XOasis 4.6 manual

XOasis 4.6 manual

1. Getting started

1.1 Files needed in addition to science exposures

At least **once during the run** (but it is safer to get several), you should have obtained bias files :
- One for the spectrographic frames, which is a full frame, unbinned one. It should be a median of at least 20 (50 is better) biases. But as long as LORAL3 exhibits the offset problem, you may have preferred to come back home with N individual biases. If this is the case, check them with the Check Overscan tool, then use the Build bias function on each of them, then median them outside XOasis (as this function is not yet provided in the current release) to obtain the final median bias file. But if you perform on Mauna Kea a median over, say, 40 or 50 individual biases, you would have to be very unlucky to get a polluted median.
- One for the imagery frames, which is a full frame, 4x4 binned one. It should be a median of at least 20 biases.

For each scenario, you should have obtained, with the telescope in zenithal position, with no beamsteerer rotation in between, and in time-adjacent periods :
- A micropupil exposure.
- A very high signal-to-noise ratio GUMBALL white light spectroscopic flat field frame, which must be a median of at least 10 exposures (or a set of at least 10 exposures, not medianed (see the bias paragraph above, relative to LORAL3 issues).
- A GUMBALL spectral lamp calibration exposure, associated with the preceding one.
- A high signal-to-noise ratio dawn or dusk skylight spectroscopic flat field frame. Sky must be quite bright to get a good SN within a reasonable time; remember the spatial sampling...
- A GUMBALL spectral lamp calibration exposure, associated with the preceding one.

For each scenario, you should have obtained at least one spectrographic exposure of a spectrophotometric standard star. It is much better to obtain several exposures at various zenithal distances.

For each spectroscopic exposure, you should have obtained an associated spectroscopic GUMBALL spectral lamp calibration exposure without moving the telescope, and without rotating the beamsteerer between the two.

1.2 Preparing the scenario's data file set

Use some standard UNIX tool to download, from the CFHT data save tape, the set of files needed to reduce this scenario's data :
- All the science imagery exposures (object, stars for seeing evaluation).
- All the science spectroscopy exposures (object, standard star(s)) pertaining to this scenario, each one with its associated wavelength calibration exposure.
- The micropupil exposure from this scenario.

- The GUMBALL flat field exposure from this scenario, with its associated wavelength calibration exposure.
- The dawn or dusk sky flat field exposure from this scenario, with its associated wavelength calibration exposure.
- The two bias frames, for spectroscopy and imagery exposures.

Note that **separate directories must be used for various scenarii**. It may even be a good idea, if you reduce data for several objects observed with several configurations, to separate completely the service files and the objects files. This is an example of a directory structure which may prove to be quite handy (forget the 3C numbers, chosen at random!) :
- Radiogal98 : Father directory, with the following sub-directories :
- CCD : All the basic CCD files (raw bias frames and final biases)
- mr2f8 : All the files (from raw frames to datacubes and correction spectra) to be used to reduce objects observed with configuration MR2 f/8. The standard stars, sky and Gumball flats, flux correction curves, etc... are there.
- mr4f20 : All the files (from raw frames to datacubes and correction spectra) to be used to reduce objects observed with configuration MR4 f/20 (AOB). The standard stars, sky and Gumball flats, flux correction curves, etc... are there.
- 3C248mr2f8 : The object files (from raw frames to final merged datacubes) of 3C248 observed in this configuration.
- 3C253mr2f8 : The object files (from raw frames to final merged datacubes) of 3C253 observed in this configuration.
- 3C285mr4f20 : The object files (from raw frames to final merged datacubes) of 3C285 observed in this configuration.

It is easy, in XOasis, to navigate in the directory structure, and so to pick up the configuration files, when they are needed, in a parallel branch. More, after you point once to, say, the flux correction curve, XOasis remembers the full path, and you no longer have to bother about that, as long as you stay in the same work directory.

IMPORTANT WARNING : in the present release (4.5) it is mandatory to have the preprocessed calibration frame associated with the continuum exposure used to create the extraction mask (ouch...) visible from every directory where data are to be reduced which belong to the same configuration. This is because this frame is used as a reference for the wavelength calibration. This is not nice, and will be corrected in futur releases. Just *link* the original file to every work directory concerned. For example, in the above example of directory structure, assuming the preprocessed calibration frame of the MR2 f/8 continuum is p438445c.fits :
cd ...../3c248mr2f8
ln ../mr2f8/p438445c.fits .
cd ../3C253mr2f8
ln ../mr2f8/p438445c.fits .
And so on...
And remember not to compress inadvertently this frame before all the wavelength calibrations are done !

1.3 Running the program

Go to the work directory of your choice, and start the reduction program by typing :
XOasis

See the installation instructions (readme file) to know how to make XOasis executable on your system.

1.4 Setting the general options

Click on **Settings** then on **Preferences** in the main horizontal menu. A window pops up, which offers you to modify ten classes of parameters :
File format
File overwrite
Debug
History
Misc
Log
Assistant
User
Help
Windows

All these possibilities are fully described in the Settings section of this manual.

1.5 Observation logbook management

This tool is provided to take advantage of original CFHT file information to ease the reduction process. Click on **Tools** in the main menu bar, then on **Observation Log Book**. In the new window, all the FITS files present in the current directory are listed, with their main characteristics (identifier, type, integration time, alpha, delta, airmass, and so on...). Please note that file names can be **Activated** (left click anywhere on their line in the display, the line is now displayed on a light blue background), then **Dragged** and **Dropped** in any of XOasis file name input fields (with the left mouse button kept depressed). By clicking the middle button on an activated file, you get a small pop-up menu which displays first the complete name of the file, and offers you to :
View : the image is displayed using RTD.
Info : as you expect...
Descriptors : the image descriptors, written into the file, can be read or modified (be careful...).
Top : the display is shifted to the beginning of the list.
Bottom : the display is shifted to the end of the list.

The main menu offers three possibilities :

File
Load Logbook :
Build Logbook : This has to be done at the very beginning of the reduction process, once the original files have been copied to the work directory, before preprocessing them. All the FITS files present in the current directory are listed, with their main characteristics (identifier, type, integration time, alpha, delta, airmass, and so on...). This reminds the user of the content of every file he brought back from the OASIS run.
Print Logbook : Does what you expect, if you correctly filled in the [Settings/Preference/Misc] print command] field...
Close Logbook : The Observation Logbook management window is closed; if you reopen it later, you will retrieve the Logbook as it was at closure time.
Patch all FITS files : This must be used if you are to reduce data obtained before December 1998; since then, an inproper keyword writing on original CFHT files have been corrected, and this function is no longer useful. Before, it is mandatory !

Selection
In this window, the *selection* of a file is made by checking the button in the [S] column, at the beginning of the line, or by using mass-selection functions :.
Select all
Deselect all
Invert selection

Actions
Download from tape : Not yet implemented...
On selected files : Once some files have been selected, the following functions will do what you expect...
Move
Copy
Delete
Compress : using the program you specified with the Settings function. In my opinion, this is definitely NOT a well-tested feature of XOasis :-(
Uncompress : same remark as above.
Display options are set up with :
Scenarii
You may choose here to list all the frames pertaining to any of the sceanrios, or to restrict the display to some specific scenario(s).
Exposure types
In the same way, you may choose here to list al the classes of exposures, ore only a subset.
Comment for the current exposure
Enter here any text you want to be associated with a particular frame. This frame must be first activated by clicking on its line in the display window (the line is then written on a

light blue background); the comment will be displayed each time the file will be activated.

1.6 User's reduction logbook management

Click on [Log] in the horizontal menu. A window pops up, which offers you to edit a logbook :
You may enter any comment you need to be recorded, and use the usual editing commands on any block of text.
In addition, if you have checked that option in the Log section of the [Settings] function (see above), an ASCII report of every command you apply to your data is continuously appended to the reduction logbook file.
Lastly, at the end of each reduction step, you may record an ASCII summary of the results in the logbook.
Just below the main menu bar, appears a set of buttons (refering to the HTML text profiling syntax) that can be used to format your text.

The main commands from the **File** pull-down menu are :
Open a **new** Reduction Logbook : used to start a new logfile, to be saved under some another name, or to flush the entire contents of the present logbook, by keeping the same name when saving.
Open file (open) an already existing Reduction Logbook.
Update the Reduction Logbook : the contents of the file is replaced by the current contents of the editing window.
Save the Reduction Logbook (under the same name).
Save as the Reduction Logbook as... (under a new name).
Print File : print the Reduction Logbook (what you expexted, if you correctly filled in the [Setup/Misc/Local print command] field...)
Quit or Close window the Reduction Logbook editing; before leaving, the user is offered to save or not the current contents of the editing window.

1.7 Reduction folder : processed files management

Principles :

This tool is provided to maintain an (hopefully) clear list of the already processed files, their nature, their relationships. Click on [**Tools**] in the main menu. then on [**Reduction Folder**].
Note first that, as in the Observation logbook, you may obtain information about, display, and so on... or drag and drop an activated file; read the Observation logbook paragraph for details. It is possible to perform multiple-selection *à la Windows* : click on a file, then shift-click on another; all the in-between files are selected, too.
At the very beginning of the reduction, only the original CFHT files are shown, with minimal information : identifier, type, file name.

Every time you process a file, the display is automatically refreshed; use [**File**]/[**Scan Directory**] to force an update of the display.
The files are displayed arranged into folders. For instance, all the datacubes and reconstructed images or spectra which derive from the same raw frame remain in the same folder, whose head is the mother table created at spectrum extraction. Folders maybe opened or closed at will : click on the [+] or [-] sign to the left of the folder icon to open or close the folder. Once open, the folders show the files identified by their name, and by a specific icon telling which kind of data they hold.

Menus :
File :
Scan directory : refreshes the display, including the newly created files.
Close folder : closes the window.

Selection
In this window, the *selection* of a file is made by checking the button in the [S] column, at the beginning of the line, or by using mass-selection functions :.
Select all
Unselect all
Invert selection

Actions
On selected files : Once some files have been selected, the following functions will do what you expect...
Move
Copy
Delete
Compress
Join to catalog
This is a very useful function. If you select a number of files by checking their [S.] button, at the very left of the reduction folder's lines, and then use this [Join to catalog] function, a new window pops up. You may then either select an existing catalog file (ending in *.cat*), or give a name for a new one in the [Selection] field, then click on [Accept]. The files you selected are then appended to this catalog. The nice point is that the name of the catalog can now be given in place of the name of the [Input frame] in the [Preprocess], etc... windows, and all the files in the catalog are processed in a row.

Display options are set up with :
Classes
You may choose here to list all the frames pertaining to any of the scenarios, or to restrict the display to some specific scenario(s).
File types
In the same way, you may choose here to list all the types of exposures, or only a subset.

2. A TUTORIAL :

2.1 Getting the data ready

You have a set of files, obtained while observing the famous Messier 105 object, with the MR3 spectral configuration; they all pertain to the same scenario, which you wisely christened M105MR3. Note that it is important to **keep each scenario file set into a separate directory**. Naming the work directory from the scenario to be reduced would be good practice; so, *m105mr3* would be a nice choice.

To start with, let us suppose you just downloaded the following CFHT files to the *.../m105mr3/* work directory :

999001B.fits	Bias frame (median of N)
999100c.fits	Micropupil exposure frame
999200F.fits	GUMBALL continuum lamp exposure frame ("Flat", median of N)
999201c.fits	GUMBALL neon spectral lamp exposure frame associated with the above GUMBALL Flat exposure
999300o.fits	Messier 105 object exposure frame
999301c.fits	GUMBALL neon spectral lamp exposure frame associated with the above Messier 105 exposure
999400o.fits	HD88888 standard star exposure frame
999401c.fits	GUMBALL neon spectral lamp exposure frame associated with the above HD88888 standard star exposure
999500c.fits	Dawn or dusk sky exposure frame
999501c.fits	GUMBALL neon spectral lamp exposure frame associated with the above sky exposure

If you did not get an already medianed bias file, but a collection of individual frames, see the note at the end of the Check overscan section below.
You are now willing to go from this set of raw data files to a set of one thousand and one hundred Messier 105 spectra, wavelength- and flux-calibrated, don't you ?.. Easy ! Well, almost...
Read first the Getting Started section of this manual, to understand the various tools provided, specially the various logbooks and file management tools.

Then, go to directory *.../m105mr3/,* and type : **XOasis**. When the XOasis interface is displayed :

In the main menu, click on **Setings**, then on **Preferences** and set any option which you may think useful for you. See the Settings section of chapter **Getting started** to know everything about that.
In the main menu, click on **Tools**, then on **Observation logbook**. In the new window, click on **Files**, then on **Build logbook**. Once the display is updated, if your data files have been obtained before the end of 1998, click on **Files**, then on **Patch fits files**. This is mandatory IF you are to reduce data obtained before December 1998, as the native CFHT files were then missing some essential header information; this has since been fixed. It is useless (but harmless) to patch later files. You may go to the Observation logbook section of the **Getting started** chapter to know more about the Observation logbook; but at this point, this is not essential.
You may now close the Observation logbook, which will no longer be used.
From the main menu, choose **Tools**, then **Reduction folder**. All the files in directory *m105mr3* are listed, with their identifications, file names, and file classes. See the Reduction folder to know more about that tool.
From the main menu, choose **Tools**, then **Reduction logbook**. You may save there various messages during the reduction. See the Reduction logbook section of the **Getting started** chapter to learn more about that.

2.2 Check overscan zone on the raw CCD frame

As described in the Check overscan help section, some LORAL3 frames suffer from a non-uniform offset. This may be cured here, or you may as well skip the present step if you have good reasons to trust the LORAL3 controller (maybe one of your relatives)... If you do not, click on **CCD** in the main menu, then on **Offset**, then on **Check overscan**. In the new window, enter the name of a raw frame, for instance 999001B.fits. Click on **View strip average**, and a graphics window opens, showing you the [x] average of the overscan strip. If you think this mean will be fitted correctly by a polynomial, check the **Polynomial fit** option as **Correction method**; it should be already checked, as this is the "normal" default. If the mean looks like a staircase, check **Median filter** instead. Click on **Accept**. Your choice is now recorded in the file header, for later use. Repeat the same operation for all the raw files, from 999100c.fits to 999401c.fits, in the directory.

Biases : CFH medians or not ?
As long as LORAL3 exhibits the offset problem, you may prefer to get back home with N individual biases, check them with the present tool, then use the **Build bias** function described just below on each of them, then median them outside XOasis (as this function is not yet provided in the current release) to obtain the final median bias file. But if you perform on Mauna Kea a median over, say, 40 or 50 individual biases, you would have to be very unlucky to get a polluted median.
But : we encountered some data encoding problem on CFHT (that is IRAF in fact) medians, with image values being strangely quantified in the integer-to-floating point conversion at the end of the median process. One way of going around this is to convert, from within XOasis, the median bias file to some non-FITS format (MIDAS for instance), then back to FITS format, using the **Convert**, **Convert tools** function. In the

process, the image data is correctly converted to floating point format, and some descriptors are set to the right value. You may delete the intermediate MIDAS file after completion. This procedure is strongly recommended if you wish to use CFHT medians.

In the following text, we shall suppose the raw bias is named 999001B.fits, regardless of its past history, double-converted or not.
A complete description of this process can be found in the Preprocessing section of the manual.

2.3 Build the bias frame

First, read carefully the last paragraph of the preceding section...
Click on **CCD** in the main menu, then on **Build bias**. Enter the name of the raw bias frame (which should have been checked for offset characteristics, as described in the Check overscan zone section), here 999001B.fits, then click on **Accept**. A new file named B999001B.fits is created, which is 999001B.fits subtracted from its offset value, and clipped off from the overscan strip.
A complete description of this process can be found in the Preprocessing section of the manual.

2.4 Build the high frequency flat field frame

This step is to be performed only if you have at hand a continuum frame which is a high signal-to-noise ratio median. As far as we know, nobody has yet obtained such a marvel. So, this step is skipped in this simple cook book.
A complete description of this process can be found in the Preprocessing section of the manual.

2.5 Preprocess the raw CCD frame

In this step, you clip the CCD frames from the unused "offset band", and correct from the electronic constants added to the signal by the readout electronics (the "offset", the "bias"). For that, click on **CCD** in the main menu, then on **Preprocess**. In the new window, enter B999001B.fits as the bias file name, and, to start with, 999100c.fits as the input frame. Click **Accept**. A new file is created : p999100c.fits, displayed as a *Preprocessed micropupil frame* in the **Reduction folder**. Preprocess in the same way the other non-bias files, creating :
p999200F.fits (*Preprocessed continuum lamp frame*),
p999201c.fits (*Preprocessed wavelength calibration frame*),
p999300o.fits (*Preprocessed object frame*),
p999301c.fits (*Preprocessed wavelength calibration frame*).
p999400o.fits (*Preprocessed object frame*),
p999401c.fits (*Preprocessed wavelength calibration frame*).

p999500c.fits (*Preprocessed sky frame*),
p999501c.fits (*Preprocessed wavelength calibration frame*).
Later on, you will learn to use the [Catalog] function, which eases mass work. More on this point in the Reduction folder section of this manual.
A warning : The preprocessed wavelength calibration frame associated with the continuum lamp frame (p999201c.fits in our example) is to be kept until ALL the object frames are lambda-calibrated. It is used to calculate, for each of these frames, any residual shift to be taken into account before calibrating. More, it should be accessible from the work directory. If you work with an elaborate directory structure, one for each object/scenario, one for service exposures, and so on (a good idea, to keep your reduction folder readable), this can be easily made outside XOasis with a simple link like :
ln {access path}/p999201c.fits ./p999201c.fits, after going (cd) to the place where the object frames are; the same can be done with the (built) bias frame.
A complete description of this process can be found in the Preprocessing section of the manual.

2.6 Find the positions of the spectra "centers" and the spectra "ridges" on the CCD

You are now about to create the information which will allow the retrieval of the spectral data on the CCD frame. You have first to find the positions on the CCD of the undeviated rays originating from all the microlenses, grism removed. This is done on the micropupil frame, using the **Mask** function of the main menu, then the **Search lenses** subfunction. In the new window, input the name of the *Preprocessed micropupil frame* : p999100c.fits, and the name you like for the future extraction mask (it is a "table", that is a file where data are arranged in rows and columns, not an image, that is the reason for the **Output mask table** request). As the scenario is M105MR3, you may choose M105MR3mask as a name, and the table full name will be M105MR3mask.fits.
A complete description of this process can be found in the Mask section of the manual.
Find the positions of the spectra ridges on the CCD
One step further : you will now find the lines along which all the spectra are stretched on the CCD, and their limits. For that, use the **Mask** function of the main menu, then the **Find maxima** subfunction. Enter the *Preprocessed continuum frame* p999200F.fits as **Input contiunuum frame**, and set the **Output maxima file** as M105MR3max, for instance. Click on **Accept**, and a file M105MR3max.max will be created to record the results.
A complete description of this process can be found in the Extraction mask section of the manual.

2.7 Create the spectra extraction mask

You have now at hand all the information to complete the extraction mask creation. For that, use the **Mask** function of the main menu, then the **Create mask** subfunction. Enter the **Maxima file** name (M105MR3max.max), the **Calibration frame**, which is the

GUMBALL spectral lamp frame associated with the GUMBALL continuum lamp frame just used to find the spectra ridges (the *Preprocessed continuum frame* p999200F.fits). The **Reference wavelength table** is a special data file located somewhere within the directory structure of XOasis; just use the browse folder icon at the end of the input field, and choose the file of the element you used in the spectral lamp illumination. For instance, here, Neon_ref.fits. Enter the name of the **Mask table** already created (M105MR3mask.fits). Click on **Accept**. This last file is updated (may be quite long on slow systems), and the extraction mask is now ready.
A complete description of this process can be found in the Extraction mask section of the manual.

2.8 Extract the spectra from the CCD frame

It is now time to obtain the set of one thousand and one hundred M105 spectra; this is called **extracting the spectra**. Click on **Extract** in the main menu, then on **Extract spectra**. Give the **Input frame** name, that is the name of the frame from which you are going to extract spectra, the **Input calibration frame** name, which is the name of the calibration frame associated with the **Input frame**. For instance, here, the input frame maybe p999200F.fits, and the associated calibration frame p999201c.fits, if you want to extract the continuum spectra. They may be p999300o.fits and p999301c.fits, to extract the Messier 105 spectra, p999400o.fits and p999401c.fits to extract the HD88888 spectra, p999500c.fits and p999501c.fits to extract the sky spectra. And it may be p999201c.fits and p999201c.fits (twice the same name) if you want to extract the spectra from the calibration frame, which should be considered as associated with itself. Same thing with p999301c.fits and p999301c.fits, p999401c.fits and p999401c.fits, p999501c.fits and p999501c.fits. In fact, the eight extractions must be conducted. Give the name of the **Extraction mask table**, which has just been constructed during the previous step : M105MR3mask.fits. Give the name you like to the spectra set produced; it may be *continuum* for the continuum spectra file, *M105MR3* for the Messier 105 spectra file, *HD88888* for the standard star file, *sky* for the sky file, *cont_neon* for the calibration associated with the continuum, *M105_neon* for the calibration associated with the M105 exposure, *HD_neon* for the calibration associated with the HD88888 exposure, and *sky_neon* for the calibartion associated with the sky exposure, for instance. Keep the **Summation full width** at the default value (5), uncheck the **Optimal extraction** option (better but slower) and click on **Accept**. The files holding the resulting spectra set are special TIGER files called **datacube**, as they contain (alpha,delta,lambda) information. They belong to four new classes : *Raw continuum datacube*, for continuum.tig, *Raw calibration datacube*, for cont_neon.tig, M105_neon.tig, HD_neon.tig and sky_neon.tig, *Raw object datacube*, for M105MR3.tig and HD88888.tig, and *Raw sky datacube* for sky.tig. Here, "raw" means "not yet fully wavelength calibrated". At the same time, auxiliary "mother" tables are created, one for each created datacube, bearing the same name but with extension .fits against .tig for the datacubes. They are displayed as *Table*, nothing more.
A complete description of this process can be found in the Extraction mask section of the manual.

2.9 Prepare the wavelength calibration coefficients

The just extracted spectra are already wavelength-calibrated to the first order. There remains only a second order calibration to be made to get the final calibrated spectra. The first step is to compute, for each spectra the numerical coefficients of this second-order correction. Click on **Wavelength** in the main menu, then on **Compute calibration**. Give the name of the **Associated calibration datacube**, that is the name of the *Raw calibration datacube* just obtained which is associated with the *Raw object datacube* (or *Raw continuum datacube*) you are going to calibrate. Here, it may be cont_neon.tig, if you plan to calibrate the continuum spectra set, or M105_neon.tig, if you plan to calibrate the Messier 105 spectra, or HD_neon.tig, if you plan to calibrate the HD88888 spectra, or sky_neon.tig if you plan to calibrate the sky spectra; bad luck, you must do all of them. Give the name of the **Input wavelength reference table**. This is a table with a single column containing all the wavelengths of the lines produced by the spectral lamp(s) used for the calibration exposures. Here, it may be the celebrated *Neon_ref.fits* table. A set of such tables is provided with XOasis, and you may use the browse icon at the end of the field to choose the right one. Keep the other parameters to their default value (remember that we are dealing with *second order* corrections) and click **Accept**. For each calibration The calibration coefficients are recorded into the mother tables created during the preceding step, ready for use in the next step; To keep trace of that, those tables change class, and each become a *Calibrated table*.
A complete description of this process can be found in the Wavelength calibration section of the manual.

2.10 Calibrate the spectra

It is now time to get the wavelength-calibrated spectra. Click on **Wavelength** in the main menu, then on **Calibrate spectra**. Give the name of the **Input datacube**, that is the spectra set to be calibrated; here, it may be continuum.tig, or M105MR3.tig, or HD88888.tig, or sky.tig. Give the name of the **Input calibration table**, that is the name of the calibrated table associated with the input datacube. Give the name of the **Output datacube**, that is the name of the resulting wavelength-calibrated datacube. It may be, here, continuum_lbda, or M105MR3_lbda, or HD88888_lbda, or sky_lbda, for instance. Keep the other parameters at their default values, and click **Accept**.
A complete description of this process can be found in the Wavelength calibration section of the manual.

2.11 Prepare the low frequency flat field reference spectra

The next step is the correction of the spatial variations of the spectral response, that is flat-fielding. For TIGER data, this is divided into two steps : the first one corrects for the high frequency variations, that is the CCD pixel-to-pixel variation, the second corrects for the low-frequency variations, that is the local spectral response variation. The first step has marginal effects on TIGER data, and is **not yet implemented**. The second is dealt

with here.
In the main menu, click on **Flat**, then on **Compute flat field**. Give the name of the **Input continuum datacube**, that is the wavelength-calibrated datacube holding the GUMBALL continuum spectra set. It is displayed as *Wavelength calib continuum datacube*; in our example, it is continuum_lbda.tig. Give the name of the **Input skyflat datacube**, that is the wavelength-calibrated datacube holding the dawn or dusk sky spectra set; in our example, sky_lbda.tig. Give the name you want for the **Output reference flat datacube**, that is the datacube which will be created to hold the low-frequency correction spectra set. Here, it may be FLAT_M105MR3, for instance. Let the other parameters as they are, click on **Accept**; this new datacube is created, and displayed in the Reduction folder as *Flat correction datacube.*
A complete description of this process can be found in the Low frequency flat-fielding section of the manual.

2.12 Apply the low frequency flat field spectra correction

In this step, all the spectra recorded into the object datacube are multiplied by their corresponding correction spectra stored into the just created flat datacube. In the main menu, click on **Flat**, then on **Apply flat field correction**. Give the name of the **Input datacube to be flat-fielded** (here, M105MR3_lbda, or HD88888_lbda; both must be corrected), of the **Input flat correction datacube** (just created, FLAT_M105MR3 in the present case), of the **Output datacube** to record the result; it may be M105MR3_flat or HD88888_flat in the present case. Click on **Accept**, and M105MR3_flat.tig or HD88888_flat.tig is created, and displayed as *Flat-fielded object datacube* in the Reduction folder.
A complete description of this process can be found in the Low frequency flat-fielding section of the manual.
Remove cosmic ray impacts from the spectra
Tiger data are affected by cosmic ray impacts, and correction is done after spectra extraction and calibration. Cosmic ray signatures are searched into the spectra, and separated from possible emission lines using spatial and spectral morphological arguments.
In the main menu. click on **Cosmics**, then on **Remove cosmics**. Give the **Input datacube** name, that is the name of the datacube to be cleaned from cosmic impacts (here, M105MR3_flat or HD88888_flat), and of the **Output datacube** which will hold the result (for instance, M105MR3_cosm or HD88888_cosm). Let the other parameters at their default values, and click on **Accept**. The datacube M105MR3_cosm.tig or HD88888_cosm.tig is created, and displayed as *Cosmics removed object datacube* in the Reduction folder.
A complete description of this process can be found in the Cosmic ray impacts removal section of the manual.

2.13 Obtain the sky spectrum

The sky spectrum can be retrieved from the object exposure using a statistical algorithm; this was how it was done with TIGER (before OASIS) data. This feature is provided in XOasis since release 4.2.
To obtain the night sky spectrum as recorded on exposure 999300o.fits (Messier 105), click on **Sky** in the main menu, then on **Compute sky spectrum**. Enter M105MR3_cosm.tig as the name of the **Input datacube**, and, for instance, skyM105MR3 as the name of the **Output sky spectrum**. Click then on **Accept**. Looks like magic... In fact, you will learn later how to restrict the area devoted to sky spectrum computation to some clean one; but the histogram algorithm used is able to find the sky spectrum even in a quite populated field. Let suppose M105 fills only, say, 20% of the field, and there is no point in bothering about clean areas for sky determination, in this tutorial... A complete description of this process can be found in the Sky spectrum subtraction section of the manual.

2.14 Apply the sky spectrum subtraction

In the main menu, click on **Sky**, then on **Apply sky subtraction**. Give the name of the **Input datacube**, which is M105MR3_cosm in our example, of the **Sky spectrum**, which is skyM105MR3, and of the **Output sky-subtracted datacube**, say M105MR3_sky. Click on **Accept**.
A complete description of this process can be found in the Sky spectrum subtraction section of the manual.

2.15 Prepare the absolute flux reference spectrum

One step remains to be completed : obtain the absolute outside atmosphere spectral flux of every spatial element. This is done using the observation of the flux standard star (HD88888 in our example). In the main menu, click on **Flux**. The **Copy reference table** sub-function allows you to import into your work directory a copy of one of the absolute star flux tables provided with XOasis. Use it if you observed one of these stars. If not, you have to provide a fits file of your own, and copy it into your work directory right now. Then, click again on **Flux**, then on **Compute throughput**. Give the name of the **Input reference star datacube**, which should be the cosmic-cleaned datacube (we suppose that the night sky spectrum is absolutely negligible on the short exposure frame of the standard star. Remember that each spectrum receive only the energy from a fraction of an arcsecond squared); here, HD88888_cosm. Give the name of the **Input flux reference table** we refer to at the beginning of this section. In this table, you must specify the **Wavelength column** and the **Flux column** names. In the tables provided with XOasis, they are *lambda* and *f_lambda*. Give the name of the **Output flux correction spectrum**, where the result will be recorded; it may be M105MR3flux_corr, for instance. Keep the **Spatial summation radius** at the default 3 arcsec value; it is the radius within which the standard star spectra will be summed up, and 3 arcsec seems enough for the

usual CFHT seeing. In the same way, the **Summation center** at 0,0 should be OK, as the standard star is supposed not to be wildly off-centered into the CCD frame. Click on **Accept**, and spectrum M105MR3flux_corr.fits will be created, and displayed as *Flux calibration spectrum* in the Reduction folder.
A complete description of this process can be found in the Flux calibration section of the manual.

2.16 Apply the absolute flux spectral calibration

Click on **Flux** in the main menu, then on **Apply flux correction**. Give the name of the **Input object datacube**, which should be the cosmic-cleaned object datacube obtained earlier. In our example, it will be M105MR3_sky. Give the name of the **Input flux correction spectrum** just computed (here, M105MR3flux_corr), and of the **Output flux-calibrated object datacube** to hold the final result. For instance, M105MR3_abs. Click on **Accept**, and the datacube M105MR3_abs.tig will be created, and displayed as *Flux calib object datacube*.
A complete description of this process can be found in the Flux calibration section of the manual.
That's all, folks...

2.17 More...

You have now at hand a set of one thousand and one hundred Messier 105 spectra, wavelength- and flux-calibrated. You may wish to SEE some physics. For that, use for instance the following possibilities :
Sum energy over a spectral interval
This feature is provided to allow the user to map the energy content of a datacube within a certain wavelength range. Click on **Analyse** in the main menu, then on **Integrate spectra**. Give the name of the **Input datacube**, that is the object datacube from which you plan to extract the data; here, it must be M105MR3_abs. Choose the **Spectral range**, either complete or within given limits. Choose the **Output** medium : either a table column, and the column (you have to give the name) will be created in the table associated at spectra extraction with the original datacube (table M105MR3.fits in this case), or an image, and you will be given the opportunity to display this image by using **View result**. Note that you may choose to compute, over the wavelength interval you specifiy, either the **Integral**, or the **Average**, or the **RMS dispersion** of the datacube.
A complete description of this process can be found in the Analyse section of the manual.
Average spectra over an aperture of any shape
This function allows you to compute the global (average) spectrum of an area of any shape within the observed field. It may be used as a side bonus of the sky subtraction function, and will be implemented cleanly in next releases. For the moment, let us compute the average spectrum of a circular zone within the observed field, Click on **Analyse**, then **Sum aperture**. Give the name of the **Input datacube**, for instance M105MR3_abs in the present case, of the **Output spectrum** where to put the sum of the

spectra of the given aperture. Enter the **Center** of the circular zone, and the **Radius**. You may choose between pixel coordinates (CCD) or alpha,delta coordinates (Sky) (sky coordinates are recorded in the associated table, they have been computed at spectra extraction time). Click **Accept**, and the sum spectrum is created. It may be displayed using the **Display**, **Spectrum or Table** function from the main menu.
A complete description of this process can be found in the Analyse section of the manual.
Separate the continuum and line components in the spectra
Click on **Analyse** in the main menu, then **Continuum subtraction**. Choose **Spectrum** as the object to be processed, and enter the name of the **Input datacube**. Here, it may be M105MR3_abs, as Messier 105 is a well-known Seyfert 4.23 galaxy. Give the name of the result datacube, for instance M105MR3_line. Let the polynomial degree which is supposed to fit the continuum at the default value of 7, choose **Subtract** for the **Operation** option, and click **Accept**. The resulting spectra, held into M105MR3_line.tig, may be displayed using the **Display**, **Spectrum or Table** function from the main menu.
A complete description of this process can be found in the Analyse section of the manual.
Compute any spectral quantity
This feature is *not yet implemented*, but will be part of some future (4.4?) release. It will allow to combine freely columns in the tables associated to datacubes to map quantities such as line widths, line ratios, line-to-continuum ratio, and so on, as it was done for years with TIGER data... You may use the reduction hotline to know the present status of the next release implementing this feature.
Reconstruct an image from any spectral quantity
In the main menu, click on **Image**, then **Reconstruct image**. Give the name of the **Input table** where is the data you want to map, and the name of the **Output image**. Choose **Sky** coordinates. Give the **Input independant column name**, that is the name of the column containing the quantity to be mapped. Click on **Accept**, and the image is created. It may be displayed using the **Display** function>
A complete description of this process can be found in the Image reconstruction section of the manual.

3. Preprocessing of raw CCD frames

3.1 Flip frames ? (EEV CCD ONLY)

Flip, if needed, frames around the center column (EEV CCD only)

Principles

Some CCDs like EEV at CFHT are read the opposite way than the old LORAL 3 CCD (for which the OASIS data reduction routines were designed). In this case, to retrieve the conventional optical layout, the raw frames are to be flipped around the centre column. If the FLIP FRAME flag is set, the frames will be flipped during preprocessing. The preprocessed frames will be stored the conventional way and a warning message displayed, to remember you the difference of storage between the raw and preprocessed frames.

Use

Click on [CCD] in the main menu, then on [Flip frames ?]. The [Flip frames ?] window pops up.
A message appears indicating in which conditions the flag is to be set. Read the conditions ans click on the radiobutton if conditions are yours.

3.2 Offset correction

Check frame overscan strip

Principles:

There have been some occurences of underlying offsets, in some image frames, showing a clear banded structure. It seems that this is a random LORAL3 controller issue. A display/diagnosis tool is provided here to check the smoothness of the offset of any image, and set the offset correction method accordingly.
This is done by column-averaging a strip situated in the CCD overscan zone; it has been found that the strips polluting the offset are parallel to CCD lines, and extend all the way through the CCD including the overscan zone. The average spectrum N = N(y) of the overscan strip is thus a good image of the underlying offset [y] profile.
According to the aspect of this overscan strip spectrum, the user has to choose between two offset correction methods : *Polynomial fit* for smooth shape spectrum (that is sloping, bent, but not stepped; often present, but to a very low level), *Median filtering* for a stepped spectrum (yes, it happens; the biggest step we have ever seen was approximately 2-3 counts high, versus a 1152 counts regular offset level...); The default setting is *polynomial fit*.

Use

Click on [CCD] in the main menu, then on [Check frame overscan]. The [Check frame overscan] window pops up.
Enter the name of the frame to be checked. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
The resulting [y] offset spectrum of the frame is displayed.
Options description
Overscan limits :
The user has to choose here the [x] limits of the overscan zone to be used for the evaluation, that is the left [X1] and right [X2] limits of the sub-frame. The full height is used, and the default [X1,X2] limits are [2060,2080].
Offset correction method :
Choose between **Polynomial fit** and **Median filter** for the smoothing method you want to check on the offset [y] spectrum.
Save values :
All the input values (files names, coordinates) are saved, and become the new default values for the current window. They can be retrieved, and used each time the *Check frame offset* job window is opened.
Recall values :
The values (files names, coordinates) saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to pre-defined defaults values; for instance, the file names are reset to blank.

Set defaults

Principles

This function is used to set :
the default method to be used to smooth the [y] offset spectrum of the currently preprocessed frame. Use the [Check frame overscan] function described above to choose between polynomial fit and median filter, or let things as they are (polynomial fit) if you trust blindly the CCD controller electronics; not always wise with the present LORAL3.
the default x limits of the overscan zone used to compute the [y] offset spectrum. The default defaults are X1=2060, X2=2080, which is usually OK.
The present defaults will be used for all subsequent frame preprocesses.

Use

Click on **CCD** in the main menu, then on **Offset**, then on **Set defaults**. The [Set defaults] window pops up.
Check the method you prefer (see *Check frame overscan* section for details, and enter the limits of the overscan zone to be used (see same preceding section for details). You must have 2049 <= X1 < X2 <= 2088.

Options description
Save values :
All the input values are saved, and become the new default values for the current window. They can be retrieved, and are used each time the [Set defaults] window is opened.
Recall values :
The values saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the method is set to *Polynomial fit* and the overscan limits are set to [2060,2088].

3.3 Build bias frame

Principles

The [y] offset spectrum is evaluated over the [X1,X2] overscan zone strip (see section Offset correction above).
This spectrum is smoothed using the method set in the same preceding section.
The smoothed spectrum is subtracted from every [1...2048] column of the raw bias frame. The resulting frame is clipped from the [2049,2088] overscan strip. The new frame is named B{*original bias name*}.

Use

Click on [CCD] in the main menu, then on [Build bias].
In the [Build bias] window which pops up, enter the name of the raw bias frame. It should have been already checked using the [Offset correction] function, subfunction [Check frame overscan], which allows the user to set the offset correction method; if not, the default (*Polynomial fit*) method is used. Regarding the name of the bias frame, you can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder...
We encountered some data encoding problem on CFHT (that is IRAF in fact) medians, with image values being strangely quantified in the integer-to-floating point conversion at the end of the median process. One way of going around this is to convert, from within XOasis, the median bias file to some non-FITS format (MIDAS for instance), then back to FITS format, using the **Import/Export**, **File format conversion** function. In the process, the image data is correctly converted to floating point format, and some descriptors are set to the right value. You may delete the intermediate MIDAS file after completion. This procedure is strongly recommended if you wish to use CFHT medians.

Options description
Debug :
No action; forget it...
Save values :
All the input values (file name, coordinates) are saved, and become the new default

values for this user. They can be recalled at will, and are used each time the [Build bias] window is opened.
Recall values :
The values (file name, coordinates) saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file name is set to blank.

3.4 Preprocess of spectrographic exposures (bias and dark subtraction, frame clipping)

Principles

The offset is subtracted from the input frame. For that :
The [y] offset spectrum of the object frame to be processed is obtained from the [X1,X2] overscan strip.
This spectrum is smoothed using the method set in the Offset correction preceeding section, subfunction [Check frame overscan]. If this is not set, the default (*Polynomial fit* method is used.
The smoothed spectrum is subtracted from every [1...2048] column of the raw object frame.
The bias is subtracted from the offset-corrected object frame. The bias used is the one which has been offset-corrected in the Build bias frame section. If no bias frame is available, enter *none* as the bias frame name and this step is skipped.
The frame is clipped according to the limits given.
The frame is dark current-subtracted, that is the specific dark current constant of the CCD used, scaled to the integration time achieved, is subtracted from the frame. Observers are strongly advised to obtain series of dark frames of integration times similar to those of their scientific exposures.

Use

Click on [CCD] in the main menu, then on [Preprocess]. The [Preprocess] window pops up.
Enter the name(s) of the [Frame to be preprocessed], of the [bias frame], of the high frequency [Flat frame] (only for image frames). You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.

Options description
Clip & Bias selected means that the input frame will be clipped according to the values given in the four fields to the right of the window, at least to get rid of the offset strip. This will be done after the bias correction.
Dark selected means that you want your frame to be corrected from CCD dark signal. If the frame to be processed has not yet been clipped and de-biased, it will be before beeing

dark-subtracted. The dark is taken as a scalar constant, as it is extremely low (0.85 e.hour^{-1} for instance for the LORAL3 CCD), to avoid introducing noise into the reduction process; medians of several tens of one-hour dark's would of course be better, but are clearly unobtainable. The value used is shown in the [Dark current] input field, and should not be changed unless you get some very good reason to do so.

Debug :

This switches the program to verbose mode, and more informations are recorded into the history file (see section [Start] in the left reduction menu).

Clipping :

This gives the lower left (X1,Y1) and upper right (X2,Y2) corners of the frame area which will be kept after clipping. Some do not trust the edge pixels of the CCD, and like to clip off a few pixels on three edges, and of course the offset strip on the fourth. Default values are for a 2048x2048 frame.

Save values :

All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Preprocess] window is opened.

Recall values :

The values (files names, coordinates) saved by the user, are loaded to the various input fields.

Default values :

The input fields are set to the general defaults values; for instance, the file names are set to blank.

3.5 Compute CCD fringing correction (EEV CCD ONLY)

Principles

In the red, (typically beyond 6300 A), the EEV CCD exhibits fringes. Their amplitude increases as we go redward and **we recommend to use the LORAL 3 CCD (not affected by fringing) for all spectral configurations beyond 7000 A (e.g. MR3).** For spectral configurations MR2, HR3 and HR4 the EEV CCD can be used but one has to correct the exposures for CCD fringing.

The CCD fringing correction frame is obtained from a preprocessed continuum frame (e.g. the one that will be used later on to build the mask and compute the falt-field correction). The contrast of the fringes can be relatively low, **we therefore recommend to use a continuum frame which is the median of at least 10 individual continuum frames**. The procedure used to compute the correctin frame is the following :

The preprocessed continuum frame is scanned column after column.

Each column is fitted using a spline. The smoothness of the fit must be tuned to fit only the low to medium frequencies (i.e. the oscillations due to the fringing should not be fitted).

Each column is then divided by the low/medium frequency spline fit, and the result (which, ideally, should contain only the oscillations due to the fringes) is used to build the CCD fringing correction frame.

Use

Click on [CCD] in the main menu, then on [Compute CCD fringing correction].
In the [Compute CCD fringing correction] window which pops up, enter the name of the [Preprocessed continuum frame] (input) and the name of the [CCD fringing correction frame] (output). You may type in the names, or use the browse icon at the end of the field, or drag and drop them from the Reduction folder.
In debug mode (check the [Debug mode] button), adjust the [Smoothness parameter] to make sure that the oscillations due to CCD fringing are NOT fitted. To do that, type a value for the [Smoothness parameter] (typically between 0.5 and 5), click on the [Accept] button. Once the program task is finished (answer yes if you are asked the authorization to overwrite debug frames like dbg_spline.fits), click on [View results]. A graphic window will pop up showing you the intensity along the column specified in [Column number] (defaulted to #4) and the spline fit (see figure below).

N/A

The next figure shows a zoom on a section of the graph displaying the oscillations due to the fringing (HR4 configuration, red line) and the low/medium frequency spline fit (blue line).

N/A

Once you are happy with the value of the [Smoothness parameter], uncheck the [Debug mode] button and click [Accept] to launch the full-scale computation of the CCD fringing correction frame. You can get an idea of what to expect for the correction frame by looking at the image below, which is a zoom on a small region of a CCD fringing correction frame (HR4 configuration).

N/A

Options description
Smoothing parameter :
This value controls the smoothness of the spline fit. The default is 3, which means that two consecutive nodes are separated by at least 3 pixels.
Debug :
If you check this option, computations are done on a single column, which the user specifies in the [Column number] input field; this may be used to check the dependency of the results on the smoothing parameter value.
Save values :
All the input values (file name, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Build bias] window is opened.
Recall values :
The values (file name, coordinates) saved by the user, are loaded to the various input fields.

Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

3.6 Apply CCD fringing correction (EEV CCD ONLY)

Principles

Apply the CCD fringing correction to a preprocessed spectrographic exposure.
Due to the non-repeatability of the wedge positionning, the position of the spectra along the y-axis can change from one spectrographic exposure to the other. The fringes will follow the spectra, it is therefore necessary to compute the shift between the exposure to be corrected and the continuum exposure which has been used to build the CCD fringing correction frame. This is done using the wavelength calibration exposures associated to the two spectrographic frames. We first compute the correlation of the two wavelength calibration frames. A Gaussian is then fitted on the correlation image to get the location of the peak, which traces the shift between the two frames.
The CCD fringing correction frame is shifted accordingly. The correction is then applied by dividing the input frame by the shifted CCD fringing correction frame.

Use

Click on [CCD] in the main menu, then on [Apply CCD fringing correction].
In the [Apply CCD fringing correction] window which pops up, enter the name of the [Input frame to be corrected], of its associated calibration frame ([Calibration frame for input frame]), of the CCD fringing correction frame and of its associated calibration frame [Calibration frame for correction frame]. You may type in the names, or use the browse icon at the end of the field, or drag and drop them from the Reduction folder.
Click on the [Accept] button.
In the program output, ALWAYS check the derived value of the shift between the input and correction frames. The value along the x-axis should be lower than 1 pixel. If the value of along the y-axis is close to the [Fit window radius] along the y-axis (defaulted to 20), you may want to either to move the center of the fit window (Fit Window Center] parameter box) or increase the size of the fit window ([Fit Window Radius] parameter box). In case of problem, run the program in Debug mode (check the [Debug mode] button) and take a look at the dbg_correlation.fits (correlation between the two calibration frames, center is at pixel 127,127) and dbg_fitwindow.fits (subset of the correlation image used for the Gaussian fitting of the correlation peak) frames. It may help you to determine the actual position of the correlation peak and to adjust the fit window parameters accordingly.

Options description
Fit window radius :
The location of the peak in the 256x256 correlation frame is computed using a Gaussian fitting. Due to the presence of multiple sub-peaks in the correlation image, it is necessary

to perform the fit on a subframe. The [Fit window radius] parameter can be used to change the default values (10,20, i.e. 10 along the x-axis and 20 along the y-axis) for the size of this subframe, either along the x-axis or the y-axis.

Fit window center :

In addition to changing the size of the window for the Gaussian fitting (see above), it is possible to change the position of this window. A fit window center of 0,0 (default values) corresponds to the "no shift" position (pixel 127,127 in the correlation window).

Save values :

All the input values (file name, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Build bias] window is opened.

Recall values :

The values (file name, coordinates) saved by the user, are loaded to the various input fields.

Default values :

The input fields are set to the general defaults values; for instance, the file names are set to blank.

4. Spectra extraction mask creation

4.1 Copy reference table

The wavelength reference tables are provided with XOasis, and are stored in a special directory. First use this function to make a work copy into you work directory, so that you will be able to try modifications without impairing basic data integrity.
The following files are provided, as of February 20001 :
Neon
Argon
Ar+Ne
Ne+Ar+Hg

4.2 Search the lenses positions

Principles

The special micropupil frame is used for that; it has been obtained with OASIS set to the spectroscopic configuration, but with the grism and the beam steerer removed from the optical path.
The background noise is computed, and a suitable low threshold is applied to the frame.
The micropupil images are detected, and a gaussian fit gives the center of each image.
The cross-dispersion profile of each spectrum is evaluated from the mean profile of the micro-pupils integrated in the dispersion direction and the local actual width of the profile.
A special algorithm is used to compute the lens array characteristics and the distorsion coefficients of the spectrograph optics.
The values obtained are stored in a new table, which is the **Mask table**, to be used later to extract the spectra, after it has been completed in forthcoming steps.

Use

Click on [Mask] in the main menu, then on [Search lenses].
Fill in the **Input micropupil frame** (find its name in the Observation Logbook) and **Output mask table** (use some nice clear significant name like {*scenario name*}mask or mask{*scenario name*}) input fields. You may want to use the "Browse folder" icons at the end of the zones, you may also drag and drop a file from the reduction folder.
Click **Accept** to start the process, **Cancel** to leave without any action.
The procedure should find 1125 +/- 5 lenses (displayed as *centers of classes*). The distortion center should be near [1050,1050] pixels and the distortion parameter should have a small value (something like 7.e-9). The calculated array step should be near 57 pixels and the lens position angle near +/- 6.6 degrees. Finally the RMS residual of the fit should be of the order of 1 pixel or less. If any of the displayed values are wildly different from this ones, check the input file and the resulting table. Note also that the [display

result] button will display the lenses positions : you can thus check for missing lenses if any.

Options description
Debug :
This switches the program to verbose mode, and more informations are recorded into the history file (see section [Start] in the left menu).

Action buttons
Save : All the input values (files names, coordinates) are saved, and become the new default values for this user.
Recall : The input fields are reset to the last saved values.
Default : The input fields are reset to the general defaults; for instance, the file names are set to blank.
Accept, Cancel, Help do what you think.

4.3 Find the maxima (spectra ridges)

Principles

A nice proprietary algorithm is used to take care of noise, emission lines, and so on... Transverse scans of the spectra pattern allow to trace the spectra crossings along each scan line.
A local (see preceding paragraph) profile fit is performed on each line to find the position of each spectrum ridge.

Use

Click on [Mask] in the main menu, then on [Find maxima].
In the window which pops up, fill in the **Input Continuum frame**, **Mask table**, and **Output maxima data file** zones; you may want to use the "browse folder" buttons at the end of the zones, you may drag and drop a file name from the reduction folder. One usually chooses a name which is a close relative of the one of the mask table, replacing the "mask" substring by "max".
Fill the **Line step** zone, if blanked. You may choose here the step for the scan lines across the spectra pattern. For instance, "5" means that the scan is performed once every 5 lines in the frame. This is a good value, and it is the general default.
Click **Accept** to start the process, **Cancel** to leave without any action.
The program must detect a large number of maxima (around 100000 with the default value of 5 lines per step). Check also the threshold value; it depends on the flux level of the exposure, a typical value being 1000 for a 20000 counts maximum level. Finally the fraction of rejected points should be quite small, at worst a few percent. In case of doubt, you can play with the [search only one line] button and display the fit with the [display result] button. Note that some maxima are not fitted because they are above the threshold;

note also that the wings of the profile are not well fitted. Only the coordinate of the crest is important at this point, and a few missing values will not affect the final results.

Options description
Debug :
This switches the program to verbose mode, and more informations are recorded into the history file (see section [Start] in the left reduction menu).
Search only line ... :
The scan is performed on this particular line only.
Save values :
All the input values (files names, lines) are saved, and become the new default values for this user.
Recall values :
The input fields are set to the last saved values.
Default values :
The input fields are set to the general defaults; for instance, the file names are set to blank.

4.4 Create the extraction mask

Principles

This function adjusts the optical parameters of OASIS spectrographic stage so that, for a regularly spaced set of wavelengths lambda inside the scenario limits, a ray of wavelength lambda emitted by lens *N* will end on spectrum *N*, as detected by the previous **Find maxima** function.

Use

Click on [Mask] in the main menu, then on [Create Mask]; you must before have used successfully the [**Search Lenses**] and [**Find Maxima**] functions.
The program fits a model of the spectrograph to get precise values of optical parameters (position of the beam steerer, grism, lens array, camera and collimator distortions). It uses the lens positions derived by [Search lenses], the positions of the continuum maxima obtained by [Find maxima] and a comparison between the true positions of the selected lines in the reference calibration table and the measured ones in the calibration exposure. The key parameter is the final RMS residual displayed at the end of the process; it should be less or equal to 0.1 pixel. A significantly larger value (e.g. 0.25 pixel) indicates a problem. In that case the user should look at the data. Check that all the important lines are present in the reference table: if some are missing, edit the table to add them. The presence in the table of very faint lines may also lead to some difficulty. There is a flag (column MASK) in the table to set or unset lines for the mask creation process. Just flag it to 1 (use) or 0 (skip) and rerun the mask creation. Another helpful value to check is the number of rejected points; it should be less than a few percent. Finally the exit status of the routine should be 0. If not, the routine has failed to converge or you required too

much precision (the default value of 0.01 is usually good enough if you do not want to spend nights and days waiting for the iteration process to end).
A visual look at the quality of the fit is given by the [Plot mask and max] function described in the next section of this page. You can simultaneously plot the full mask and max (be prepared to a long display time on a slow machine), which show up in two contrasting colors, and then make use of the zoom, or restrict the plot to a narrow window of e.g. 100 pixels width (keeping the full 2048 pixels, or at least some large value like 500 or 1000, in y). Repeating this for the 3 other corners and the center will give you a reasonable idea of the fit quality; zooming is important as you are looking for subpixel deviations. For that, click on [zoom] in the bottom row of buttons, define the window with the left mouse button, un-zoom by right-clicking.

Standard parameters

You are invited to enter here :
The **Input Maxima datafile** name : this is the file just created by the [**Find Maxima**] function.
The **Input Calibration frame** name : this is the GUMBALL spectral calibration lamp frame associated with the GUMBALL continuum exposure you are using to create the extraction mask.
The **Wavelength Reference table** name : this is to be selected among the ones offered when you use the browse icon at the end of the input zone. Names are self-explanatory, and you must choose a reference file according to the spectral lamp you used for the calibration frame.
The **Input/Output mask** name : this is the mask table which has been first created during the **Search Lenses** process, and already updated during the **Find maxima** step.

Optional parameters

There is only one : you may specify the **Tolerance**, that is the mean cross-dispersion error between a real spectrum crest and the polynomial path describing this crest.

Debug parameters

If you check the [**Debug**] button, you may choose to skip first, or second, or both, pass(es), or just save the initial values found before any adjustment is performed. If they are really crazy, do not hope any result...

4.5 Plot the maxima and mask figures

Principles

The maxima and the mask "lines" are plotted to allow the user to check visually the relative fit of the two.

Use

Click on [Mask] in the main menu, then on [Plot max & mask].
Options description
Standard parameters
The mask, or the maxima lines, or both, will be plotted, according to the (left) Max and Mask buttons beeing checked or not.
Max : enter here the name of the mask file pertaining to the current scenario. You may want to use the browse icon at the end of the input field, or to drag and drop the file from the reduction folder.
Mask : same thing for the mask file pertaining to the curent scenario.

Optional parameters
Full frame plot : as you expected. But this may be lonnnnnnnnnng on some machines...
Xaxis / Yaxis : you may chose here to restrict the plot to a region containing few spectra, making thus the plot much more readable. A good idea is to plot three regions, one central, two on both sides of the CCD frame; for instance :
[50,50 ; 100,2000], [1000,50 ; 1050,2000], [1950,50 ; 2000,2000].

5. Extraction of the spectra

Principles

For every lens (spatial element), for every wavelength, the optical parameters of the spectral stage of OASIS are used to find the CCD position of this wavelength in this spectrum. If the *Optimal extraction* is not selected, the CCD column pixels are then summed up over a total width of W_{spec} centered on this position. If it is selected, the effects of the inter-spectra pollutions, of the CCD pixel sampling of the cross-dispersion profile, as well as the varying S/N ratio of the five pixels across this profile are taken into account by :
Using a 3-gaussians fit of the focal reducer PSF.
Using a theoretical cross-dispersion profile obtained through a modelling of OASIS + telescope optics.
Adjusting its parameters on the actual cross-dispersion profile of the 30 central micropupils.
Obtaining the best overall fit to the actual cross-dipersion profile of this particular spectrum.
Computing the weighted sum of the W_{spec} pixels centered on this particular spectrum ridge at this particular wavelength. The weights are computed as 1 over the total signal variance over this pixel, using the photon noise of the fraction of the signal on this pixel (from the cross-dispersion profile obtained previously, integrated over the pixel) and the known CCD readout noise.
The value obtained is taken as the value of the N^{th} spectrum for this wavelength, if N is the number of the lens involved.
The user is given here the additional possibility to have the optimal extraction *Handle inter-spectra pollution*. It is of course strongly recommended to let this option ON, and this is the default. But this extraction algorithm is quite young for us; we got a limited background on the subject, and the "do not take care of pollutions" option is a relic of debugging work. It should disappear from subsequent releases of XOasis; but, just in case...
All the [lambda,$N_{1...m}$] sets are arranged in a **datacube**, with two spatial dimensions (the coordinates of the N^{th} lens), and lambda, which will later be more finely calibrated.
As there is only a single mask for all the exposures of one configuration (in the same run), usually obtained with the telescope in a zenithal position, one must take care of any slight displacements due to flexures or small uncertainties in the repositioning of the beam steerer used to shift the grism null deviation to the central wavelength of the filter. The program thus computes the precise offset between the mask and the object via a correlation between the calibration exposures associated with the continuum and with the object exposures respectively. It is therefore essential to select the right calibration exposure; and remember that the object and associated calibration exposures must have been obtained with the same telescope position without any change in the configuration (save the exposure duration...) between them. The first thing to check is that they bear consecutive exposure numbers ! If not, this must be explained in the *Comments* of the file header, to be read with the Reduction logbook function.
When you run the spectra extraction, check that the computed offset between the mask

and the object is reasonable; you should get a subpixel value in x (less than 1 pixel) and a few pixels in y (less than 20 pixels). Larger values indicate a problem... Note that the procedure used to compute the shift between the frames is the same than in the [Apply CCD fringing correction] task ([CCD] menu) and if you had to change the fit window parameters during at the CCD fringing correction stage, you should do the same at the extraction stage.
A warning is issued if the calibration frame and the object frame are separated by more than one hour. This may happen to the most well-intentioned astronomer, if the integration time on the object is one hour.

Use

Input frame : the name of the preprocessed frame from which you want to extract the spectra. It may be an object exposure, or a GUMBALL continuum flat, or a sky flat, or a wavelength calibration exposure (neon, argon, Perot-Fabry, ...). You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input wavelength calibration frame : the name of the wavelength calibration exposure associated with the input frame; that means obtained in exactly the same TIGER configuration, same telescope position, obtained just before or just after, with no beamsteerer movement between the two. If the input frame is already a wavelength calibration exposure, it is associated with itself, and the same name is to be input in the two zones. Same input possibilities as above).
Extraction mask table : the name of the mask table built during the **Mask creation** reduction step. Same input possibilities as above.
Output datacube : the name of the file which will hold the tridimensional data. Do not specify any extension, a ".tig" will be added to the name you give. This is a special format, specific to TIGER data. Same input possibilities as above.

Options description

Summation full width : the width across which pixel values are summed up to obtain the spectrum intensity at this point. Default is 5, it may be lowered in some special cases to get rid of any possible inter-spectra pollution; 7 is the extreme width possible, as this is the spectra separation on the CCD. It is usually wise to stay with 5, which has always given good results.
Optimal extraction : Checking this button switches the algorithm from brutal cross-dispersion summation to clever weighted summation (see principles for a brief summary). You must use that is you are afraid of possible inter-spectra pollution; they are usually at a very low level, due to the spacing of the spectra on the CCD, but may become noticeable in some cases (strong spatial gradients and huge emission lines). It is supposed, too, to improve slightly the local S/N ratio.
Handle inter-spectra pollution : This option is ON as a default, but may be deactivated for debugging purposes. See principles for details.
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started). In debug mode, you are allowed to extract a

single spectrum, identified by the **Lens** number, to check the quality of the result, for instance regarding possible pollution by neighbours in a high gradient area.

Fit window radius :

The location of the peak in the 256x256 correlation frame (which is used to compute the shift between the mask and the spectrographic exposure) is computed using a Gaussian fitting. Due to the presence of multiple sub-peaks in the correlation image, it is necessary to perform the fit on a subframe. The [Fit window radius] parameter can be used to change the default values (10,20, i.e. 10 along the x-axis and 20 along the y-axis) for the size of this subframe, either along the x-axis or the y-axis.

Fit window center :

In addition to changing the size of the window for the Gaussian fitting (see above), it is possible to change the position of this window. A fit window center of 0,0 (default values) corresponds to the "no shift" position (pixel 127,127 in the correlation window).

Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Extract spectra* window is opened.

Recall values : The values (files names, coordinates) saved by the user, are loaded to the various input fields.

Default values : The input fields are set to the general defaults values; for instance, the file names are set to blank.

6. Wavelength calibration

6.1 Edit reference table

Principles

This function is provided to allow the user to remove one or several calibration line(s) from the set used to calibrate the spectra. It is generally used after the end of the reduction, when it is found that the calibration failed for a particular lens. This maybe due, for instance, to a residual cosmic ray misidentified as (or blended with) a line of the spectral lamp. By removing this false line (for this particular lens only), the calibration may be redone, and give a satisfactory result; remember that this should be done on the copy of the reference table which has been made at the beginning of the extraction mask creation, not on the original one.
As a first step, do not use this function, and jump to the next paragraph : compute calibration parameters.

Use

Click on [Wavelength] in the main menu, then on [Edit reference table], and set the following parameters:
Input associated calibration datacube : the name of the spectral lamp (or Perot-Fabry) datacube associated with the object datacube you intend to calibrate later. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input wavelength reference table : one of the table provided with the software, and copied into the work directory at the extraction mask creation step; You can type the name in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder. Choose the right element (neon, argon, Perot-Fabry, ...).
Lens number : the lens/spectrum you will to work on.

By clicking on [Accept], a graphics window pops up, showing the *Nth* spectrum just selected (in black), with the theoretical line positions (in red) overimposed. Ctrl-left-clicking on a line deselect it for the *Nth* spectrum (it is now blue, another technological miracle), and the next wavelength calibration computation for the associated datacube will not use this particular line for this particular spectrum.
Options :

Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Edit reference table* window is opened.
Recall values : The values (files names, coordinates) saved by the user, are loaded to the various input fields.
Default values : The input fields are set to the general default values; for instance, the file names are set to blank.

6.2 Compute calibration parameters

Principles

Upon creation, the spectra contained into the *raw datacube* are already wavelength-calibrated to a good approximation (from a fraction of an Å to 2 Å, according to the configuration), thanks to the extraction algorithm. In the present reduction step, the small residuals are fitted by a second-order polynomial to refine the calibration.

Use

Click on [Wavelength] in the main menu, then on [Compute calibration], and set the following parameters:
Calibration data cube : this is the spectral lamp or Perot-Fabry datacube associated with the object data cube you are willing to calibrate. You can type the name in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder. In the reduction folder, it is displayed as a *raw calibration datacube*. A new file will be created, with the same name as the calibration data cube, but with extension .fits; it will be shown into the reduction folder display as a *Table*. In the next reduction step, this table will be used for the final wavelength calibration of the raw object datacube.
Wavelength reference table : it has been choosen into a data folder provided with the software, and copied into the work directory at the extraction mask creation step; You can type the name in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder. Choose the right element (neon, argon, Perot-Fabry, ...).
If all goes well, something like 1120 spectra should have been successfully calibrated (watch the message in the output window). Use the **View result** button to check the quality of the calibration. It plots the error as a function of the spectrum number; all the points should lie near to zero (below 0.05). If a few points show up as a small cloud at the upper right of the plot, this is the trace of some problem with this configuration, but they maybe neglected if they lie at the very edge of the CCD frame; you may loose a few spectra over the theoretical 1128 ones. Or you may play with the optional parameters to try to reduce the size of the cloud. For instance, identify the poor points by using the **File**, **Plot table**, function in the **Graph Display** menu of the plot window. Open the table you just write into, and **Plot with label** after specifying **X Col** = a, **Y Col** = error, **Label** = no. Play with the zoom to get things readable in the cloud. Left mouse button to define the zoom area, release button to apply.
Options :
Polynomial degree for continuum subtraction : this is for the fit of the underlying continuum of the spectral lamp or Perot-Fabry, if any. Usually, there is no such continuum.
Polynomial degree for calibration : this is for the fit of the second-order calibration polynomial. Note that upon creation, the spectra contained into the *raw datacube* are already wavelength-calibrated at (roughly) the Å level, thanks to the extraction algorithm. This explains that a degree 1 is in fact enough for the present step, 2 (the default) beeing a conservative value.

Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started). More, checking this option validates the following ones :
Perform over : this allows to compute the calibration coefficients over **All lenses**, or for **One lens**. This last possibility is given to allow the user to play with the optional parameters, in case of tricky data (never happens, of course...) to find the right combination. The lens number must then be provided, and **1** is a default, not worse than any other value, as this is the central lens. Change it at will.
Debug table, **Debug spectrum** : in debug mode, a special *debug table* will be created, as well as a special *debug spectrum*, to hold the temporary results of the user's tests.
Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Edit reference table* window is opened.
Recall values : The values (files names, coordinates) saved by the user, are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the file names are set to blank.

6.3 Apply wavelength calibration

Principles

The parameters computed for each spectrum in the preceding section are used to transform the pre-calibrated spectrum into a wavelength-calibrated spectrum. A new datacube is created to hold the spectra set.

Use

Click on [Wavelength] in the main menu, then on [Apply calibration], and set the following parameters:
Input data cube : this is the data cube created by the Extract spectra function. It is displayed as a *Raw object datacube*. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input calibration table : this is the table created during the Compute calibration (just preceding) step. It bears the same name as the *Raw calibration datacube* associated with the *Raw object datacube* you are to calibrate, but with extension *.fits*. After the calibration, it will keep the same name, but will be displayed as *Calibrated table*. Same input possibilities as above.
Output datacube : from the input datacube, a new datacube file will be created, holding a full set of wavelength calibrated spectra; the input one was made with a set of pre-calibrated spectra.. After completion of the calibration, it will be displayed as *Wavelength calib object datacube*. Same input possibilities as above.
Wavelength scale : you may choose to get the wavelength coordinate of the spectra to be **linear** (lambda) or **log** (log of lambda).
Options :

Quality threshold : you may choose here to set it to **None**, and all the spectra are calibrated, regardless of the quality of the calibration fit (default, and good choice, as least as a first step), or to **Very good**, or even to **Excellent** if you prefer that; note that there is no option like **Terrible**, as OASIS data are supposed to be at least good ;-) ... You may come back to this step later if you discover a lens exhibiting a strange behaviour.
Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values, **Recall values**, **Default values** : as in the previous section.

7. Low frequency flat-fielding

7.1 Compute flat field correction curves

Principles

The spectra which are now recorded into the TIGER datacube suffer from two (related, but separated in the present reduction program) flat field effects. One is the high frequency flat field effect, due to pixel-to-pixel response variation on the CCD, the other is the low frequency flat field effect, due to non-uniform transmission of the optics, both spatial and spectral. This last one is dealt with here. To get rid of this effect, a GUMBALL continuum lamp exposure is used (usually the one which has already been used for the spectra ridges search). The continuum spectra must be wavelength calibrated, and the cosmics impacts removed. They are divided by the median integrated value of the central spectra to get them near unity. To eliminate any non-uniformity of the GUMBALL field, a sky spectroscopic frame is optionally used as a reference to spatially flatten the data. For that, each sky spectrum is summed over -almost, see below- the entire wavelength range, and the resulting scalar used to normalise the corresponding GUMBALL spectrum. All the spectra are then medianed, and a spline fit is performed on this median to eliminate any high frequency residual. The resulting spectrum is the low frequency flat field reference spectrum. A new set of spectra is then computed, by dividing each continuum spectrum by this reference spectrum. A new datacube is so created, holding the low frequency flat field correction curves (somewhat misleadingly called spectra); it will appear as *Flat correction datacube* in the reduction folder display.

Use

Click on [Flat] in the main menu, then on [Compute flat field correction]. In the new window :
Input: continuum datacube is the name of the wavelength calibrated (and cosmics-removed) GUMBALL continuum datacube,i which should be displayed as *Wavelength calib continuum datacube*. or *Cosmics removed continuum datacube*. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input: skyflat datacube is the name of the wavelength calibrated sky flat datacube, which should be displayed as *Wavelength calib sky flat datacube*. or as *Cosmics removed sky flat datacube*. It is used if you check the left button. Same input possibilities as above.
Output: reference flat datacube is not a topologist's nightnare, but the datacube to be created to record the flat reference curves ("spectra"). Same input possibilities as above.

Options
Smoothness parameter : to be used by the spline fit of the reference spectrum. Zero means that each pixel is taken as a node; that is, there is NO filtering. Five to seven is usually OK, with the fit going nicely through the noise. Greater values increase the level of smoothing. Note that non-integer values are allowed.

Fraction of wavelength range : the central fraction of the wavelength range to be summed up to compute the sky flat spatial correction factor.
Filter radius in pixels : window size (in pixels) for the smooth filter.
Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Select lens : if set, only lens/spectrum *N* will be treated; this is very useful as a first step, to tune the parameter values before performing the complete operation.
Lens number : lens/spectrum number to check, if *select lens* is set.
Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Compute flat field table* window is opened.
Recall values : The values (files names, coordinates) saved by the user, are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the file names are set to blank.

7.2 Apply flat field correction

Principle

The low-frequency flat field correction computed during the preceding step is applied to the wavelength calibrated object datacube, and a new datacube is created, holding the wavelength calibrated and flat fielded spectra of the object. This new file appears as *Flat fielded object datacube* in the reduction folder display.

Use

Click on [Flat], in the main menu, then on [Apply flat field correction]. In the new window :
Input datacube to be flat fielded : enter the name of the wavelength calibrated object datacube. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input flat correction datacube : enter the name of the datacube holding the low frequency flat field correction curves, which was created during the previous reduction step. It is displayed as *Flat correction datacube* in the Reduction folder display. Same input possibilities as above.
Output datacube : enter the name of the datacube where you wish to get the final result recorded. Use some significant name, where the object name, exposure number if there are several, do show up; same input possibilities as above.

Options
Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values, **Recall values**, **Default values** : as in the previous section.

8. Cosmics suppression

Principles

Spectra are searched for cosmic impacts; an algorithm does its best to discriminate them against emission lines using morphological criteria, both spatial and spectral :
A spatial radius is chosen, which defines the region over which spectral similarity is supposed to be true. Over this region, usually twice the size of the spatial sampling, the integrals of the spectrum to be checked (and cleaned) and of the neighbours are computed. For instance, if the radius is given as 1, this means that one ring of neighbours will be used; then, seven integrals are computed : one for the central lens, six for the ring made of the six nearest lenses. They are used to normalise the peripheral spectra to the central spectrum integral value.
The median of the spectra from the above normalised disk region is computed. This normalisation is used to be able to correct cosmic impacts even within high spatial gradient regions.
The differential spectrum [Central spectrum]-[Median spectrum] is computed. This difference is then median-filtered, with a radius given by the user.
The second-order difference between this filtered difference and the un-filtered difference is computed. This final spectrum is now holding only high spectral frequencies, and cosmic ray pixels may now be detected quite safely.
A sigma-clipping is performed on this spectrum, and pixels above *N* times sigma are flagged.
All the flagged pixels are replaced, in the original central spectrum, by the median value (the one obtained in the second step above). In fact, not only the flagged pixels, but their immediate neighbours too, up to a correction radius specified by the user, are replaced.

Use

Click on [Cosmics] in the main menu, then on [Remove cosmics].The [Remove cosmics] window pops up.
Enter the name of the **Input datacube**, which is the object datacube you have wavelength-calibrated and flat-fielded in previous steps. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Enter the name of the **Output datacube**, that is the new datacube to be created after cosmics removal. Same input possibilities as above.
Click on [Accept]. The result window opens, and trace the various steps of the elimination process. As a rule of thumb, expect to find something like 3000 cosmics over a one-hour exposure.

Parameters

Spatial filtering radius : the radius of the neighbourhood over which similarities are to be searched for discriminating between lines and cosmics in spectra; to be given in lens ring units. Usually, it is wise to use a radius of 1, and the neighbouring disc will span

over seven lenses, the one which is to be checked for cosmics, and the ring of six around it. Default value is 1.
Spectral filtering radius : the radius (in pixels) used for the median filtering of the first order difference between the central spectrum and the disc median. Default value is 3.
Sigma clipping factor : this is the factor to be multiplied by sigma to find and flag the bad pixels in the second-order difference. Default value is 10.
Relative threhold : Default value is 1.5.
Maximum number of iterations : for the sigma-clipping operation. Default value is 5.
Spectral correction radius (pixels) : the radius around flagged bad pixels within which all pixels of the spectrum are to be replaced by median values. Default value is 2.
Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see button [Start] in the left menu). In addition, the following options become then available :
Polynomial datacube :
Median datacube : the name of the datacube which will hold the normalised disc medians for each of the original datacube.
Residual datacube : the name of the datacube which will hold the second-order difference spectra for all the original spectra.
Save values :
All the input values (files names, etc...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Remove cosmics* window is opened.
Recall values :
The values saved by the user are loaded into the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

9. Sky substraction

9.1 Define the empty sky area

Principles

Using an image of the observed field, reconstructed from spectroscopic data from a datacube of the same family, the user selects graphically a zone which is supposed to be empty, and so is able to give the pure sky spectrum. This region is then translated into a selection to be made over the spectra set, in fact into the table associated with all the datacubes which derive from the same raw object datacube. This selection will be used during the next step (Compute the sky spectrum) to restrict the spectra summation over the selected spatial area.

Use

Prepare an image of the field, to be used as a coordinates reference. For that, from the main menu, click on [Analyse], then on [Integrate spectra], and output an image corresponding to a whole wavelength range spectral integration; use a cosmics removed datacube as data base, to avoid spatial ghosts due to cosmic impacts. The datacube chosen must be of the same "family" as the datacube you will to subtract the sky from; that is, it must be associated with the same initial fits table. It is not necessarily the datacube to be sky-subtracted, although this one is usually a good choice. But you may, after a complete reduction, discover on a line-only reconstructed image that in the area you elected as empty, there are some emission blobs. It is then time to redefine the sky area on this last image, and redo the computations, starting with the sky subtraction function. For image reconstruction, see details in the Analyse data section of this manual.
In the main menu, click on [Sky], then on [Define sky area]; the RTD interface pops up.
Click on [File], [Open], and chose the image you just computed.
Click on [Auto set cut levels] to get a better view of the low levels, and play with the zoom button to make things completely comfortable.
Click on [Select], then on [Select an area]. A square, with four conspicuous nodes at the summits, appears. You can move it (center mouse button depressed), drag a node (left button depressed), add more nodes (right click). Envelop the empty zone you whish to choose as the sky area.
In the RTD menu, click on [Select], then on [Apply selection to]. In the window which pops up, enter the name of the table you whish to select. It is usually the table which has been created at the spectra extraction time, and is associated with all the datacubes created since that by the various reduction steps already done. Select the [New] action, which means that you are going to get rid of any previous selection. Then click on [Accept].
You may want to enclose in a polygon the object itself, and define as the clear sky region the outside of the polygon. In such a case, select the **Use area outside polygon** option.
If you want to use as sky area the reunion of several zones, repeat the above point (no need to close RTD, just drag the previous polygon to the new location, reshape it, and reselect the table; if you think there are too many nodes on this previous polygon, erase it

with the [Select]/[Clear selection] function and define a new one by [Select]/[Select an area]). But when you are about to **Apply selection to**, choose the **Join** action, so the previous selection will be kept, and the new zone added to the previous one(s).
You may combine at will **New**, **Join**, **Intersect**, **Inside** and **Outside** to satisfy your needs. For instance, in a field where appear only two stars, enclose one of the stars in a polygon, apply selection with **New** and **Use area outside of polygon**; then, enclose the other star in a polygon, and apply selection with **Intersect** and **Use area outside of polygon.**. Another interesting possibility is to define two selections on two different images (for instance one reconstructed from the continuum, the other from lines), and **Intersect** them.
To **Check** the final selection, it is a good idea to plot the selected table, just to be sure you did not get lost in the Inside/Outside/Join/Intersect jungle... In the main menu, use the **Display** item, choose **Spectrum and table**. Then **File**, **Plot table**. Give the name of your table, and chose **X taken from column** = A, **Y taken from column** = D. If you like to see the lens numbers, choose **Label each data point using column** = no, and click [Plot with label] instead of [Plot].

Important warnings :
RTD is unable to make a selection over a very small area, due to the fact that each polygon summit has a circular "dead" neighbouring where no point can be selected. If the summits get very close, their neighbourings overlap and no place is left for any lens to be selected. Approximately ten lenses must be enclosed within the polygon to get them selected.
Once the table has been selected (*i.e.* restricted) by the above described function, it remains so. If you want to regain access to the complete son datacubes, you must deselect the table, using the [Tools]/[Table access]/[Table select] function. Otherwise, some selection-aware functions may perform on the fraction of the datacube corresponding to the table selection. Not all functions take into account table selection, but this is a risk.

9.2 Compute the sky spectrum over the empty sky area

Principles

The "mean" spectrum of the area selected in the previous step is computed, and will become the sky spectrum from the corresponding frame.

Use

Click on [Sky], then on [Compute sky spectrum]. The input window pops up.
Enter the **Input datacube**, which is usually the datacube with cosmic rays removed, and the **Output spectrum** name, that is the name of the resultant sky spectrum. You may type them, or use the browse icon at the end of the field, or drag and drop them from the Reduction folder window. Click then on [Accept].

Options

Method : You may choose **Histogram**, **Median**, or **Mean**. The two last do what you expect, and the first one (default) uses an algorithm which, to make short a long story, computes the common part of the spectra in the area, which is by definition the true sky spectrum. If you choose this option, you may fill two additionnal values, **q** and **rejection factor**. They are defaulted to 1 and 5 respectively.
Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Compute sky spectrum* window is opened.
Recall values : The values (files names, coordinates) saved by the user, are loaded to the various input fields.
Default values : The input fields are set to the general default values; for instance, the file names are set to blank.

9.3 Subtract the sky spectrum from the object spectra

Principles

Straightforward : the sky spectrum, evaluated from the same frame, is subtracted from every spectrum of the input datacube. This datacube must have been flat-fielded before.

Use

In the main menu, click on [Sky], then on [Sky subtraction]; the *Subtract sky spectrum* window pops up.
Enter the name of the **Input datacube** from which you will to subtract the sky spectrum, the name of the file holding the **Sky spectrum** (which you obtained in the preceding step Compute sky spectrum), and the name of the **Output sky-subtracted datacube** which will be created.
Click on [Accept].

Parameters
Normalisation factor : provided here just to allow the formerly hopeless astronomer to use a sky spectrum computed on another frame, with a different exposure time; not very clean, but as long as nobody knows ;-)... Should stay at 1.0 for "normal" operation...
Debug : the program is switched to verbose mode, and more informations are recorded into the history file (click on *Getting started* in the left menu for details).
Save values : All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Subtract sky spectrum* window is opened.
Recall values : The values (files names, normalisation factor) saved by the user, are loaded to the various input fields.
Default values : The input fields are set to the general default values; for instance, the file names are set to blank.

10. Flux calibration

10.1 Copy reference table

Principles

The fits table containing the absolute spectrum of the reference star will be copied into the work directory; this ensures that the original will remain in a safe place, and will not be inadvertently tempered with. To be reachable, this table must be stored into directory {*Local XOasis installation path*}/oasis-4.x/user/flux_ref/
where a few ones are already provided with the present software (the ones which had already been of interest for the Lyon team, in fact...). The following stars are already included, as of February 20001 :
HR 1544
HR 4963
HR 5501
HD 23733
HD 93521
BD+332642
Hilt 600
G 191b2b
Feige 110
If you need another one, you must provide it. You may for instance download the corresponding ASCII file giving F=F(lambda) for this particular star from the ESO spectrophotometric standards library. With your favourite package (MIDAS, IRAF, ...), create then from that a FITS table with at least two columns, one for LAMBDA, the other for F_LAMBDA. But you may create more data columns, with different flux units, which you may use later into XOasis. The standard name of such a table is {*Name of the star*}abs.fits in XOasis, but you may use any denomination of your own, like Georgette or MonEtoileStandard.

Use

Click on [Flux] in the main menu, then on [Copy reference table].
In the new window which pops up, check the table(s) you want to copy to your work directory, then click on **Copy**.

10.2 Compute flux correction

Principles

A standard star observation is used to calculate the spectral transfer function of OASIS/TIGER. The total flux of the star is obtained by summation of all the spectra obtained within a certain radius, larger than the seeing disc radius. In the case where it is suspected that the wings of the star image may be lost due to CCD size limitation, it is

possible to use the AOB theoretical PSF (two gaussians) to estimate the amount of light lost. The correction curve, or spectrum, obtained is stored in a *Throughput spectrum*.

Use

Click on [Flux] in the main menu, then on [Compute flux correction]. In the new window:

Input reference star datacube is the preprocessed datacube of the standard star. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder. This must be a FLUX standard, and the absolute spectral flux of this star must be provided as a table :
Input flux reference table is the table where is recorded the outside atmosphere absolute spectrum of the standard star; same input possibilities as above.
In this table the **Wavelength column** and **Flux column** are the labels of the two columns of interest, usually *LAMBDA* and *F_LAMBDA* (case unsensitive).
Output flux correction spectrum is the name the user wish to use for the file holding the flux correction curve.
Spatial summation radius is the radius inside which all the spectra will be summed up to get the total flux of the reference star .
Summation center (X,Y) is the center of this spatial summation. This center must have been obtained before. Use for instance the **Analyse**, **Integrate spectra** function (see the Analyse data : spatial or spectral integration section of the present manual). Integrate the spectra over the whole wavelength in the cosmics-removed standard star datacube, make an image with the resulting data, choose **View result**, and note the cursor coordinates of the centroid of the star image, as well as the radius of this image; an eye estimate is good enough for the present purpose. This will give you the **Summation center**, as well as the **Spatial summation radius**, which should be large enough to enclose ALL the star energy.

Options

Use PSF estimate : checking this option means that the user is willing to use an AOB PSF estimate (made from two gaussians) to take care of light from the star possibly falling outside the CCD. It validates the following sub-options :
Core sigma is the sigma you adopt for the inner peak of the AOB PSF, while **Halo sigma** is the same parameter, but for the outer PSF halo. **Halo/core ratio** is the peak-to-peak ratio of the two.
Polynomial smoothing may be used on the correction curve, with a given **Ncoeff**.
Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (files names, coordinates) are saved, and become the new default values for the user. They can be recalled at will, and are used each time the *Compute throughput* window is opened.
Recall values : The values (files names, coordinates) saved by the user, are loaded into the various input fields.

Default values : The input fields are set to the general defaults values; for instance, the file names are set to blank.

10.3 Apply flux correction

Principles

Each spectrum in the object datacube to be flux-calibrated is multiplied by the flux correction spectrum obtained during the preceding step. Care is taken of the different integration times and air masses.
If the PSF estimate is used, an extra geometrical correction factor *"corr"* is taken into account, and the following formulae apply :
If halo/core ratio = 0 (PSF taken as a single gaussian) :
corr = 1/[erf(r/(sqrt(2)*core_sigma))]
If halo/core ratio > 0 (PSF is a sum of two gaussians) :
A = halo_sigma + ratio*core_sigma
B = halo_sigma*erf(r/(sqrt(2)*core_sigma))
C = core_sigma*erf(r/(sqrt(2)*halo_sigma))
corr = A / (B + ratio*C)

Use

Click on [Flux] in the main menu, then on [Apply flux correction]. In the new window :
Input object datacube : the object datacube as it is after wavelength calibration, flat-fielding, and cosmic ray removal. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Input flux correction spectrum : the correction spectrum just generated during the *Compute throughput* phase. Same input possibilities as above.
Output flux-calibrated object datacube : the result will be recorded there. This will be the end product of the reduction process for this exposure.

Options

Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values, **Recall values**, **Default values** : as in the previous section.

11. Mosaic - Merging intersecting fields

11.1 Coordinates recentering

Principles

To be able to mosaic fields, that is to be able to merge datacubes obtained from different exposures, the exposures must, first, belong to the same scenario. Second, one must know exactly the spatial shift between the exposures, to avoid any field distorsion in the resulting datacube. This shift has to be evaluated by the user. The usual way is the following :
Use the [Analyse] / [Integrate Spectra] function to produce, for each exposure to be merged, an image of the field integrated over the whole spectral range.
On these images, find some eye-catching detail (a star is best, but any peaked maximum will do the job, or at least some small morphological feature), and note the [$alpha_{center}$,$delta_{center}$] position of this point. One may use the [Display] / [Datacube explorer], and note visually the displayed coordinates, or use the [Image] / [Fit PSF] function. You may want to use then the *Display result* function to have a look at the image with the fitted point overlaid.
The point thus defined must be present on ALL the images, that is it must belong to the intersection of the reconstructed images. If some exposure have an empty intersection with some other ones, it may not be possible to merge it, except if it can be first properly merged with another exposure which in turn intersects the rest of the data.
On each exposure to be later merged, this particular point will become the [alpha=0,delta=0] point, after the completion of the coordinates recentering function.

Use

In the main menu, click on [Mosaic], then on [Coordinates recentering].
In the new window which pops up, enter the name of the Tiger **datacube** to be recentered, and the point which should now be considered as the **Center** of the field.
Click on [Accept].

Standard parameters

Tiger datacube : the name of the datacube to be recentered; it will keep its name after the operation (which in fact affects only the associated table). May be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.
Center (A, B) : in these fields, enter the [$alpha_{center}$,$delta_{center}$] you just obtained on the images reconstructed from this datacube (see *Principles* above) for the common reference point.

Optional parameters

Debug : This switches the program to verbose mode, and more informations are recorded

into the history file (see Getting started).
Save values : All the input values (datacube name, coordinates, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Coordinates recentering* window is opened.
Recall values : The values (datacube name, coordinates, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the datacube name is set to blank.

11.2 Truncate datacubes

Principle

The datacubes to be merged must hold spectra which share the same wavelength range. If you went all the way through the complete reduction process, the flat (and/or the flux) calibration has done the job, and all the spectra are already truncated to the same wavelength limits. But if (and only if) you want to merge datacubes which have not been flat (or flux) -calibrated, then you must use the present function.
Spectra which do not contain entirely the wavelength range are discarded in the new datacube.

Use

In the main menu, click on [Mosaic], then on [Truncate datacubes].
In the new window which pops up, enter the names of the **Input datacube** and **Output datacube**.
Choose the wavelength range to be kept, either as a couple of numerical values, or to be taken from some template spectrum.
Click on [Accept].

Standard parameters

Input datacube : the name of the datacube to be lambda-truncated. May be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.
Output datacube : the name of the resulting truncated datacube. May be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.

Optional parameters

Wavelength range : give here the wavelength limits (Å) for the truncate operation. Keep in mind that "short" spectra, which do not contain entirely the wavelength interval, will be discarded.

Get wavelength range from spectrum : you may decide to take some particular spectrum as a template for the operation. The name of the spectrum may be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.
Display only removed lenses : by checking this button, you get a display of the discarded lenses.
Save values : All the input values (datacube names, wavelengths, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Truncate datacubes* window is opened.
Recall values : The values (datacube names, wavelengths, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the datacube names are set to blank.

11.3 Compute Norm and Weight Factors

Principles

This application is to be used before merging datacubes. It derives all the parameters needed for the merging process, i.e.
the normalization factor to apply to the datacubes to correct for the flux differences.
optimal weights to be used during the (average) merging process to maximise resultant S/N.

These values, once computed, will be displayed on the top of the "Merge datacubes" window, for the next merging step.

Use

In the main menu, click on [Mosaic], then on [Compute Norm & Weight Factors].
In the new window which pops up, enter the names of the **Input datacubes**.
Click on [Accept].

Standard parameters

Input datacubes : the names of the datacubes to be merged. They may be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.

Optional parameters

Output spatial sampling : it may be set to **Auto**, and it is computed after the spatial sampling of the original datacubes, or set to the desired value (in arcsecond).

Output spatial limits : if **Auto** is set, the whole spatial region where valid spectra can be defined is used. If not, you have to specify the [X1,Y1] and [X2,Y2] bottom left and top right angles of the rectangular region you want to cover.
Fraction of wavelength range : fraction of the spectrum to keep, given in percent of the full width (wavelength range). The value must be between 0 and 1, default value is 0.8.
Save normalized frames : if option is set, then, intermediate normalized frames are save as *norm_xxxx.fits* files, where *xxxx* stands for the datacube filename from which it is derived.
Debug
This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (datacube names, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Merge datacubes* window is opened.
Recall values : The values (datacube names, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the datacube names are set to blank.

11.4 Merge datacubes

Principles

A new datacube is created, holding the reunion of the data contained in the datacubes merged (under some conditions, see below). The new datacube has a square spatial sampling grid, instead of the original hexagonal grid.

Use

In the main menu, click on [Mosaic], then on [Merge datacubes].
In the new window which pops up, enter the names of the **Input datacubes**, the weight and norm factors. If you have first ran the previous command (Compute norm & weight factors), all these fields are preset.
Give the name of the resultant **Output datacube**.
Click on [Accept].

Standard parameters

Input datacubes : the names of the datacubes to be merged. They may be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in. You may specify a **Weight** factor, if you think the data quality is widely different among exposures. You may also specify a **Normalization factor**, for the case you are about to merge exposures which are not flux-calibrated, and have, for instance, different exposure times. Usually, accept the unit default value for both.

Output datacube : the name for the datacube resulting from the merge action. It may be dragged and dropped from the *Reduction folder*, selected using the browse icon at the end of the field, or just typed in.

Optional parameters

Output spatial sampling : it may be set to **Auto**, and it is computed after the spatial sampling of the original datacubes, or set to the desired value (in arcsecond).
Output spatial limits : if **Auto** is set, the whole spatial region where valid spectra can be defined is used. If not, you have to specify the [X1,Y1] and [X2,Y2] bottom left and top right angles of the rectangular region you want to cover.
Merging options :
Adjust center : if you check that, a spectrum will be created by interpolation at the [0,0] point, and the grid will be built around this.
Bicubic interpolation : once checked, the interpolation process is no longer bilinear, but bicubic. Better, specially for high-gradient regions, but time-consuming, and needs a good S/N ratio; otherwise, the algorithm may be fooled by local noisy gradients.
No fill : no interpolation at all.
Merging method :
Weighted summation : the weighted average is used for the interpolation of spectral data.
Median : the median is used for interpolation; this needs a lot of high S/N ratio exposures to work.
Minimum number of spectra : you may ask to discard a point if there are not at least *N* neighbouring spectra to interpolate; but usually, people want to keep every point where there is at least one spectrum nearby... So, 1 is a good default value.
Debug
This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (datacube names, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Merge datacubes* window is opened.
Recall values : The values (datacube names, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the datacube names are set to blank.

12. Spatial or spectral integration

12.1 Integrate over a wavelength interval : integrate spectra

Principles

For a given datacube, for all the spectra, a given wavelength interval is scanned. Output may be the integral of the energy over this domain, or the mean, or the RMS dispersion. The data are recorded into a column created into the table associated with the datacube, or directly mapped as an image of the spatial repartition of the computed parameter.

Use

Click on [Analyse] in the main menu, then on [Integrate spectra]. The [Integrate spectra] window pops up.
Enter the name of the **Input datacube**. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Enter the **Spectral range** (Å) of the spectral region to be scanned, or check the **Full range** button to scan the complete spectral domain.

Options description

Save output as a table column : the computed result will be recorded into a column which will be created (if it does not already exist) in the table associated with the input datacube; remember that this table is known by the software, as it has been associated with the creation of the datacube. the user is requested to fill in the **Column** name input field.
Build an image from output : the computed result will be recorded into an image. This image may be later viewed (button [View result] in the result window). The user is requested to give the **Image** name.
Compute : the user may choose between :
Sum (the spectra will be integrated over the given wavelength range),
Average (the spectra will be averaged over the given wavelength range),
RMS dispersion (the spectra will be scanned over the given wavelength range, and the RMS dispersion computed).

Optional parameters : the user may filter the spectra using a given wavelength transmission curve. This curve may be given as a 1D-image (a spectrum) or as a couple of table columns [lambda,transmission].
Save values :
All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Check frame offset* job window is opened.
Recall values :
The values (files names, coordinates) saved by the user, are loaded to the various input fields.

Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

12.2 Integrate over a circular spatial area : sum aperture

Principles

A circular area is defined in the field, by giving a center and a radius.
The spectra of the spatial pixels included into this area are summed up and the resultant spectrum recorded as a 1D-image.

Use

Click on [Analyse] in the main menu, then on [Sum aperture].
Give the names of the **Input datacube** and of the **Output spectrum**. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Give the **Center** and the **radius** (in arcsec or pixels, according to the type of coordinates) of the area of interest.
Click on [Accept].

Options description

CCD coordinates : the **Center** and **Radius** are to be interpreted as CCD pixel units.
Sky coordinates : the **Center** and **Radius** are to be interpreted as arcseconds on the sky [alpha,delta].
Other coordinates : the **Center** and **Radius** are interpreted as given in the same units as the **X** and **Y** columns of the associated table. The names of the columns are entered by the user in the two **X** and **Y** input fields.
Save values :
All the input values (files names, coordinates) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Check frame offset* job window is opened.
Recall values :
The values (files names, coordinates) saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

12.3 Integrate over a spatial area of any shape

The user's interface for this function is *Not yet implemented* (please wait for release 4),

BUT the function may already be used. Yes, that is XOasis... Just use the **Sky**, **Define sky area**, and **Compute sky spectrum** functions described in the Sky subtraction section of this manual to build the spectrum of any part of the field, connex or not.

12.4 Continuum subtraction

Principles

This operation can be performed either on an isolated spectrum or on all the spectra recorded into a datacube.
A polynomial fit is performed over the spectra, and the resulting fit is taken as an image of the underlying continuum of each spectrum.
The continua maybe saved as such, or used to divide the associated spectra, or subtracted from the spectra.

Use

Click on [Analyse] in the main menu, then on [Continuuum subtraction]. The [Continuum subtraction] window pops up.
First choose between options **On spectrum** and **On datacube**.
Give the **Input spectrum** (or **Input datacube**) and **Output spectrum** (or **Output datacube**) names. You can type them in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Choose the **Polynomial degree** for fitting the continuum. The default, 7, is often OK, but it depends on the object observed.
Choose what to do with the **operation** options :
Save : The computed continuum(a) is(are) saved in the output spectrum (datacube).
Subtract : The computed continuum(a) is(are) subtracted from the spectrum(a) of the input spectrum(datacube), and the result is stored into the output spectrum (datacube).
Divide : The spectrum(a) of the input spectrum(datacube) is (are) divided by the computed continuum(a), and the result is saved in the output spectrum(datacube).
Divide-1 : The spectrum(a) of the input spectrum(datacube) is(are) divided by the computed continuum(a), then 1 is subtracted from every spectrum to bring the mean around zero, and the result is saved in the output spectrum(datacube).

Click on [Accept]. The continuum (continua) is (are) computed, and recorded into the output spectrum (datacube).

Options description

Rejection factor : this is the N-sigma rejection level for the polynomial fit over the number of iterations given (see below). The user may introduce a dissymetry in the rejection process by choosing different absolute values for the **Inf** and **Sup** coefficients.
Maximum number of iterations : does what you expect regarding the continuum fit with bad points rejection...

Discarded intervals table : the user may tell the program to ignore some wavelength intervals for the continuum fit. The interval limits are recorded in a table which is most conveniently created using the standard XOasis graphical display :
Using function [Display]/[Spectrum and Table]/[Open]/[Plot spectrum], plot some nice typical spectrum where the zones to be avoided are clearly spotted.
Click on [Curs] at the bottom left of the window, then, starting with short wavelengths, left-click on all the points where you would like the program to switch from *take care of the following pixels* to *ignore the following pixels*; you note that the cursor positions are displayed in the result window.
When done (there must be an even number of points), click on [Save], and give a name for the save file.
Debug parameters :
As a first step, the user may choose to fit a single spectrum of the datacube, to fine tune the above parameters. For that purpose, check the **Debug** button. The **Lens number** has to be given, as well as the filename of the the temporary **Lens spectrum** (which will be automatically extracted from the datacube), and those of the **Output spectrum**.
Save values :
All the input values (files names, numerical values) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Check frame offset* job window is opened.
Recall values :
The values (files names, numerical values) saved by the user, are loaded into the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

13. Image tools

13.1 Reconstruct image

Principles

A quantity has been computed for any spectrum in a datacube, and this quantity has been recorded, for each spectrum, into a table. It maybe a line intensity, a line width, a line ratio, and so on... The tool provided here reads this column and the columns holding the coordinates of each spectrum, and constructs the 2D-image of the repartition of this quantity. It is so an straightforward way to obtain intensity, line ratio, velocity, ... maps of the field.

Use

Click on [Image] in the main menu, then on [Reconstruct Image]. The [Reconstruct Image] window pops up.
Enter the **Input table** name, the **Output image** name and the **column** to be mapped.
Click on [Accept]. The result window opens, the image is built, and recorded under the name you gave.

Parameters

Input table : the name of the table from which the numerical data will be extracted to build the image. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Output image : the name of the resultant image. Same possibilities as above...
Input coordinates columns : the user have to choose among three possibilities :
CCD [row,column] and the pixel coordinates will be taken automatically from the right columns in the input table, if they exist.
Sky [alpha,delta] and the relative *alpha,delta* will automatically be taken from the right columns in the input table, if they exist. Remember that in the Extract spectra function, the [alpha=0,delta=0] point has been arbitrarily set at the center of the CCD frame, and the coordinate units as arcseconds.
Others : if this option is checked, the user is intented to supply the labels of the columns which contain the **X** and **Y** values to be used as coordinates for the image reconstruction.
Input data column to be mapped (Z) : this is the label of the column holding the numerical values the user wants to map over the spatial field.
Options
Image sampling : the user may check the **Auto** button to let the program decide which sampling to use for the reconstructed image. In such a case, the sampling is given by :
$S = \min [(X_{max}-X_{min}) / (2 * \mathrm{sqrt}(N_x)) , (Y_{max}-Y_{min}) / (2 * \mathrm{sqrt}(N_y))]$
where X_{min}, X_{max}, Y_{min}, and Y_{max} are the field limits, and N_x, N_y the number of spatial elements in X and Y in the original data. Otherwise, the final image sampling (arc seconds) has to be specified.

Coordinates limits : if set to **Auto**, the image is built over the spatial rectangle which exactly contains the original data. If not, the user is asked to specify the limits to be used; do not trust extrapolation, and avoid larger-than-original images!
Interpolate options :
Adjust center : if the user checks this option, the interpolation grid is adjusted so that there is a [0,0] pixel in the reconstructed image.
Extrapolate : if the user checks this option, and if there are "holes" in the map, the polynomial surface is locally extrapolated until it fills all the image rectangle.
Bicubic interpolation : once checked, this constrains the interpolation to be bicubic; default is bilinear.
Fill triangle : in the interpolation process, a grid of spatial triangles is created over the data. A "mean step" size is computed for this mesh, and the algorithm does not interpolate over triangles much larger than this mean. If the option is checked, the sizes are no more checked, and every region surrounded by three data points is treated. This produces big area differences if original data present some "angled holes", like after a datacube merge (function Mosaic). This option is superseded by the [Extrapolate] one.
Blank value : this is the value to be used in the reconstructed image if the user did not check **Extrapolate**.
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (names, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Reconstruct Image] window is opened.
Recall values : The values (names, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the names are set to blank.

13.2 Fit PSF (gaussian fit)

Principles

A gaussian (or a sum of gaussians), plus a polynomial background, are adjusted on the reconstructed image. The computed gaussian parameters may be taken as an estimate of the actual PSF of OASIS data at the time of the exposure.
Use
Click on [Image] in the main menu, then on [Fit PSF]. The [Gaussian PSF fit] window pops up.
Enter the name of the image to be fitted, and the number of gaussian components you wish to fit simultaneously.
Click on [Accept]. The parameters of the fit are displayed in the result window. A [View result] button allows to display the result. For that, sections of both the gaussian fit (in red) and the underlying image (in black) are overplotted for several radial lines.

Parameters

Input image : the name of the image on which you want to fit a sum of gaussians. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Number of gaussians : the user may specify here to fit a single gaussian, or a sum of *N* gaussians.
Fit window : the user specifies here if the fit should work on the whole frame, or on some restricted area. In this last case, the **Center** [X,Y] and **Size** [dX x dY] must be specified; there are default values : center in [0,0], size 2x2 arcseconds.
Save PSF as an image : an image is created from the sum of gaussians + background. The user may specify a name for this new image. There is a default name : dbg_ima.
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Output fit table : created automatically, do not care... It holds the radial profiles which are to be used later by the [View result] function. If you are a table collector, you may specify some *ad hoc* name to keep it.
Max nb of iterations : as one expects...
Tolerance : as one expects...
Save values : All the input values (names, coordinates, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Reconstruct image] window is opened.
Recall values : The values (names, coordinates, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the names are set to blank.

13.3 Extract trace

Principles

A rectangular aperture (trace) is superimposed over an image. The intensities are summed up across the trace, and the resulting intensity profile along the trace is saved as a 1D file.

Use

Click on [Image] in the main menu, then on [Extract trace]. The [Extract trace] window pops up.
Enter the name of the **Input image** you will to cut across. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Enter the name of the **Ouput spectrum**, that is the name of the 1D file recording the cut. Same input possibilities as above.
Enter the **Center** (arc seconds) of the trace, the **Width**, the **Step** of the sampling along the trace, the **Position angle** of the trace.

Click on [Accept]. The result is stored as a 1D file. A [View result] button allows to display the "spectrum".

Parameters

Trace Length : you may chose to cut across the **Full frame**, or to restrict the length of the trace to **Length**.
Subsampling : subsampling factor (relatively to the step of the input frame).
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (names, coordinates, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Reconstruct image] window is opened.
Recall values : The values (names, coordinates, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the names are set to blank.

14. Import/export: file conversion

14.1 Export spectra from datacube

Principles

As you may guess, with this tool, a single spectrum maybe extracted as a 1D image from a given datacube. But it may also be used to export simultaneously ALL the spectra from a given datacube rearranged in a 2D image.

Use

Click on [Import/Export] in the main menu, then on [Export spectra from datacube]. The [Export spectra from datacube] pops up.
Enter the name of the **Input datacube**. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Enter the name of the **Output object**, which will be either a single spectrum (a 1D image) or a 2D image holding all the spectra from the datacube, according to the user's choice.

Parameters

Way to export : the user may choose between :
Single lens (#) : a single spectrum will be extracted, stored into a 1D file, and the user is requested to identify this spectrum by its lens number.
Single lens (arcsec) : a single spectrum will be extracted, stored into a 1D file, and the user is requested to identify this spectrum by its (approximate) relative [Alpha,Delta] position on the sky; remember that [0,0] is the center of the field.
Single lens (CCD) : a single spectrum will be extracted, stored into a 1D file, and the user is requested to identify this spectrum by its approximate CCD pixel coordinates.
Single lens (columns) : a single spectrum will be extracted, stored into a 1D file, and the user is requested to identify this spectrum by the values which are to be found in two columns [X label] and [Y label] in the mother table associated with the datacube.
2D output, and a 2D image is created, where [x] is the lambda coordinate, and where each spectrum occupies a line of the image. The spatial field is explored starting from the lower left corner, and a line of lenses is followed to its upper end. These lines are the ones which are tilted ~6 degrees relative to CCD columns on a micropupil image. Each lens (spectrum) found on this line of lenses is written in the image above the preceding one. Once at the end (top) of the line, the process is repeated with the line of lenses which is next to the right, and so on until all the field has been covered. So, each lenses column gives a strip in the output image. In this strip, spectra are spatially correlated. Optionally (recommended) spectra strips maybe separated in the image by a black (zero) line. This tool is provided to give ARGUS user's a more familiar touch-and-feel... A table is created, too, to hold the {*N*,*X*,*Y*} triplets, that is the coordinates of the lens *N* which produced the *Nth* spectrum, or the *Nth* line of the image. It is named after the name the user gives for the output object, by adding the prefix string *tab_* and changing the

extension from *.tig* to *.fits*. The user is asked to give the labels [X label] and [Y label] of the columns which hold, in the mother table associated with the input datacube, the coordinates he whishes to keep track of.
Type of the exported spectrum : you may export the **Signal** (i.e the spectrum itself; this is the default), but also the **S/N ratio** (the S/N ratio spectrum is recorded), or the **Noise** spectrum (thus the noise spectrum is recorded).

Options

Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see button [Start] in the left menu).
Save values :
All the input values (files names, etc...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Export spectra from datacube* window is opened.
Recall values :
The values saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

14.2 Import spectra as a datacube

Principles

This feature is provided to allow the user to reimport into a datacube the spectra he exported from another datacube as a 2D image using function [Export spectra from datacube] (see preceding paragraph). The idea is that the exported 2D image has been worked on using some external package, and is now ready to be reimported to continue with XOasis reduction process.

Use

Click on [Import/Export] in the main menu, then on [Import spectra into datacube]. This window pops up.
Enter the name of the **2D Input image**, the associated **Input table**, and of the **Output datacube**.
Click on [Accept].

Parameters

2D Input image : this is the image which has been built using the Export spectra from datacube function, option [All spectra]. No other input format is supported. You can type the name in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.

Input table : this is the name of the table, holding the spatial information, which has been automatically created at the same time as the 2D image; it was named *tab_{name-of-the-2D-image}*. But it may be any table as long as it holds the right information in the right place. Same input possibilities.
Output datacube : the name of the new datacube which will be created using the 2D image and table data. Same input possibilities.
Input type : in the current release, there is only one possibility, that is **2D input**; so, do not mind...
X label, Y label : the labels of the table columns which are to be taken as the coordinates for the reconstruction of the datacube from the 2D image. It is usually [A,D], but it may change if the user worked on the table to perform some coordinate transformation, creating new columns.

Options

Debug : this switches the program to verbose mode, and more informations are recorded into the history file (see button [Start] in the left menu).
Save values :
All the input values (files names, etc...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Import spectra as a datacube* window is opened.
Recall values :
The values saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

14.3 File format conversion

Principles

This tools allows the user to convert to and from MIDAS, IRAF, ASCII FITS and binary FITS format.

Use

Click on [Import/Export] in the main menu, then on [File exchange]. The [File exchange] window pops up.
Enter the name of the **Input file**, and of the **Output file**. You can type them in directly, or use the browse icons at the end of the field, or drag and drop them from the Reduction folder.
Choose the **Input format** and the **Output format**.
Click on [Accept].

Optional parameters

Save values :
All the input values (files names, etc...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Convert file format* window is opened.
Recall values :
The values saved by the user, are loaded to the various input fields.
Default values :
The input fields are set to the general defaults values; for instance, the file names are set to blank.

15. Tools

15.1 Observation logbook

This tool is provided to allow the user to manage the original data file set, as it came from the OASIS observing run. Usually, one uses it to separate the various files in separate directories, according to their scenario and to their exposure class. Then the observation logbook is closed for ever...
It makes use of the observation logbook built on-line during the observing run. This file has unfortunately to be saved separately, as the PEGASUS interface offers to save data files, but *not* the logbook. This is unfortunate, but if you find, later in your institute, that you have lost the original observation logbook, do not panic : the [File]/[Build logbook] button allows the reconstruction of its contents, provided you are able to download ALL your files in a single directory (may need several gigabytes).
The **Observation logbook** features are fully described in the getting started section.

15.2 Reduction folder

This tool is the one which is used throughout the reduction process to keep track of the various input/output files. One instance is opened in each of the various directories where the different scenario and service exposures are reduced separately. The display is updated each time a new file is created, or each time a file is deleted; an update may be forced anytime.
The **Reduction folder** features are fully described in the getting started section.

15.3 Compute expression

Principles

This tool is provided to compute mathematical expression, using data from images, tables or datacubes.

Use

Click on [Tools] in the main menu, then on [Compute expression]. The [Compute expression] window pops up, displaying currently available mathematical functions in the mathematical expression.
Enter the name of the **file** where the result is to be stored (just before the equal sign).
Enter the **expression** to evaluate.
Click on [Accept].

Parameters

File name : the name of the file where to store the evaluated expression, or the filename

followed by the name of the column if the result is stored in a table *(i.e. table,column)*.
Expression : the expression to evaluate. Frames, column from tables and spectrum from datacubes can be combined as long as the dimensions are consistent.

How to write the mathematical expression ?

Use *standard mathematical operators* and *functions*,
Use the *file names* for frames or datacubes,
Use the *file name*, followed by the *column name* for tables, i.e. *table,column* (see examples below),

The available functions are :
log neperian logarithm
log10 decimal logarithm
exp exponential
sqr square
sqrt square root
abs absolute value
sin sine of an angle
cos cosine of an angle
tan tangent of an angle
asin arc sine
acos arc cosine
atan arc tangent
sinh hyperbolic sine
cosh hyperbolic cosine
tanh hyperbolic tangent

Examples of valid syntax :

556755t.fits / (3*log(556756t.fits))
pHIINa.tig - sky.fits
table1.fits,I1 - table2.fits,I2

Note : Currently, filenames cannot contain '/' characters since it figures the divide operation. The syntax analysis need to be improved so such filenames could be entered.

15.4 Get info regarding ...

15.4.1 A datacube :

Principles

This function allows the user to display either characteristics and statistics regarding a given datacube, and/or the complete characteristics of a given spectrum.

Use

Click on [Tools] in the main menu, then on [Get information about...], then on [Datacube]. The [Display datacube information] window pops up.
Enter the name of the **Input datacube**.
Check, if you want, the **Statistics** button.
Identify the spectrum from which you want to get information.
Click on [Accept].

Parameters

Datacube : the name of the datacube you will to get information about, or where the spectrum you are interested in is recorded. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Statistics : if you check that, a statistics is performed and the results displayed (min, max, wavelength limits, and so on...).
Display information about a spectrum identified by... : you may choose to use :
Lens # : and you are requested to give the number of the lens>
CCD pixels : and you are requested to give the pixel X and Y coordinated (approximated) of the point you are interested in.
Sky arcsec : and you are requested to give the relative sky coordinates [alpha,delta, measured in arcseconds from the center of the field] of the point.
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (datacube name, coordinates, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Display datacube information] window is opened.
Recall values : The values (datacube name, coordinates, ...) saved by the user are loaded to the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the datacube name is set to blank.

15.4.2 A 1D or 2D file

Principles

This tool is used to get information regarding a 1D or 2D frame, either a single spectrum, or a CCD image, or a reconstructed image. This is done by reading the descriptors in the file header.

Use

Click on [Tools] in the main menu, then on [Get information about...], then on [1D or 2D file]. The [Display image information] window pops up.
Enter the name of the **Input frame**.

Choose the amount of information you want.
Click on [Accept].

Parameters

CCD frame or image : the name of the 1D or 2D frame to be read. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Informations to be displayed : you may choose to display :
All : all the descriptors are displayed.
Telescope/AOB/OASIS/CCD/Object/Process : the display is restricted to the category (categories) of your choice. [Process] means that you get an idea of the processing steps which conducted to this 2D-frame.
Debug : This switches the program to verbose mode, and more informations are recorded into the history file (see Getting started).
Save values : All the input values (frame name, display options, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Display datacube information] window is opened.
Recall values : The values (frame name, display options, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the frame name is set to blank.

15.4.3 A table

Principles

This tool is used to get information regarding a table, and this is done by reading the descriptors in the file header, or to plot column data from this table.

Use

Click on [Tools] in the main menu, then on [Get information about...], then on [Table].
The [Display table information] window pops up.
Enter the name of the **Input table**.
Click on [Accept]. Information is displayed in the two formerly blank **Table information** and **No**/**Label**/**Type**/**Format**/**Unit** windows.
You are then allowed to specify a couple of table column names, say X and Y, and click on [Plot] to see Y = Y(X).

Parameters

Table name : the name of the table to be read. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.

Columns to plot : if the user wants to plot, he must give here the names (labels) of the X and Y columns to be plotted. He may specify a **Symbol** to be used in place of the regular cross. For instance, one may want to specify *Symbol = NO*, where NO is the label of the column holding the lens numbers, and each point on the plot will be identified by the lens number.
Save values : All the input values (table name, columns to print, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Display table information] window is opened.
Recall values : The values (table name, columns to print, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the table name is set to blank.

15.5 Access to file descriptors

Principles

Theses utilities allow the user to read/write or delete descriptors into file headers. The write action allows both to change the value of an existing descriptor, or to add a new descriptor to a file.

Use

Click on [Tools] the main menu, then on [Descriptor access]. The [Descriptor access] window pops up.
Enter the file name, choose the action to be performed.
If necessary, give the name and the value of the descriptor.
Click on accept.

Parameters

File name : the name of the file where to read/write the descriptor(s). You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Mode : you want to read, or write, a descriptor? Check whatever you want...
Descriptor name : either an existing name for read of modify actions, or a new descriptor name. In a read action, if you enter ALL, all the existing descriptors are displayed along with their values.
Descriptor value : for the modify or read action, specify here the value to be written in this descriptor. Be careful to fit the descriptor type (char, short, int, long, float, double).
Descriptor data type : if you create a new descriptor, specify here the data type which it is supposed to hold : char, short, int, long, float, double, with the usual meanings.
Save values : All the input values (file and descriptor names, values, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the [Descriptor access] window is opened.

Recall values : The values (file and descriptor names, values, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; for instance, the names are set to blank.

15.6 Access to table data

15.6.1 Table selection

Principles

This tool is provided to allow the user to restrict a table (and, if there is one, the associated datacube) to a defined subset. This subset may be defined, for instance, as a sub-field, or as "all the lenses with a S/N ratio above N", and so on... No datum is deleted from the table, but some lines are flagged as "to be skipped". It is always possible to go back to the "no selection" status.

Use

Click on **Tools** in the main menu, then on **Table access**, then on **Select**. The *Table selection* window pops up.
Give the name of the **Input table**.
Give the **Selection criterium**.
Click on **Accept**.

Parameters

Table name : the name of the table to be selected (restricted). You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Selection criterium : the criterium to be applied. By entering "ALL" (case-unsensitive), every selection is cancelled. Valid conditions are entered as in the following examples :
No.gt.500
X0.lt.123.and.N2.gt.2.856
Cont.lt.2563.12.or.A.gt.D
(Cont.lt.2563.12.or.A.gt.D).and.(No.le.1089)
etc...
where No, X0, N2, Cont, A and D are supposed to be valid (existing) column labels; the labels are case-unsensitive. The comparison operators allowed are :
.gt. (greater than)
.ge. (greater than or equal to)
.lt. (lower than)
.le. (lower than or equal to)
.eq. (equal to)
.ne. (non equal to)

Boolean operators .and. and .or. are supported. No arithmetics is allowed in the criterium string.
Save values : All the input values (table name, selection criterium, ...) are saved, and become the new default values for this user. They can be recalled at will, and are used each time the *Table select* window is opened.
Recall values : The values (table name, selection criterium, ...) saved by the user are loaded into the various input fields.
Default values : The input fields are set to the general defaults values; the table name and the selection criterium string are set to blank.

15.6.2 Table reading

Principles

This tool displays the data contents of a table, entirely or over a given subset of columns. The table display may have been previously line-restricted using the Table selection function described in the previous section.

Use

Click on [Tools] in the main menu, then on [Read]. The [Table reading] window pops up. Enter the **Table name**, and the name(s) of the **Column(s) to be displayed**, separated by blanks.
Click [Accept]. The contents of the column(s) is displayed in the same [Table reading] window.

Parameters

Table name : the name of the table to be read. You can type it in directly, or use the browse icon at the end of the field, or drag and drop it from the Reduction folder.
Column(s) to be displayed : the list of the labels of the columns to be displayed, separated by blanks, as in this example :
No X0 A J
Note that columns labels are case-unsensitive, and that you can use the special identifier ALL to mean that you want all the columns to be read.

15.6.3 Table edition

Principles

This tool allows the user to make any modification on any FITS table. This is a powerful tool, to be used with caution, as one may guess...

Use

In the main menu, click on [Tools], then on [Table access], then on [Edit].
The *Table editor* window pops up.
Use the **File** menu item to open, save, close, ... FITS tables.
Use the **Edit** menu item to add or remove lines or columns.
Use the **View** menu item to sort the table according to some criterium.
Click on any cell to activate it and modify the corresponding datum.

15.7 Set/unset super class

Principles

Each file created in XOasis has a special descriptor, which is **File class**. This descriptor allows the software to know that this file is a raw datacube, or a calibrated table, or a sky-subtracted datacube, and so on... Before performing any operation on any file, XOasis checks that this file pertains to the right class, and will so behave correctly during the processing. But for some odd reason it may be useful to perform some operation on a file which has not been previously processed in the usual way. This may be done by temporarily promoting the file to a special file class named **Superclass**. If the corresponding flag is found in the descriptors of a file, XOasis no longer checks the adequation of the file to the process called. Note that the regular *File class* is not destroyed by this action, and is restored as soon as you unset **Superclass**.

Use

In the main menu, click on [Tools], then on [Set/unset superclass]
In the *File class* window which pops up, enter the **File name**, and set/unset the **Superclass mode**.
Click on **Accept**.

15.8 New shell window

Principles

A new shell window is opened, with the tremendous advantage that it is opened in the directory you are currently working with XOasis. Handy to list, move, rename, etc... files, and messing up everything.

Use

In the main menu, click on **Tools**, then on **New shell window**. Magic ! A new shell window pops up ! Unless you manage to put into your .login file some command like "cd ~user", of course...

16. Display tool

16.1 2D image display

Click on **Display** in the main menu, then on **Image**. The RTD (Real Time Display, from ESO, but customized to meet XOasis needs) window opens :
The main window displays the image which has been chosen using the **File** menu, **Open** function.
The instantaneous sky coordinates of the cursor are displayed in the upper part of the display.
The **Color Map** and **Intensity Transfer Table** are selected using the **View** menu, **Colors** function.
The **Cuts** of the display are set using the **View** menu, **Cuts** function.
The upper left windows shows the displayed zone as a small rectangle, whose size shrinks if you apply a zooming factor on the display.
The **Zoom** factor is chosen using the [**Z** : zoom in] and [**z** : zoom out] buttons.
The upper right windows allows to **Pan** the display window over the object, using the mouse.
Always click on **Auto set cut levels** to get a good view of low-level details.
The **Select** menu item is used to graphically select a spatial region of a datacube. This function is described at length in section Sky subtraction of this manual.
The user is encouraged to play with RTD to discover the various possibilities; and we are sorry about the color palette management, which is old-Unix style, with colors complemented as soon as the mouse cursor leaves the RTD window...

16.2 Datacube explorer

Principles

This tool is provided to allow the user to check any particular spectrum recorded in a given datacube, with the help of a graphical interface allowing the easy selection of the spectrum of interest. An image, built using the Analyse data or the Image main menu item, must be provided to be used as a spatial reference frame.

Use

Click on [Display] in the main menu, then on [Datacube]. The [3D explorer] window opens; it is based on RTD (see above section).

Menus

Three menus items are provided to handle data exploration :
File :
Open frame : Give the name of the 2D frame you reconstructed (see above paragraph) from data recorded into the datacube to be explored, or from data recorded into any other

datacube linked to the same mother table. This frame will be used as spatial reference. As soon as it is loaded, the instantaneous sky coordinates of the cursor are displayed in the upper part of the display.
Open datacube : Enter the name of the datacube you want to explore. Once the datacube is opened (the idle time at the beginning of the operation is normal : it corresponds to the time needed by the program to build a correspondance table of sky coordinates and lens numbers). Then, if you left-click on any point of the displayed reference image, the nearest lens is found, and the corresponding spectrum is displayed in a graphic window. The number of the lens/spectrum is displayed, too. This is extremely useful, at the end of the reduction process, to identify any spectroscopic defect (abnormal spectra, undetected cosmics, ...) which shows up on reconstructed images, or to wander through the local spectral characteristics of an extended object...
Clear : Clears the display windows of the reference image frame.
Exit : As you may guess...

View :
Colors : Use this to set the lookup table (LUT) of your choice. *Rainbow* and *heat* are among the favourites... Some of the ugliest LUTs ever designed are provided, you will soon notice. You may, too, choose the intensity transfer table (ITT), and use either *Log* or *Ramp* (linear) level encoding.
Cut levels : Use this to set the cut levels of the display.
Cuts : Use this to draw a line through the 2D frame, and then plot the object profile along this line.
Pick object : Use this to isolate a square area of the 2D frame (set the window size with the **Sampling size** cursor), and obtain various informations regarding this area : position, intensity and FWHMs of the peak enclosed in the window, statistics, ...
FITS header : Use this to list the FITS header of the displayed 2D frame.
Lenses : an XOasis special. If you click on that, you get the complete figure of the lens array, superimposed over the reference image. Usefull, but may be a complete mess if you forget to zoom in before.

Graphics :
Toolbox : Use this to improve the image displayed adding lines, arrows, shapes, labels, and so on, before capturing the display.
Clear : As you guess, clears the graphic overlay.

16.3 Spectrum and table display

Principles

This tool is used to plot various quantities stored into datacubes, 1D spectra, or tables. Note that successive plots may be overlaid using the **Overplot** function instead of **Plot**. Line styles and colors may be changed at will.

Use

Click on [Display> in the main menu, then on [Spectrum and table]. The *Graphics display* window pops up.
Use the **File** menu item to load the file you want to plot :
Plot datacube : as a datacube cannot really be plotted as a whole, the user is asked to give also the number of the **lens** for which the spectrum will be displayed. Instead of the spectrum data, one may ask for a display of the **Noise variance**, or of the **S/N** ratio.
Plot spectrum : this time, the name of the 1D file has to be entered into the input field.
Plot table : enter the name of the **Table to plot**, of the two data columns (**X taken from column...** and **Y taken from column...**). You can choose the **Symbolshape** and the **Symbol size** used for each data point, or use the content of a column to mark the position of each data point (**Label each data point with column...**). As an example, to plot the lenses map, use as **Table to plot** the main table of the exposure, created at spectra extraction. Set **X taken from column...** = A, **Y taken from column...** = D, **Label each data point using column...** = no, and click on **Plot with label**. May not be very readable, as there are quite a lot of lenses... Use the zoom function : left click and hold, drag the window, release the left button; yes, like in Windows (Ooooops! Copyright, trade mark, and so on...). Right click to de-zoom.

Use the **Options** menu item to set the following :
Set bounds : enter the four values which must limit the plot area. Click on **Done**. Click then on **Man X** and **Man Y** at the bottom of the *Graphics window*, and replot.

17. Reduction logbook

Principles

This tool provide the user an easy-to maintain, easy to edit, log file. If the right option is checked into the [Setup] / [Log] function, all the commands are logged to this file.
The contents of the [Result] window, after completion of any reduction step, may be exported to this file.
In addition, it remains freely editable, and can be saved, renamed, and so on, at any moment. Since the 4.5 version of XOasis, the file is stored in HTML format, so is now compatible with a great among of tools.

Use

The use is straightforward, and is described into section Reduction logbook of this manual.

18. Setting user’s preferences

18.1 Current work directory

It is possible to change the work directory without leaving XOasis.
Enter the name of the new one, or browse the directory tree displayed on the left of the window, and click on your choice, then on **OK**. After that, do not forget to use **File**, **Scan directory** to update the Reduction folder display.
Note that the various parameter files holding the default values of the user inputs to the various programs are specific to each directory.

18.2 User's general preferences

18.2.1 File format and overwrite

File format : For files just brought back from an observing run at CFHT, this is **Binary FITS**. [MIDAS] is for ESO's MIDAS package. In a future release, [IRAF] format will be supported too.
Note that the XOasis implementation of binary FITS format is able to read ASCII FITS format as well.
File overwrite : You get here the choice to have the existing file being silently overwritten, or to be warned each time an overwrite is needed.

18.2.2 Debug mode

Activating the **XOasis debug mode** mode switches the program to verbose mode. All the actions are traced, and written in an ASCII file named .XOasis-debug.{user}, situated in the current directory. The **XOasis kernel information** button opens a window where release dates of the various programs are displayed, easing remote debugging.

18.2.3 History file

As the reduction process goes on, a trace is kept of every action performed, with all the optional parameter values, to allow later inspection. Everything is written in a file named .XOasis.history.{user}, situated in the current directory. As this file may grow really big, it is possible at this point to select automatic erasing of the file once it reaches a given size.
This file is for debugging purposes only, and stays unreadable for within XOasis. It can later be read, off-line, with any text editor.

18.2.4 Miscellaneous settings

You may specify here :
The command to be used to compress files.
The command to be used to uncompress files.
The local command to be used to print plain ASCII files.
The circumstances where a **beep** should be emitted. **On error or at end** is a good choice, which will warn you upon completion of long lasting operations.
Reset XOasis parameters to default values does what it means. This applies to all the parameters which may be set in the File format, File overwrite, Debug, History, Misc, Log, Assistant, User and Help folders.

The icon to the right of the text zones allows you to apply the given command to the file of your choice.

18.2.5 User's reduction logfile

If you check **Store commands into log file**, every command you give is logged into the *Reduction logbook*. If you want the output information to be logged too, you have to click on the save button in the result window of the action just performed.

18.2.6 User's assistant : concise on-the-fly help

If the XOasis on-line assistant (nicknamed Calvin) is loaded, then the following options become effective :
Pop-up main menu information : when the cursor is set on a main menu button, a few (supposed) informative words are displayed.
Stay with me through each reduction step : the Calvin windows permanently displays information during the reduction process.
Pop-up input field information : when the cursor is set on an input field, a few informative words are displayed.
Display help on entering each reduction step : when a main menu function (i.e. CCD, Mask, Extract, and so on) is selected, the Calvin windows displays a short help about this function.
Load Calvin at XOasis start : the assistant window is opened by default, giving you easy access to the short-form help.

18.2.7 User's name

Enter here the name to be used for personnalised messages...

18.2.8 On-line help browser

You may provide here :
The name of your favourite **WEB browser**
The **OASIS help path**

This information will be used to display the in-line help while using XOasis. Note that this concerns the full scale help, the one which you are reading, not the short form built-in XOasis help. See the download section of the present manual.
Note, too, that it is visually MUCH better to start your browser BEFORE starting XOasis, to avoid this greedy program to steal all the color resources of the system, leaving a very strange-looking WEB manual. So, not using this otherwise nice possibility of XOasis is a good idea !

18.2.9 Windows size

You may specify here the default size of the reduction folder window; but it can be resized anytime with the mouse... Later, it may work on other windows too; that is for the "s"...