or the userlist’s symbol will be removed or added according to your selection.

Reference period

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

The reference period is only used as a filter if the Userlist is used as an aggregate.

The reference period can be shown directly in the Dimension Selection Dialog:
Weight

Similarly, a weight can be assigned to each element of a Userlist. The weight only used if the Userlist is used as an aggregate during the extraction. The weight can be shown directly in the Dimension Selection Dialog:

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; \[ \text{rate} = \frac{\text{total1999} - \text{total1998}}{\text{total1998}} \] 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; \[ \text{XW} = \frac{x}{\text{world}} \] in the “partner” dimension
3. For a given product, what is the relationship between the value and the quantity exported? Using the formula; \[ \text{indexvq} = \left(\frac{\text{val}}{\text{quantity}}\right) \times 100 \]

Formulas cannot reference themselves.

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

\[ \text{Aggr1} = x1 + x2 \quad \text{and} \quad \text{Aggr2} = x3 + x4 \]

and the formula \( Z = \frac{\text{val}}{\text{quantity}} \) then

\[ (\frac{\text{aggr1}}{\text{aggr2}}) \times Z = \frac{(x1 \times \text{val} + x2 \times \text{val})}{(x3 \times \text{quant} + x4 \times \text{quant})} \]

The creation and insertion procedure is similar to that for the Userlists.
To use formulas:

1) From the ‘Edit Dimension’ window, select ‘Insert Formula’ option or click the icon.

   ![Insert Formula Icon]

   The following window appears:

   ![Inserting a formula]

   Figure 87: Inserting a formula

2) Select an existing Formula

To create a formula:

From the Edit Dimension window, select ‘Insert Formula’ or click the icon.

1) Give a name to the formula

2) Click on ‘Insert New’
3) If you insert a new formula at the Indicators dimension, you can define either a constant or an expression of type of String, Numeric or Status. At any other dimension, only as an expression is allowed.

4) Insert the expression into the "function" box

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

5) Use the ‘Check’ button to validate the syntax

6) Click ‘Apply’ to confirm the formula kind, constant or expression

- **DECLARANT**: You may select elements from other dimensions and use them in the formula expression
- **EXPRESSION**: You may select existing templates for your formula expression.

The following functions are can be typed in a formula:
ASIN, ACOS, ATAN, CEIL, COS, COSH, DRAND48 (generate random numbers), EXP, FABS, FLOOR, FMOD, LOG, LOG10, MAX (and list of components), MIN (idem), Null, Nvl, POW, SIN, SINH, TAN, TANH, SQRT.

Please do not use the call of a function inside herself nor the special characters // or /* inside a function.

All trigonometric and hyperbolic functions receive one argument. Example: sin(myExpression).

POW and FMOD receive 2 arguments: POW(<base>, <exp>), FMOD(<x>,<y>).

Only the logical operators AND, OR are accepted.

For example: $INDICATORS['VALUE_1000ECU'] > 100 AND $INDICATORS['QUANTITY_TON'] < 2
Set last periods

The user can add in the Period dimension the element Set last Periods from the top panel.

The panel User objects will be opened. The user can insert already created Last periods or Insert new.

Press Insert new button and type the new name and the window Set last periods function will be opened. The user can set last periods parameter and press Ok and Last periods function will be added to the Period dimension.
Double click on the Last periods function in Selected elements will allow to edit it. The window Set last periods function will be opened. The user can update the parameter and press Ok. The changes will be saved.

**Note:** The system will calculate the last periods based on the periods available at the execution time. The order is according to the server (based on the last available periods) and not according to the order in the Available elements panel.

**Manage User Objects**

The User can open the panel Manage User Objects from the top menu Tools.
At the top of the panel Manage User Objects the user can select Dataset or Nomenclature, User or System, Explicit or Implicit, Regular, Status or Missing. The user chooses the dataset and dimension (or Nomenclature) in the right part of the panel. Aggregates and Userlists created for the selected dataset (or Nomenclature) and selected dimension will be shown. The user can create the new aggregate when Dataset or Nomenclature and type of the user object are selected otherwise the button Create is deactivated. Press the button and type the name of the new aggregate and press OK. The new aggregate will be added. Double click on the added aggregate and it will be opened and the user can add elements in it (the same logic as in the chapter User Objects). Or select the aggregate and press button to edit it, to set the label.

When the user presses the button the valid successors will be added and the original element will be deleted. When there are no valid successors no change will be performed on the selected aggregate. The user will be informed about the result via the message.
Note: Upgrade an aggregate means that all the successors (of the codes belonging to the aggregate) valid for the current year (the current year is included into the validity period of the successor) will be added to the aggregate. It means also that an aggregate will be upgraded if its parent is upgraded: it involves the recursivity of the upgrade functionality.

The user can delete the selected aggregate. Press the button. The system asks for the confirmation. When the user confirms the action the system checks if the aggregate belongs to the queries but the system will check only the first level and not if the aggregate is inside other aggregates in the queries. If the aggregate is in the queries the system shows the message and asks for the confirmation again.

The predecessors and successors:

The nomenclatures can vary in time. For instance, the Combined Nomenclature is revised annually, resulting in the end of validity of certain codes, code changes for certain products, or re-arrangement of groups. To allow users to keep track of code changes, the XTNET system provides information on Successors/Predecessors.

This time-based code tracking system is also used for some Country nomenclatures. When the dimensions of a query or a Userlist are being edited, an arrow pointing to the right precedes each code containing a successor. An arrow pointing to the left precedes each code containing a predecessor. An arrow pointing in both directions precedes each code containing a predecessor and a successor. The possibility to show predecessors/successors depends on the domain.
To use the successors/predecessors functions

1) From the edit dimension window, double click on one code marked with a green arrow

2) From the Successors/Predecessors window, click on the ‘Add’ button to select the highlighted code

3) Click on the ‘Add All’ button to select all the codes available in the graph

4) Click on the ‘Add MSA’ button to select all the codes available in the graph plus all the codes related to one of the code of the diagram

The "Add", "Add all" and "Add MSA" buttons are activated in the window containing the diagrams.
Note: ‘MSA’ mean Minimum Stable Aggregate. It enables selection of all codes involved in the evolution of the requested code. However, *this function must be used with the greatest caution.*

When "Add MSA" is selected, the system selects all the codes in the diagram ("Add all" function), but also the successors/predecessors of the successors/predecessors. To ensure that this function is correctly used and that the user can create a real Minimum Stable Aggregate, the selection must be made using a code that has **successors only**. In this case, the MSA function selects a group of codes from which the user is able to create an aggregate.
EXTRACTION OPERATIONS

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

The advanced functions enable advanced data manipulation.

Filter Data

This function will enable you to apply a filter on a saved extraction.

The filter will be applied on the Indicators. The “Filter Data” performs almost like the formula but data are treated by their order and we cannot access a previously treated value.

The filter returns the logical “TRUE” or “FALSE”.

If the filter returns the logical value “TRUE” then the related code will be displayed in the spreadsheet. Else, it will be not. It means that if you put $INDICATORS['VALUE_IN_EUROS'] without any other operator (>;<, !=, etc.), then this expression will be evaluated as TRUE (the value_in_euros exists) and all data will appear.

The filter shows the defined indicators but we can join other dimensions too:

REPORTER['TT'] AND $INDICATORS['VALUE_IN_EUROS'] ! = 100000

Other example: $INDICATORS['VALUE_IN_EUROS'] >1000000 AND (PARTNER['XX'] OR PARTNER['YY'] )

Another one $INDICATORS['VALUE_IN_EUROS'] >1000000 AND NOT REPORTER['AA']

To apply a filter from an extraction:

1) Select an output file from the Completed Work window

2) Click “Filter Data” button

3) Select an existing Filter or insert a new one from the following dialog:
4) To insert an existing filter, select the filter in the list and press the ‘Insert’ button.

**Note:** If the selected filter cannot be applied (for example, it contains indicators not present in the extraction), an error message will be displayed.

5) To create a new filter, give a name to the new filter in the **Selected:** box and use the **Insert New** button to define the filter’s options.
6) From the dialog displayed, select the available indicator (double click to select in the text box and enter your filter).

7) Click the **OK** button to confirm the filter and to launch the filtering.

8) Confirm the filtering and select the processing option (Batch or Interactive mode)

The data filtering is also accessible through the Extraction window under **Advanced Functions**.

---

**Transpose**

This function will enable you to transpose dimension(s) **vertically** or **horizontally**.

▶ **To Transpose Dimension(s):**

1) Select an output file from the Completed Work window

2) Click “Transpose” button

3) Select the type of Transposition and display the associated parameters to be selected:
   a. **Horizontal Transposition:**
The horizontal transposition will proceed to a concatenation of the indicators and of the codes of the selected dimensions. It is mostly used to generate time series.

The above dialog will enable user to:

- Change the order of the dimensions using the buttons
- Select the Indicators and dimensions delimiters symbols

b. Vertical Transposition:

Two kinds of vertical transposition can be generated, the Pure and the Tuple ones. The selection of the type of transposition will be done on the top of the dialog:

i. Vertical Transposition Pure:

In the vertical transposition we create new dimension with the indicator’s name.

1) Select the type of vertical transposition as Pure

2) Define the structure of the transposition:
Under the section “Indicators to transpose”, the system will display all the existing indicators of the extraction. User must select one or several indicators.

3) On the right side, give the name of the new dimension and of the codes for the type of indicators (String, Numeric and Status).

4) Confirm the transposition by a click on the **Ok** button.

For example, if user select “VALUE, US and UC” in the list of Indicators, in the extraction generated, a new dimension called “Dimension” will contains the 3 codes (Value, US and UC) and the Indicators dimension will contains “String, Numeric and Status”:

**ii. Vertical Transposition Tuple:**

5) Select the type of Vertical transposition as **Tuple**

6) Define the structure of the transposition:
Please note that the names corresponding to the String, Numeric and Status Indicators must be unique.

**Note:** A Tuple is composed of “at least” two indicators. The first indicator in the list will become a code of the new created dimension and the second one will be the indicator for which the value will be displayed.

User can add as many tuple as he/she wants. If a Tuple is composed of 3 indicators (For ex: US, UC and VALUE), two new dimensions will be created. The first one will contains the code US and the second one, the code UC.

**Note:** Using this option, it is possible to create a new dimension for which the Value (numbers) will become codes of the new dimension by creating a Tuple having VALUE in last position.

7) Define names for the new vertical dimensions which will be created

   **Vertical Dimensions**

   - unit

8) Confirm the transposition by a click on the **Ok** button.
Interpolate Time Series

This function will enable you to generate figures in the extraction for the “missing” periods. For example, if an extraction is composed of 3 months (January, May and August), this function will enable the creation of figures for the months February, March, April, June and July). The values will be generated according to an interpolation function which will be selected by the user.

To Interpolate a Time series:

1) Select an existing extraction from the Completed Work window
2) Click “Interpolate Time Series” button
3) Define the parameters of the Interpolation in the following dialog:

![Figure 93: Interpolate Time series dialog]

In the above dialog, users will have to select a calendar (predefined in the system) and a interpolation method. Three methods are available (in addition to the NONE which is the default one and will not proceed to any interpolation):

- **NEAREST**
  - This option will copy the value of the nearest filled month. If we take the example mentioned above, the value of January will be assign to the “created” period of February and March. The value for May will be used for the months of April and June, and the value of August will be used for July

- **LINEAR**
  - This option will compute the missing periods by using a linear regression between two “available values”.
CONSTANT

- This option will use the value for the first period available and will assign it to the missing period, until the next “extracted” value.

**Values Types**

This function will enable you complete your extraction by associating indicators names and types.

► To Add Values Types:

1) Select an output file from the Completed Work window
2) Click “Values Types” button
   
   ![Values Types](image)

   **Figure 94: Value type dialog**

3) Fill the following dialog:

4) Click on the OK button to generate a new extraction. The new extraction will contain the original dimensions and two new dimensions (Value Name and Value Type), while the previous indicators will be available in a new dimension called according to the name provided under the (Value Name) text box of the “Get Values Types” dialog.

**Drop**

► To drop dimension(s) in an extractions:

1) From the extraction window
1) Or from the ‘Completed Works’, click on **Drop**.

This option will display the **Drop** dialog, enabling the setting of **Drop** parameters.

![Figure 95: Drop dimension dialog](image)

From this dialog, user can select one or several dimensions to be dropped. When selecting one or several dimension(s), the parameters for the Indicators are set by default as follows:
Figure 96: Aggregation function under the Drop dialog

For each indicator of the dimension, user will be able to select the aggregate function (from the list of the available operations), the name of the Indicator and the use or not of the Zeros/Nulls.

1) Click on the **Ok** button to generate the new extraction
Reshape

To Reshape an extraction:

1) Select an output file from the ‘Completed Work’ window
2) Click “Reshape Extraction” button

This option generates a new extraction (reshaped extraction) using an existing one

Figure 97: The reshape dialog

Under Reshape, user will be able to Split and/or Concatenate Indicators and Dimensions.

Concatenate dimensions

This operation is similar to the “Horizontal Transposition” and can be used for time series.

1) Select the ‘Concatenate’ button in order to open the following dialog:
2) Define the parameters for the Concatenation of dimensions:
   a. Define a name for the new dimension
   b. Add a description (not mandatory)
   c. Select the type of indicators to be taken into account (Numeric, String or Status) and the default and missing value.
   d. Select the delimiter from the drop down list
   e. Select the dimensions to be concatenated
   f. Click on OK button to close the concatenation parameters definition dialog. The dimensions selected to be concatenated are highlighted in blue.
   g. Click on OK to view the concatenated extraction.

Figure 98: Concatenate options dialog
The split can be used in various ways: a split by length (upper value limit is excluded), split of a previous concatenation (done on the delimiter). It creates new dimensions named as shown below.

1) Select the dimension to split from the ‘Reshape’ extraction dialog. This action will open the SPLIT definition dialog:

![Split definition Dialog](image1)

**Figure 99: The Split option Dialog**

1) Select how the dimension will be split (on Length or based on Separator)
   a. Splitting parameter. Define a name for the new dimension
   b. Add a description (not mandatory)
a. Click on **OK** to view the split dimension statement.

If the user wants, he can suppress this split by the button “Delete”: all split parameters will be deleted.

Please note that in case the “FROM-TO” values are identical for different rows, an error message will appear:

If the user wants to split by “Separator” then he must indicate the separator and the name of the new dimension to appear. The ‘Missing value’ and the ‘Default value’ are set by default values, the user can change.
Swap a dimension to Indicator

1) Select the dimension to be swapped and click on button
2) Fill the parameters in the displayed dialog:

![Swap dimension to indicator dialog](image)

Figure100: Swap dimension to indicator

a. Define the new indicator’s Name
b. Define the Description (optional)
c. Define the precision (nbr of decimals)
d. Click on OK to create the new indicator.

Please note that it is not allowed to swap all dimensions to indicators.

1. To make samples on the Indicator values:

1) Select the Indicator that you want to make samples on and click on button
2) Fill the parameters in the displayed dialog:

![Make samples on Indicator values dialog](image)

Figure101: Make samples on Indicator values
a. Define the new Dimension’s Name  
b. Define the Description (optional)  
c. Define the precision (Nb. of decimals)  
d. Define the fallback values:  
   The fallback values will enable the splitting of the data, according to a range of value. A code for each range can be defined as well as a default fallback code. These codes will be the “nomenclature” of the newly created dimension.  
e. Click on OK to create the new dimension
Subcube Functions

The subcube functions are decomposed in four panels:

- Aggregation operations
- Time series operations
- Rank operations
- Filter operations

Figure 102: Subcube function dialog

Aggregate

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

2) Figure 74: The Subcube Element dialog
3) You can change the order of the dimension in the list by selecting the dimension and using the Up or down buttons.

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

   ![Result extraction name](image)
   
   test-0

5) Click on Add button to select the function to apply on the data. Several “Aggregates functions” are available:

   ![Figure103: The Aggregate functions selection Element window](image)

6) From the “On Indicator” list, select the indicators on which the calculation will be done

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

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

9) A panel for trim parameters will be displayed for functions where trim is available (First quartile, Third quartile, Trimmed average, Median absolute deviation, Trimmed standard deviation). The user selects trim parameters (MIN Values and Percentage).

   The user can select check box to set up also Right Trim.
Figure 104: Panel for trim parameters

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

2) Click on 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.

1) Click on 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.

**Advanced aggregate functions**

Trim functions are available in Aggregate tab.

**First Quartile**
Median of the lower part of the value set – the size of the “lower part” is specified as a percentage.

**Third Quartile**
Median of the higher part of the value set – the size of the “lower part” is specified as a percentage.

**Trimmed Average**
Function involves the calculation of the mean after discarding given parts of the sample at the high and low end, and typically discarding an equal amount of both. The number of values to discard is usually given as a percentage.
**Median Absolute Deviation**

For a data set \( X_1, X_2, \ldots, X_n \), the MAD is defined as the median of the absolute deviations from the data's median:

\[
\text{MAD} = \text{median}_i \left( |X_i - \text{median}_j(X_j)| \right),
\]

that is, starting with the residuals (deviations) from the data's median, the MAD is the median of their absolute values.

**Trimmed Standard Deviation**

The standard deviation of a variable is defined as

\[
s = \sqrt{\frac{\sum_{i=1}^{n} (x - \bar{x})^2}{n - 1}}
\]

with \( n \) and \( \bar{x} \) denoting the sample size. The standard deviation can be heavily influenced by extreme values (i.e., it is not robust). The trimmed standard deviation compensates for this by dropping a certain percentage of values from the tails. For example, the 50% trimmed standard deviation is the standard deviation of the values between the upper and lower quartiles. The 90% trimmed standard deviation is the standard deviation of the values after truncating the lowest and highest 5% of the values.
Time

This function allows you to apply calculations based on the time dimension. The Time tab is only available when the Reshape option has been 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:

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: offset, period, filter length.

**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:

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

2) Select the “Nulls” and/or “Zeros” boxes if you wish to use them in the calculation.

3) 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.

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

The button **Delete** 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’ windows, 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:

Figure106: 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.

**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.

1) Select the Order (Ascending or Descending)

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

3) 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.

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

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

4) Click on 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.

![Result extraction name]

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 108: The Filter “Ranking” dialog]

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

On Indicator: VALUE_1000ECU

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.

Note: The application does not allow defining simultaneously Aggregate, Time, Rank and Filter Subcube functions. Only one type can be defined and saved.
SERVICES

This function allows you to launch predefined extractions on a regular basis and to receive notification and/or output files in your mail-box.

Services functions

1) From the Remote menu, select Services

![Services window](image)

Figure 107: The Services window

The Service window displays the list of existing services. In addition, several buttons enable you to create new services, to edit existing services, to delete existing services or to define output objects.
Create New service

1) From the Services window, click ‘New’ button
   This gives access to the ‘Get Service’ window:

   ![Get Service window]

   **Figure 108: The Get Service window**

2) Fill out the fields in the ‘Get Service’ window:
   a. Give a name to the Service
   b. Give a description (optional)
   c. Select whether the Service is active or not (YES/NO option)
   d. Select a period (periodicity) for the extraction (Daily, Weekly or Monthly)
   e. Give a number to the period (this will define the period of validity of your service. For example, 12 with a Monthly periodicity will select the service for one year)
   f. Select the Query to be used for the extraction
   g. Select the next extraction date
   h. Enter a syntax (optional) for conditional services
   i. Click the OK button

   **Note:** The above procedure defines the service. The XTNET application will extract data according to the definition provided.

The result is stored in your XTNET account as an extraction and is available via the Completed work menu.

Edit an existing service

From the Services window, Select an existing service and click the ‘Edit’ button

This gives access to the Get Service window.
From the Get Service window, change the parameters that have to be updated and click the ‘OK’ button.

▶ To Delete an existing service:

From the Services window, Select an existing service and click ‘Delete’ button.

A confirmation window will ask you to confirm the deletion.

Click on ‘Yes’ and the service will be deleted.

Subscribe to a service:

This option enables you to specify the delivery location for your service. It can be a simple notification and/or an output file.

From the Services window, click ‘Subscribe’ button
This gives access to the ‘Subscribe for Service’ window.

![Subscribe for Service window](image)

Figure 109: The Subscribe for Service window

1) Complete the fields in the Subscribe for Service window:
   a. Select if the Service is active or not (YES/NO option)
   b. Select the period of the subscription (From DD/MM/YYYY to From DD/MM/YYYY Y)
c. Give a delivery address (mail address where the deliverable should be sent) (Optional)

d. Select the Delivery type (Account only or Account and Email)

e. Give a maximum size for the attachment to be sent

f. Click the OK button

**Outputs**

This option is used to specify the output Object for your service. All the previously mentioned output formats (Print, Dump, Table, etc.) can be selected.

1) From the **Services** window, click the ‘Output Object’ button

This gives access to the Output Object selection window:

![Output Object selection window](image)

**Figure 110: The Output Object selection window**

2) From the **Output Object Selection** window, select the required format.

**Note:**

1. According to the selected output object, an appropriate definition window displays so that you can define your output structure. For example, if you select the ‘generate Table’ output, the ‘Generate Table selection window’ will be open so that you can define the output options for your table.

2. The **MTX** is the native file format of the XTNET server. The MTX holds an XTNET extraction.

3) Click ‘OK’ to confirm the Service and to close the Services window.
OUTPUTS

Print

The ‘Title’ is set by default by the application. The user can change it. The max length (spaces included) is of 255 characters.

The user must define the layout of the report by specifying which parameter(s) are for the X-Axis and Y-Axis. He can also set the language of the labels and choose to print the codes (e.g. the country code for a ‘Partner’), the labels (e.g. the country name for a ‘Reporter’) or both. By default, the application sets the ‘Codes only’.

The default printing orientation is ‘Landscape’ but it can be modified by the user.

The printers are set by the application administration. The user can also choose to print the null values and/or the zeroes. This information is not mandatory. All other checkboxes are set by default and can be modified by the user.
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:

1) Select an extraction from the Completed work window.
2) Click on “Dump” button

(This option can also be launched via the Extract section of the Query Context menu, Dump option)

The following window appears, allowing configuration of the 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 only) or B (Codes and Labels) from the “Dump” option windows

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

![Language Option](image)

3) Choose whether or not to print the “Nulls” or “Zeros”, by checking or un-checking "Print Zeros" or "Print Nulls" boxes

![Options Option](image)

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

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

6) 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.

7) Select the Remote Host Name and fill the Username and Password.

Launch the Dump, click “OK” button

![OK Button](image)

8) 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 section** of the Query Context 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:

![Generate Table configuration window](image)

**Figure 113: 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 **Line** in the Table

If you want to reorder the dimensions in the Dimension text box, then use the right mouse button, select one dimension and “drag & drop” it at the requested position.

➢ **To define the printout options:**

Give a title to the Tables in the Title text box,

| Title: Table Generation of Extraction from Plan:k001 5846.mtx |

1) Select the appearance of the headings from the **Print heading**

The "PRINT" heading gives the option to display:

<table>
<thead>
<tr>
<th>codes only</th>
<th>Code (C)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Labels</strong></td>
<td>Label (L)</td>
</tr>
<tr>
<td><em>abbreviated codes and labels</em></td>
<td>Codes and Labels (B)</td>
</tr>
</tbody>
</table>

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

| Language | English |

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.

<table>
<thead>
<tr>
<th>Text Identity</th>
</tr>
</thead>
<tbody>
<tr>
<td>☐ No</td>
</tr>
<tr>
<td>☐ Excel</td>
</tr>
<tr>
<td>☐ Lotus</td>
</tr>
</tbody>
</table>
5) Select the output from the ‘Output’ selection box for:

- [ ] Nulls as *
- [ ] Print Nulls
- [ ] Print Zeros

6) Click “OK” 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 an 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 section of the Query Context menu, Out Data option)

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

![Out Data configuration window](image)

Figure114: The "Out Data" configuration window

► To define the presentation of your File:

Note: This option will only be available for some output formats.. (XML, HTML, HTML_TOC, RTF, Tabular CSV et Tabular FIXED.

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 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.

**To define the printout options:**

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

The "PRINT" heading gives the option to display:

<table>
<thead>
<tr>
<th>codes only</th>
<th>Code (C)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Labels</strong></td>
<td>Label (L)</td>
</tr>
<tr>
<td><strong>abbreviated codes and labels</strong></td>
<td>Codes and Labels (B)</td>
</tr>
</tbody>
</table>

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

3) Select the output format from ‘**Text Identity**’. This option defines the output format.

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

**Note:** As mentioned above, several formats are available under the **Out Data** selection, such as:
Figure 115: List of available formats under the Out Data option

Note: TFIX and TCSV output have the following restrictions:
- all the dimensions must be set on X or Y axis (no dimensions should remain "free")
- at least one dimension should be set on the X and Y axis.
SDF

The following dialog will appear, to specify where the parameters of the SDF output file. The SDF format is suitable for Eurobase system.

![Parameters dialog for the SDF extraction](image)

Figure116: Parameters dialog for the SDF extraction

The SDF format expects only 1 indicator (value) in addition a flag can be added (not mandatory).

According to the dataset structure, it will be possible to apply transposition when defining an SDF output. The transposition will enable the use of an indicator as a dimension in the SDF file. For example, if a dataset contains the unit of a value (for ex unit codes of a specific value), this unit can become a dimension for Eurobase, using the transposition option.
SDF Without Transposition

To generate a standard SDF (Eurostat Standard Dissemination format) file:

1) Select the name and the order or the dimensions

<table>
<thead>
<tr>
<th>Dimension Name</th>
<th>Renamed Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>SPDEPRENAMED</td>
<td>SPDEPRENAMED</td>
</tr>
<tr>
<td>GEO</td>
<td>GEO</td>
</tr>
<tr>
<td>TIME</td>
<td>TIME</td>
</tr>
<tr>
<td>UNIT</td>
<td>UNIT</td>
</tr>
<tr>
<td>DPFUNC</td>
<td>DPFUNC</td>
</tr>
</tbody>
</table>

- To change the name of the dimension, click on the dimension in the Renamed Dimension column in order to enable the editing.

- To change the order of the dimensions, select a dimension and the arrows on the right side of the dialog.

2) Select the update Path (ID_Key) and the update Mode

The ID_Key: is the name of the table in Eurobase.

The Update mode will determine which kind of update will be done.

3) Select the indicator (mandatory) and the flag (optional)

4) Provide optional information to the SDF file:
5) Click **OK** to save the SDF definition.

As mentioned here above, the SDF format is composed of dimensions, 1 value and optionally Flags. According to the XNET dataset structure, it might be necessary to transpose a dimension. The transposition will enable the definition of an indicator (or a set of indicators) to become a dimension.

**To generate a transposed SDF (Eurostat Standard Dissemination format) file:**

1) Select the option Vertical Transposition

2) Select the type of transposition (Pure or Tuple).

Two types of transpositions are available, Pure and Tuple:

### SDF with Pure Transposition

The Pure transposition will enable the transposition of one indicator. Using this option will include, in the list of the dimensions of the SDF format, the label of the indicators. For example, creating an SDF file from a dataset having the indicator Value_1000 Euros transposed will generate a new dimension having the content "Value_1000 Euros" in addition to the value itself (which will contains the figures)

When selecting the Pure option in the Transposition Type box, user will have to provide the following information:

3) Select the Indicator to be transposed and provides the associated options

<table>
<thead>
<tr>
<th>Indicators</th>
<th>Renamed Indicator</th>
<th>Status Indicator</th>
<th>Special Value</th>
<th>Flag Qualifier</th>
<th>Special Value Qualifier</th>
</tr>
</thead>
<tbody>
<tr>
<td>VALUE</td>
<td>VALUE</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLAG</td>
<td>FLAG</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

4) Name the new created dimension:

**New Dimension:** Dimension

5) Click on the **OK** button to save the SDF format definition
SDF with Tuple Transposition

The Tuple

The selection procedure is the following:

1) Select a Tuple from the dialog:

A tuple will be the association of several indicators (one indicator being the value and the other one the unit. The order of the selected elements in the tuple is important as it will be followed in the final SDF file. The indicator which will be set as unit must be in the first position.

In the example, IEA_ITEM has been set before Value.

2) Define the parameters (Missing values, Status Indicators, Special Value, Flag qualifier and special Value for the flag Qualifier)

3) Define a name for the “new dimension” in the vertical element text box. In the event of a tuple having two components, a Value and a Unit, the name of the Unit will have to be entered. If the tuple contains 3 components, two new dimensions will be given. The Clear button will clean the entered name.

4) Select/unselect the option to print the missing Tuple. By default, the option will not be selected.

5) Click on the OK button to launch the generation of the SDF file
SDF With EUROBASE

SDF is the file format used to load in the Eurobase system in Eurostat. It is possible to load directly in Eurobase from the WEB Analytical Client.

**Prerequisites:** This option is only available for specific users in Eurostat, by default no user is allowed to load in Eurobase. If needed, please ask the Comext support to add such feature to your account properties.

**Actions**

1. There is an optional check box Eurobase to enable Eurobase specific fields. Click on this check box.

2. A report file will be generated when loading the SDF File: it stores a trace of the actions when sending the SDF file to the Eurobase loading services. You have to set a name for the report, the report will be a new kind of output attached to your SDF output in the Completed Works dialog. Please note, that it is in general a good idea to keep such report: if there is an error raised during the loading of the file, you will be able to notify persons in charge of the system with information contained in the file.

3. Set the kind of Action performed:
   a. Either the Eurobase loader will SEND the file to Eurobase.
b. Either the Eurobase loader will VALIDATE the file before the actual SENDING. Any error will be written inside the report file as detailed previously.

4. Set the Destination: there are different Eurobase systems deployed, you will have to perform a choice:
   a. STAGING Eurobase
   b. PRODUCTION Eurobase

5. Set remaining properties on the SDF output if needed as detailed before and then click on Ok.
Graph

This option formats the results of the extraction into a Graph. XTNET 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
(This option can also be launched via the Extract section of the Query Context menu, Graph option)

The following window appears, allowing configuration of the parameters for the Graph:

![Graph configuration window]

- Select the type of Graph from the list available in the “Type” box on the top right of the window. Several options are available:
 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).

If you want to reorder the dimensions in the Dimension text box, then use the right mouse button, select one dimension and “drag & drop” it at the requested position.

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). The appearance possibilities are shown (enabled /disabled) when the selected graph type changes. E.g. 3D is not available for Scatter and Bubble graphs

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.

**Note:** Please keep in mind that the ‘Code’, ‘Label’, ‘Code &Label’ buttons are related to the dimension or the indicator you choose. They do not apply to all the elements.

Please keep in mind that line, bar and column graphics need at least 2 columns of data.

**Only one row** is admitted on **Pie Charts**.

**Only one row** (mapped dimension) is admitted for **Maps**.

**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 other 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-Axis nor Page) one graph is produced.
• **Pie chart** displays ratio of indicator values. There is no notion of the scale. (See Figure 120)
  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 121).
  Dimensions from X-Axis list become dimensions of horizontal axis of the graph. Indicator value is displayed as height of point or bar above the code. One dimension may be listed as Page. Codes of this dimension are displayed in various colors 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.
  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:

2. 0001.Feb. 2003
3. 0001.Mar. 2003
5. 0002.Feb. 2003

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

Map

This option formats the results in a Map. Only datasets with mapped dimensions can be used to produce Map graphics. Usually, the DECLARANT and/or the PARTNER dimensions can be defined as mapped.

► To Generate a Map:

3) Select an Extraction from the Completed work window
4) Click on 'Map' Button

(This option can also be launched via the Extract section of the Query Context menu, Map option)

The following window appears, allowing configuration of the parameters for the Map:
This option formats the results in a XTTABLE.

**To Generate an XTTable:**

5) Select an Extraction from the Completed work window

6) Click on 'XTTABLE' Button

(This option can also be launched via the Extract section of the Query Context menu, XTTABLE option)

The following window appears, allowing configuration of the parameters for the XTTable:
By default, the ranges are not explicitly specified and all the elements are used (see print screen). If you choose to define ranges, you will have to click on ‘Ranges’.

Please keep in mind that:

1. they must cover all elements
2. they ranges should not overlap each other
3. they must start by 0

It is possible to use the ranges on one of the axes or on both.

**EUROBASE**

The Eurobase button enables the user to load SDF files directly in the Eurobase system.

**Prerequisites:** This feature is only enabled for Eurostat users that are allowed to load in the Eurobase system. You can send an email to Comext support to modify your account to enable this feature.
Actions:

1. Select an SDF Output to enable the Eurobase button.
2. Click on the Eurobase button, it will show you the following dialog.

3. Set the report name, for any invocation of the Eurobase loading service a report will be generated and attached as a child to the SDF output to be loaded.
4. Select the Action to be performed:
   a. Either select SEND that will ask the Eurobase loading service to perform the loading of the SDF output.
   b. Otherwise select the VALIDATE that will ask the Eurobase loading service to perform some sanity check on the provided SDF output.
5. Set the Destination of the loading:
   a. Either to load in the STAGING Eurobase environment.
b. Or to load in the PRODUCTION Eurobase environment.

6. Choose either to run the loading in batch or in interactive mode.

7. Click on Ok.

8. On completion of the Job, a report will be shown in the Completed Works dialog.

![Completed extractions, prints and dumps dialog]

**Note:** A report will be generated in any case, it is a good idea to always keep such reports for bookkeeping, or if an error is raised to know what was the issue during the Validation/Loading in Eurobase.

### CFX

The CFX button enables the user to generate CFX outputs. The CFX output can be set on a Query, Operation or Extraction Groups.

**Prerequisites:**

- This feature is only enabled for Eurostat users that are performing DVD Production. You can send an email to Comext support to modify your account to enable this feature.

- This feature is only available for limited set of datasets: the CFX output is only available for a predefined list of datasets. The following error dialog will be shown:
Actions:

1. Select an Extraction and click on the CFX button.
2. The following dialog will be shown:
   ![Dialog](image1)
   3. Select the name of the object.
   4. Choose to extract in batch or in interactive.
   5. Click on OK.
Macros

Macros general user options

Macros are available in the Developer tab in the main ribbon menu and will give you the option to define a new macro for automating your tasks. A macro is a series of commands and instructions you can perform within the system to group together as a single command to accomplish the task automatically.

In order to achieve this goal we have introduced in the system the Developer tab. Within it we have a number of possible actions for the users.

- **Record Macro** the system will start recording your actions.
- **Pause Recording** you can pause recording by pressing the pause recording button.
- **Resume Recording** you can continue recording by pressing the resume button.
- **Stop Recording** you can stop recording by pressing this button.

**Note:** It is important to close all windows before recording a macro, so there is no interference of the flow of actions you want to perform and are recorded from the system.

Stopping your macro recording will need to be confirmed when you are ready:
As soon as you have stopped recording the system will prompt you to enter a name for your macro, according to your preference:

![Macro name input dialog]

**Note:** A predefined automatic name for your macro will be suggested, following the below naming convention: `name_date_time`.

### Viewing new macros

A finished macro recording process will lead you to the next part of viewing and saving the macro, as well as schedule its future execution.

![New macro drop window]

The following options are available:

- “Save” your macro, this option enables you to store the macro in your list of macros;
- “Save as” your macro, this will enable to create a copy of the current macro;
“Submit” your macro, this will enable you to run the macro, or schedule it in the calendar for automatic execution according to your preferences;

“Close” will close the new macro window panel and return you to the main view.

**Note:** At this stage you still have the option to save the macro without scheduling it for execution. If you wish to define the date then you can select “Schedule to run on” and select the date, and periodicity, and if you want to repeat this execution of up to a limit of 100 times.

### Scheduling macros execution

With the macros functionality you have the option to schedule your macro to run on specific predetermined times with a daily, weekly, monthly and yearly frequency.

To achieve this you will need to press **Submit** during the view new macro window.

The user can select the following options:

- On submission (macro will be executed immediately),
- On macro scheduled date (system will execute macro if you have defined a date specified in the new macro view window)
- On specific date. You can override the previous two settings and define a new date and time for your macros, with the same periodicity options and up to a max limit of 100 times repetition of the macro.
To submit a macro and end the process you need to select the **Submit** button. The system will record all the specifications and perform the needed operations either immediately or at the scheduled dates.

**List of already defined user macros**

The list of macros can be opened from the Developer tab by pressing the “Macros” option in the left side of the tab.

The following options are available:

- **Edit** (reserved for system administrators)
- **Delete** (select and delete the macro you wish)
- **Import** (import a macro from another source, in txt format)
- **Export** (the macro will be exported in a txt format file)
- **Submit**.
Notes: Column Status informs the user if the macro is Invalid, Saved, Submitted for the execution or Completed. When the user double clicks on the macro the Edit macro form will be opened, selected macro will be shown and can be edited if the user has System Admin rights. When the user presses job icon ![button] in the Last job column Jobs dialog will be opened and the job corresponding to the selected macro will be inverted.

Examples of using macros

Example 1: How to record a macro (simple)

Step 1: Select Developer tab and press the Record Macro button (the system starts recording).

Step 2: Open the query and press the Extract button.

Step 3: Confirm the extraction (query can be extracted only interactively during recording session).

Step 4: Press the Stop Recording button after extraction is finished.

Step 5: Put the macro name (e.g. Extract1) and press OK (New Macro form will be opened).

Step 6: Press Save or Save as to save the new recorded macro.

Note: When the macro Extract 1 is executed the same plan will be extracted, new job will appear in Jobs window and new extraction will appear in the Completed works when the job finished successfully.

Example 2: Defining the Drop operation (advanced macro recording)

Step 1: Select Developer tab and press the Record Macro button (the system starts recording).

Step 2: Open Completed Works.

Step 3: Select one extraction and press the Drop button.

Step 4: Define Drop operation, press Ok and confirm online extraction.

Step 5: Press the Stop Recording button after extraction is finished.

Step 6: Put the macro name for ex. Drop1 and press OK (New Macro form will be opened).

Step 7: Press Save or Save as to save new recorded macro.
Important Macros information - Troubleshooting

The following information can be considered while working with the macros functionalities:

My macro show no result in the application

If a macro recording is unfinished (e.g. opening the query only, or opening the CW) and doesn’t contain any extraction operation, the new job will appear in the Jobs dialog but nothing will happen in the application when it is finished.

My macro recording session is expired

The macro recording session allows for 10 minutes of inactivity, after that the user will be prompted to stop recording or continue the operation as it consumes resources of the system if left open.

I cannot use function X in my macro recording

Macros are not currently supporting the following functions as they are not relative to the extraction process, and are disabled in this mode:

1. Disk usage information
2. Notifications
3. Services
4. Options
5. File Download.

The macros recorder will record operations related to queries (new, open, execute) and extraction operations (except operations, which need the availability of the user’s PC like download and Excel direct output).
Can I do other operations while recording a macro?

It will not be possible to perform extractions in batch mode while submitting a macro interactively. Scheduled macros will always run in batch mode.

Error X during recording process

When there is an error during the recording process you will be notified at the top of the macro by the red icon 🔄 Edit Macro: View_vector_12112015_150358_667 🔄. Invalid macros can’t be submitted to the execution. Please ask the assistance of the administrator in this case.

Common examples where an error can be received is with the following user actions:

- When the user records some operations contained opening a spreadsheet. There can be hidden error which is not shown but affects the macro because invalid calls were recorded.

Execution of macro terminated with an error

The User can submit valid macro but this macro job terminates with error. Common examples where an error can be received is with the following user actions:

- When the main extraction is deleted from Completed Works and user recorded macro of operation from that extraction, the macro ran will always terminate with an error;
- When a macro contains opening a query and extract it, but this query was deleted the macro execution will produce an error.

Editing a macro

Editing a macro is only possible if the Advanced mode is enabled, this function is currently reserved for the system administrators.
Frequently Asked Questions

What is the difference between a query and an extraction?

The query contains the set of instructions given to the system to extract the data. The extraction contains both the query and the “raw data” extracted from the XTNET Dataset.

The query is stored on the server, and the extraction is also stored 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, I have an issue, 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: “Query/Export ” Any file saved in this form (.sde) can be imported to the same or other XTNET installation by using the command: “Query/Import ”.

What is the difference between a Null and a Zero?

In XTNET, as in all databases, a null means 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’.
Green arrows pointing to the left or to the right. What do they mean?

We will try to explain with an example:

Until recently (end of 1998) Belgium and Luxembourg were producing common statistics under the country code 002. After '98, they supplied separate statistics with codes 017 (Belgium) and 018 (Luxembourg): A kind of code “evolution” happened. This is not only true for country codes but also for product codes (new, discontinued products etc.).

If we double click on code 002 (arrow pointing to the right) we will see a graph showing this split of the code. An arrow like this reminds us that this code has evolved into something else.

The same graph for code 017 reveals that a left-pointing arrow, denoting the existence of an “old” code, from which the current emerged.
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).

**Important**: when the periods of validity and / or weight have been changed on some codes belonging to a user object (list or aggregate), it is important to know that during the extraction only an aggregate will show these changes. If you choose the list, then these changes will have no impact.

Extracting a user list provides a complete list of codes that will be displayed according to their periods of validity and initial weights.

Why is the software disconnected every now and then?

*The XTNET software disconnects itself from the server after half an hour of inactivity. This is done to unload the server from the burden of inactive connections. After being disconnected, you can use the menu command “Remote/Connect” to reconnect to the system again.*
# TABLE OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>The domains accessed by the XTNET client</td>
<td>8</td>
</tr>
<tr>
<td>2</td>
<td>Define a new query</td>
<td>9</td>
</tr>
<tr>
<td>3</td>
<td>The dimensions of the “EU25 SINCE 1999 CN(SIMULATED)” dataset</td>
<td>10</td>
</tr>
<tr>
<td>4</td>
<td>Save as (Remote or Local)</td>
<td>12</td>
</tr>
<tr>
<td>6</td>
<td>The login/password input window. (example for External user)</td>
<td>14</td>
</tr>
<tr>
<td>7</td>
<td>Inactive Account message</td>
<td>15</td>
</tr>
<tr>
<td>8</td>
<td>Main User Interface</td>
<td>16</td>
</tr>
<tr>
<td>9</td>
<td>The main Ribbon Options</td>
<td>17</td>
</tr>
<tr>
<td>10</td>
<td>Query options from main Ribbon</td>
<td>19</td>
</tr>
<tr>
<td>11</td>
<td>The option from main Ribbon</td>
<td>19</td>
</tr>
<tr>
<td>12</td>
<td>Tools options from main Ribbon</td>
<td>20</td>
</tr>
<tr>
<td>13</td>
<td>Download files window</td>
<td>21</td>
</tr>
<tr>
<td>14</td>
<td>Help options from main Ribbon</td>
<td>22</td>
</tr>
<tr>
<td>15</td>
<td>Status bar</td>
<td>22</td>
</tr>
<tr>
<td>16</td>
<td>Interface Language selection</td>
<td>23</td>
</tr>
<tr>
<td>17</td>
<td>Set Query Language</td>
<td>23</td>
</tr>
<tr>
<td>18</td>
<td>Contextual menu “Dataset”</td>
<td>24</td>
</tr>
<tr>
<td>19</td>
<td>Comments window with files</td>
<td>25</td>
</tr>
<tr>
<td>20</td>
<td>The download directory</td>
<td>25</td>
</tr>
<tr>
<td>21</td>
<td>Query contextual menu</td>
<td>27</td>
</tr>
<tr>
<td>22</td>
<td>select the option when creating a new query</td>
<td>28</td>
</tr>
<tr>
<td>23</td>
<td>The Query” dialog</td>
<td>30</td>
</tr>
<tr>
<td>24</td>
<td>The “Edit” dimension menu</td>
<td>32</td>
</tr>
<tr>
<td>25</td>
<td>Dimension dialog</td>
<td>32</td>
</tr>
<tr>
<td>26</td>
<td>Selecting Code(s)</td>
<td>33</td>
</tr>
<tr>
<td>27</td>
<td>Filter dialog with filter activated on Code</td>
<td>34</td>
</tr>
<tr>
<td>28</td>
<td>Example of hierarchical searches with filters</td>
<td>40</td>
</tr>
<tr>
<td>28</td>
<td>Under &quot;CODE&quot;. ‘##’ produces all length 2 numeric codes</td>
<td>41</td>
</tr>
<tr>
<td>29</td>
<td>010 followed by #### will produce all digits following 010 in a 8-digit code</td>
<td>41</td>
</tr>
<tr>
<td>30</td>
<td>The string of characters <em>anes</em>I<em>mulets</em> will produce all codes containing &quot;anes&quot; or &quot;mulets&quot;</td>
<td>42</td>
</tr>
<tr>
<td>31</td>
<td>Sort options</td>
<td>45</td>
</tr>
<tr>
<td>33</td>
<td>Display of codes validity periods</td>
<td>48</td>
</tr>
</tbody>
</table>
Figure 34: editing validity periods and weight ................................................................. 49
Figure 35: The label type dialog ....................................................................................... 50
Figure 35: Dump Nomenclature dialog ............................................................................. 51
Figure 37: Query information dialog ............................................................................... 52
Figure 38: Print query dialog ......................................................................................... 53
Figure 39: Save dialog (for exported query) ................................................................... 54
Figure 40: Open dialog (to import query) ....................................................................... 55
Figure 41: Reorganize button ......................................................................................... 63
Figure 42: Reorganize dialog ......................................................................................... 63
Figure 43: Renaming a dimension ................................................................................... 64
Figure 44: Renaming an Indicator .................................................................................. 65
Figure 45: define Formula execution order ..................................................................... 66
Figure 46: Mirrored Matrix - step 1 .............................................................................. 77
Figure 47: Mirrored Matrix – Selection of dimensions .................................................. 78
Figure 48: Mirrored Matrix - selection of Codes .............................................................. 79
Figure 49: Mirrored extraction - Spreadsheet ................................................................ 80
Figure 50: Batch Extraction confirmation dialog ............................................................ 81
Figure 51: Batch extraction confirmation dialog for the user with Production role ......... 82
Figure 51: Maximum Batch extractions .......................................................................... 83
Figure 53: The list of batch jobs executed by the central site. ........................................ 84
Figure 54: The On-Line extraction confirmation window ............................................... 86
Figure 55: The Spreadsheet .......................................................................................... 87
Figure 56: Completed extractions, prints and Dump window .......................................... 90
Figure 57: The spreadsheet and its contextual menu ...................................................... 95
Figure 58: Filtering codes of a dimension of the spreadsheet ......................................... 100
Figure 59: The Extraction context menu ....................................................................... 101
Figure 60: The indicators format definition .................................................................... 102
Figure 61: Save from Spreadsheet window .................................................................... 103
Figure 62: Data Editing - Virtual dimension definition .................................................. 104
Figure 63: The spreadsheet in Edit mode - 1 ................................................................. 105
Figure 64: The spreadsheet in Edit mode - 2 ................................................................. 106
Figure 65: Tabular View parameters window .................................................................. 107
Figure 66: Product dimension dialog ............................................................................ 110
Figure 67: Save Userlist / Aggregate ............................................................................ 111
Figure 68: Insert Userlist .............................................................................................. 112
Figure 69: Insert Aggregate ......................................................................................... 113
Figure 70: Defining a label of a Userlist ....................................................................... 114
Figure 71: The Insert Implicit dialog .............................................................................. 116
Figure 72: Insert / Insert New Implicit object dialog using the Pattern ......................... 118
Figure 111: The Output Object selection window ................................................................. 176
Figure 112: setting the parameters of a print ................................................................. 177
Figure 113: setting the parameters of a dump ............................................................... 178
Figure 114: The "Generate Table" configuration window .................................................. 180
Figure 115: The "Out Data" configuration window ............................................................. 183
Figure 115: List of available format under the Out Data option ........................................ 185
Figure 117: Parameters dialog for the SDF extraction ....................................................... 186
Figure 118: The "Graph" configuration window ............................................................... 192
Figure 119: The "Map" configuration window ................................................................. 196
Figure 120: The "XTTable" configuration window ............................................................. 197