

BRUKER MICROCT

NRecon User Manual

Bruker microCT

July, 2016

User manual for SkyScan reconstruction program NRecon**Index**

- Introduction: [NRecon in a nutshell](#)
- Chapter 1. [Installation, configuration and license](#)
- Chapter 2. [General layout and usage of the software](#)
- 2.1 [Start the cluster reconstruction program](#)
- 2.2 [Layout and walk around](#)
- 2.2.1 [The *Start* page](#)
- 2.2.2 [The *Settings* and *Advanced* page](#)
- 2.2.3 [The *Output* page](#)
- 2.2.4 [The *Summary* page](#)
- 2.2.5 [The *batch manager*](#)
- 2.2.6 [Optional functions](#)
- 2.3 [Customize the application](#)
- Chapter 3. [Data format and conventions](#)
- Chapter 4. [How to import a non-SkyScan dataset](#)
- Appendix I. [GPUReconServer installation guide](#)

Introduction: NRecon in a nutshell

SkyScan NRecon package is used to reconstruct cross-section images from tomography projection images, mainly the cone-beam X-ray projections. The algorithm implemented is the Feldkamp algorithm. The native SkyScan data are recognized automatically. This program can also reconstruct non-native tomography data, if the file format and file names follow the conventions described in [chapter 3](#), and a corresponding log file is created following the instruction described in [chapter 4](#).

The program is provided together with a help file with detailed information on how to use the software. This additional document serves as a manual for quick startup, and it overlaps largely with the online help. There are also a number of related technical notes available upon request.

The reconstruction software package consists of 2 programs: the user interface program NRecon and a reconstruction engine, either a CPU version NReconServer or a GPU version GPURconServer. The two programs can run either on the same computer or on a cluster of computers. The communication between NRecon and NReconServer/GPURconServer uses a client-server model (a common port). If windows firewall is enabled, you will be asked to unblock it for the server program. Data are shared among the PCs using shared directory via network: please make sure that all PCs involved have access to both input and output directories.

A typical configuration (see Figure 1) is a combination of 1 host computer and N ($N=2,3,4,\dots$) reconstruction nodes, as *config. 1* shown below. Or, you can run it as in *config. 2*, where the host PC and node PC can refer to either the same machine or different machines. Ideally, the node PCs should be high-definition: dual (or more) processors, large RAM (minimum 1GB if 8Kx8K reconstruction will be performed), fast FSB, minimum 40GB hard disk (excluding data storage) if 8Kx8K reconstruction will be performed. For the GPUserver, the requirements for the node PC are similar, but the PC has to be 64-bit, and the GPU's should be CUDA-enabled cards from NVIDIA.

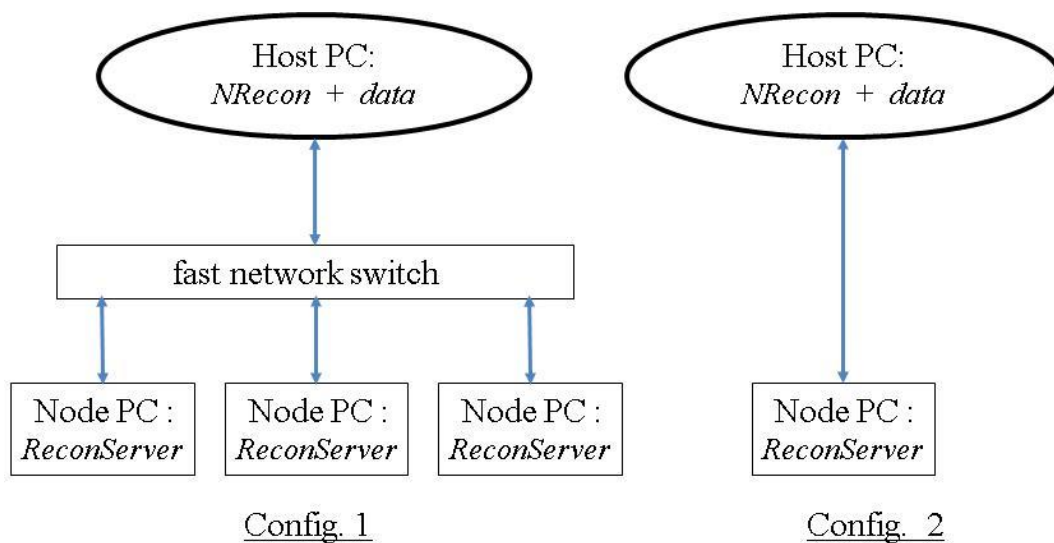


Figure 1 Typical layouts of NRecon/ReconServer: the ReconServer

Chapter 1 Installation, configuration and license

It is rather straightforward to install and configure the reconstruction program NRecon/NReconServer: simply copy the programs to your destination folder. More details can be found in the installation guide which is part of the online help.

For the installation of GPURconServer and a quick guide, please refer to [appendix I: GPURconServer installation guide](#).

The programs run under Windows operating systems (32-bit and 64-bit). It is important to install the 64-bit server program on a 64-bit computer to ensure maximum performance. GPURconServer is only provided in 64-bit, therefore it will not run on a 32-bit PC. For the user interface program NRecon, only 32-bit version is provided, which runs on both 32-bit and 64-bit systems.

There are 2 versions of NRecon/NReconServer/GPUReconServer available: the local and distributed/cluster version. The local version is free.

The local/free version: please download and install the local NRecon/NReconServer/GPUReconServer. There is no limitation in functionality. As the name suggests, both programs have to run on the same computer. For each NRecon, only one server can be connected to. For GPUReconServer, the maximum number of GPU's for reconstruction is limited to two for the free version.

The distributed/cluster version: you need a license to run this copy. To apply for a license, please note down the product ID (found in menu Help->About NRecon ...; or in the warning message when no license or an invalid license is found by a distributed version), and send the ID together with a user name and your institution/company name to your distributor or to SkyScan. More information can be found on our website: <http://www.skyscan.be>. Note that there are 2 types of license: CPU license and GPU license.

To install license: the NRecon.lic has to be in the same directory as NRecon.exe. By the first startup, you need to fill in the product ID, name and company name exactly as you received from SkyScan (case-sensitive; please use copy-and-paste to avoid typing errors). The file name for both CPU and GPU licenses are identical, so that you use either CPU license or GPU license, not both together.

You can also run the cluster version software without a license. Then you will be operating in 1-to-1 cluster mode. This allows you to run NRecon on one PC, and the server on another one. As the name suggests, only one server is allowed to be connected to NRecon at any given moment. The advantage of this configuration is that you can use a dedicated powerful node PC for reconstruction, while running the user interface program NRecon on your own desktop. For GPU server, the rule of maximum 2 GPU's still applies. Attention: if you have a CPU license, you will not be able to run in 1-to-1 cluster mode for GPU server. In this case, simply set up NRecon in a different folder without the license file.

Chapter 2 General layout and usage of the software

If you have a local version, just start up NRecon: the server program will be started up automatically if it was not yet on. You may skip 2.1 which is specific for distributed/cluster version.

The local version of NReconServer/GPUReconServer will only accept connection requested by NRecon running on the same computer.

In case of multi-user environment, it is recommended to run only one copy of the server program at the same time: the computation resources are then used more optimally.

2.1 Start the cluster reconstruction program and the client-server connections

In general, the server program should be started first, but it is not strictly necessary. If you exit the server program, turn off one of the node PCs or unplug one of the network cables while NRecon is on and connected, NRecon will be informed and show the status

correctly. However, if you start up a server program while NRecon is already on, you need to re-establish the connection.

Neither NRecon nor NReconServer/GPUReconServer allows more than one copy running on the same computer. In case of multi-user workstation (multiple users can log on simultaneously), only one copy of the server program can be started at a fixed port number: it is then recommended to install the server program only on one (common) account, so that you do not have to worry about different port number or multiple instances of the server program on a single machine.

The server only "serves" one NRecon at one time (the active client), but it does hold a numbered waiting list and it will inform all connections (the connected NRecon programs) whenever this status changes. An active client gives up its "privilege" only when it disconnects from the server, and the first waiting client will be promoted to be the active client. In case of multiple NRecon instances, communication among the users may be required to use the limited computer resource efficiently. For a smooth operation in a multi-user environment, it is recommended to disconnect the servers whenever they are not needed any more.

2.2 Layout and walk around

Figure 2 shows the typical appearance of NRecon and NReconServer. Little explanation is needed for NReconServer: only the communication port number and the number of threads can be altered by the users. The port number should be the same as that used in NRecon (default to 1) and usually does not require modification. The number of threads influences reconstruction speed.

The layout of NRecon is more complicated. There are a few windows on the application window of NRecon: The main display where the images are shown, the reconstruction window which hosts a few pages/tabs (start, settings, advanced, output, summary) for reconstruction purposes, the connection window which shows the status and progress of all servers, the color window which allows you to view the images in the main display in different color and scale. Except the main display, all other windows are so-called dockable windows: you can drag them out of the main window, or dock them differently than they are shown here. You may customize the appearance of NRecon to your taste and convenience like most windows programs: the last used layout will be saved and loaded automatically at next startup. You can also show/hide these dockable windows by closing the individual windows or via the *View* menu.

Since there is only one display area, a few display modes are used: navigation mode, alignment mode, preview mode, image mode and idle mode. These display modes are coupled to the different pages/tabs on the reconstruction window. In other words, switching from one display mode to another is done by switching from one page to another on the reconstruction window.

The color window is also coupled to the display modes, i.e., each display mode has got its own settings for display.

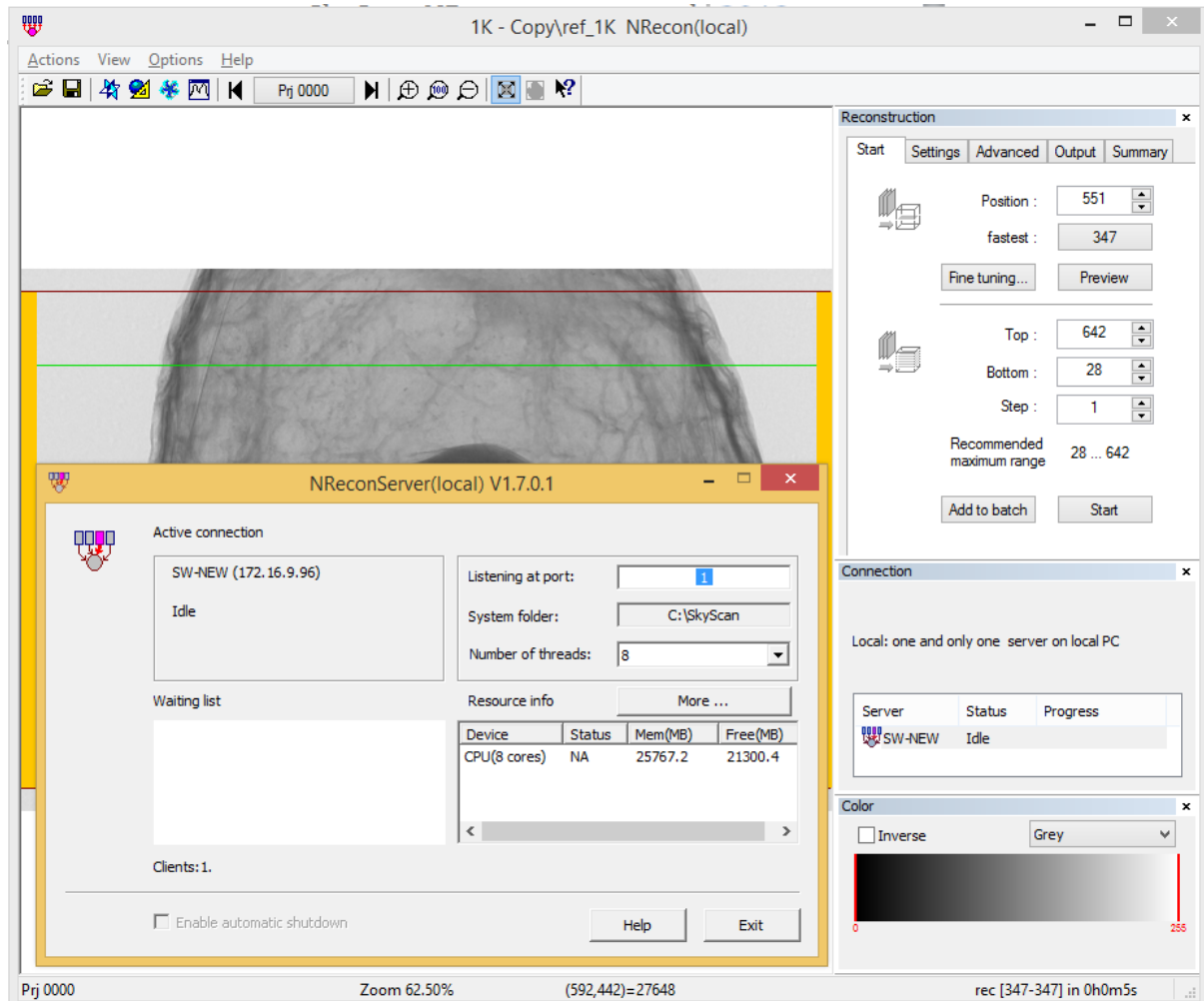


Figure 2 Typical appearance of NRecon + NReconServer: the local version

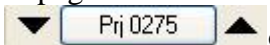
A general scenario of reconstruction is the following: after loading a dataset, NRecon will be set at the [Start page](#) and the display will be in the navigation mode, where the projections can be viewed; go to the [Settings page](#) to check the parameters, usually the display will be in navigation mode to allow you to check the alignment; go back to the [Start page](#) to do a preview; NRecon goes automatically to the [Output page](#) after reconstructing a slice for previewing, the display will be set at previewing mode, and you can modify the parameters concerning final output image; You may go the [Advanced page](#) for further parameters adjustment, or to the [Summary page](#) to view the scan parameters and reconstruction parameters; once all parameters are set, go back to [Start page](#) to start the full reconstruction or send the reconstruction task to the batch manager.

The sections below will give more details on the different pages on the reconstruction window and the batch manager.

2.2.1 The Start page

This is the default page when a dataset is loaded, if no post-alignment is set automatically at loading time.

This window/page offers possibility to select axial reconstruction range, to start a reconstruction or to submit a batch job. It is coupled to the main display: setting this page active will set the main display to *navigation mode* in which a shadow image will be shown.

Figure 2 is a snapshot taken when this page is set active. To view other shadow images, you can click on the up/down arrows  on the toolbar; or invoke a popup window by clicking on the second button.

The green line on the main display in the navigation mode indicates the slice for previewing. The dark-red lines indicate the slice range for volume reconstruction. These lines can be dragged to a desired location (left mouse button). The corresponding slice numbers are also shown on the start page, and can be altered from there as well, by either editing or scrolling. Please note, this range indication is only a rough guide due to the cone-beam geometry. Therefore, one should foresee a safe margin or do a preview to confirm when selecting the slices.

The functions provided on the *Start* page are described here.

➤ **Preview:**

This function is provided to adjust the various parameters. It basically reconstructs one slice of choice. To change the slice on which a preview reconstruction will be performed, you can drag on the green line with mouse move, use the spin button, or simply type in a slice number.

In preview mode, only the parameters shown on settings/advanced page will be taken into account, while the parameters shown on Output option page will be ignored silently. For example, the region-of-interest option is ignored so that you can draw ROI on the full preview image. When preview reconstruction finishes, the Output option page will be automatically set active, offering you the chance to modify a few output parameters for the final results, such as the dynamic image range, output format, scales, color and etc.

The major aim of previewing is to help you adjust parameters. You can always switch back and forth between the pages to adjust them. To help you associate the parameters with result, the slice number (e.g., *Prv 0514*) is shown on the status bar whenever the preview image is in display. If any change occurs which might result in different image quality, a * will be shown (e.g., **Prv 0514*).

➤ **Fine tuning:**

A few reconstruction parameters have to be adjusted manually in a try-and-error fashion using the preview function. It requires quite some effort to save the preview images, view them and then make a decision. The "fine-tuning" function is designed to make this adjustment somewhat easier by launching a series of previews. You may adjust one parameter at a time, while keeping all other parameters fixed. In this way, you may tune 4 parameters: post-alignment, beam-hardening correction, ring artifacts correction level and smoothing level.

Figure 3 shows the popup window appearing upon clicking on the *fine-tune* button. You may choose the parameter to tune, and the parameter values are given such that they are centered on the current value (the valid range of the parameters are taken into account as well) with given step size. Once you have set this, you can click on *Start* to start a series of preview reconstructions.

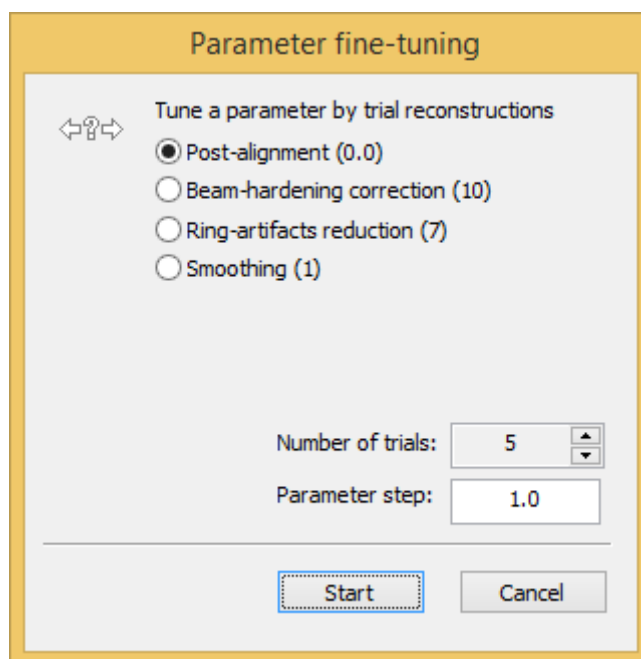






Figure 3 Fine-tune window

Once the preview reconstructions are done, you may use the   buttons on the toolbar to scroll. The parameter is adjusted automatically when you scroll. In other words, simply concentrating on the images until you find a satisfied one, and the fine-tuning function is done. In case of doubt on the consistency, there is text information at the top-right of the image, indicating which preview you are viewing; or go to the **Settings** page, switch the alignment image off, and you will be able to view the preview image and observe how the parameter updates while scrolling. If, accidentally or not, you have changed any parameter which does not correspond to any of the preview images, a * will be shown (e.g., *Prv 0514) on the toolbar, just as in the normal preview case.

➤ *Start*

Obviously, this is the button to start a reconstruction. It checks for disk sufficiency and whether you want to overwrite files if there are already there.

When the first resulting image is available on the disk, it will be loaded automatically and the Overview page will be set active, showing the first reconstructed slice in *image mode*. Same as in *navigation mode*, you may walk through the resulting images by using the   buttons. Please note that the images may appear non-sequentially due to work division among multi-threads and multi-computers.

➤ *Add to batch*

Instead of reconstructing the volume immediately, you can send the reconstruction task to the batch manager. You can start the batch jobs later. [Section 2.3.5](#) has more details about the batch processing.


2.2.2 The *Settings* and *Advanced* page

There are two windows/pages for reconstruction parameters. These two windows offer possibility to change reconstruction parameters for the current dataset. The most frequently used parameters are put in page *Settings*, and other more advanced parameters are grouped together in page *Advanced*.

The *Settings* page can be coupled to the main display in 3 ways: *alignment mode*, *preview mode* or *navigation mode*. The *alignment mode* will be switched on if both the post-alignment option and the show option are checked; otherwise the *preview mode* (if previewing image is available) or the navigation mode will be switched on. You may switch between two modes by check/uncheck the show option: convenient when you are adjusting parameters by looking at the existing preview image.

The *Settings* page is set automatically active after loading if the post-alignment option is checked. Figure 4 shows the settings page with alignment display mode. This display mode is switched on when both the post-alignment option and the show option are checked. The image shown in display is the overlaid image (alpha-blending) of 2 shadow images at opposite positions (180 degrees apart). One of the 2 images is flipped. The intention of this option is to help you choose the correct alignment value at the first stage. However, the 2 images may differ a lot due to the cone-beam geometry and different magnification factors. Therefore, it is always recommended to use the previewing choose the post-alignment value.

The initial/default post-alignment value is estimated at loading time. Instead of comparing the original cone-beam projections, we compare pseudo-parallel projections obtained from the cone-beam projections at 0 and 180 degrees. Only the central portion of the projections are used (you may change the portion of data used for post-alignment in Preferences.). If you interrupt/cancel the rebinning, the original version as described above will be used instead.

The profiling function is partially enabled: clicking on  will show you 2 curves (profiles along the central horizontal line, average over the full vertical length). Together with the profiles, the overlaid image provides a rather interactive way of adjusting the post-alignment value.

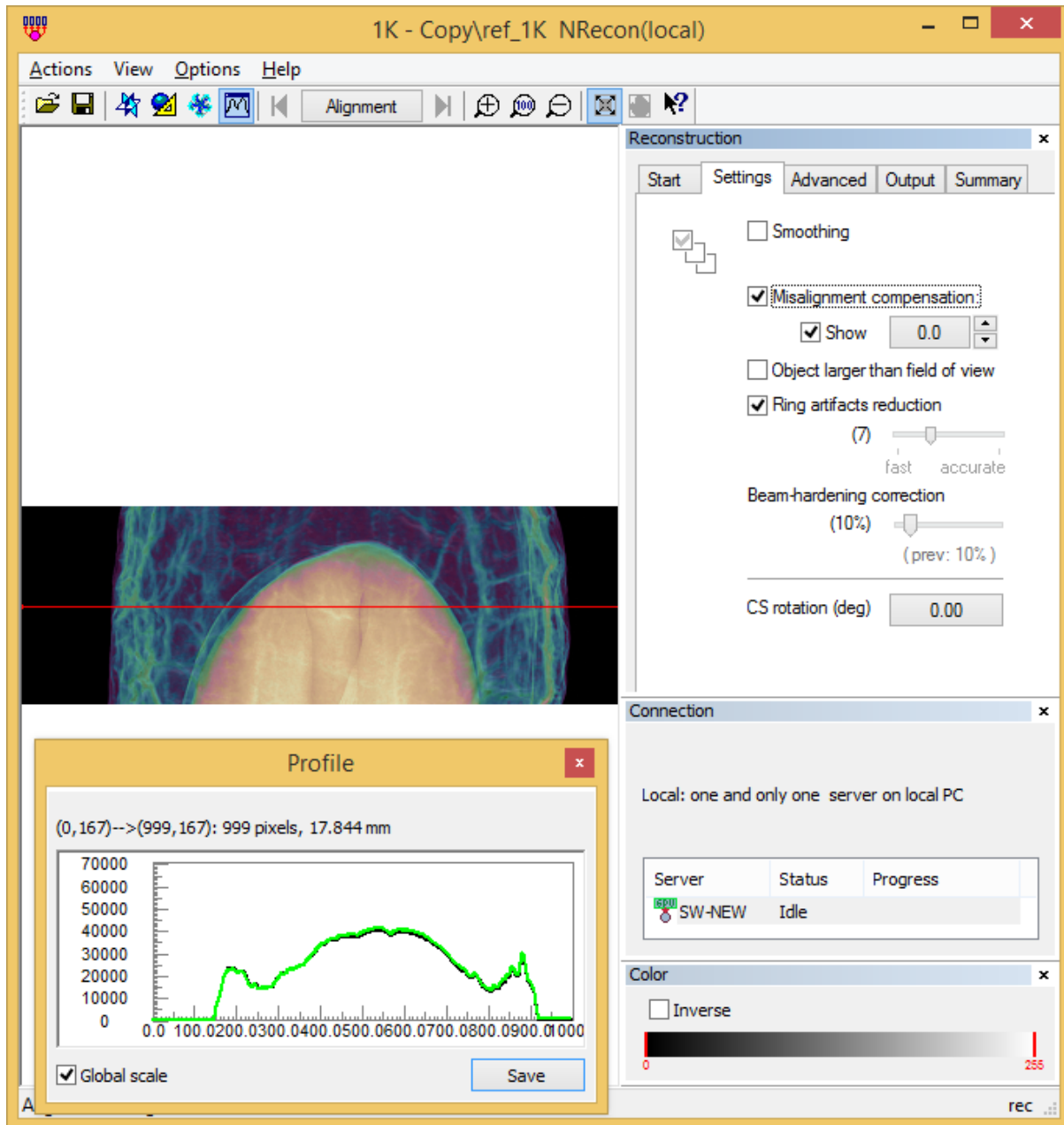


Figure 4 The Settings page with alignment display mode. The profile window is also visible

Figure 5 shows the *Advanced* page coupled with navigation display mode.

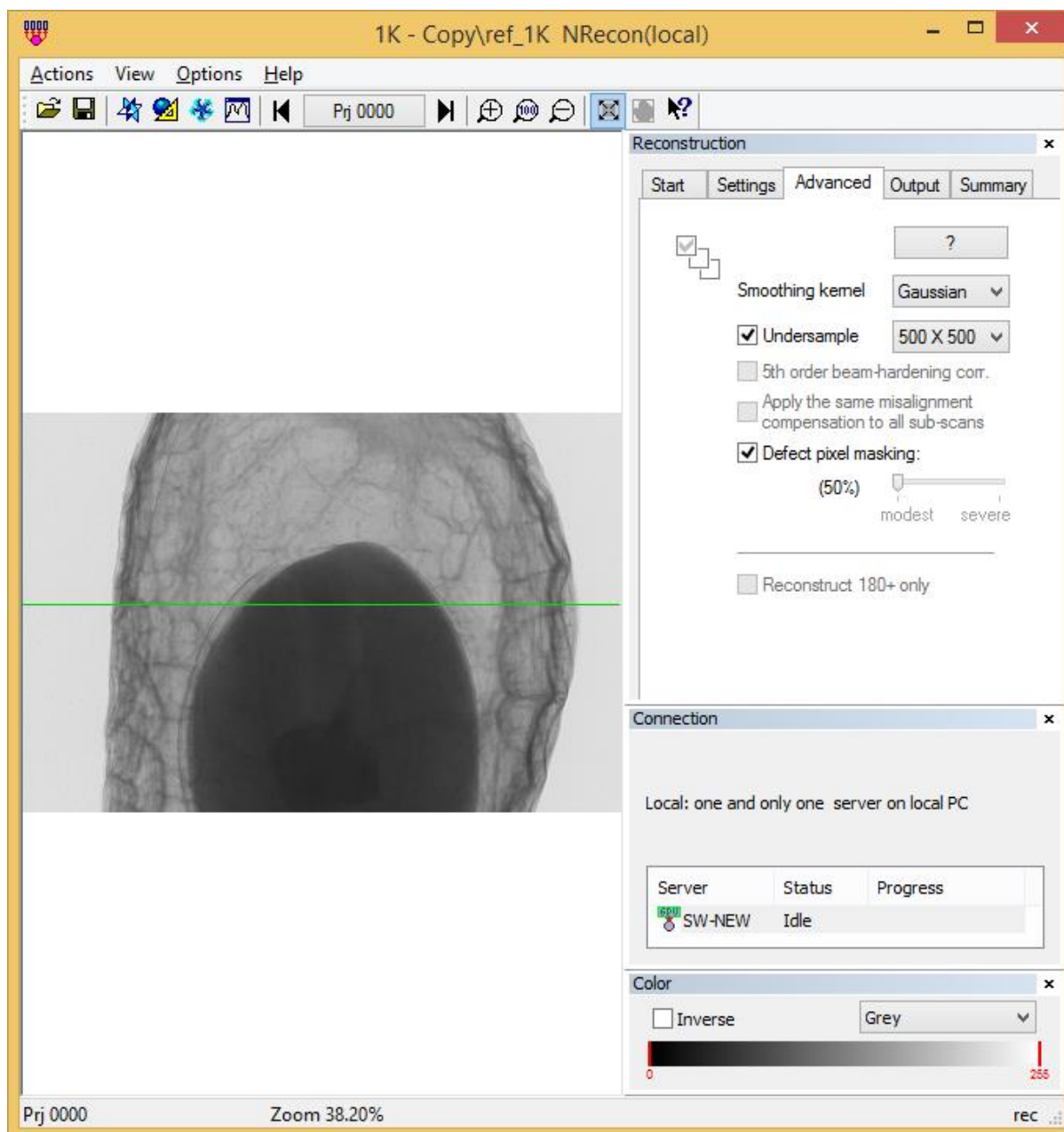


Figure 5 The Advanced page with navigation display mode

When loading a dataset, the default value of each parameter is determined in various ways. Firstly, a few parameters (recommended beam-hardening correction, recommended HU calibration factor) are read from the corresponding configuration file accompanying each dataset if any, and the default post-alignment value is calculated if applicable. The static cross-section rotation and output image format are also assigned according to the settings in *Options->Preferences*. Secondly, an attempt is made to search for a matching entry in the batch manager. If a matching entry is found, the parameters in the batch manager will be loaded exactly as they are and you will be asked if you want the entry in the batch file to be removed or not. If no matching entry is found in the batch manager, an attempt is made to search the corresponding log file (<dataset>.log) for previously used parameters. They are loaded if found.

If no parameter found so far, and the check box "load last-used parameters instead of default protocol" in the preferences window is checked, the last-used parameters will be loaded (except the post-alignment value ---which always takes the calculated value for this dataset, the output format, the scale option and the cross-section rotation), if any at all; otherwise the default protocol will be loaded.

Alternatively, there are a few ways to alter reconstruction parameters altogether rather than modifying them one by one. You can load a set of reconstruction parameters from a valid log file (generated by NRecon), a valid ini file (generated by ConeRec), a valid protocol template file *.rcp (created by NRecon, see [Summary page](#)), or load the last-used parameters specifically. These functions are available through menu Actions.

A detailed description of each parameter is given here.

➤ **Smoothing**

This is applied to the projection. It smoothes each pixel with a MxN neighbourhood, where M is the horizontal dimension and N is the vertical dimension. It reduces noise but may introduce blurring for fine structures. The smoothing level d (number shown on the slide bar) has a range of 1...10. You may use the fine-tuning function to search for a suitable smoothing level. Three smoothing kernels are provided: Box(asym.), Box(sym.) and Gaussian. This can be modified in the *advanced* page. In the preferences, the default smoothing method can be specified. The asymmetrical box function has been the default and the only method in the early versions: the smoothing kernel is a box of size MxN, while $M = 2 \cdot (d/2) + 1$, and $N = 2 \cdot d + 1$. The symmetrical box function: $M = N = 2 \cdot ((d+1)/2) + 1$, and when d is even, the edge pixels in the box have a weight of 0.5, while the inner pixels have a weight of 1. The 2D Gaussian functions are defined as: $\sigma = 0.25 + d \cdot 0.25$ for $d > 0$. In general, the Gaussian kernel is preferred.

➤ **Misalignment compensation (post-alignment)**

This compensates possible misalignment during acquisition. It is an important parameter: wrong alignment compensation would cause tails, doubling or blurring in the reconstructed image; and it may differ slightly from one dataset to another on some scanners. Though the estimation accuracy is improved a lot, it is still a parameter to be adjusted in try-and-error fashion: the fine-tuning function makes this task somewhat easier.

More details can be found in the FAQ list of the online help file coming with NRecon program.

➤ **Object larger than field of view**

This option indicates that the object is larger than field of view. This function only influences the way how the reference (air) intensity is defined. As well known, the shadow image is converted before reconstruction: $\text{value} = \ln(I_0/I)$, where I is the intensity in the shadow image and I_0 is the reference (air) intensity. If this option is not checked (default), I_0 is searched line per line (maximum of the line); otherwise a theoretical maximum (for 16 bit data, $I_0=65535$) is used. Please do not check this option unnecessarily: the resulting image might

be less sharp otherwise. Check this option only when your object extends beyond the FOV boundary at both sides in any of the shadow images!

➤ **Ring artifacts reduction and defect pixel masking**

This option reduces ring artifacts. It is applied on the projections before any other pre-processing steps. The mean projection image in <DatasetPrefix>_arc.tif is used for this purpose: if absent, NRecon will create it automatically. You can select the depth of correction in the slider with 20 positions (1, 2, ..., 20). The value is determined in try-and-error fashion: you may use the fine-tuning function to adjust this parameter. It is generally not recommended to use a unnecessarily high value: blurring may occur.

This method is sufficient for most scans with small variations in camera responses, but has nearly no effect if one camera pixel has varied sensitivity during the same scan (e.g., due to beam-hardening), or is completely dead (0 sensitivity). We call these pixels defect pixels.

For SkyScan X-ray CT scanners, one may specify a threshold to detect the defect pixels in the *Advanced* page by checking button *Defect pixel masking*.

For more details on how a defect pixel is found automatically and how it is corrected, please refer to the FAQ list on the online help.

➤ **Beam-hardening correction**

This option compensates beam-hardening effect by linear transformation in the software. Depth of correction (0, 1, ..., 100) can be selected according to object density. Same as post-alignment, this parameter needs manual adjustment (and experimenting) by visualizing the resulting images (the profiling function will help you during visualization). You may use the fine-tuning function to adjust this parameter.

A higher order (up to 5-th order) polynomial function may be used to do beam hardening correction, but this is considered as a non-standard option. This option can be switched on in the *advanced* page by checking button *5-th order beam hardening corr.* For more details refer to the FAQ list on the online help.

➤ **CS rotation (in deg.)**

This option rotates the resulting cross-section image by a fixed angle. It does not influence the image quality in any way. The value shown is only valid for current dataset: the value may be taken from your preferences, or from the default reconstruction protocol.

Tip: if you want the image to be re-oriented in a certain way, first do a preview, then on the preview image you may draw a line which you wish to be rotated to the horizontal position. On the profile window, the orientation of the line (angle between the line and horizontal direction: 0 - 180 degrees) will be given. The cross-section rotation angle should then be: the current value + line orientation.

➤ **5-th order beam hardening correction**

This option becomes available when it is enabled in the *Options->Preferences*. You may view the current values by clicking on the ? *button*.

➤ **Undersample**

Once checked, a drop-down list will allow you to select the undersampling factor. The undersampling is applied during reconstruction in the back-projection stage.

➤ **Apply the same misalignment compensation to all sub-scans**

This option is only available for connected/oversize scans. It forces to use the same misalignment value for all sub-scans.

➤ **Defect pixel masking**

It is recommended to use this option only in case of "hard" rings. For more details on how a defect pixel is found automatically and how it is corrected, please refer to the FAQ list on the online help.

➤ **Reconstruct 180+ only**

This option instructs the reconstruction to use 180+ degree data only. It is only available if the current dataset is scanned over 360 degrees. You can specify the starting angle (in degree) from which the data can be used.

2.2.3 The Output page

This window/page offers possibilities to adjust parameters which influence the output images in format, size, color or dynamic range. It is coupled to the *preview mode* in the main display if any preview image is available. It is automatically set active after a preview reconstruction is performed.

The color control is also coupled to this page: the color palette shown will be the color palette in the output image (BMP format only). Confusion often arises concerning the option *Inverse*: it is the color palette which inverts, not the pixel values.

The standard output format is 16-bit TIFF, 8-bit BMP, compressed 8-bit PNG and JPG. There is also a possibility to have additional output in one of the 2 selected format (DICOM or F4F). To enable this option, go to Options->preferences.

Figure 6 is a snapshot when the output option page is active and the main display is in preview mode. The preview image at slice 532 is available in full image size, and the user has selected a Region Of Interest (ROI), which can be resized and moved by left-mouse button. The ROi can be in elliptical or circular shape.

The individual options are explained here below.

➤ **Histogram window --- choose the dynamic image range**

This function determines the data dynamic range when converting from real numbers (reconstruction is done using floating-point real numbers) into integers determined by the output file format. The histogram is obtained from the previewing slice only. Therefore, if the object is very non-uniform, it is important to choose a slice which goes through the dense parts of the object; otherwise higher values may be truncated in the final image.

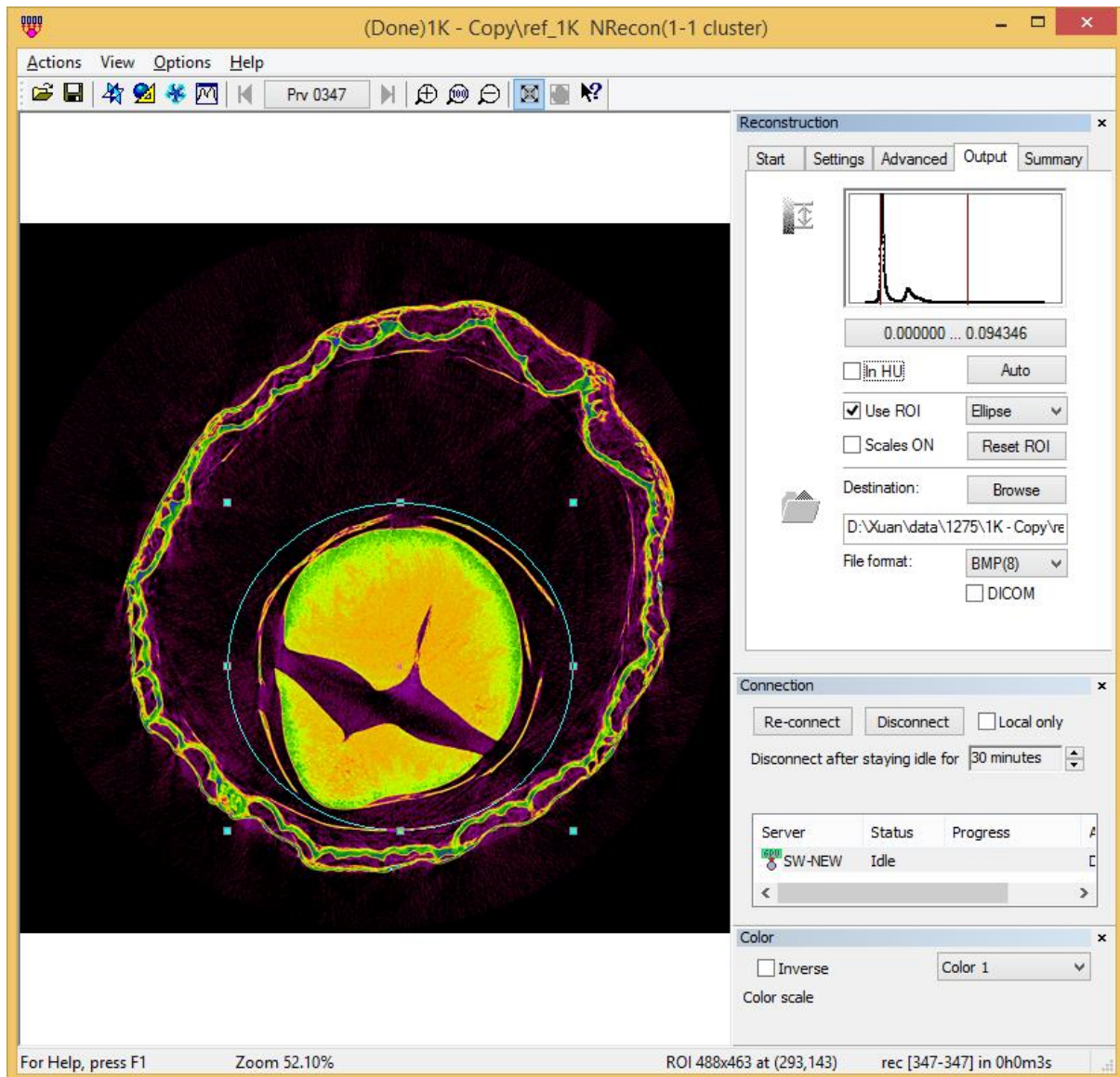


Figure 6 The Output page with preview display mode

The histogram can be displayed either in linear or logarithm scale: double-clicking with left mouse button on the histogram will switch between the 2 modes. The image range can be altered in 3 ways: dragging one of the 2 dark purple lines on the histogram with mouse move (holding left mouse-button); press button *Auto* to get default range; or change the values manually by clicking on the range itself. By manual modification in the range (the 3rd way),

you are allowed to give a range beyond the boundaries of the histogram. If the higher limit is set to be lower or equal to the lower limit, the range will be indicated as "range not defined". In case of "range not defined", a preview reconstruction will be performed automatically to find the "automatic" range before doing any reconstruction.

The "auto" range is determined in an empirical way. The upper limit is defined as 140% of the maximum in the preview image. The lower limit is defined as 3% of the maximum in the preview image. However, if you preset "positive image" in the *Options->Preferences*, the lower limit will be fixed at 0, which is a very reasonable choice in many applications.

➤ In HU

Checking the box *In HU* switches the units from absolute value (attenuation coefficients) into Hounsfield unit.

If this box is checked, HU calibration can be performed. On the preview image, draw a ROI in the area where HU value is known (e.g., 0 for water) by clicking and holding the right-mouse while the CTRL key is pressed, and drag. By right-mouse-button release a window will pop up (see Figure 7): fill for "material #2" the known HU value for this region to calibrate HU. Click on OK to apply it. Please note: it is important to press CTRL key, hold-and-drag to start HU calibration; a right-click (without dragging) will activate the view menu. If you want to use this calibration on the control software, the calibration scaling factor can be found on the overview page.

For more information about pixel unit (grey value, attenuation coefficient, HU), please refer the FAQ list in the online help. For more information on HU calibration, there is a technical note available.

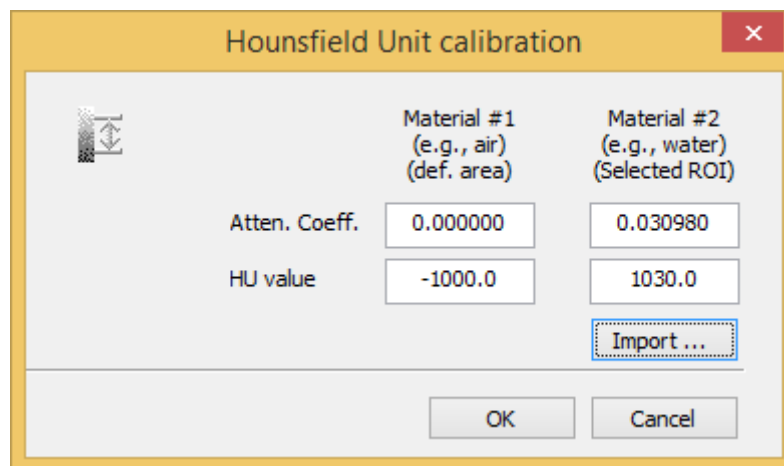


Figure 7 The pop-up window for HU calibration

➤ Reconstruct ROI

This option enables reconstruction of a region of interest. The default ROI is a rectangle. To force it to be a square, check *Square ROI*. You can also specify a circular or elliptical ROI. For technical reasons, the width and height always a multiple of 4 (For example, you

have chosen 403x403 as a ROI, but it will be forced to be 400x400). You can resize or move the ROI using left mouse button.

You can edit ROI manually: right-click in the ROI region; or double-click on the ROI info on the status bar.

➤ ***Scales/Labels on***

Check this box to have the scales and labels hard coded into your final images. It does not take extra time to draw them, but you might lose information at the edges in case of reconstruction of a small ROI.

➤ **Destination**

Specify the output dataset. The filenames of cross-section images are fixed: <dataset>_rec.bmp, <dataset>_rec.tif, <dataset>_rec.png or <dataset>_rec.jpg. You can change the output directory. The default output directory is determined by your settings in Preferences: either the input folder itself or a sub-folder in it. Click on button *Browse* to make modifications.

➤ **File format**

Four output formats are provided: 8-bit BMP (0-255), 16-bit TIFF (0-65535), 8-bit JPG (0-255) and 8-bit PNG (0-255). To convert the floating-point cross-section image to the limited precision of these formats, the dynamic range showing in the histogram section is used: values within the dynamic range are mapped into [0,255] in BMP/JPG/PNG format or [0, 65535] in TIFF format, and values out of range are assigned 0 (if below lower limit) or the maximum value (if above maximum limit). For the JPG format, you may specify a compression factor between 50% and 100% in *Options->Preferences*: it is recommended to use 100% to keep the most details. PNG format is added later: it allows lossless compression and yet provides a good compression factor.

A non-standard format F4F is provided. More information can be found in the FAQ list: What is the non-standard float format F4F and how/why to use it?

To retrieve pixel values, please refer to the FAQ list in the online help: How to convert pixel value from gray-level to its physical unit: linear attenuation coefficient in 1/mm?

2.2.4 The Summary page

This window/page offers the possibility to view all parameters currently in use at one glance on the first panel. The 2nd panel displays acquisition information. The page is coupled to the *image mode* in the main display. Note that the color scale shown (you may change it at will) has no correspondence to the used color palette in the final image.

This page is automatically set active during reconstruction after the first cross-section image is reconstructed and found. Sometimes this means that you will only see the image towards the end of the reconstruction. To use as little computer resource as possible, only the first image is shown.

You can save the current settings into a template/protocol file (*.rcp) by clicking on the button *Save settings as* Note that you can alter the values for some of the parameters by double-clicking on them.

See Figure 8 for a snapshot image when the overview page is active and the main display is in *image mode*.

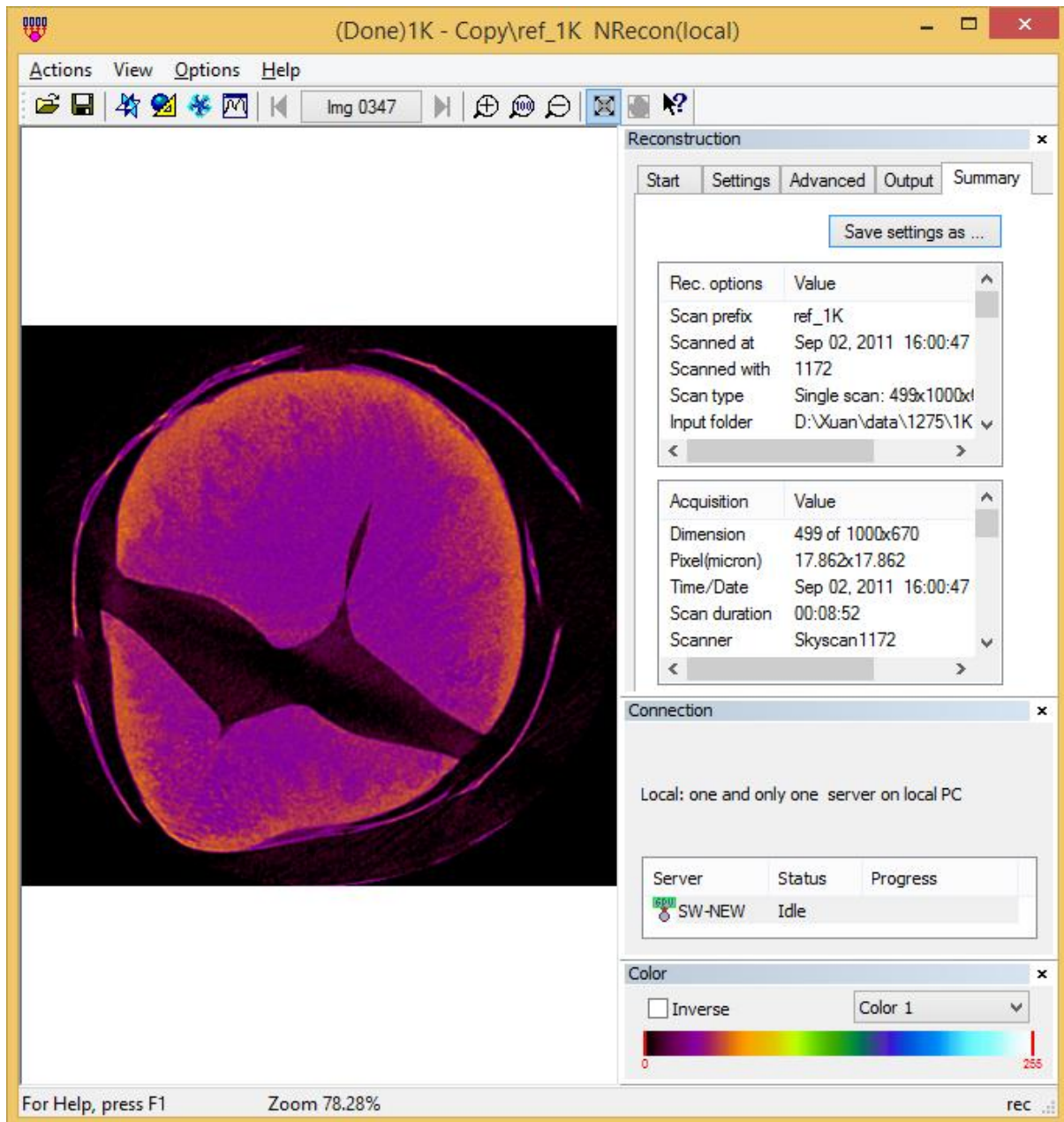


Figure 8 The Summary page with image display mode

2.2.5 The batch manager

The batch manager (see Figure 9) enables you to make a list of jobs to be launched later. It is part of NRecon and can be only launched from within NRecon. This window is part of the layout and can be toggled visible/invisible, resized or repositioned. It is shown automatically

at startup of NRecon if there are entries in the batch file. But you can show/hide it at any time via the view menu.

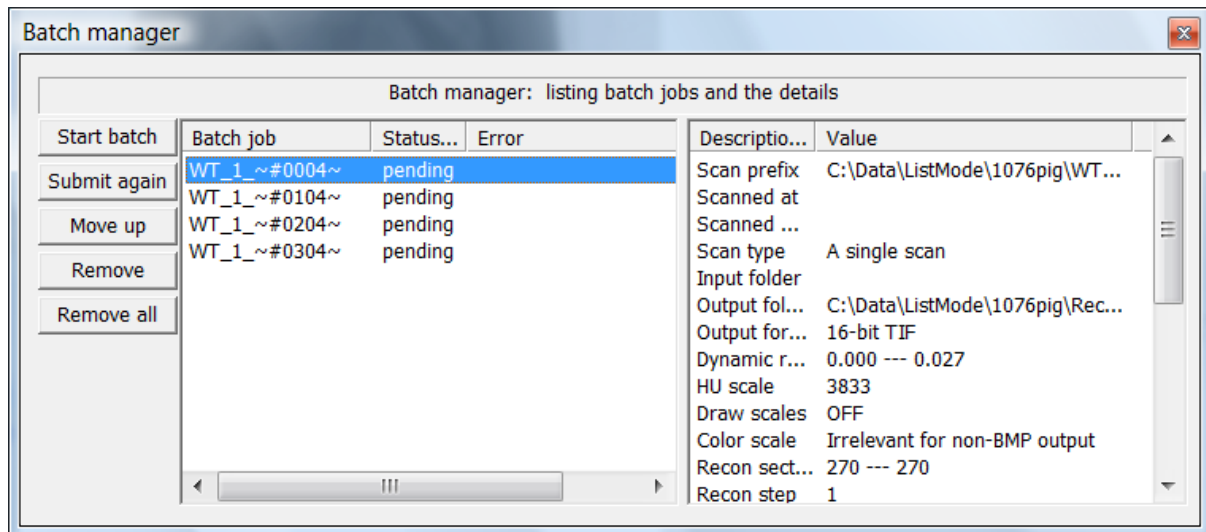


Figure 9 The batch manager

An entry in the batch file is identified by two items: the input dataset and the output directory. It is not possible to have two identical entries in the list.

To add a job into the list: load the dataset and adjust the parameters accordingly, press *add to batch* button on the [Start page](#). You will be reminded if an identical entry is found in the batch.

The entries in the batch are not removed automatically, even if the job is done. You have to remove them manually.

To start the batch, click on *Start batch*. Only those entries with status *pending* will be processed. If one of the entries fails, you can submit it again (error information is saved in NRecon_error.log in the system directory): select it with left-mouse click, and click on button *Submit again*.

To view details of a batch entry, click on it and its contents will be shown at the right panel. You can modify the parameters by double-clicking on the items (some items are not modifiable, such as the dataset or output directory which define the entry), but no consistency check is applied here. To modify parameters safely, it is recommended to load the dataset and submit it again: double-clicking on a batch entry will load the dataset and its parameters to the main window.

2.2.6 Optional functions

A number of optional functions have been added to the software. Most of these functions can be found in the menu *Actions* as shown in Figure 10. Some of the functions are only available for a certain type of scan, or when certain information is available. Only a very brief description is given here. For more details, see either the online help or technical notes.

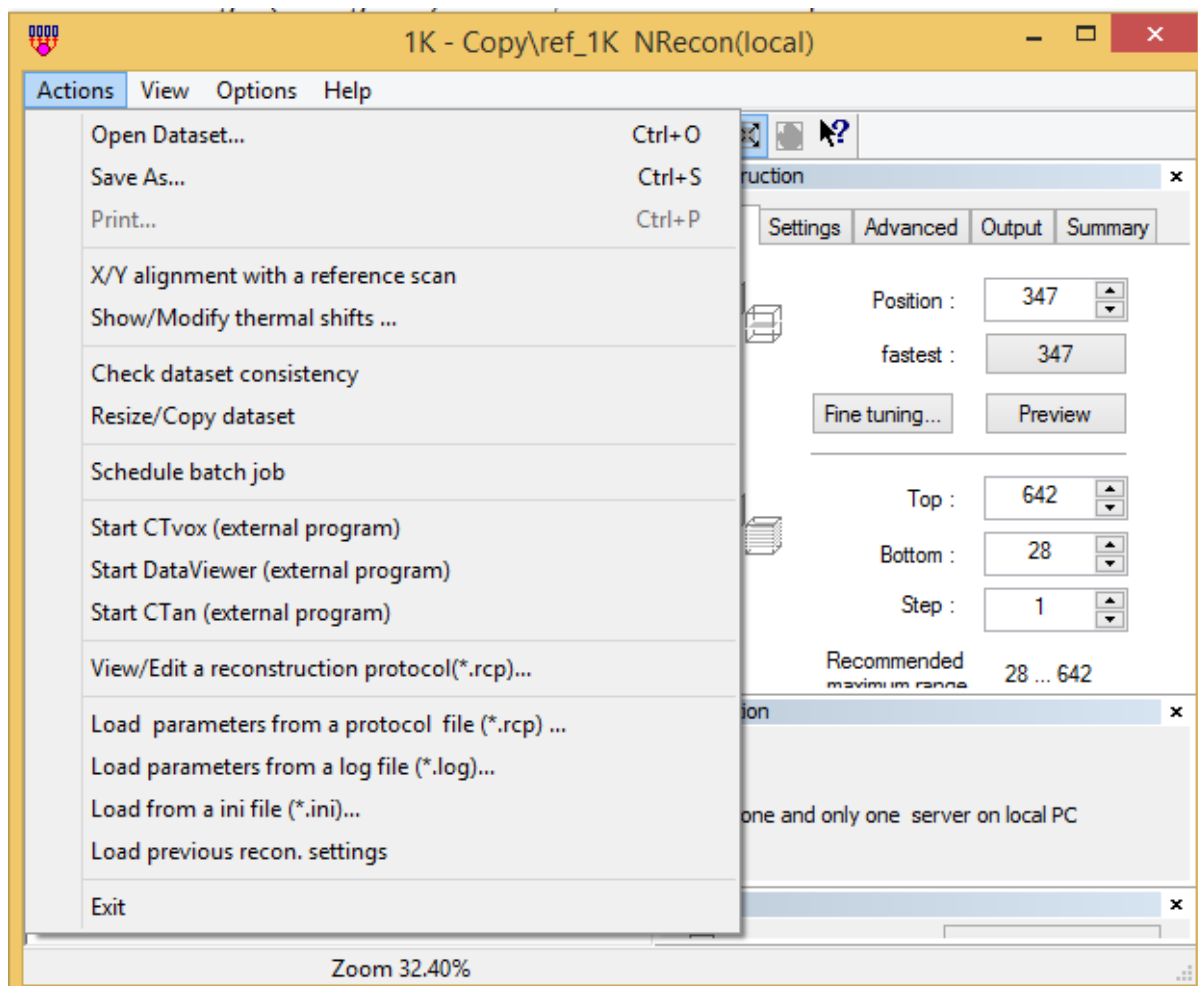


Figure 10 Optional functions available in menu Actions

X/Y alignment with a reference scan: this option is available only when a corresponding pre- or post-scan can be found. This is to compensate possible stage/source movement due to thermal changes. An extended function allowing rotation/zooming can be found in advanced mode.

Show/Modify thermal shifts: it is a helping tool when the x/y alignment option is used. The shifts are recorded as tags in TIFF header, therefore invisible. This tool reads the tags, display them and allow modifications.

Check dataset consistency: a useful debug tool to report more information about the dataset.

Resize/copy dataset: a tool to resize/copy the loaded dataset. Log files are modified consistently so that reconstruction will be possible after such an operation.

Schedule batch job: a tool to run batch jobs at a specified time.

2.3 Customize the application

NRecon can be customized in a few aspects: the layout, the default behavior and the set of reconstruction parameters.

To customize the layout, drag and drop are the obvious ways to get the layout to your desire. Once this is done, the layout is saved automatically.

If you have a set of scans with similar acquisition conditions and you wish to reconstruct them in similar way, there are 2 ways to achieve this. The easiest way is to use the log file accompanying each set of cross-section images: take one typical scan, adjust the reconstruction parameters accordingly and then reconstruct it, a log file will be created automatically together with the reconstructed images (*_rec.log); load this log file for all other scans to be reconstructed via menu *Actions->Load parameters from a log file*. A second way is to use the so-called reconstruction protocol files: take one typical scan, adjust the reconstruction parameters, go to the *Summary* page on the reconstruction window, and save the parameters as a *.rcp file. This RCP file can be either loaded individually via menu *Actions->Load parameters from a protocol file*, or can be set as the default protocol file via the *Preferences*.

You can customize the default behavior via the menu item *Options->Preferences*. More details can be found in the online help.

Chapter 3 Data format and conventions

As well known, a reconstruction program takes the projection images (also called shadow images) as input, and gives the cross-section images as output. The projection images are taken during acquisition over 360 degrees or 180+ degrees. For a reconstruction to succeed, this projection dataset has to be intact and is accompanied with necessary information about acquisition.

Only one input data format is recognized by NRecon: 16-bit TIFF format.

All projection image file names use the same convention. For each dataset, you can give it a name, or a prefix. For example, we have a 207-projection dataset named "sample". The convention for a projection file is: dataset name followed by a 4-digit (or 8-digit) index, starting from 0, namely, *sample0000.tif*, *sample0001.tif*, ..., *sample0206.tif*.

The acquisition parameters are saved in a text file (*sample_par.txt*, or *sample_cfg.txt*, or *sample.log*). The *log* format is the standard format, while the other formats are from earlier versions of acquisition software and are only recognized for backwards compatibility. It is worth mentioning that NRecon first searches for *sample_par.txt*, then *sample_cfg.txt*, and at last *sample.log*, as a *log* file might be created later on, either manually or by other software. A dataset cannot be loaded or reconstructed successfully if any of the following occurs: 1. Any projection file is missing; 2. The acquisition parameter file is missing or corrupted; 3. Inconsistent information between the text file and the projection image. SkyScan control programs also produce an additional TIFF file (*sample_arc.tif*) for ring artifact reduction. If this file is absent, NRecon will make one at the time of reconstruction. Therefore, the *_arc.tif is considered as part of input, but is not strictly necessary. Contrary to what many users believe, the *sample_spr.bmp* and *sample_sp2.bmp* are not considered part of input and are not used by NRecon, but they are used for display purpose by DataViewer and CTan.

A special case is the so-called connected or oversize scan. Suppose we have a scan with 3 bed positions (i.e., 3 sub-scans), we would expect the following set of TIFF files:

(sub-scan 1) *sample~000000.tif, sample~000001.tif, ..., sample~000206.tif;*

(sub-scan 2) *sample~010000.tif, sample~010001.tif, ..., sample~010206.tif;*

(sub-scan 3) *sample~020000.tif, sample~020001.tif, ..., sample~020206.tif.*

For each sub-scan, there is a text file (*sample~00_par.txt, sample~00_cfg.txt, or sample~00.log*). In case the text file is a *log* file, a global *log* file may also exist: *sample.log*.

The output cross-section datasets use the same convention as the input, but the index now is the cross-section number. To distinguish the input and output, "_rec" is appended to the dataset name. Take the example above, the resulting cross-sections would be:, *sample_rec0100.bmp, sample_rec0101.bmp,*. For an oversized scan, the sequence number is automatically shifted to form one complete volume.

Chapter 4 How to import a non-SkyScan dataset

It is possible to import a non-SkyScan dataset to NRecon, if a few simple rules can be followed.

As described in chapter 3, the file name convention should be followed. Each projection file should be a 16-bit TIFF image. If there are more than 10000 projections, the number of digits for the sequence should be 8, instead of 4. The pixel size information (for those who use Visual C++, biXPelsPerMeter and biYPelsPerMeter; for those who use MATLAB, tag resolution) should be filled correctly.

A corresponding log file should be available. If you have created a series of TIF files: MyScan_0000.tif, ..., MyScan_9999.tif, the corresponding log file should be named MyScan_.log. An example is given here. The comments are given by lines starting with ";". These comment lines can be retained in the log file: they are ignored by NRecon.

Begin of log file

```
#####
```

[System]

Scanner=SkyScan1172

; A string with at least 11 characters. Typically, the first 7 characters refer to the company, and the last 4 characters are 4-digits ---

; scanner model (in case of non-digits, it is interpreted as 0) . Model number 0-2999: X-ray systems;

; Model 3000-3999: Optical scanner systems

; Please respect the number of letters: hardcoded inside NRecon.

Instrument S/N=2

; info only: serial number of the scanner

[Acquisition]

Optical Axis (line)= 256

; Important parameter: the central slice in which the x-ray hits the detector perpendicularly;

Object to Source (mm)=100

; Important parameter: distance between image center and the x-ray source. If the distance is > 10m, it is assumed that it is a parallel beam scan.

Number Of Files= 401

; Info only. Number of projections, dimension and pixel . information are actually obtained/confirmed from the TIFF files

Number Of Rows= 512

; info only

Number Of Columns= 512

; info only

Image Pixel Size (um)=15.98

; info only. The pixel size is actually read from the header of the 16-bit TIFF files.

; Important! For cone-beam geometry: the concept of "virtual detector" is exploit. More specifically, the detector is

; assumed to be centered at the FOV.

; Therefore, the pixel size in the TIFF header should have been scaled accordingly.

; Alternatively, one may use the non-standard key *Scaled Image Pixel Size (um)* to indicate pixel size scaled at image center.

Linear pixel value=0

; Non-standard key. Default: 0, exponential attenuation case; 1: Linear case ; -1: inverse linear

Scaled Image Pixel Size (um)=15.98

; Non-standard key. When present, it has higher priority than pixel information in TIF header

Exposure (ms)=100

; info only

Rotation Step (deg)=0.900

; Important parameter

Table Feed (micron)=0.0

; Spiral CT specific: table movement length per rotation step

; Positive: table movement direction coincides with projection vertical direction

Use 360 Rotation=YES

; Important parameter: YES or NO (360 or 180+)

Flat Field Correction=ON

; info only

Rotation Direction=CC

; Important parameter: TRY IT OUT TO SEE IF IT SHOULD BE cc OR cw. Almost all SkyScan scanners have rotation direction CC.

Type of Detector Motion=STEP AND SHOOT

; info only

Scanning Trajectory=ROUND

; info only. Spiral scan is recognized by entry "table feed"

Study Date and Time=Aug 10, 2005 15:13:11

; info only

End of log file

#####

Appendix I. GPUReconServer installation guide

The Graphics Processing Unit (GPU) accelerated reconstruction server is available as GPUReconServer from version 1.6.6.0 (release).

How to install:

Simply download and copy all files into the folder where NRecon is located. GPUReconServer is supplied together with a few DLL's which should stay in the same folder as the program itself.

The GPUReconServer requires CUDA-enabled NVidia GPU's. You can find a recent list of these GPU's on NVidia website <http://www.nvidia.com/>. The GPU reconstruction has been evaluated on a number of these GPU's with compute capability from 1.3 onwards. If you intend to update your computer with a faster GPU, we can advise you for suitable models. If you intend to upgrade your computer for fastest reconstruction by more powerful graphic card with two GPUs (like GTX590), check with computer producer that capacity of power supply is enough for such graphical card and that your power supply unit has compatible power connectors (dual-GPU card requires two special 8-way power connectors). Single GPU graphical cards (like GTX580) can be installed in most computers in proper slot with enough space around for graphical card. If connectors and space allow, two (identical) graphical cards can be installed in your PC.

The current version GPUReconServer can make use of all CUDA-enabled graphical cards found on the PC. However, it is not suitable if the GPU devices have different computation power. If you do have different GPU devices on one PC, it is advisable to disable the less powerful devices. This can be done by double-clicking on the GPU on the program window, see fig. 1.

Before running GPUReconServer, it is advisable to update your GPU driver. This should be done via NVIDIA website. A not up-to-date GPU driver may cause program failure: unable to find a proper device or the program simply cannot be started. Figure A1.1 shows a typical error window when the driver is not up-to-date.

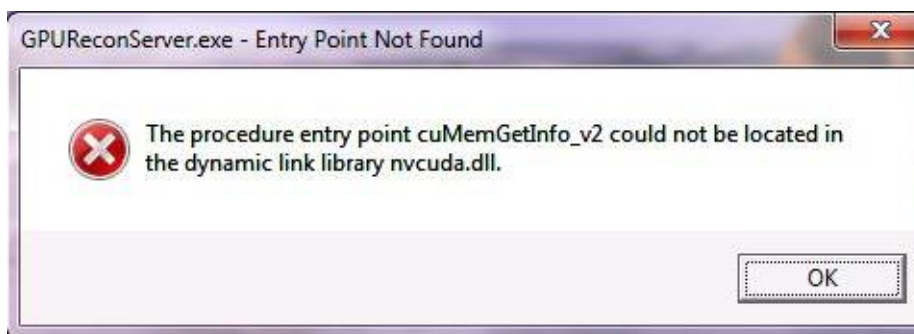


Figure A1.1 Typical error window when the GPU driver is not up-to-date.

If the reconstruction GPU is also used for display, you need to disable the automatic time-out of the display device by modifying the registry. Otherwise you will experience black screens and failures during reconstruction. This can be done by double-clicking on the supplied program `gpu_reg.reg` and allows it to perform the modification in the registry of your computer (this step creates a new entry *TdrLevel* with value 0 at registry section `HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\GraphicsDrivers`). You might need to restart your computer.

For GPU card with more than 1 device on it, e.g., GeForce GTX590, one has to disable the multi-GPU option as shown in figure A1.2. Otherwise, you may experience very slow reconstruction for larger volumes. The NVIDIA Control Panel can be started by right-clicking on desktop.

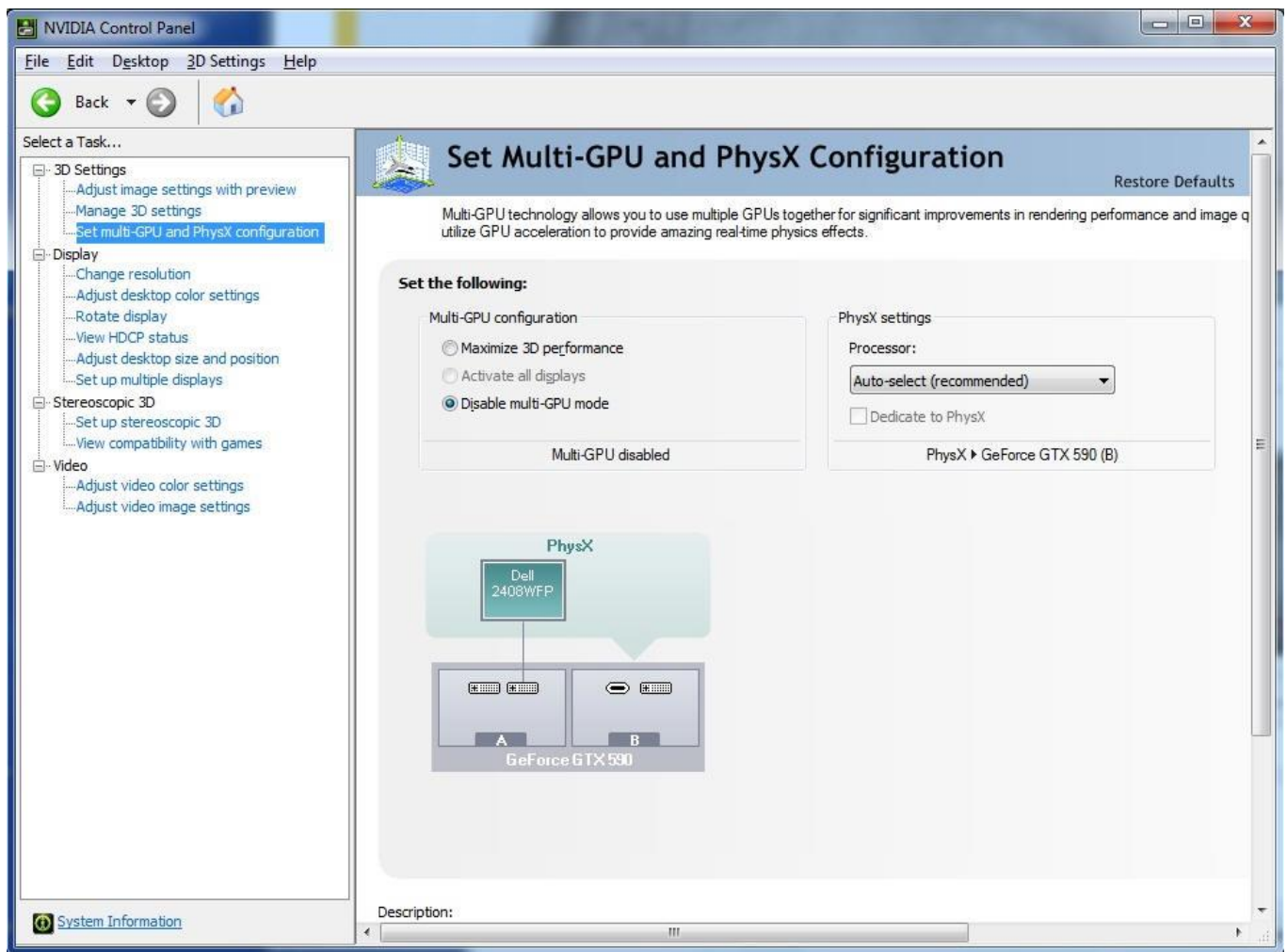


Figure A1.2 NVIDIA control panel

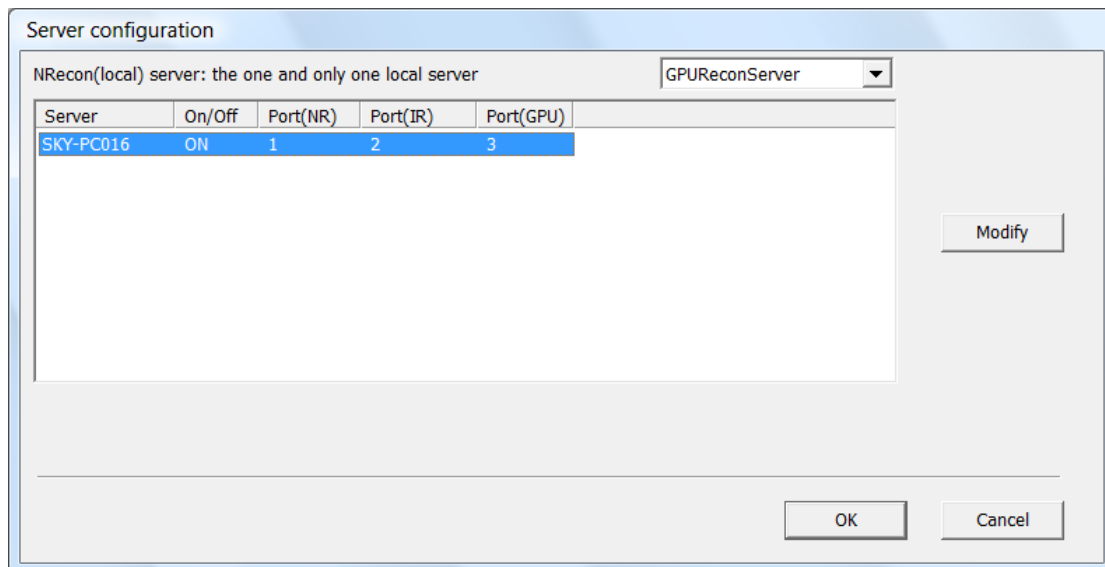
How to use:

This server works in the same way as the CPU version NReconServer: it has a local version and a cluster version. However, the GPU server is only provided in 64-bit. The usage of this server is thus straightforward for users who are already familiar with NRecon/NReconServer.

However, there are a few points one should pay attention to, if the GPU for reconstruction is also used for display. Simultaneous acquisition and GPU reconstruction is not impossible, but you may experience slow responses, and the chance for crashing due to exhausting resources is real. Running GPU reconstruction simultaneous with CTvox/CTvol should be avoided at all: both programs may crash. If possible, a safer work-around is to install 2 GPU's: one for display, one for reconstruction. Or, yet better, to have a dedicated reconstruction computer.

GPURconServer makes use of GPU to accelerate the core part of reconstruction. Although it is the GPU counterpart of SkyScan's NReconServer on CPU, very small image differences do exist between the CPU and GPU version. The differences come mainly from 2 sources: NReconServer does the convolution in space domain, while GPURconServer does the convolution in Fourier frequency space; NReconServer has slight approximation to accelerate back-projection, while GPURconServer does exact back-projection. GPURconServer and NReconServer should not be mixed in a cluster configuration: if this does happen, only the GPU server will be used.

For ease of comparison, it is possible to run GPURconServer, NReconServer or InstaReconServer on the same PC by using different communication port, but they cannot be connected simultaneously. If you want to use multiple servers, firstly change "Listening at port" value in the server programs to get all ports different, for example port 1 for NReconServer, port 2 for InstaRecon and port 3 for GPURconServer. Use the same port numbers in "Network configuration" of NRecon program. By selecting a server type (in the shown screenshot, the GPURconServer is selected), the given ports are used by NRecon to connect to the servers. You can switch between the different server types via menu "Options->Network configuration ...", as shown in the "Server configuration" window below.



By selecting a server type (in the example, the GPURconServer is selected), the given port (port 3 in the shown screenshot) is used by NRecon to connect to the server. You can change the port numbers here using button "Modify" after having highlighted the line. If all servers are running, then you can easily switch between the CPU server and GPU server (or InstaRecon server) by selecting the current server type in the "Server configuration" window.

How to provide feedback for a problem report:

As for all software programs, a screen image of the error message will be very helpful for troubleshooting.

Similar to our CPU server, an error log file exists also for the GPU server: GPUReconServer_error.log. This file records useful information for debugging when something goes wrong. It is located in the system folder as shown on the program window, see fig. A1.3. If NRecon has been run on the PC, the server shares the same system folder as NRecon (can be modified in the menu Preferences ...); otherwise, the folder is the same as the folder from where the server program was started. Please note: if this system folder does not exist for some reason, you need to create it manually, or redirect it to an existing folder via the Preferences menu on NRecon; otherwise no error information will be recorded.

You can provide your feedback by mailing to info@skyscan.be. In case of problems, in addition to a short description of the problem, please also include the error log file, a screen image if possible, and the log file of the dataset you are having trouble with.

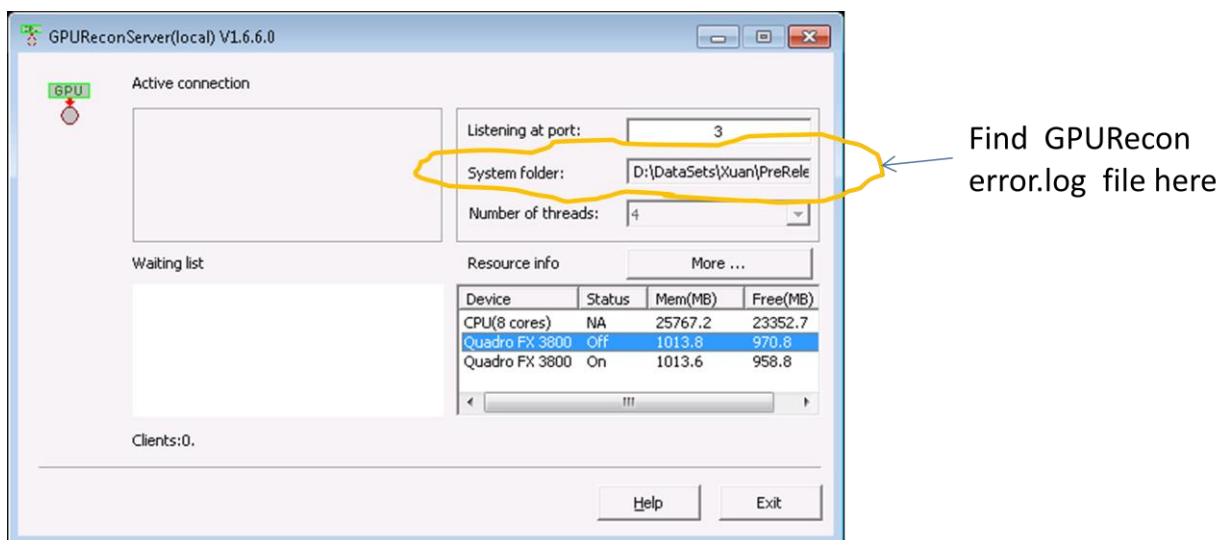


Figure A-1.3 GPU reconstruction server.