aRTist User's Guide

This manual is intended as a practical guide for the use of the radiographic simulator aRTist. aRTist is a software package for computer simulation of radiographic testing developed by Bundesanstalt für Materialforschung und -prüfung (BAM), Berlin, Germany.
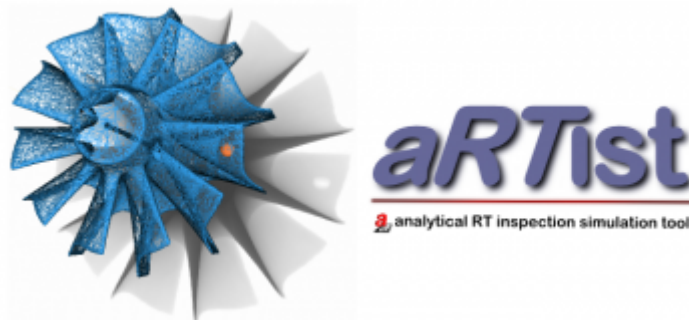
This manual refers to aRTist version **2.12**.

The information in this manual is for reference and informational use only. We assume no responsibility or liability for any inaccuracies or errors in this document.

Contact information:
aRTist@bam.de
http://www.aRTist.bam.de

Installation

System Requirements

The *aRT*ist program comes as a Microsoft Windows application. The system requirements narrow down to the required platform: Microsoft Windows 7 or superior. The program is provided as 64-bit application.

It is possible to use *aRT*ist on low prize netbook hardware. But it is recommended to run it on computers with multi (core) processor and accelerated graphical processing unit (OpenGL 2.1) to have the necessary calculation speed for treating ambitious simulation tasks efficiently.

aRTist Licenses

The *aRT*ist program is equipped with software protection mechanisms and needs license activation for full functionality. Without activation, the program runs in Lite mode of reduced functionality. In case of licensing errors, e.g. broken license file, the program might still be usable to arrange and review virtual scenes, but it is not possible to make any simulations (viewer mode).

> ⚠ An **evaluation period** of 30 days is granted without any license activation (DEMOMODE).

An *aRT*ist license is shipped as serial number, which can be used to activate an *aRT*ist installation.

> ⚠ Licenses can be bound to a specific PC (**node-locked license**) or shared among different installations (**network license**).

The *aRT*ist licenses can be used either node-locked or shared within a network. A node-locked license is assigned to one specific PC. With network licensing the *aRT*ist program can be installed on several PCs with access to the same license file. In this case *aRT*ist can be alternately used on different PCs (floating license). To run several *aRT*ist installations at the same time it is possible to assign more than one license to the shared license file.

Installation instructions

The radiographic simulator *aRT*ist comes with an installation routine 'aRTistSetup-X.X.X.exe' (where X.X.X is replaced by the version number). For installation execute the setup routine you got with your distribution. A dialog will appear (Figure 1) and guide you through the installation process.

> ⚠ If *aRT*ist is already installed, the installation routine will ask for de-installation of the current

version before proceeding with the installation process, if necessary. Consequently, installations with the same version number will be replaced, while installations with different version numbers can coexist on the same PC.

During the installation process, you can choose options like the generation of a Windows Start Menu entry and a Windows Desktop icon for convenient program execution. After installation *aRTist* is ready to use. Without an active and valid license *aRT*ist will run with the restricted functionality of Lite/Viewer mode only.

In the next section you will find information on reqesting a evaluation or permanent license activation.
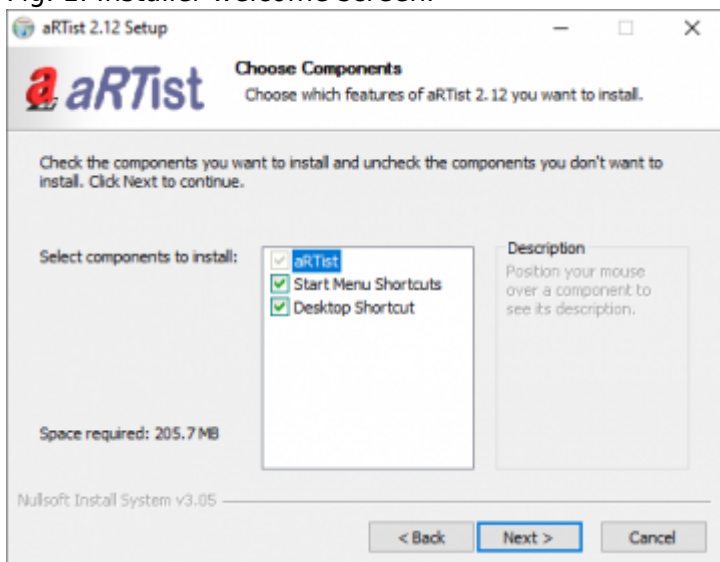

Fig. 1: Installer welcome screen.

Fig. 2: Installer option selection.

License Activation

*aRT*ist needs to be installed before proceeding with the license activation. The activation is a two-step process: first, a dedicated activation request has to be sent to the supplier of the software and secondly, the activated license file provided in response has to be installed.

The actual license status can be displayed by item `License Information` at the Help menu (Figure

[3](#)).

Activation request

The activation request requires information about your *aRTist* installation. The necessary information can be collected at the license information dialog of *aRTist* (menu bar: Help). There are options to apply for node-locked or network licensing and to request an evaluation period. E-mail the request to aRTist@bam.de.

In response to an activation request, you will get back an activated license file within the next business days.

> ⚠ To request a network license, a shared network folder has to be prepared in advance. It is required to store the common license file.

> ⚠ All users of the network license need to have write permission for the license file.
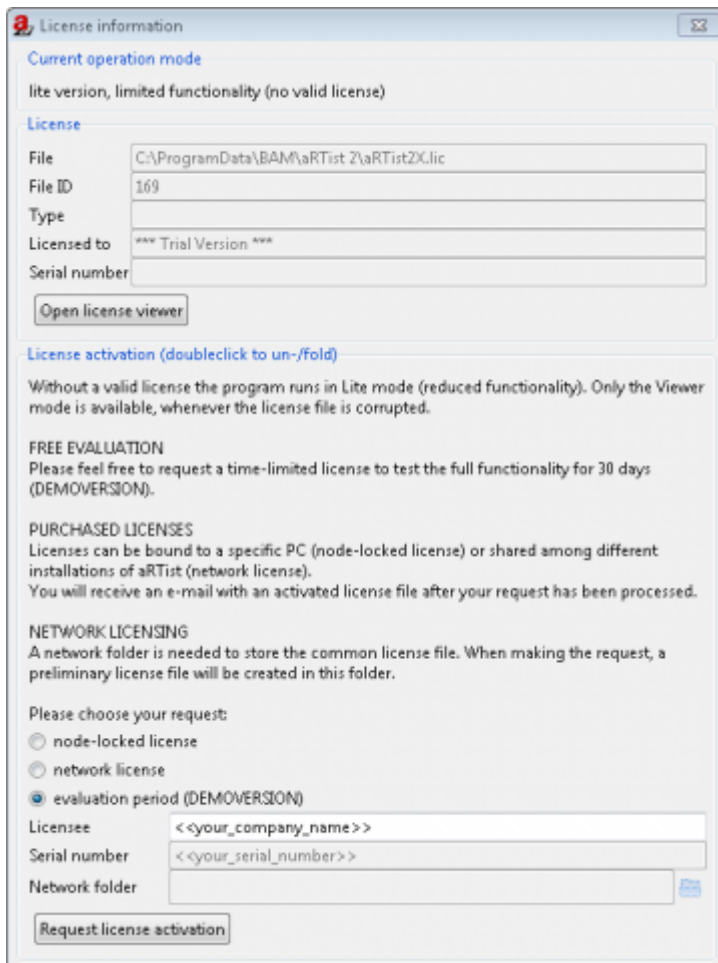

Fig. 3: License information dialog (menu bar: Help⇒License Information).

License File Installation

The final step to activate a license is to install the license file provided in response to the activation request. Generally, this means overwriting the temporary license file by the activated one.

The license file installation can be done just by opening the license file in *aRT*ist.

> ⚠ Just Drag'n'Drop the license file over the *aRT*ist window to install it.

> ⚠ The same applies to use an already activated network license at additional PCs: Just Drag'n'Drop the license file at the network folder over *aRT*ist window. Complete the subsequent dialog "Use (not install) this network license?" with "Yes".

Getting Started

This manual describes how to operate the *aRT*ist software package for computer simulation of radiographic testing.

The first chapter *Getting Started* will give you a short overview of the user interface with its different areas.

User Interface

The *aRT*ist user interface, the main window, includes the following areas:

- *Menu bar*
- *Toolbar*
- *Parameter panel* with tabbed pages of modelling settings (left)
- *Virtual scene*, a 3D scene view of the radiographic setup (right of parameter panel)
- *Docking area* to integrate additional program windows within the main window. By default, the *ImageViewer* showing the simulated radiograph is placed here (right or bottom, configurable).
- *Status bar* (bottom)

Figure 4 shows the default view of the user interface. The scene already contains two items: source and detector. The three main areas (*Parameter Panel*, *virtual scene*, *docking area*) can be scaled by clicking and dragging the edges of the respective areas.
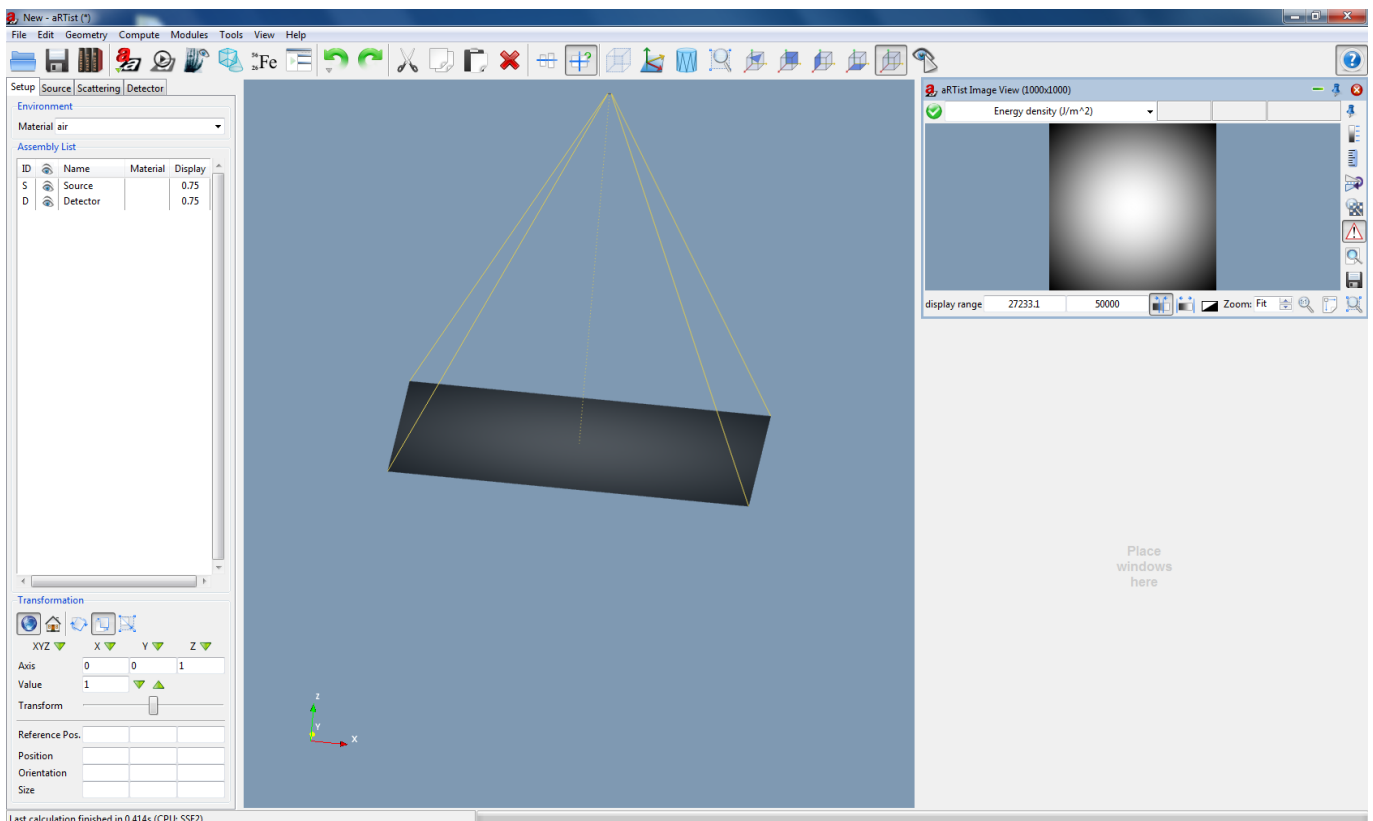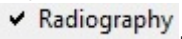

Fig. 4: User interface

Menu Bar

The *Menu Bar* is a horizontal "bar" anchored to the top of the screen (Figure 5). It consists of different drop-down menus and contains generally known commands like *Save*, *Copy* or *Undo* and program-specific commands to adjust and modify the assembly. A black tick next to an option indicates that it is currently activated: ✔ Radiography .

Further information regarding the *Menu Bar* can be found in the separate chapter.

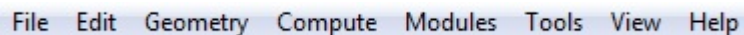File   Edit   Geometry   Compute   Modules   Tools   View   Help

Fig. 5: Menu bar

Toolbar

In the toolbar you can find some of the menu bar commands for easy access (Figure 6). All of those commands indicated by its toolbar icons will be explained in the *Menu Bar* section of this guide. The toolbar can be found directly below the *Menu Bar* and contains icons, some of which have a drop-down menu that can be opened by clicking and holding for a second or with a right-click. Hover over an icon with the mouse pointer and a little tag with a short explanation of its function will appear. An activated icon can be identified by a black border around the symbol: . While a deactivated icon does not have one: (The highlighting of selected checkbuttons may depend on OS version and settings).

You can hide the toolbar by clicking on *Toggle Toolbar Visibility* at the *View* menu.

Fig. 6: Toolbar

Parameter Panel

The *Parameter Panel* on the left contains four tabs: *Setup, Source, Scattering* and *Detector* (Figure 7). Each tab offers several options to modify parameters of the simulation. You can rotate, translate or scale the items of your assembly list, make adjustments to the source and the detector or take scattering into account. Hide/Show the *Parameter Panel* by clicking on *Toggle Setup Visibility* in the *View* menu or by pressing Ctrl-w on your keyboard. Further information about the *Parameter Panel* can be found in the separate chapter.
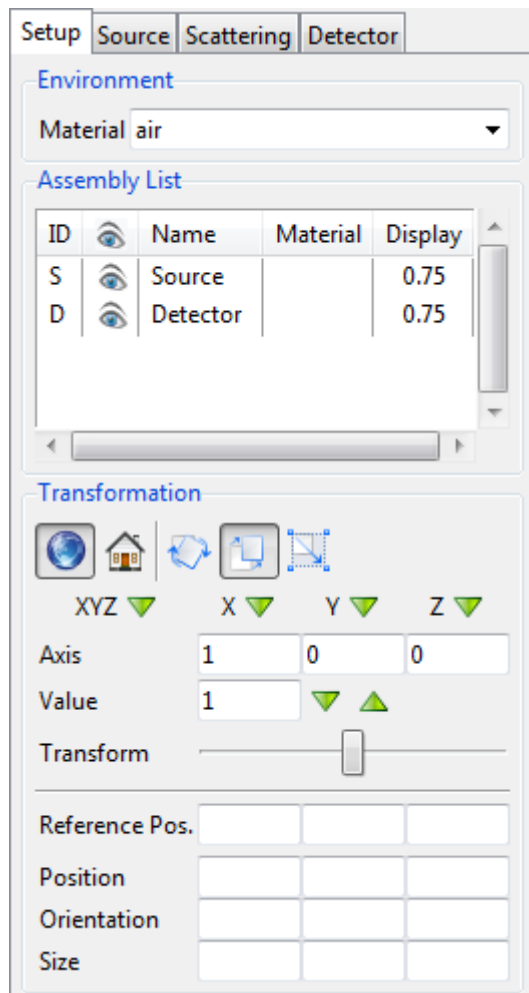
Fig. 7: Parameter Panel

Virtual Scene

The 3D visualization area shows the virtual scene view, a 3D representation of the setup (Figure 8). The default view only contains source and detector. In the assembly, virtual parts (geometric objects) can be appended in the scene view. The virtual scene can be freely rotated, scaled and shifted in space in the following ways:

- Hold down the left mouse button over the empty space of the scene view and drag it. The scene will rotate.
- Hold down the right mouse button and drag. The scene view will scale.
- Hold down the middle mouse button and drag. The scene view will move with the mouse.

A coordinate system pictogram is shown in the bottom left corner indicating the actual orientation of the scene view.

In this guide *items* define all parts, detector and source in the scene. While *parts* are only the geometric objects in the scene.

For selecting items in the scene view, use the Ctrl key on the keyboard and click on them with the left mouse. The first selected item can be recognized by its yellow corners. All subsequent items will be marked with white corners. The order of selection can be important for *Geometry* applications.
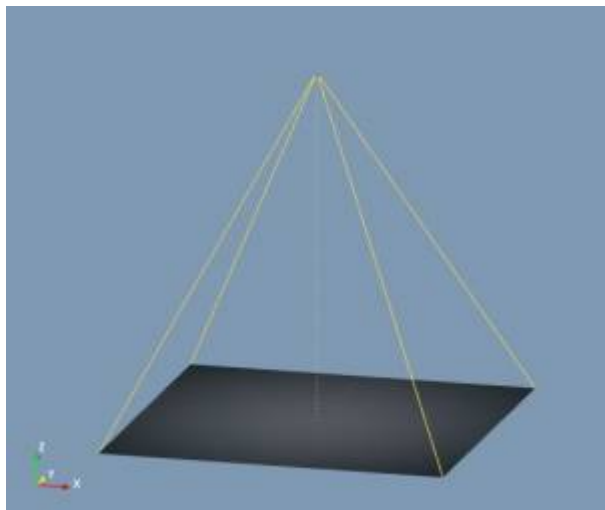
Fig. 8: Virtual scene, scene view

Docking Area

In the docking area is written "*Place windows here*". Several commands throughout the program open separate windows (mostly modules) that can be docked in the docking area (indicate to chapter → *Modules*). You can minimize or close a window from the docking area or (un)dock it by clicking on the blue pin between the minimize and close symbols: . An undocked window can be relocated anywhere on the user surface. The default docking area contains only one window, the *ImageViewer*, which displays the radiograph according to the assembly. You can hide the docking area by clicking on *Toggle Dock Visibility* in the *View* menu or by pressing Ctrl-d on your keyboard.

Status Bar

The status bar at the bottom of the user interface displays information when hovering over certain areas with the mouse pointer (for example the *ImageViewer* or the scene view) or when a calculation is finished (Figure 9).

Fig. 9: Status bar

General Simulation Approach

When you first start up a*RT*ist, you will see the screen as in Figure 4. If you make any mistake, you can use the undo icon in the main toolbar.
In the following, the general approach for a successful radiographic simulation with a*RT*ist will be explained.
First of all, you need a part in the scene, which you want to image. The detector and source are already available.

- Click on the *Open library* icon ![open library icon] or on the *Solid* icon ![solid icon] in the main toolbar and choose a part or project (in *library*).

- The part appears in the scene center, and a new entry appears in the *Assembly List* of the *Parameter Panel* → *Setup* (see Figure 10, left side).

- To move the part you have to select the part first by clicking on it in the scene (recognized by its yellow corners) or by selecting it in the *Assembly List* (highlighted blue). An arrow shows the *manipulation axis*.

- The *Transformation* settings of the part can be done below the *Assembly List* (Figure 7). You can select each axis by clicking on the axis button arrow. Type the *Value* or use the arrows next to it.

- The *Transformation* buttons allow to choose between *world coordinate system* (world button) and *local coordinate system* (house button). Choose a button next to it to *rotate, translate* or *scale* the part. The part can be moved with the slide control or directly in the scene with the left mouse button. You can monitor the movement/modification of *Reference position, Position, Orientation* and *Size* over the display fields in the bottom of the *Setup* page. You can alternatively type the values yourself. A big axis with a red ball at the original position appears when you translate the part. This ball indicates the reference point. It is used as the center when a part is rotated.

- If you want to move the source and/or detector just select the item and transform analogically.

- In the *Assembly List* you can change the material of the selected part of the scene. The part is made of iron as a default. Click on the corresponding column and a drop down box will appear, where you can select desired material. (similar for *Display* → changing the *Opacity*)

- The standard detector has only a resolution of 100×100 pixels. These settings can be changed on the *Detector* panel (see Figure **A** of Tab. 1). Therefore click on the notebook header *Detector* of *Parameter Panel*. At the top of the panel, in the *Geometry* frame, there are fields with physical *size* of the detector, the size of a single *pixel* and its *pixel* resolution (*Res.*). In *Characteristic* frame you can select the detector type.

- The standard scene is equipped with a 150 *keV* monochromatic source. The source setting of the X-ray tube can be changed in the *Source* panel (see Figure **B** of Tab. 1). In the top frame labeled *Source spectrum* is a graph of the spectrum with a single line at 150 *keV*. Click on the *X-ray tube* icon ![x-ray tube icon] to open the spectrum calculator. There you can select the *General* tube to have a typical Bremsstrahlung spectrum in the spectrum graph and set the parameters like in the screenshot of Figure **C** (Tab. 1). Press *Compute* when your settings are set. You also can set the current in the *Exposure*. Open the drop-down list under *Spot type* and make a selection. a*RT*ist can consider an extended source focal spot and computes multiple rays per pixel of the final image.

- Choose the *Zoom to selection* icon ![zoom icon] in the main toolbar to see where the axis goes through. To see the full scene again, click in the empty space of the 3D view to deselect all parts. Then again, click the *Zoom to selection* icon.

- On the right side of Figure 10, the *ImageViewer* displays the preview image of the simulation

(identify by the crossed out red circle:).

- Click on the *Run* icon in the main toolbar to display the finished simulation in *ImageViewer* (identify by a green tick:).

- If your radiographic setup is finished, you should save your work as a project file. Click the *Save* icon in the main toolbar. Choose "aRTist Project" as the type and save your settings. aRTist stores everything inside the project file including the CAD files. This means you can pass the project around to colleagues who will be able to open and reproduce your simulations.
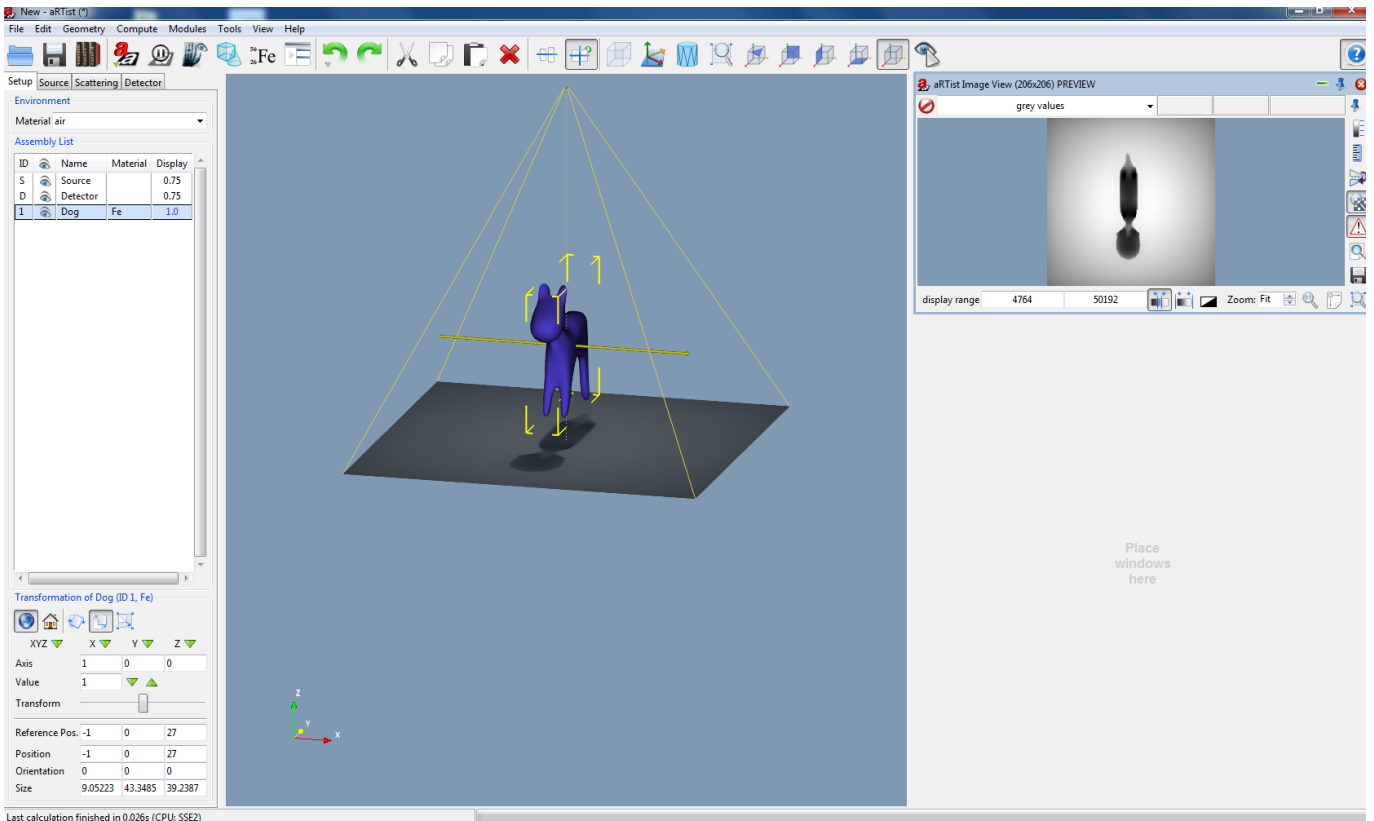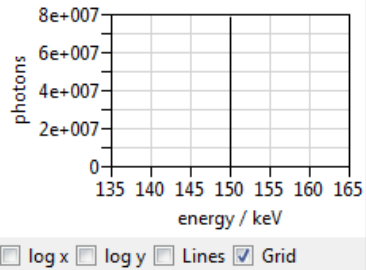

Fig. 10: Screen of a*RT*ist program, one part added into the scene

Tab. 1: Table for *General Simulation Approach*

Parameter Panel Detector settings	*Parameter Panel Source* settings	*Spectrum calculator*
(A)	**(B)**	**(C)**

(A) Parameter Panel Detector settings

Setup | Source | Scattering | **Detector**

Geometry

○ Size [mm]   ● Pixel   ○ Res. [mm]

X 100 | 1000 | 0.1
Y 100 | 1000 | 0.1

Multisampling   source dependent ▼
Curvature   off ▼

Characteristic

detector type   1:1 ▼
scan mode   off ▼

(B) Parameter Panel Source settings

Setup | **Source** | Scattering | Detector

Source spectrum

Monochromatic 150 keV, 1 / (GBq * sr)

Graph | Text

photons
8e+007
6e+007
4e+007
2e+007
0
135 140 145 150 155 160 165
energy / keV

☐ log x ☐ log y ☐ Lines ☑ Grid

Exposure

exposure [mA or GBq]   1

Focal spot

Size [mm]   Spot type
X: 1   point ▼
Y: 1

(C) Spectrum calculator

aRTist - Spectrum calculator

Settings

Tube   General ▼
voltage (kV)   150
resolution (keV)   1.0
filter material   VOID ▼
filter thickness (mm)   0

Tube Settings

Name   General ➕
Maximum voltage (kV)
Maximum current (mA)
Maximum power (W)
☐ Transmission
target_material   W ▼
target thickness (mm)   1
target angle (deg)   45
angle of e- incidence (deg)
window material   Al ▼
window thickness (mm)   4

Compute   ☐ Show principle

Menu Bar

The *Menu Bar* consists of different drop-down menus with common Windows commands and program specific functions (Figure 11) that will be explained in this section. Some of the menu entries can also be found in the toolbar indicated by the same icon. If that is the case, the icon is next to the command/option in this manual.
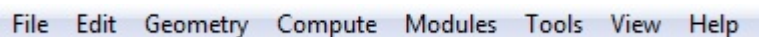
File   Edit   Geometry   Compute   Modules   Tools   View   Help

Fig. 11: Menu bar

File

The "*File*" drop-down menu comprises commands referring to the current project (Figure 12).

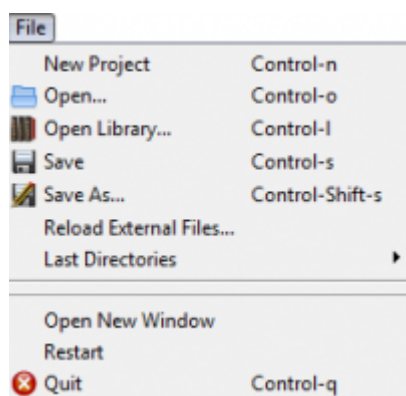| File	
New Project	Control-n
Open...	Control-o
Open Library...	Control-l
Save	Control-s
Save As...	Control-Shift-s
Reload External Files...	
Last Directories	▶
Open New Window	
Restart	
Quit	Control-q

Fig. 12: File Menu

New Project opens a new a*RT*ist scheme. This may also be done by pressing Ctrl-n. The program will ask you, if you want to save the changes on your current project.

Open is used to search an existing project. Then select a project file in the dialog boxes that follow. Otherwise press Ctrl-o.

Open Library is used to make a selection in the register of files. There example parts and projects can be found. It can also be opened by pressing Ctrl-l.

Save the current project to a single file. Alternatively press Ctrl-s.

Save as renames a project or changes the location of where you want to save it. By pressing Ctrl-shift-s the command *Save As* will be opened, too.

Reload External Files serves to check and update external dependencies (incorporated data files) of the current project. There you can choose file versions from the project or the file system in the dialog box that follow.

Last Directories shows a list of recently opened directories. By clicking on an entry, a file open dialog for this directory will pop up.

Open New Window starts a new *aRT*ist in addition.

Restart reboots the programm. Before restarting, the program will ask you if you want to save the changes on your current project.

Quit terminates the program. Before quitting the program will ask you, if you want to save the changes on your current project. This will also be achieved by pressing Ctrl-q.

Edit

The "*Edit Menu*" contains *undo/redo* functionalities with the projects history, whereas the usual commands (*cut, copy, paste* and *delete*) refer to parts in the assembly (Figure 13).
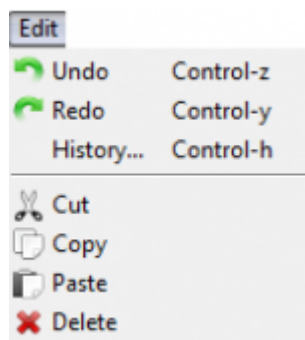

Fig. 13: Edit Menu

 Undo erases the last change done to the project reverting it to the last state it was saved as. The program will also be undone by Ctrl-z.

 Redo reverses the *Undo* or advances the buffer to its former state. Otherwise press Ctrl-y. The opposite of *Undo* is *Redo*. The *Undo* and *Redo* commands restrict you to an incremental sequence of changes.

History displays the report of the edit and lets you revert back to any previous change instantly. By pressing Ctrl-h the program will open the history, too.

 Cut removes parts from the assembly and keeps it in the clipboard. Therefore select part(s) you want to cut by using the Ctrl key on the keyboard and click on them with the left mouse either in the scene or in the *Assembly List* (of *Parameter Panel*).

 Copy can create a duplicate of the part. First, select part(s) you want to copy by using the Ctrl key on the keyboard and click on them with the left mouse either in the scene or in the *Assembly List* (of *Parameter Panel*). The selected parts are kept in the clipboard.

 Paste appends parts in the clipboard to the *Assembly List*, again.

 Delete removes selected parts from the *Assembly List*.

Geometry

The "*Geometry Menu*" includes all functions regarding the geometry application for parts from the assembly (Figure 14). Especially "*Geometry Manipulation*" (→ *Union, Intersection, Difference, Arrange* and *Pick Destination*) for creating and arranging more complicated parts are described more in-depth in the separate chapter.
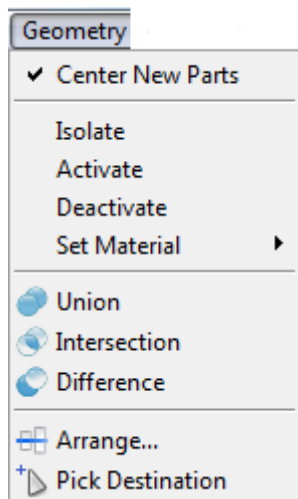

Fig. 14: Geometry Menu

Center New Parts position new parts while loading from external files (.stl or .ply) at the origin of the global coordinates, if activated. If deactivated, new parts will be loaded with its original position.

Isolate deactivates all parts in the scene except the selected ones. Alternatively you can deactivate or activate a part with a click on the eye symbol 👁 at *Parameter Panel* → *Setup* → *Assembly List* → *Activate/Deactivate* option.

Activate/Deactivate selected item(s) of the assembly. If you deactivate an item, it will be ignored during simulation and be nearly invisible in the scene. Please note, that you have to mark the respective item first. Otherwise you may use the eye symbol in the *Assembly List* of the *Parameter Panel* to achieve the same effect.

Set Material changes the material of a selected part. As an alternative, you may click on the material name of a part in the *Assembly List* of the parameter panel to open a drop-down menu with the same options.

Union/Intersection/Difference is used to create complex parts in the scene. For the commands *Union, Intersection* and *Difference*, parts have to be created (e.g. with *Solid* module) and placed in the scene, so that they overlap. Then, select parts by using the Ctrl key on the keyboard and click on them with the left mouse. The first selected part can be recognized by its yellow corners. All subsequent parts will be marked with white corners. The so-called *Geometry Manipulation* commands are explained simplified in the subsequent.

Union joins two or more parts. Thereby the order of selection is not important.

Intersection creates a new part from the overlapping areas of at least two parts. Everything else will be removed. The order of your selection is not important.

Difference is used to create a new part from the difference of at least two other parts. By selecting this command the order of selection will be important for the result. The first selected part (yellow corner) is thereby the minuend and all other selected parts (white corners) are the subtrahends.

Arrange is used to put parts in order to the assembly. Select the parts, which have to be arranged, and activate this command. With the opening dialog box four options: *none*, -, *center*, + are available for each of the three axes. The parts will be aligned with the first selected part.

Pick Destination moves item(s) from one point of the scene view to another. Click with the left mouse on desired destination to relocate selected item(s).

Compute

The "*Compute Menu*" offers different enable/disable functions referring to view attributes in *ImageViewer*, intern computations and the *Run* button for starting simulation (Figure 15).
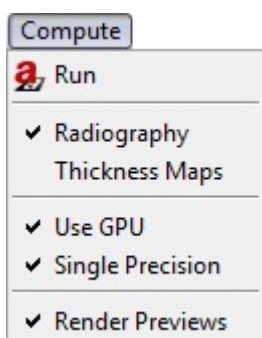

Fig. 15: Compute Menu

Run starts a simulation of the current assembly. Resulting images are presented in the *ImageViewer*.

Radiography/Thickness maps is used to display grey values or thickness maps in *ImageViewer*. You may choose between these two options to either simulate radiography (*ImageViewer* displays grey values) or thickness maps (*ImageViewer* displays thickness in *mm* for each material of the assembly list and of the environment). Otherwise, you may use the *Run* toolbar icon by opening its drop-down menu: click and hold for a second or click with the right mouse button.

Use GPU accelerates computation if activated. Thereby the computers GPU will be used alongside CPU.

Single Precision uses the available storage space more efficiently (when a check mark is next to it). The figures or calculations are thereby less precise. Where in the opposite of the so-called "*Double Precision*" (= inactive single precision, without a check mark) the computations are more precise with more decimal places.

Render Previews enables/disables live rendering of preview images in *ImageViewer*.

Modules

Different modules in this drop-down menu are available for several functional simulations. More information about the modules (Figure 16) can be found in the separate chapter *Modules*. All modules can be docked in the docking area or be handled as separate windows. In the following you can find short introduction information about the single modules.
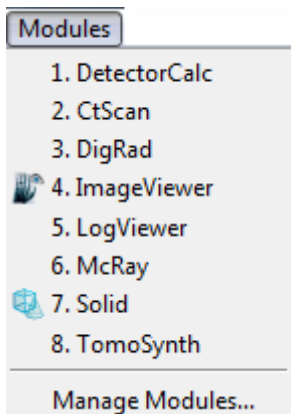
Fig. 16: Modules Menu

Manage Modules

In the Mange Modules individual modules can be selected, which will load at startup of *aRT*ist. The changes take effect at the next startup.

DetectorCalc

The DetectorCalc module is useful to create a new detector with specialised settings for flat panel detectors. Thereby the *reference shot* is necessary for calibration and calculation, also filtration over the detector is possible.
For more information see the chapter *Modules* → *DetectorCalc*.

CtScan

A simulated computerized tomography (CT) scan is enable with the CtScan Module. It combines series of X-ray images taken from different angles and uses computer processing to create cross-sectional images or slices of a part. CT scan images provide more detailed information than plain X-rays do.
In the three tabs of this module the corresponding settings can be navigated. Click on the *Run* button in the bottom of each page to start the simulation or to stop it. When the simulation is finished, click on the *Show* button of the first tab and the tomogram can be viewed in the scene. In *ImageViewer* the sectional image of the tomogram can be seen when in *Volume View* (third tab of CtScan Module) the *Show* button is clicked. For more details see in *Modules* → *CtScan*.

DigRad

This module will add new detector descriptions to a*RT*ist. It is intended to describe individual digital detectors by measured values according to system characterization standards. For more information see *Modules* → *DigRad*.

ImageViewer

The only module by default in the docking area is the *ImageViewer*. The preview image and the finished simulation are displayed there. You may choose between radiography and thickness map. The different settings and functions of *ImageViewer* (upper bar, lower bar and bar on the right) will be explained in *Modules* → *ImageViewer*. In a subitem called *Tools Menu: ImageViewer* advanced settings from the *Menu Bar* → *Tools* → *ImageViewer* are described.

LogViewer

The LogViewer displays messages generated by a*RT*ist and its components.

McRay

McRay is a Monte Carlo raytracing implementation of electron and photon transport. This module forms a graphical interface for McRay, in order to configure and run simulations using it from within a*RT*ist. The McRay module consists of six tabs: Basic, Advanced, Expert (*including: General, Photons, Electrons, Scene, Detector, Spectrometer, Objects*), Manual, Cluster and Interactive. A *Run/Stop* button is at the bottom of each tab. The different settings and functions of McRay will be explained in *Modules* → *McRay*.

Solid

With the Solid module, simple parts (respectively virtual objects) of different sizes can be created and added to the scene. Different geometric shapes can be chosen from: *cuboid, ellipsoid, cylinder, tube, cone, text, wedge* and/or *step wedge*. For more information see the chapter *Modules* → *Solid*.

TomoSynth

Simulating tomosynthesis within a*RT*ist are enabled with the TomoSynth module. On the *Laminography* tab settings for the tube, detector and output options can be set up. The *Advanced* tab offers settings for experts. More functions and information of TomoSynth will be explained in *Modules* → *TomoSynth*.

Tools

The "*Tools*" drop-down menu offers several help-functions relating to error messages (*Console, VTK*), irradiation settings (*attenuation data, detector properties, ImageViewer*) and settings for materials or spectrum (Figure 17). A detailed description about the tools topics *Attenuation Data*, *Detector Properties*, *Materials*, *Spectrum* and *Settings* can be found in the separate chapter *Tools* of this tutorial.
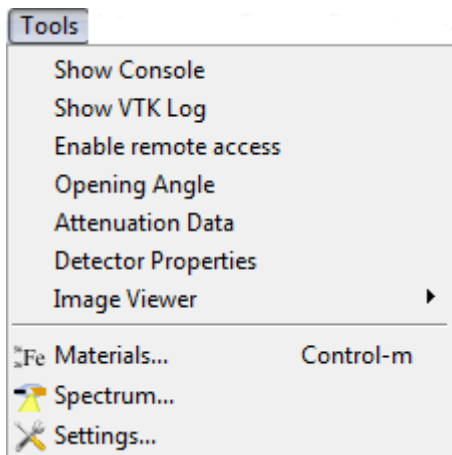
Fig. 17: Tools Menu

Show Console

This will open a console window to the command-line interface of a*RT*ist. Here detailed status information is available. Additionally, it allows to control a*RT*ist by commands in Tcl/Tk scripting language. The menus from the Console menu bar are also accessible with a right-click anywhere on the surface of Console (Figure 18).
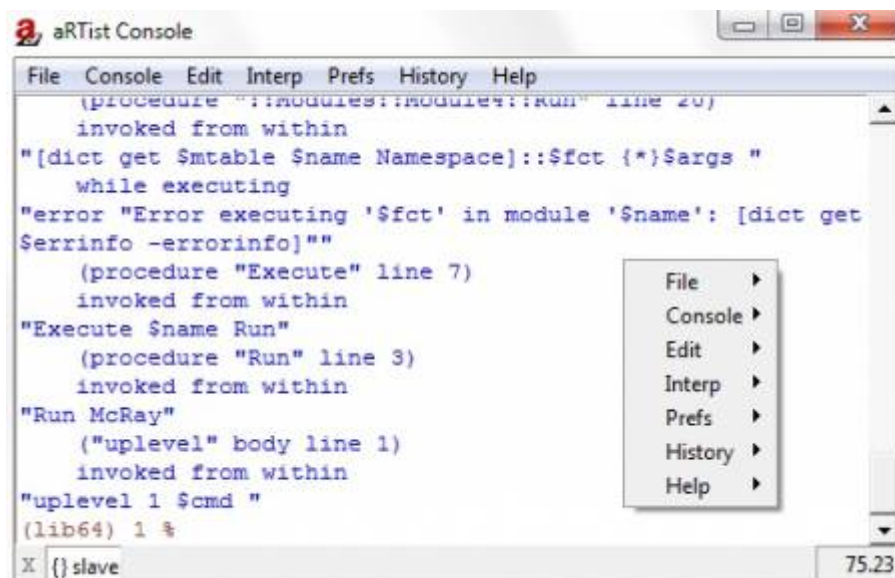
Fig. 18: Console

Show VTK Log

That command opens the *LogViewer* of the *Modules*.

Opening Angle

This command shows the actual opening angle of the cone beam of the source in the *Status Bar* (Figure 19).

0.958931 rad, 54.9° | Last calculation finished in 0.336s (CPU: SSE2)
Fig. 19: Status Bar indicates the Opening Angle (left)

Attenuation Data

Attenuation Data opens the *AttenuationViewer*, which displays attenuation-coefficient graphs as a function of the radiation's energy. A more in-depth explanation can be found under *Tools* → *Attenuation Data*.

Detector Properties

Detector Properties opens the *DetectorViewer*, which contains plots of the properties of the current detector.
Further information can be found under *Tools* → *Detector Properties*.

ImageViewer

This *ImageViewer* submenu of the tools menu is (for technical reasons) the access point to the menu of the *ImageViewer*. Therefore this topic ranks to the *Modules* → *ImageViewer* → *Tools Menu: ImageViewer*, where more information can be found. The submenu of the *ImageViewer* offers several commands for the current image/simulation. Please note that changes made with commands from the *Image* submenu are permanent (for example when saving the image). For non-permanent modifications use commands in the *View* submenu.

Materials

With the *Materials editor* all materials that are available within *aRTist* including density and composition are listed. It is possible to edit materials and to dock the editor in the docking area. For more information see: *Tools* → *Materials*.

Spectrum

Specific settings for the source can be adjusted by the *Spectrum calculator*. These settings can also be opened from the *Parameter Panel* → *Source*: . In the separate chapter *Tools* under *Spectrum* you can find more explanations about these settings. The Spectrum calculator can be docked in the docking area.

Settings

The *Settings* dialog box offers several options for the user interface. It should not be altered usually. Under *Tools* → *Settings* more information can be found.

View

This drop-down menu offers various different settings related to the point of viewing the scene and toggle functions for the items in the *Assembly List* (Figure 20).
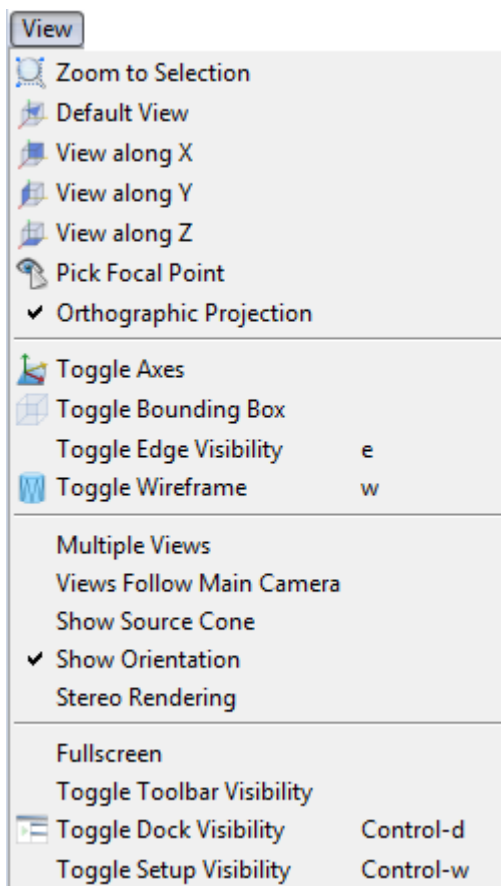

Fig. 20: View Menu

 Zoom to selection enlarges to a previously selected sector of the scene.

 Default view retrieves the standard perspective.

 View along X/Y/Z chooses either one of these options to view the scene along

different axis (*X, Y, Z*).

Pick focal point is used to select a different focal point in the scene.

Orthographic projection shows the assembly in the scene view as an orthographic projection when activated as opposed to a perspectival view. It can be helpful when you align the components of the assembly.

Toggle Axes displays the local coordinate system of a selected item in the assembly (Figure 21).
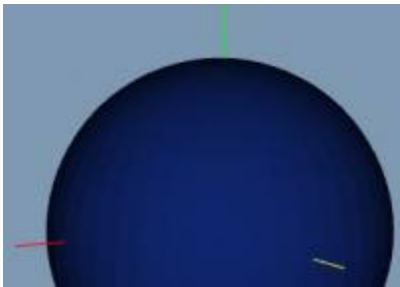
Fig. 21: Scene view showing an ellipsoid with its axes

Toggle Bounding Box displays the bounding box of a selected item in the assembly (Figure 22).
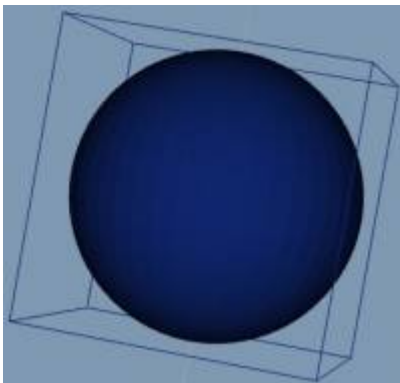
Fig. 22: Scene view showing an ellipsoid with its bounding box

Toggle Edge Visibility displays in black the edges of the triangles forming a specific item in the assembly (Figure 23). It can also be activated/deactivated by pressing *e* on the keyboard. It has only an effect if *Toggle Wirframe* is deactivated.
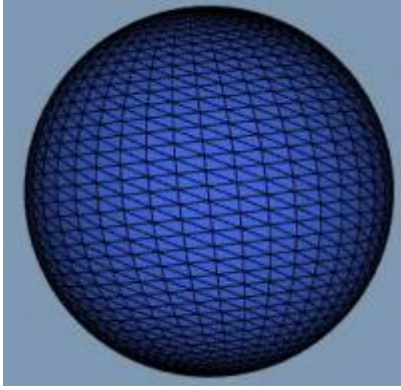
Fig. 23: Scene view showing an ellipsoid realized with edge of the facets

Toggle Wireframe displays a wire-frame model of a selected item in the assembly (Figure 24). The item will be transparent. As an alternative it can also be activated/deactivated by pressing *w* on the keyboard.
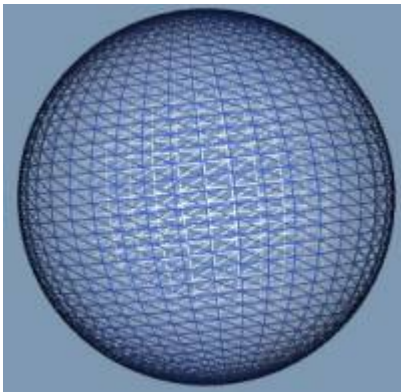

Fig. 24: Scene view showing an ellipsoid in wireframe mode

Multiple Views show the assembly from three different angles in the upper sector of the scene view (Figure 25).
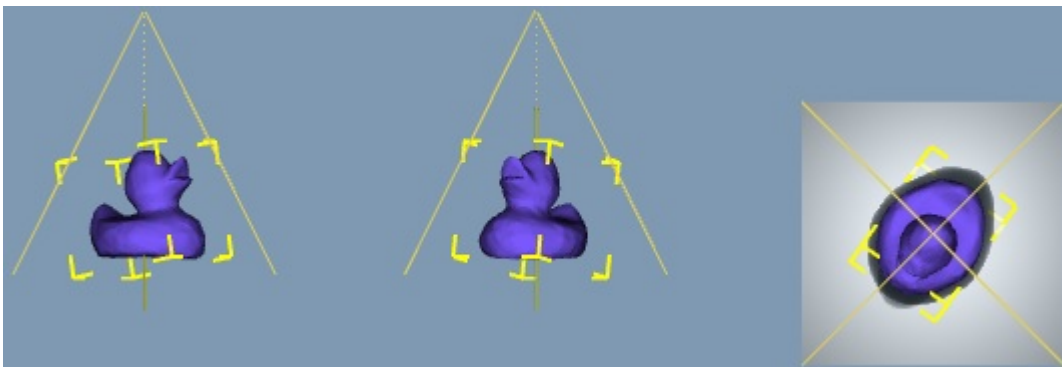

Fig. 25: Multiple Views

Views follow main camera refers to the previous one and only has an effect if both are activated. If acivated, multiple views will follow main camera when the scene is being rotated.

Show source cone enables/disables the display of the beam cone originating from the source (Figure 26).
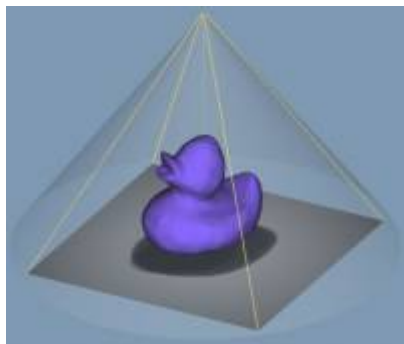
Fig. 26: Scene view showing the source cone

Show orientation activates/deactivatea the display of the coordinate system in the bottom left corner of the scene view.

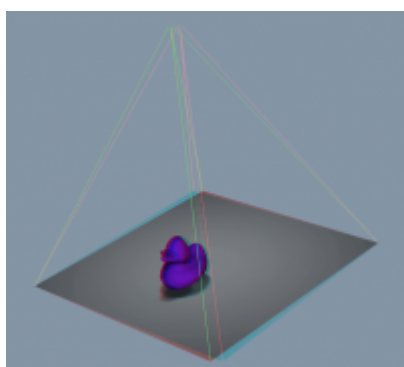**Stereo rendering** activates/deactivates stereo rendering in the scene (Figure 27).

Fig. 27: Scene view showing stereo rendering of the items

Fullscreen displays a fullscreen version of the program's user interface. Deactivate the option to retrieve normal view.

 Toggle Toolbar/Dock/Setup Visibility is used to hide the toolbar, docking area or *Parameter Panel*. Docking area and *Parameter Panel* can also be toggled by pressing Ctrl-d and Ctrl-w respectively.

Help

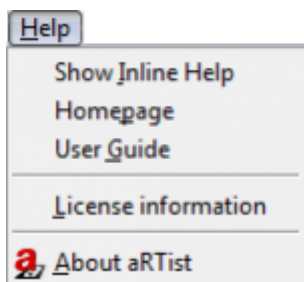This drop-down menu provides some help options and license information.

Fig. 28: Help Menu

 Show Inline Help depicts the inline help where available, e.g. at *Scattering*, or *materials*.

Homepage opens *aRT*ists homepage at *http://artist.bam.de/* .

User Guide opens the documentation file (PDF).

License information gives you the actual license status and. See also *Installation* → *aRTist Licenses*.

About *aRT*ist opens the about box where you can find information about the program, the installed version, installed modules and third-party software.

Parameter Panel

The *Parameter Panel* on the left contains four tabs: *Setup, Source, Scattering* and *Detector*. It offers a number of settings for the essential segments of a radiographic assembly to help you create a realistic radiographic simulation.

Setup

The "*Setup*" of *Parameter Panel* is mainly pertinent for positioning items in the assembly.

Default settings (Figure 29):

- *Environment: air*
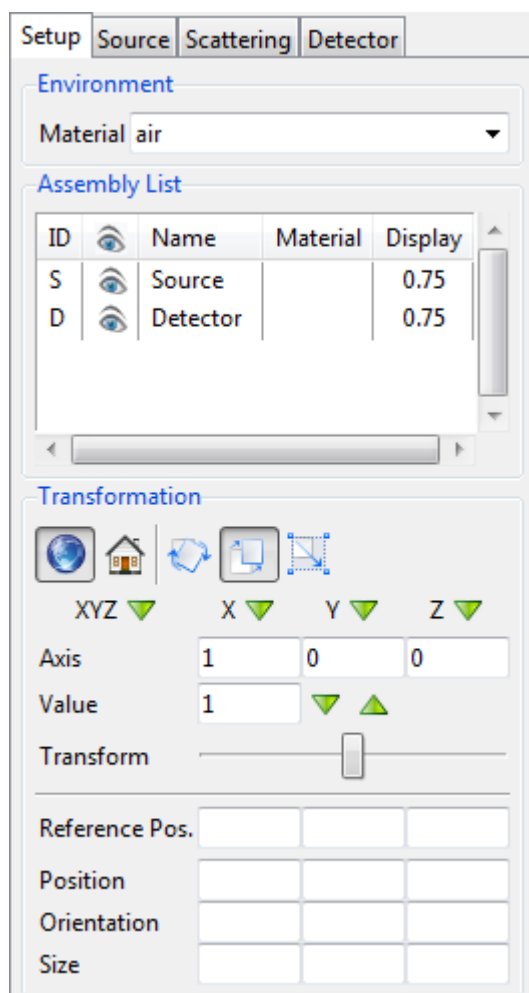- *world coordinate system*
- *translate mode*

Fig. 29: Parameter panel: Setup

Environment

Open the drop-down menu and select an environmental material of your choice. This material applies everywhere at the space of the scene outside of any part.

Assembly list

The *Assembly List* consists of five columns (Figure 30). You can select an item from the assembly either from the list or the virtual scene. Use therefore the Ctrl key on the keyboard and click on them with the left mouse. It will be highlighted blue in the list and marked with yellow corners in the virtual scene. All subsequent items in the scene will be marked with white corners.
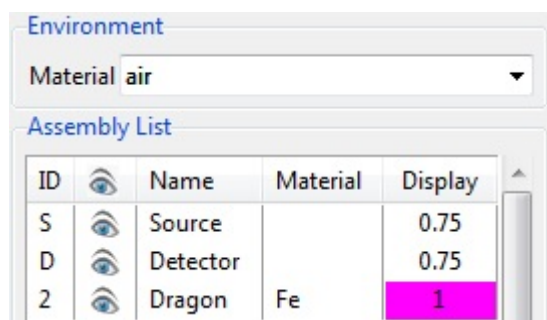

Fig. 30: Environment, *Assembly List*

- **ID** represents an unique identification for an entry.

- **Deactivates/Activates** an item with a click on the eye symbol. Alternatively, you may use the corresponding commands from the *Geometry* menu.

- **Name** changes the name of the part (by double-click). It is not possible to change the name of the detector and source.

- **Material** changes the material of a part (by double-click). Here it is not possible to change the material for detector and source.

- **Display** shows the color and opacity of each item. Double-click on the number to change the color and opacity.

Transformation

Under transformation you can find a number of options to manipulate and scale each item of the *Assembly List* (Figure 31). Proceed in the following way:

1. Select an item.

2. Choose between *world coordinate system* and *local coordinate system*.

3. Choose type of transformation: *rotate, translate* or *scale*.

4. XYZ ▽   X ▽   Y ▽   Z ▽ Select an axis or all three with a click on one of the green arrows. 0 = axis is currently not selected, 1 = axis is currently selected.

5. ▽ △ Set the step size with the green arrows or type in a value. The unit matches the type of transformation: degree, *mm* or 10 % magnification.

6. [icon] Rotate, translate or scale the item with the slide control or directly in the scene with the left mouse button. You can monitor the movement/modification of *Reference position, Position, Orientation* and *Size* over the display fields in the bottom of the *Setup* page and also put in values yourself as an alternative to the previously explained approach.
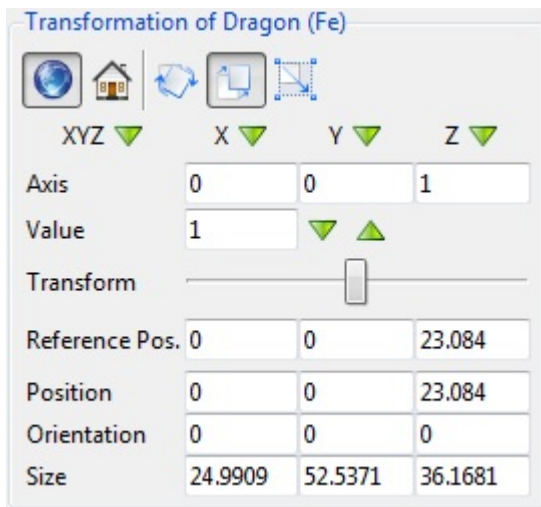

Fig. 31: Transformation of an item

Source

The "*Source*" tab of *Parameter Panel* is important for defining the radiation source settings.

Default Settings (Figure 32):
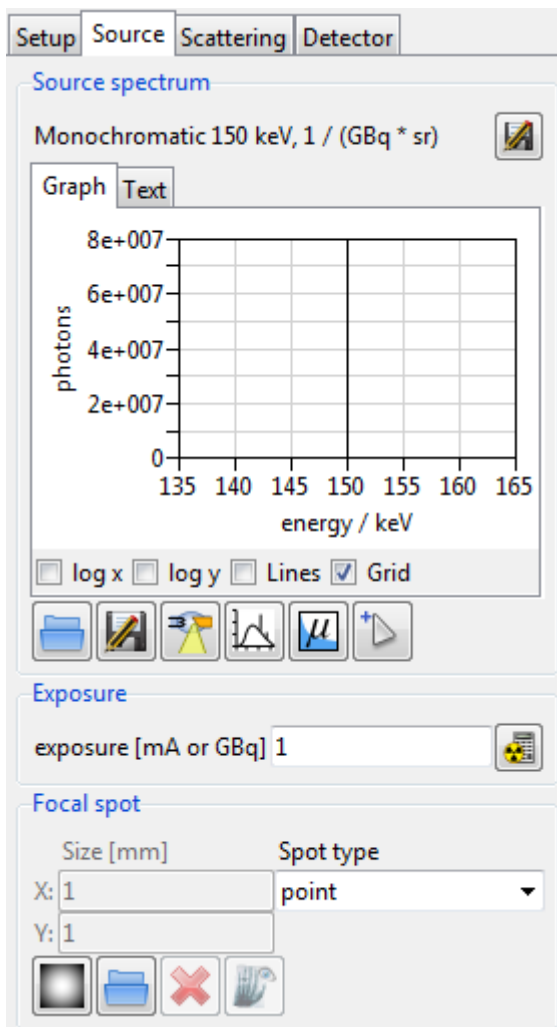
- *point source*
- *voltage: 150kV*
- *graph*

Fig. 32: Parameter panel: Source

Source Spectrum

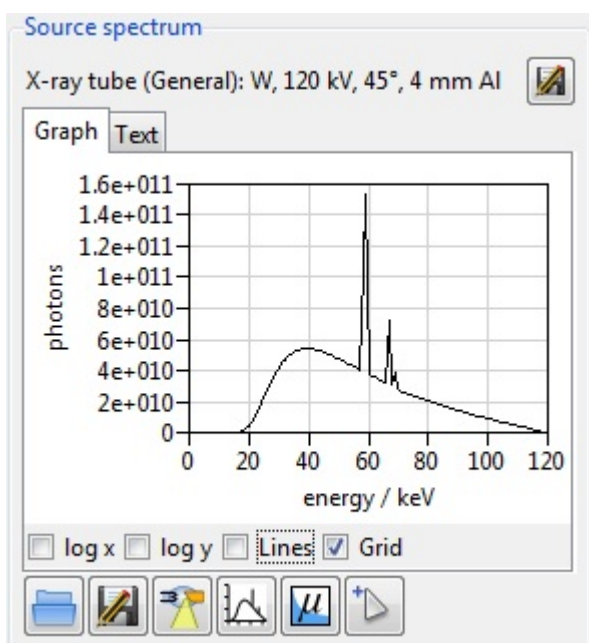The main area of this page displays the source spectrum (Figure 33).

Fig. 33: Source spectrum

<u>Top:</u>

- ![icon] *Save spectrum graph or data* as a *.txt* file.
- Choose between *graph* or *text* via tabs

<u>Bottom:</u>

- Enable/Disable *log x, log y, Lines, Grid* via checkboxes.

- ![icon] *Load spectrum* from *aRT*ists library. You can choose from four different spectra: *Co-60, Ir-192, Se-75, Yb-169*.

- ![icon] *Save current spectrum* as a *.xrs* file.

- ![icon] *Calculate x-ray tube spectrum* opens the *Spectrum calculator* where you can make detailed adjustments to the tube. Important details of the current tube settings are displayed directly above the *Graph/Text* tabs. A more in-depth explanation of the spectrum calculator can be found in the *[Tools](#)* → *[Spectrum](#)* section of this tutorial.

- ![icon] *Show current spectrum* opens the current spectrum in a seperate window. Click and hold for a second or right-click for more options (*Show original spectrum, Show reduced spectrum* or *Show preview spectrum*).

- ![icon] *Plot attenuation data* displays the attenuation data for the current settings. In the separate section of this tutorial *[Tools](#)* under *[Attenuation Data](#)* more information can be found.

- ![icon] *Show attenuated spectrum for picked position* opens the *Spectrum Picker*. The incident beam is shown in black, the attenuated beam in blue (Figure [34](#)). You can dock the window in the docking area.
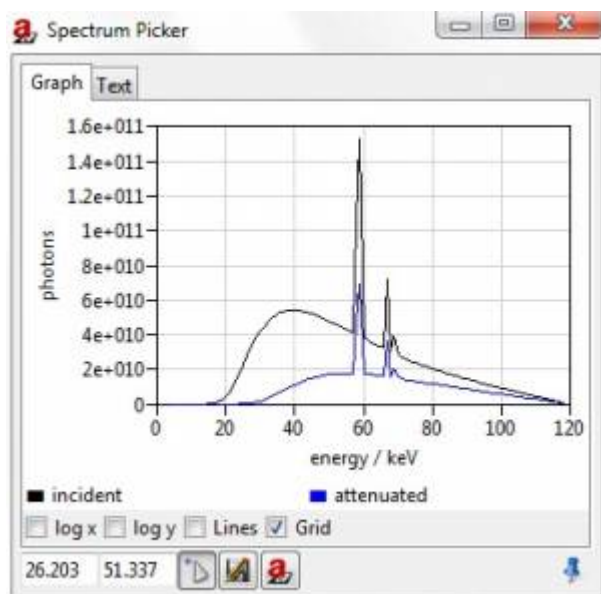

Fig. 34: Spectrum Picker

- ![icon] *Pick image position to calculate attenuation for* is used to pick a spectrum reference position in *[ImageViewer](#)* (click with the left mouse button and hold). Alternatively, you may put in the *X-*

and *Y*- coordinates manually. The beam shown in Figure 34 is attenuated by an aluminum cube.

- ⧄ *Save current data* as a *.txt* file.

- ⓐ *Recalculate attenuation data*

Exposure

Enter the tube current in *mA*, or the activity in *GBq* if you want to use a radionuclide (Figure 35). Open the *Activity calculator* (▦) to determine a actual activity from a reference activity in the past.
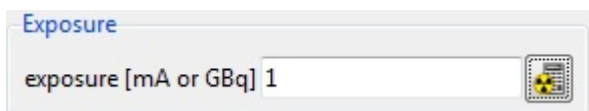
Fig. 35: Exposure by radionuclide

Activity Calculator

- ⟨⟨ ⟨ ⟩ ⟩⟩ Pick reference and exposure dates - « »: year, < >: month. To enquire the present-day activity, you may simply press the *Today* button in the lower part of the window.

- Put in activity in *[GBq]* or *[Ci]* and halflife in *[s]*. You may also convert to the respective alternative unit.

- Press *Set* to apply changes.

Fig. 36: Activity Calculator

Focal spot

Select number and arrangement of focal spot samples from the list or enter your own values (Figure 37). A single integer denotes the number of samples randomly distributed by poisson disk sampling algorithm (weighted by focal spot image if this has been set). An entry of the form *NxM* denotes a regular grid. Geometric unsharpness is negligible. A 5 point source is sufficient to check for possible unsharpness.

X, Y: Size of focal spot along *X, Y* in *[mm]*. Attention: This is the overall size, or extent, of the focal spot. If a profile image is used, the effective spot size may be smaller.
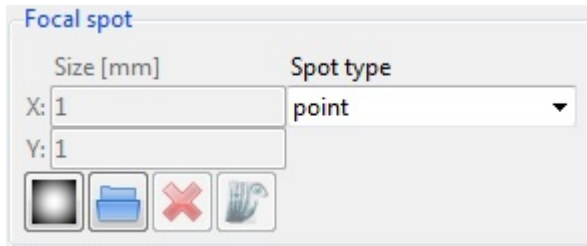
Fig. 37: Focal spot settings

 Create a spot profile image opens the *Focal Spot Profile* dialog box.

Set the Focal Spot Profile (Figure 38):

- effective spot *width* in *mm*, FWHM of horizontal profile
- effective spot *height* in *mm*, FWHM of vertical profile
- Lorentz *fraction* of pseudo Voigt profile, 0.0 … 1.0
- pixel dimension of spot profile image (*Resolution*)
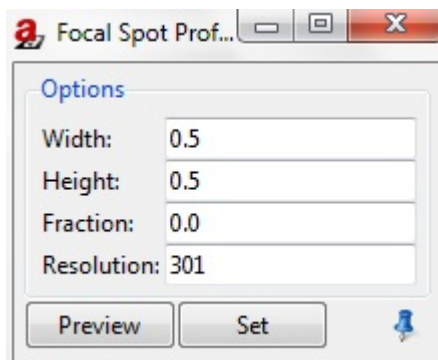
Fig. 38: Focal Spot Profile settings

 Load spot profile image from file: Supported files: *.tif, .tiff, .raw, .bin, .fld, .png, .bd, .dd*.

 Delete spot profile image

 Show spot profile image

Scattering

The "*Scattering*" tab of *Parameter Panel* is useful for different scattering settings (homogenous, McRay etc.) in the stimulated radiation.

Default settings (Figure 39):

- *Mode: off*
- *Reference point: min*
- *Set defaults: ✔*
- *# of photons: 2e+007*
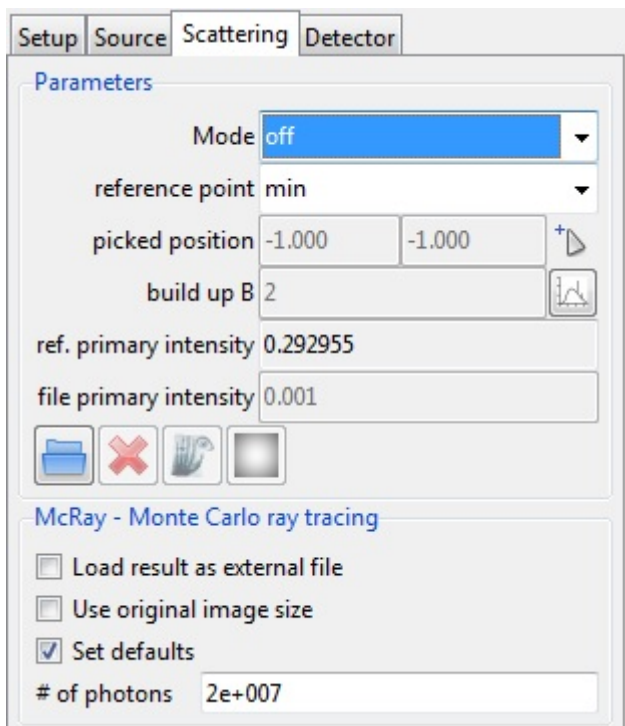
Fig. 39: Parameter panel: Scattering

Parameters

Mode

off does not take into account scattered radiation at all.

build-up factor is used to estimate a constant scatter. The build-up factor is the ratio of total to primary radiation at the reference position.
Decreased contrast due to scattering is simulated. Set the build-up factor in the *build-up B* input field. Build-up factor of 2 means as much scatter radiation (S) as primary radiation (P) impinges on the detector at the reference position.

 BuildUp Estimator estimates the BuildUp Factor for a specific part. You can dock the window in Figure 40 in the docking area.

- Set material and thickness in *mm*, the number of layers and the energy resolution in *keV*
- Enable/Disable multiple scatter
- Press *Update* (BuildUp factor will be estimated but not set) or *Set*

Layers, Energy resolution and *Multiple Scatter* are model parameters for the algorithmus function thereby. *Material* and *Thickness* are physical parameters for the geometric function. After the program restarts with a *build-up factor* additionaly *primary + scatter intensities* can be seen in *ImageViewer*.
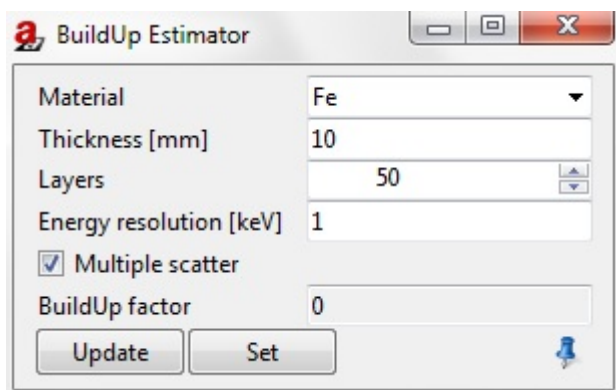
Fig. 40: BuildUp Estimator settings

external file is used for a user-supplied image, which is scaled and added. The corresponding primary radiation at the reference position has to be set (→ *file primary intensity*).

McRay is used to determine the amount of scattering with Monte Carlo simulation. It calculates fully automatic the Monte Carlo of scatter contribution. A larger number of photons results in a better estimate, but longer calculation times.

Reference Point

Select the reference point within the detector image. Available settings:
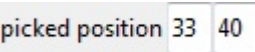
Off is used to set a value under *ref. primary intensity*, instead of using a reference point.

Min selects the image point of minimum grey value as the reference.

Max selects the image point of maximum grey value as the reference.

Center selects the image center as the reference point.

Picked is used

- to pick a scatter image reference point in *ImageViewer* (click and hold) or
- picked position 33 40 to put in the *X* and *Y* coordinates manually.

: *Load, delete, show* or *smooth* scatter image.

McRay - Monte Carlo ray tracing

Monte Carlo ray tracing (Figure 41) will contribute a image of scatter radiation based on the actual setup. This includes considerable extra computing effort. Typically, it takes seconds to minutes for one calculation. Worthwhile when inspecting castings (thick walls). Build-up factor is recommended when inspecting metal sheets and welds. More detailed settings for McRay can be done in the *Menu Bar* → *Modules* → *McRay*.

- *Load result as external file*
- *Use original Image size*
- *Set Defaults*
- *# of photons*

When McRay is finished in the upper bar of the *ImageViewer* module not only *Energy density (J/m²)* and *primary intensities* can be seen but also *scatter intensities* and *primary + scatter intensities*.
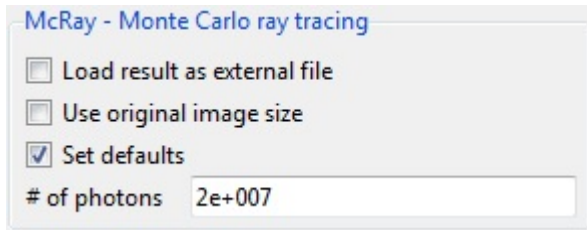
Fig. 41: Monte Carlo ray tracing

Detector

The "*Detector*" tab of *Parameter Panel* is relevant for the settings of detector properties like geometry and type.

Default Settings (Figure 42):

- *Size [mm]:* X = Y = 100 mm (square)
- *Multisampling: source dependent*
- *Curvature: off*
- *detector type: 1:1*
- *reference point: max*
- *set to [D or GV]: 50000*
- *# of frames to average: 1*
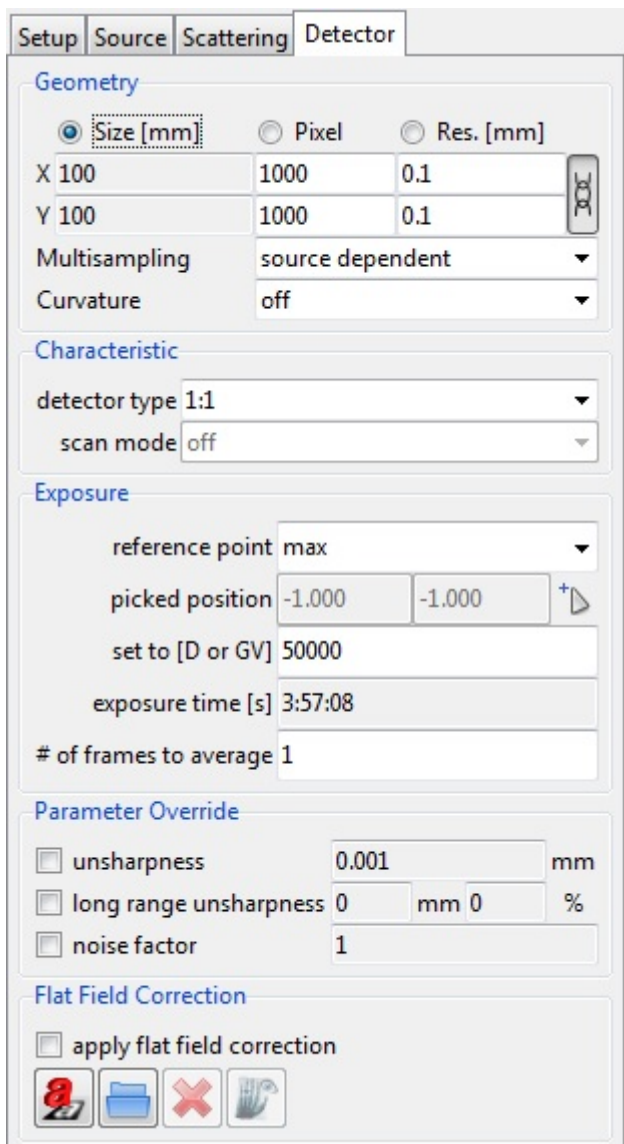
Fig. 42: Parameter panel: Detector

Geometry
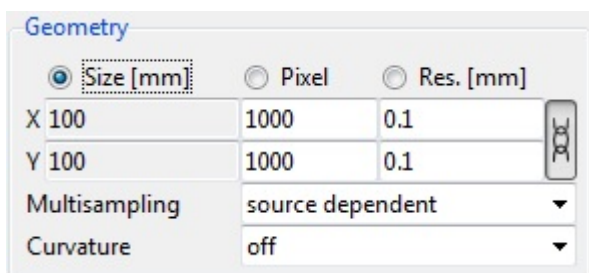
Fig. 43: Geometry settings

In this part of the Detector tab, you can adjust the geometry of the detector (Figure 43). Choose which quantity you want to update automatically (*Size [mm]*, *Pixel*, *Resolution [mm]*) and set the values for *X* and *Y*. With the button next to the input fields you can predetermine a square pixel area.

Multisampling is used for anti-aliasing. The default setting is source dependent, which means, that the source's sampling pattern will also be used for the image resulting in reduced computation time. The other available patterns of the list are: *5, 10, 20, 30* and *1×1, 2×2, 3×3, 4×4, 5×5*. Thereby you can also enter your own values. A single integer denotes the number of samples randomly distributed

and an entry of the form *NxM* denotes a regular grid.

Curvature is used to have a curved detector around its reference point along the selected axis (reference point of the detector should <u>not</u> be the same as its position, but e.g. the center axis of a cylindrical part). This can be useful if you want to test curved parts like a pipeline. Thereby you can choose between *X* or *Y* curvature.

Characteristic

Fig. 44: Characteristic settings

Detector type is used to select the type of detector you want to use in your simulation. You may choose between

- different types of film (*film D4, film D5, film D5 (Ir 192), film D7, film IX100, film IX50, film IX80*)
- two imaging plates (*ST-VI Dynamix SK708, UR-1 Dynamix SK 708*)
- a *flat panel* detector
- an *Air dose meter*
- an *1:1* detector type
- *10000:1 (4IC), 10000:1 sharp noisy* or
- load an external file (*External*)

To access the *digital detector* open the *DigRad* module (*Modules → DigRad*). The default setting for detector type is *1:1*, which means a *1:1* conversion of dose values into grey values.

For example it is possible to load (in *External*) saved detector files, which were done with the *DetectorCalc* module (*Modules → DetectorCalc*).

Scan mode takes digitization of film into account. The drop-down menu is therefore only active when a film is selected for detector type. Then you can choose between a laser-based (*Kodak LS85*) and a CCD (*CCD 16bit*) based scanner or deactivate (*off*) the option.

Exposure

Fig. 45: Exposure settings

The following settings are available for **reference point**:

- *off* sets the exposure time in seconds or as hh:mm:ss in the corresponding input field and the
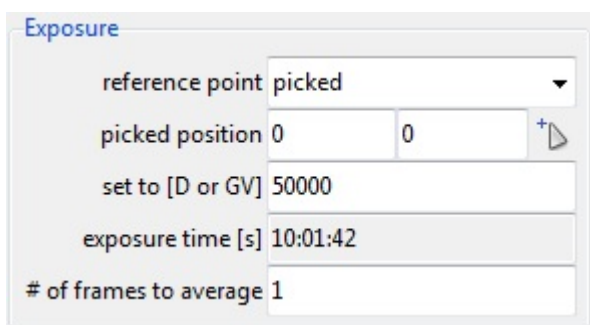
number of frames to average below.

- *min* sets the exposure time automatically for minimum grey value. Set dose or grey value to achieve at the reference point (*set to [D or GV]*).

- *max* sets the exposure time automatically for maximum grey value. Set dose or grey value to achieve at the reference point (*set to [D or GV]*).

- *center* sets exposure time automatically for medium grey value. Set dose or grey value to achieve at the reference point (*set to [D or GV]*).

- *picked* picks auto-exposure reference point either through manual input of the *X* and *Y* coordinates in the corresponding fields (*picked position*) or by selecting a position in [*ImageViewer*](#) (activate access with this button: ⁺▷).

set to [D or GV] sets dose or grey value to achieve at reference point if possible.

# of frames to average sets the number of frames to average.

Parameter Override
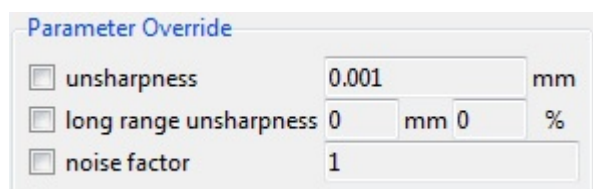
Fig. 46: Parameter Override settings

If the command is selected it is necessary to change the number of the parameters for override (Figure [46](#)):

- *unsharpness* sets the receptor unsharpness in *mm*.
- *long range unsharpness* sets a greater receptor unsharpness for imaging plates.
- *noise factor* sets → no noise factor or *1* → noise factor according to characteristic (factor is not proportionate).

Flat-field correction
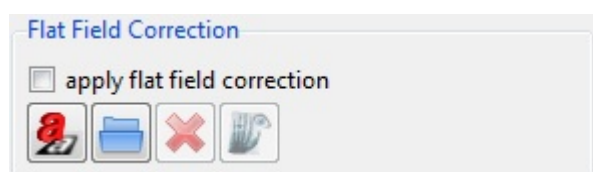
Fig. 47: Flat field correction settings

Flat-field correction is part of the calibration procedure of digital detector arrays. It is used to remove artifacts from 2-D images that are caused by variations in the pixel-to-pixel sensitivity of the detector and/or by distortions in the optical path. It also removes the shading in the image due to the different path lengths of the cone beam. For this reason, the flat-field correction is available here in *aRT*ist as well. The flat-field ("white") image is generated by an simulation without any parts in the scene. If there isn't already a flat field image present, it will be automatically generated. (Figure [47](#)).

Calculate, load, delete or *show* the flat field image.

Modules

In this chapter the modules of *aRT*ist *Menu Bar* will be explained in detail.
This drop-down menu contains different *Modules* that can all be docked in the docking area or be handled as separate windows (Figure 48).
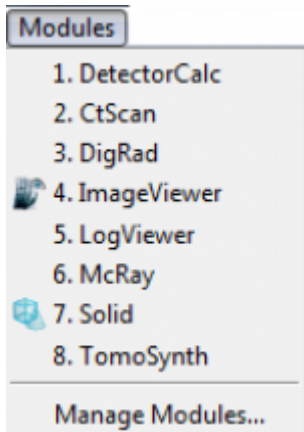
Fig. 48: Modules Menu

Manage Modules

Select the modules you wish to load at startup of *aRT*ist (Figure 49). All changes take effect only at the next startup. You can activate/deactivate: *DetectorCalc*, *CtScan*, *DigRad*, *McRay* and *TomoSynth*. Modules like *ImageViewer*, *LogViewer* and *Solid* are obgligatory and cannot be deactivated.
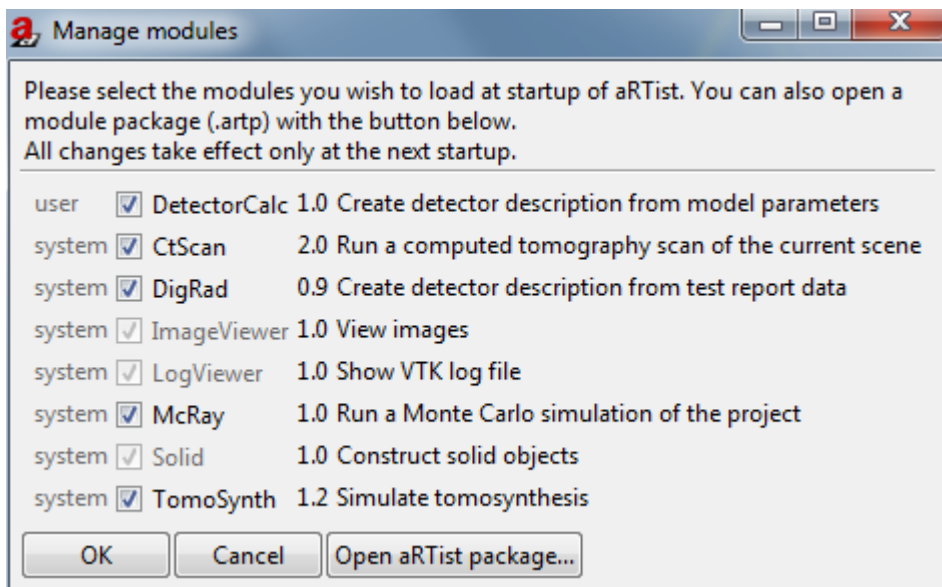
Fig. 49: Manage modules

DetectorCalc

The DectorCalc module opens the *Detector calculator* (Figure 50). It is useful for the detector's flat

panel settings and calculating the captured photons. In addition to the more general *DetectorCalc* there is a simplified detector calculator: *DigRad*.
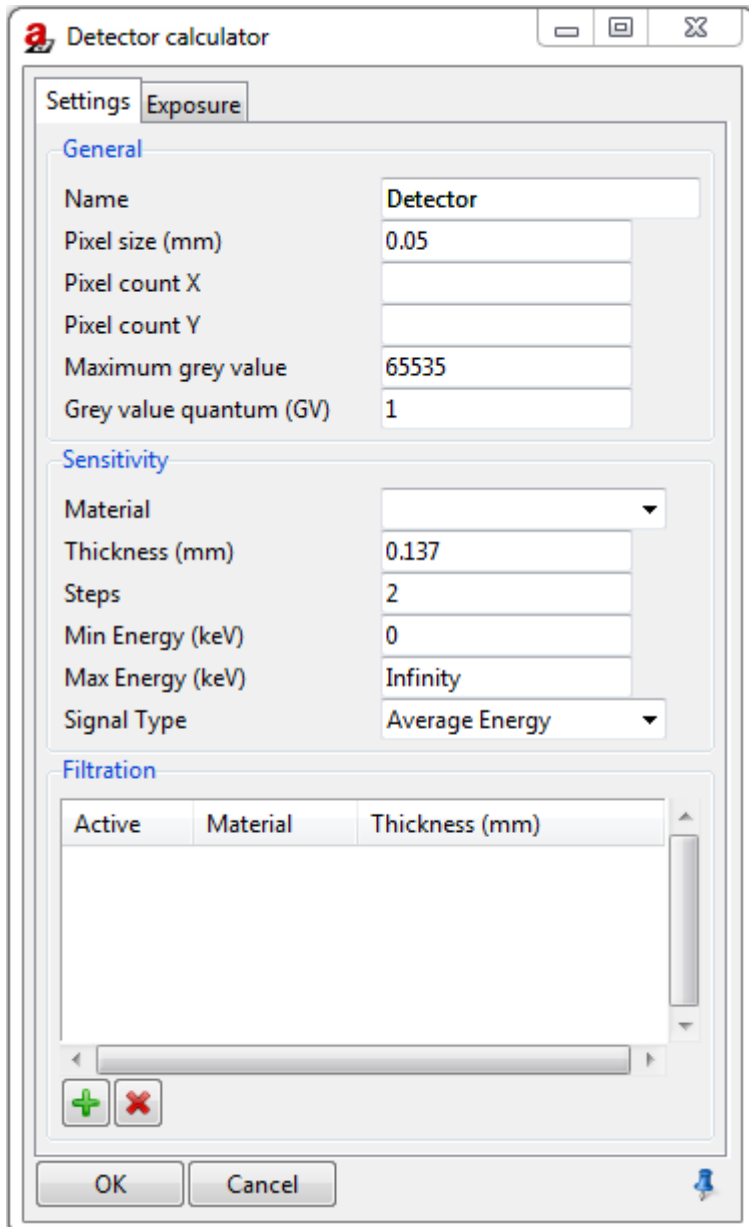
Fig. 50: Detector calculator, Settings

General Settings

Set the general settings for your new detector:

- *Name*
- *Pixel size (mm)*
- *Pixel count X*
- *Pixel count Y*
- *Maximum grey value*
- *Grey value quantum (GV)*

Sensitivity

Set

- *Material*
- *Thickness (mm)*
- *Steps*

The amount of *Steps* are relevant for the layer model. The more steps you set, the more accurate the simulation will be. Thereby the execution time will increase. Typically two steps will be enough.

- *Min Energy (keV)*
- *Max Energy (keV)*
- *Signal Type*

Filtration

It is possible to add a filtration over the new detector.

Click on the green button to add a filter (default filter is Al with 1.0 *mm* thickness).

Click on the red button to delete a selected filter.

If you want to change the settings of your filtration, click on the corresponding *Material* or *Thickness*. For the *Material* column you can choose a material of the editor list (*Tools* → *Materials*). For *Thickness* just enter your desired value (in *mm*).

The column *Active* enables to activate (*Yes*) or deactivate (*No*) the selected filter. Therefore use the left mouse button to choose *Yes* or *No*.
Press *OK* to apply or *Cancel* to quit these settings.
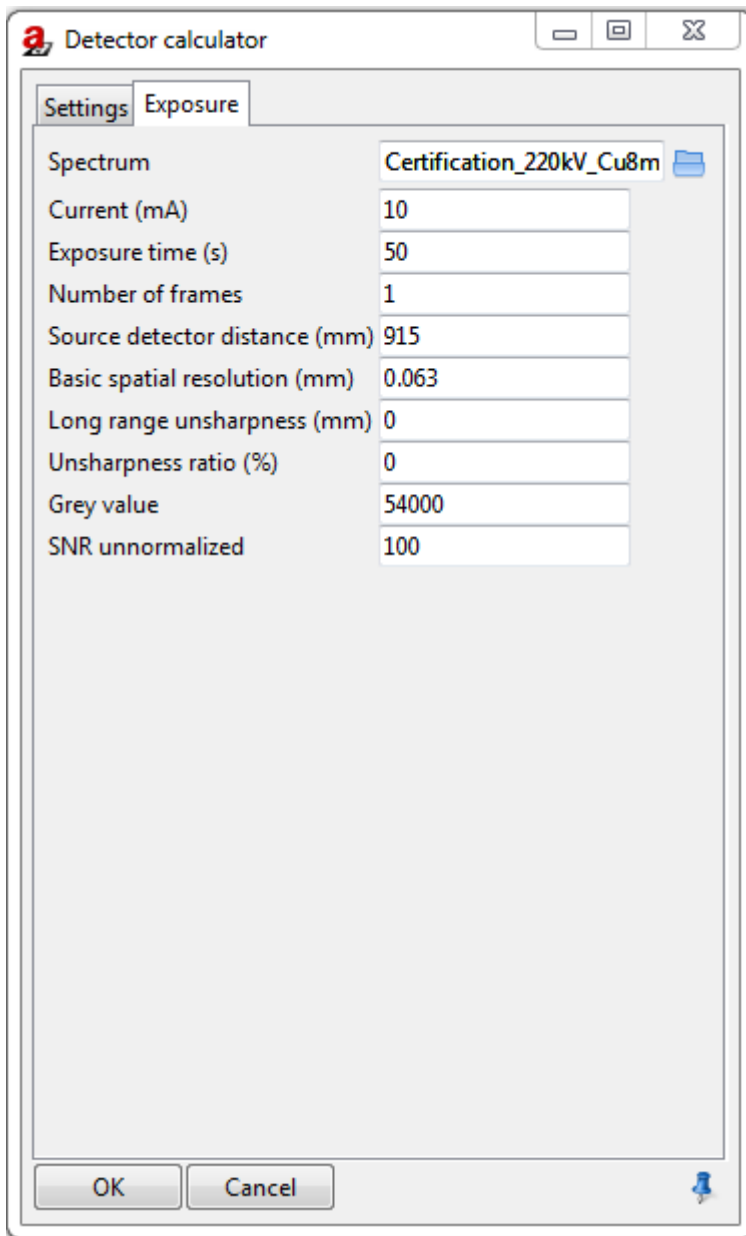
Fig. 51: Detector calculator, Exposure

Reference shot

For creating a new detector it is necessary to set reference values. With those, the new detector will be calibrated (with available detector settings) and considered for calculations.

Set

- *Spectrum*
- *Current (mA)*
- *Exposure time (s)*
- *Number of frames*
- *Distance (mm)*
- *Long range unsharpness (mm)*
- *Unsharpness ratio (%)*
- *Grey value*
- *SNR unnormalized*

Afterwards the new detector can be loaded and applied at *Parameter Panel* → *Detector* → *Characteristic* → *detector type* → *Extern*.

CtScan

With the CtScan module, a computerized tomography (CT) scan can be simulated. The module is accomplished by Feldkamp reconstruction and volume visualization. The simulated projections can be processed in the same way as experimental CT projections by external reconstruction programs. It is recommended to use the TIFF file format for easy data exchange in this case.

The CtScan module consists of three separate pages that can be navigated using the tabs at the top of the window (Figure 52). At bottom of module window, you can find a *Run* button to start or to stop the simulation and a progress bar. The first two pages offer settings for the scan and for reconstruction (*aRT*ist uses the Feldkamp algorithm for reconstruction). The finished tomogram can be viewed in the scene with the help of the first (*Setup*) or third (*Volume View*) tab by clicking on the *Show* button. In *ImageViewer* the sectional image can be seen with the *Slice View* in the third (*Volume View*) tab. All three pages will be explained in detail below.

Setup

Fig. 52: CtScan: General settings

Scan Parameters

Set

- *total angle [°]*
- *number of steps* and
- *angle step size [°]*.

Output

Choose directory and file type and put in a file name. The file type can be chosen between *BAM CT* or *TIFFs* and *16 bit* or *float*. If fields are left blank they will be filled in automatically. If you wish to start reconstruction directly after the scan is finished check the corresponding box, *Run Feldkamp*. You may set th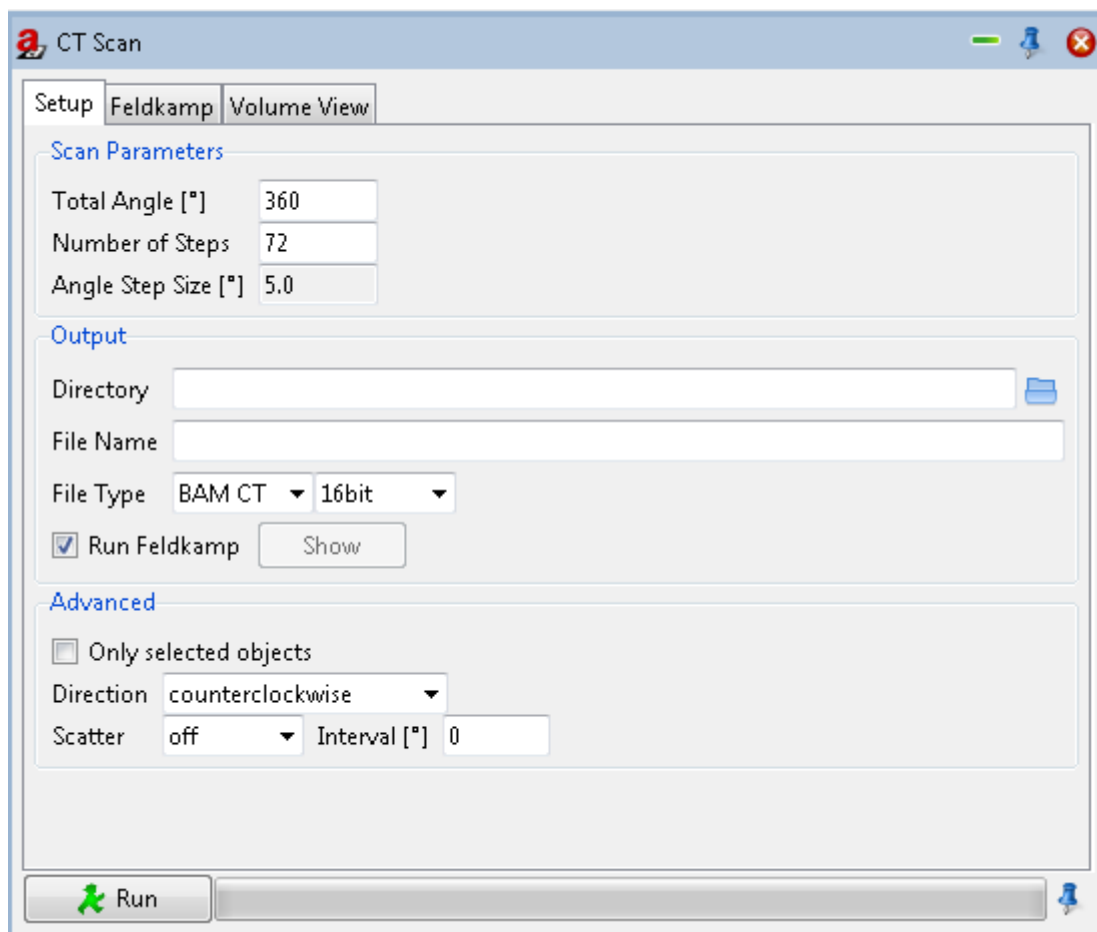e parameters for Feldkamp on the second page. Press the *Run* button on either one of the three pages when finished with the settings. If you choose not to check the *Run Feldkamp* box before starting the simulation you may initiate reconstruction manually (see: *Feldkamp*). Click on the *Show* button when the simulation is finished to see the tomogram in the scene. For an improved view deactivate the part or minimize the opacity (double-click on the number of the object to change the opacity: *Parameter Panel* → *Setup* → *Display*).

Advanced

Set

- *Direction* is used to choose between clockwise or counterclockwise direction.
- *Scatter* is used to choose between off or McRay scatter-attitude.
- *Interval [°]* sets the interval for using the same scatter image (to save some time by not recalculating for each projection).

You may also choose to only include selected parts of your assembly in the simulation.

Feldkamp

Set the parameters for reconstruction on the second page (Figure 53). Check the *Run Feldkamp* box on the first page to start reconstruction automatically after the necessary data has been acquired. Otherwise, you may start reconstruction manually by pressing the *Reconstruct* button in the lower half of the *Feldkamp* page.

- *File name* 📁: You may load previously acquired data with a click on the file symbol or use the recently acquired data.

Enable/disable:

- *Interpolation*
- *use of GPU*
    - *multiple textures*
    - *half precision*

Output options:

- *File type: BAM CT, VTK*
- *8bit, 16bit, 32bit, float*
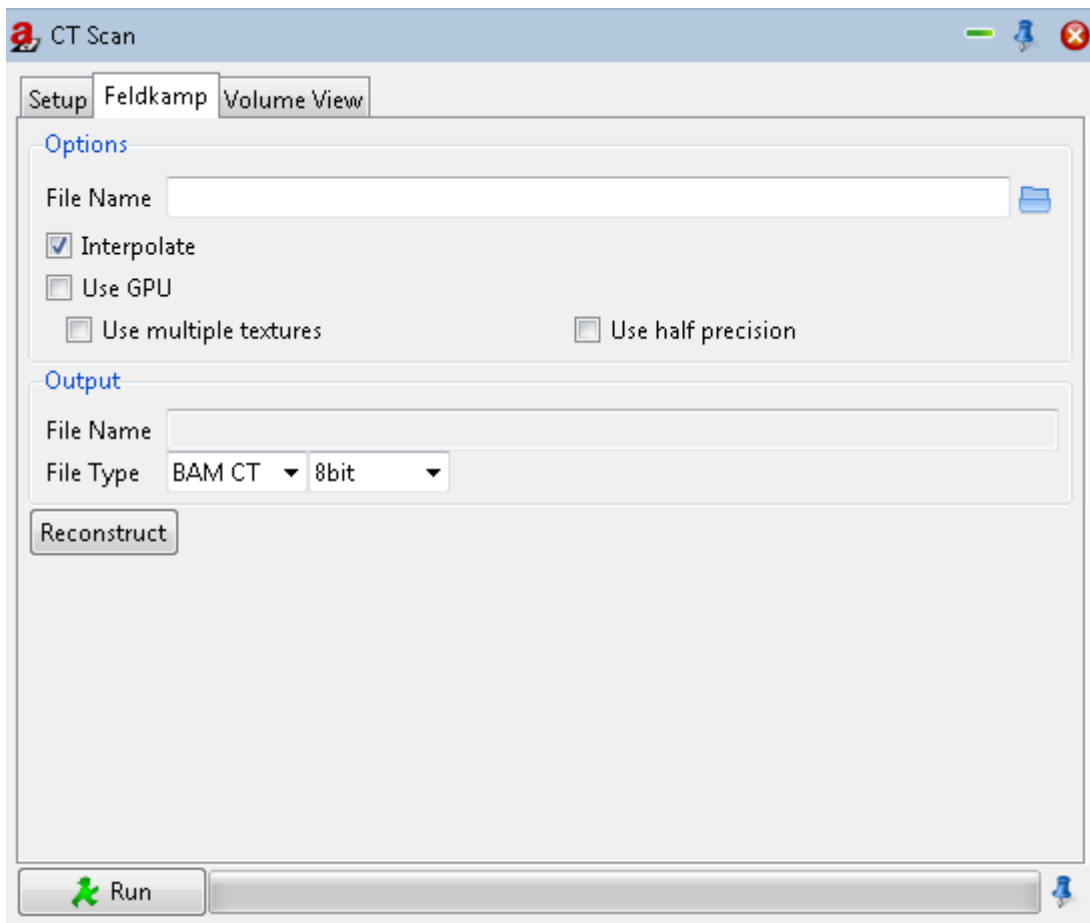
Fig. 53: CtScan: Parameters for reconstruction

Volume View

With the tab *Volume View* (Figure 54) it is possible to show sectional image of the finished tomogram in *ImageViewer* directly after reconstruction. Alternatively, you may load previously acquired data. Supported file types are: *.bd, .raw, .bin, .vtk.*

Press the *Show* button on this page or the first page to start the calculation. This will take a few seconds. Then you can view the 3D-Scan of the part in the scene. Deactivate the part or minimize the opacity for an improved view of the 3D-Scan.

With the option *Slice View → Show plane* a sectional image of the part for the respectivly axis (*X, Y* or *Z*) will appear in *ImageViewer*. If *Show slice* is activated, borders of the object can be seen in the plane. Choose an axis and move the plane with the slider to see a sectional image of the part.

Settings for **Volume View**:

- *Minimum*
- *Maximum*
- *Level*
- *Contrast*
- Activate/Deactivate *3D texture*
- *Gradient/GradientROI* → choose between *All, Min/Max* or *Level/Contrast*

In **Positioning**, the scene is affected. It is possible to position the voxel packet. With *Spacing*, the size of the scene can be influenced (in *X, Y, Z*).
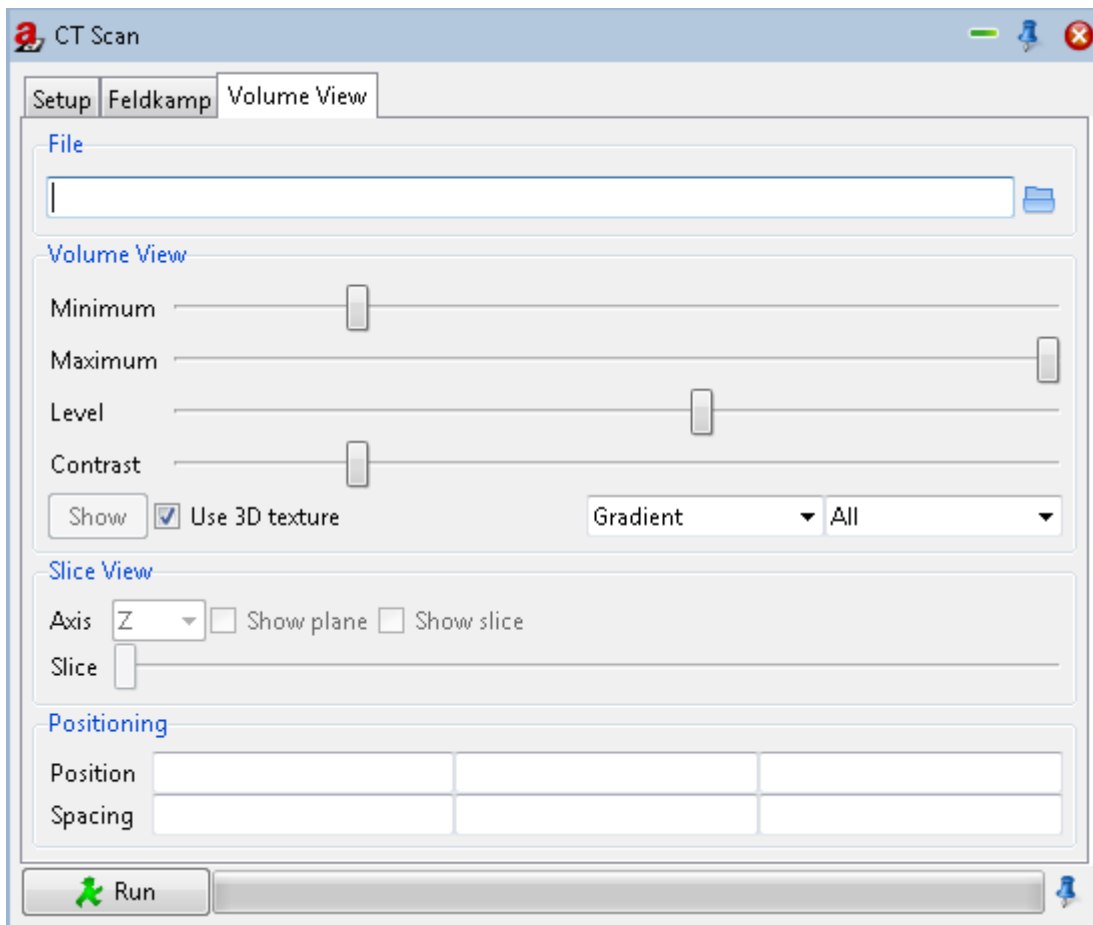
Fig. 54: CtScan: Parameter for voluming rendering

DigRad

The DigRad module refers to the type of detector used in the simulation. For example if you want to use a digital detector, open this module and set the parameters (Figure 55). Press *Apply* when finished.

- *Pixel size (mm)*
- *Basic spatial resolution SRb (mm)*
- *Internal scatter radiation ISR (%)*
- *ISR correlation length (mm)*
- *Gray response (GV/mGy)*
- *SNRn,max*
- *Efficiency (SNRn/mGy^0.5)*
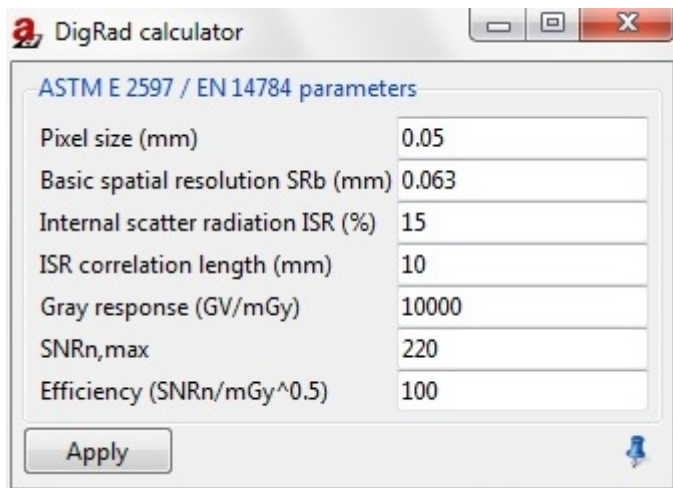
Fig. 55: DigRad calculator

ImageViewer

The *ImageViewer* module is the only module that is in the docking area by default (Figure 57). It displays the preview image and the finished simulation. You may choose between two options as mentionend before: *radiography* and *thickness maps* (see: *Compute* segment in this tutorial). The different settings and functions of *ImageViewer* will be explained in this chapter.

Upper bar

: A crossed out red circle means that you are currently viewing a preview of the simulation. The preview has a lower resolution and is calculated with different parameters for faster results.

: Run the simulation and a green tick will appear indicating that the simulation was successful.

The drop-down menu right next to that symbol offers different views from the simulation process (*Energy density J/m²* or *primary intensities*). The available options depend on the current settings.

The three grey fields next to the drop down menu display the mouse position (first grey field: *X*-axis, second grey field: *Y*-axis) and the *energy density* [in *J/m²*] or *primary intensities* in grey value or *thickness* [in *mm*] (third grey field) at that position. Those figures are also displayed in the status bar.

Lower bar

Display range indicates min. and max. *grey value/thickness* of the current image. If needed, you may change those values with your keyboard. Press *Enter* when finished.

Optimize display range (dependent on current image).

Reset display to default range (0 - 65535).

Display inverted to a negative of the image and back.

Zoom: Fit *Zoom* in and out with the up-down control of the spin box. Alternatively, you may use the mouse wheel or the right mouse button. Default: *Fit* (zoom fits window).

Show in actual size shows a *1:1* depiction of the current image.

Zoom to fit window resets zoom to fit window after zooming in or out.

Zoom to selection enlarges to a selected part of the image. Select the desired part with the left mouse button.

Bar on the right

Save the current image.

Show in external viewer opens the current image in an external viewer.

Display exposure latitude warning (de)activates warning for over- or underexposure. Overexposed areas will be marked red, underexposed areas will be marked blue.

Pixel for pixel mode changes between actual pixel size and screen pixel size. This option is only active when *Zoom = Fit*.

Change image orientation opens a menu with different options to mirror or rotate the image.
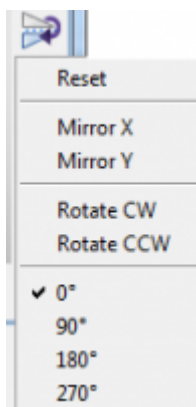
Fig. 56: Image View: Change image orientation

Reset is used for resetting the settings in the image orientation.

Mirror X/Mirror Y is used to reflect the image orientation on *X*-axis or *Y*-axis.

Rotate CW/Rotate CCW is used to spin the image clockwise or counter-clockwise.

0°/90°/180°/270° is used to orientate the image (between four different angles).

The same menu like in Figure 56 can be found in the *Tools* menu under *ImageViewer* → *View* → *Orientation*. Please note that the changes made with commands from either one of those menus do not have lasting effects, for example, when saving the image. If you want to permanently edit an image use the commands from the following menu: *Tools* → *ImageViewer* → *Image*.

[icon] *Show ruler* displays a ruler on the bottom of the image.

[icon] *Show legend* opens a legend with grey or thickness values. Minimum and maximum values are determined by the display range. Height, width and position of the legend are adjustable.
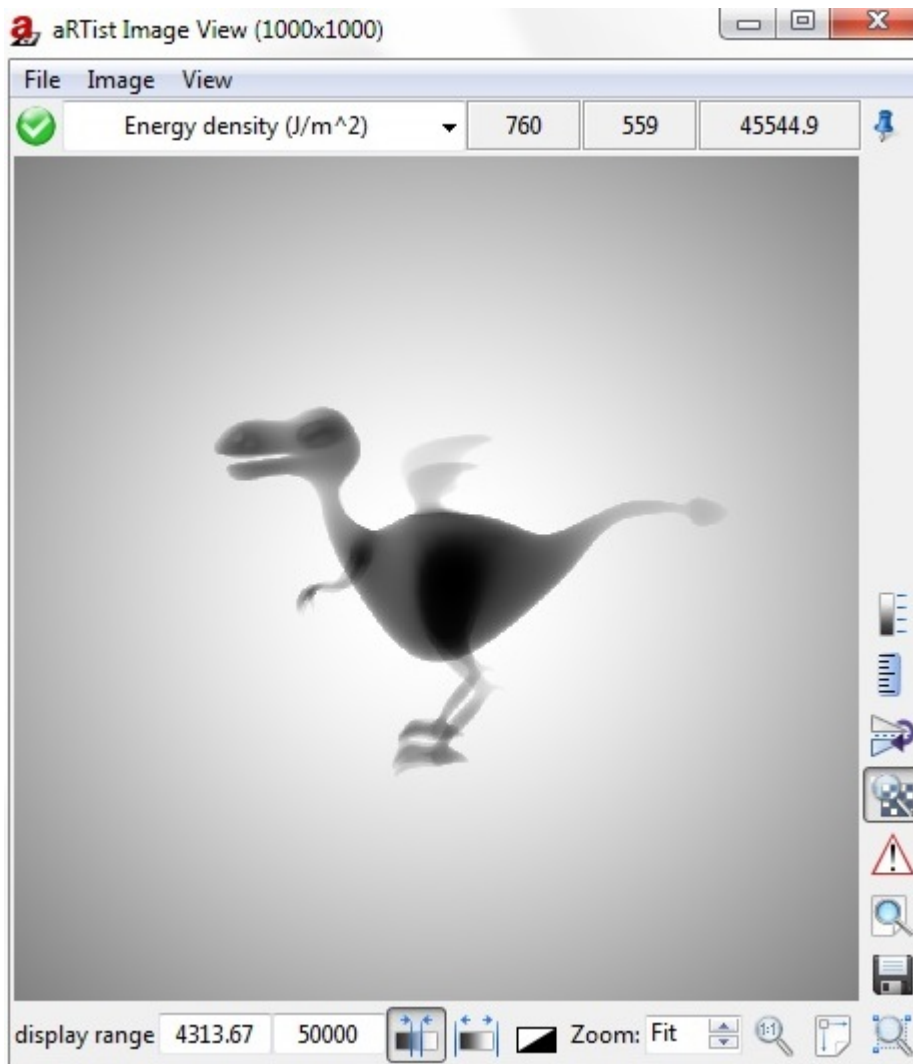
Fig. 57: ImageViewer

Tools Menu: ImageViewer

In the following every command of the *ImageViewer* submenu from the Tools Menu (Figure 58) will be explained.
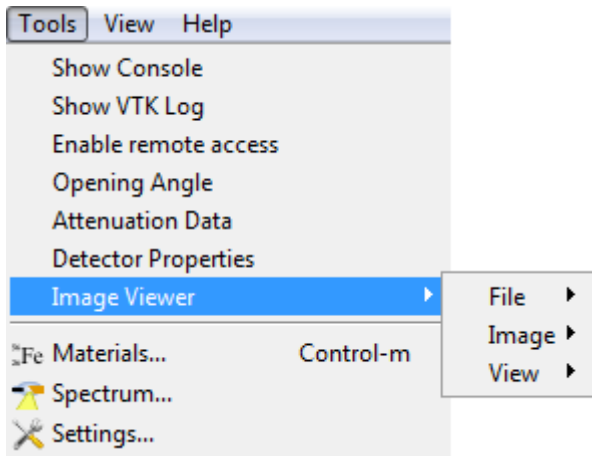
Fig. 58: Tools Menu, ImageViewer Menu

This submenu offers specific settings for *ImageViewer* and consists of different additional submenus:

- File
- Image
- View

File

It consists of three additional submenus: Under *File* you can find commands to *open* a new image file, *save* the current simulation (you may also use the *Save* button in *ImageViewer*) or *clear* the image list (Figure 59).
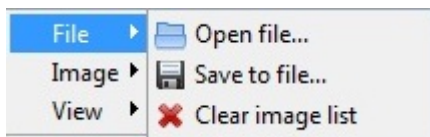
Fig. 59: ImageViewer, File Menu

Image

Under *Image* you can find different filters and options to mirror or rotate the current image (Figure 60). Please note that changes made with commands from this submenu are permanent (for example when saving the image). For non-permanent modifications use commands from the *View* submenu.
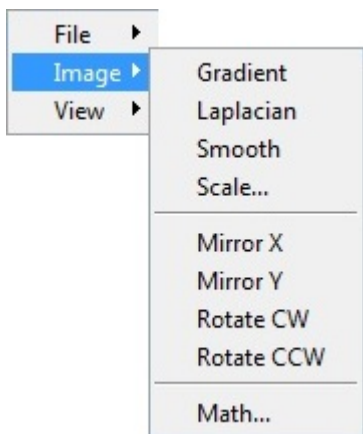
Fig. 60: ImageViewer, Image Menu

Filters are used to select between the filters: *Gradient, Laplacian* and *Smooth*.

Scale is used to fill in the output values for *X* and *Y* (Figure 61). With the button next to the input

fields you can predetermine a square image [ ] (closed symbol). If the symbol is open [ ] the pixels can be rectangular (by selecting different *X* and *Y* values). The *Mode* submenu offers specific settings. There you can choose between *linear, cubic* or *nearest neighbor* mode. Press *Cancel* for quitting these settings or *OK* for resampling the image.
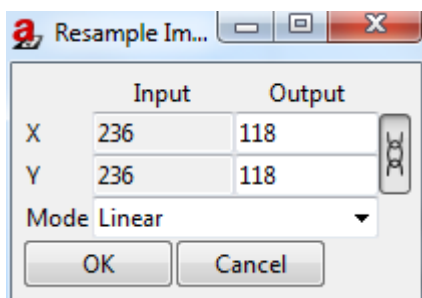
Fig. 61: ImageViewer, Image, Scale

Mirror X/Y reflects the current image on *X*-axis or *Y*-axis.

Rotate CW/CCW spins the current image in 90 degree steps clockwise or counter-clockwise.

Math opens the *ImageMath* window (Figure 62). It offers several arithmetic operations and point operations that can be applied the current image.
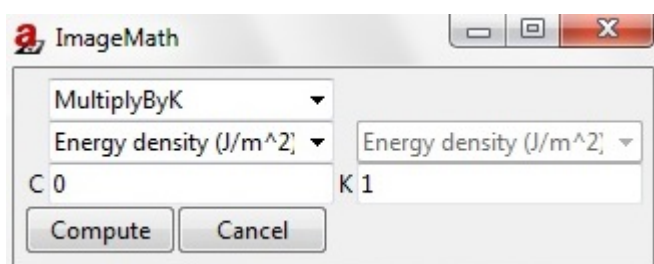
Fig. 62: ImageViewer, Image, Math

The first drop-down menu of the *ImageMath* offers the following arithmetic operations. In order for them to work the input field for *C* or *K* has to be filled in. Except for the first two functions, *MultiplyByK* and *AddConstant*, the actual number is not important. Press compute to run the operation and cancel to reverse it:

- *MultiplyByK* is used for pointwise multiplication of each pixel with *K*.

- *AddConstant* is used for pointwise addition of each pixel and *C*.

- *Exp* is point operator based on the exponential function and changes the dynamic range of the image.

- *Log* is point operator based on the log function and changes the dynamic range of the image.

- *Invert* inverts the current picture.

- *AbsoluteValue* takes the absolute value of each pixel.

- *Square* squares the value of each pixel.

- *SquareRoot* takes the square root of each pixel.

- *Sin* calculates the sine of each pixel.

- *Cos* calculates the cosine of each pixel.

- *ATAN* calculates the arctangent of each pixel.

The following operations calculate a new image out of two images therefore the third drop-down menu is enabled. *C* and *K* may be left empty.

- *Add* sums every corresponding pixel value of the two images.

- *Subtract* deducts every corresponding pixel value of the second image from the first image.

- *Multiply* multiplies every corresponding pixel value of the two images.

- *Divide* divides every pixel value of the first image by corresponding pixel value of the second image.

- *Min* shows minimum grey values of the two images.

- *Max* shows maximum grey values of the two images.

View

Under *View* you can find the same four mirror and rotate commands explained under *Image* and additional options (Figure 63). Please note that the changes will not be visible when you save the image or open it in an external viewer as opposed to the commands found in the *Image* submenu.
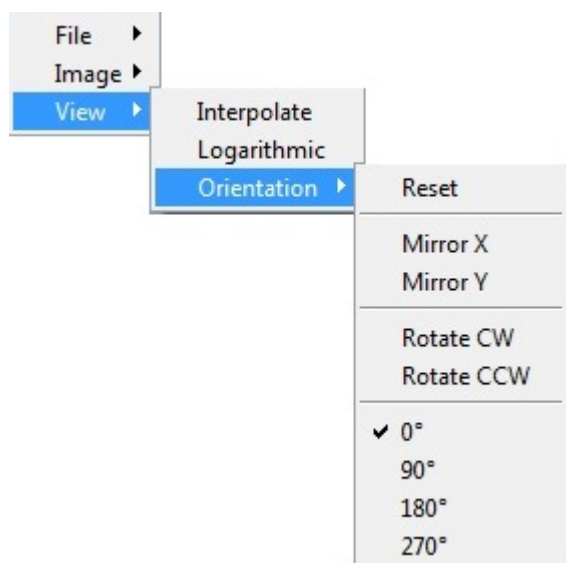
Fig. 63: ImageViewer, View, Orientation Menu

Interpolate activates/deactivates interpolation. *aRT*ist uses linear interpolation.

Logarithmic is point operator based on the log function and changes the dynamic range of the image.

Orientation is used for the following commands, which also access with this button in the

ImageViewer: :

- *Reset* removes all changes made in regards to orientation.
- *MirrorX/Y* reflects the current image on the *X*-axis or the *Y*-axis.
- *Rotate CW/CCW* spins the current image in 90 degree steps clockwise or counter-clockwise.
- *0°/90°/180°/270°* rotates the image with four different preset values.

McRay

McRay is a Monte Carlo raytracing implementation of electron and photon transport. This module forms a graphical interface for McRay, in order to configure and run simulations using it from within *aRT*ist. If you activate the inline help icon an additional tab (*Help*) will appear.
In this user guide, the McRay module topics *Expert, Manual, Cluster* and *Interactive* are not being explained.

A *Run/Stop* button and a progress bar are located at bottom of module window. The first four tabs expose the configuration in order of increasing complexity.

Basic

This page contains everything necessary to run a Monte Carlo simulation with the default settings (Figure 64). At the top there are two buttons, one to reset all settings to the default, and one to load the images normally resulting from a successful simulation run.
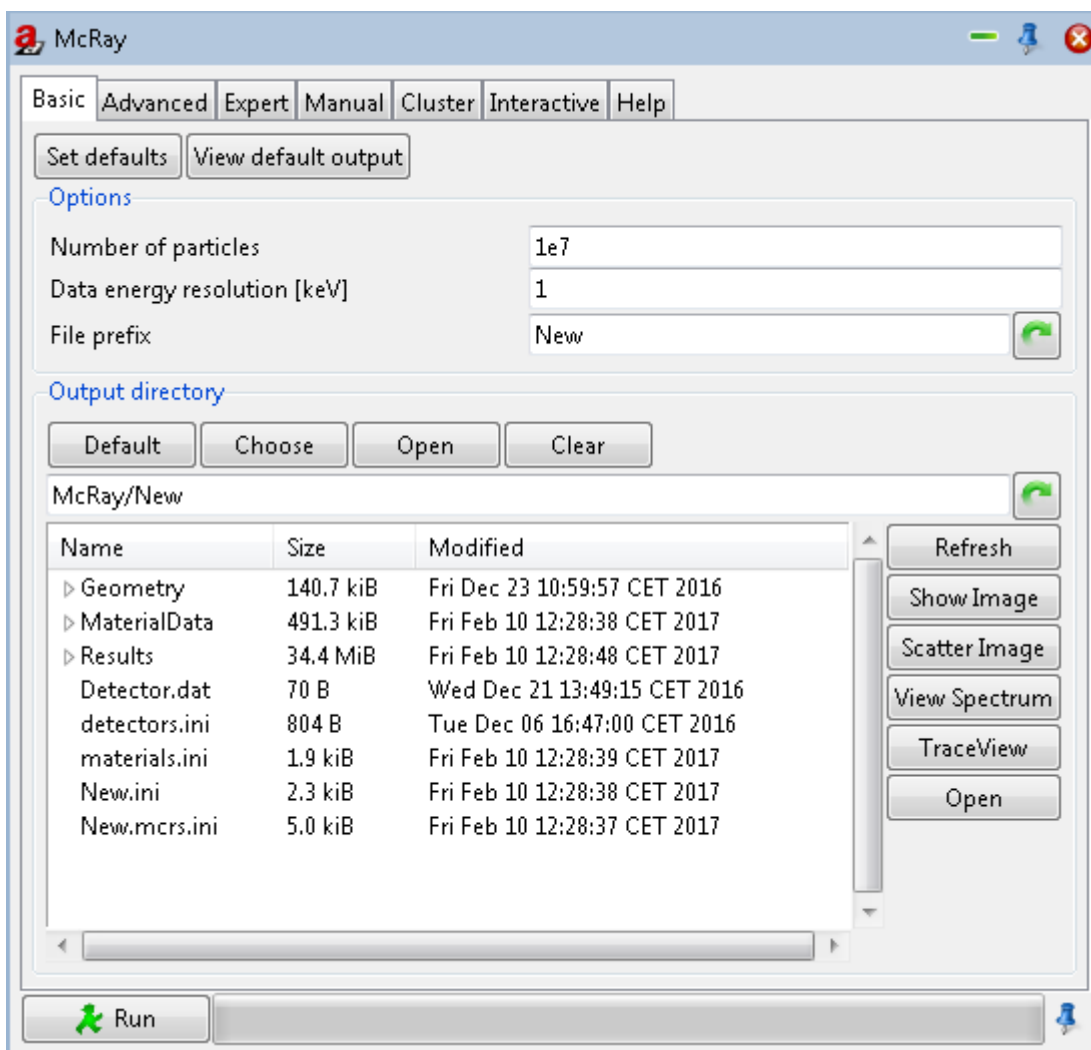
Fig. 64: McRay: Basic

Options

- *Number of particles* is used to specify the desired number of particles (photons / electrons).

- *Data energy resolution* is used to influence the energy resolution of these structures (by this parameter).

Internally most material data (attenuation-coefficients, differential scatter cross-sections, etc.) is kept in lookup structures for fast access. The default value of 1 keV does not normally need to be changed, but for low-energy calculations, especially including fluorescence, a smaller value of e.g. 0.1 keV may be advisable.

- *file prefix* is used to set a project name in this field. Default name is "*New*".

Output directory

Here you may change and inspect the output directory, which will hold the configuration files, data files, and all output files (images, spectra, etc.). The four buttons at the top allow you to change the directory via a graphical dialog, open it in an external file browser, or delete all contained files (but not subdirectories). It is possible to reset the settings to default. You may also change the directory by typing or pasting into the text field below the buttons.

The remaining space is filled by the file listing and associated buttons. A double click in the listing will open the currently selected files with their associated application, as will pressing the *Open* button. *Show Image*, *Scatter Image*, and *TraceView* will load the currently selected files in the *ImageViewer* module, as the scatter image for subsequent calculations, and in the *TraceView* module (if installed) respectively. The *TraceView* module is intended to visualize trajectories saved by McRay in the 3D scene view. *View Spectrum* will attempt to pass the selected file to the included spectrum plotter.

Advanced

Simulation Options

The options on the left hand side are mostly concerned with physical effects and geometry handling. The right hand side allows configuration of the output to produce (Figure 65).
If the command is selected it is necessary to change the number. While if the command is not selected the default setting will be set.
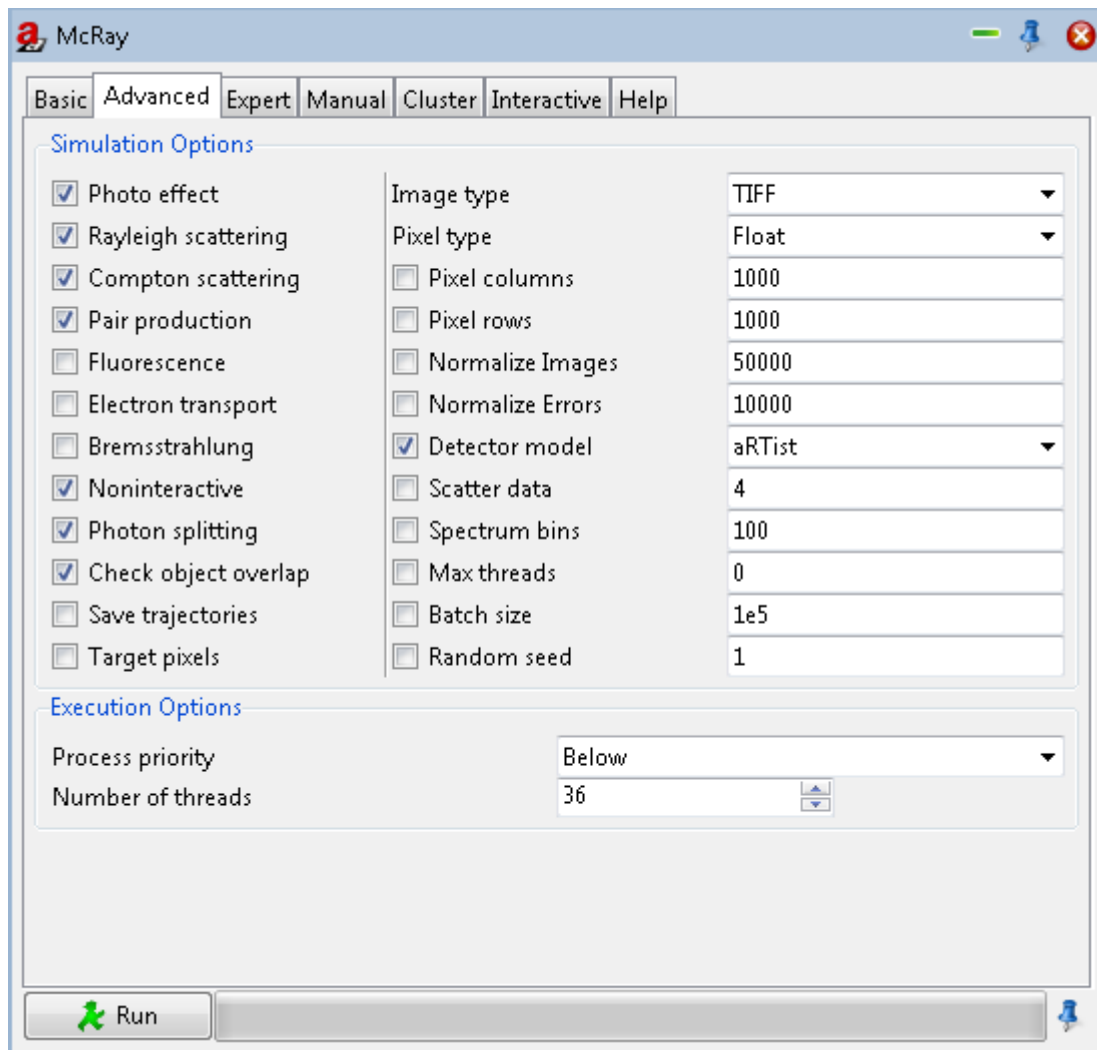
Fig. 65: McRay: Advanced

left hand side:

- *Photo effect* considers photoabsorption when elected. Should almost always be selected.

This photoemission is the emission of electrons or other free carriers when light is shone onto a material.

- *Rayleigh scattering* considers Rayleigh scattering when elected.

This electromagnetic interaction is the mostly elastic scattering of light or other electromagnetic radiation by particles (atoms or molecules) much smaller than the wavelength of the radiation. Rayleigh scattering is a parametric process.

- *Compton scattering* considers Compton scattering when elected.

Compton scattering occurs when the incident X-ray photon is deflected from its original path by an interaction with an electron. The electron is ejected from its orbital position and the X-ray photon loses energy because of the interaction but continues to travel through the material along an altered path. Energy and momentum are conserved in this process. The energy shift depends on the angle of scattering and not on the nature of the scattering medium. Since the scattered X-ray photon has less energy, it has a longer wavelength and less penetrating than the incident photon.

- *Pair production* considers pair production when elected.

Pair production often refers specifically to a photon creating an electron-positron pair near a nucleus but can more generally refer to any neutral boson creating a particle-antiparticle pair. In order for pair production to occur, the incoming energy of the interaction must be above a threshold in order to create the pair – at least the total rest mass energy of the two particles – and that the situation allows both energy and momentum to be conserved.

- *Fluorescence* considers flourescence when elected. Photons, and electrons if enabled, can excite the bound electrons of the material's atoms. During relaxation fluorescence photons, fluorescence electrons, and Auger electrons may be emitted.

Fluorescence is the emission of light by a substance that has absorbed light or other electromagnetic radiation. It is a form of luminescence and in most cases, the emitted light has a longer wavelength, and therefore lower energy, than the absorbed radiation.

- *Electron transport* considers electron transport when elected. When fluorescence or bremsstrahlung is enabled, secondary particles will be created.

The Monte Carlo method for electron transport is a semiclassical Monte Carlo (MC) approach of modeling semiconductor transport. Assuming the carrier motion consists of free flights interrupted by scattering mechanisms, a computer is utilized to simulate the trajectories of particles as they move across the device under the influence of an electric field using classical mechanics. The scattering events and the duration of particle flight is determined through the use of random numbers.

- *Bremsstahlung* (photons) are created during electron interactions when elected and requires electron transport.

The "braking radiation" or "deceleration radiation" is electromagnetic radiation produced by the deceleration of a charged particle when deflected by another charged particle, typically an electron by an atomic nucleus. The moving particle loses kinetic energy, which is converted into a photon, thus satisfying the law of conservation of energy. The term is also used to refer to the process of producing the radiation. Bremsstrahlung has a continuous spectrum, which becomes more intense and whose peak intensity shifts toward higher frequencies as the change of the energy of the decelerated particles increases.

- *Noninteractive* starts/stops calculation without further user interaction.

- *Photonsplitting* applies variance reduction scheme. Photonsplitting takes into account the possible interaction process when a photon is penetrated by a medium.

- *Check object overlap* notes the overlapping of parts when elected.

- *Save trajectories* writes trajectory segments to disk (slow).

- *Target pixels* aims photons toward each pixel (mostly for visualization purpose).

right hand side:

- *Image type* is used to choose between different image types: *TIFF, Isee, RAW, PGM* or *TXT*. The default setting is *TIFF*.

- *Pixel type* is used to choose between different pixel types: *UInt 16, Float, Double*. Usually the *Float* is used.

- *Pixel columns* enters the number of pixel in columns.

- *Pixel rows* enters the number of pixel in rows.

- *Normalize Errors* normalizes a relative error of 1 to this value (relative statistically error).

- *Detector model* is used to choose between different detector models: *aRTist, Photons, Leptons, Electrons, Positrons* or *All*. The default detector model is *aRTist*.

- *Scatter data* registers data for this many scatter events separately.

- *Spectrum bins* sets the number of spectrum bins. The maximum energy divide by the number of bins reveals the spectrum for each energy.

- *Max threads* sets the maximum number of threads.

- *Batch size* sets the batch size of trajectories.

- *Random seed* sets the random seed.

Execution Options

Process priority is used to adjust the process priority of the background process carrying out the actual simulation with this parameter. You can choose between *Idle, Below, Normal, Above* and *High*. As calculations can potentially take a long time, a reduced priority is generally advisable (default setting: *Below*).

Set the *Number of threads*.

Help

The "*Help*" tab is used to get more information to some topics.

Solid

Geometry Type

The Solid module allows you to create simple parts of different sizes to construct your assembly as desired (Figure 66). It offers different geometric shapes to choose from:

- *cuboid*
- *ellipsoid*
- *cylinder*
- *tube*
- *cone*
- *text*
- *wedge*
- *step wedge*

Dimensions

Select the part of your choice from the drop-down menu and set the *Dimensions* in *mm* and the *Phi/Theta steps* if applicable. You may predetermine equal values for *X, Y* and *Z* by checking the box at the bottom of the module. If you choose "*text*" the input field at the top of the module is enabled. When finished press the *Create* button.

The default material for new parts is Fe. A list with available materials can be accessed from

- the *Menu Bar* → *Geometry* → *Set Material* or
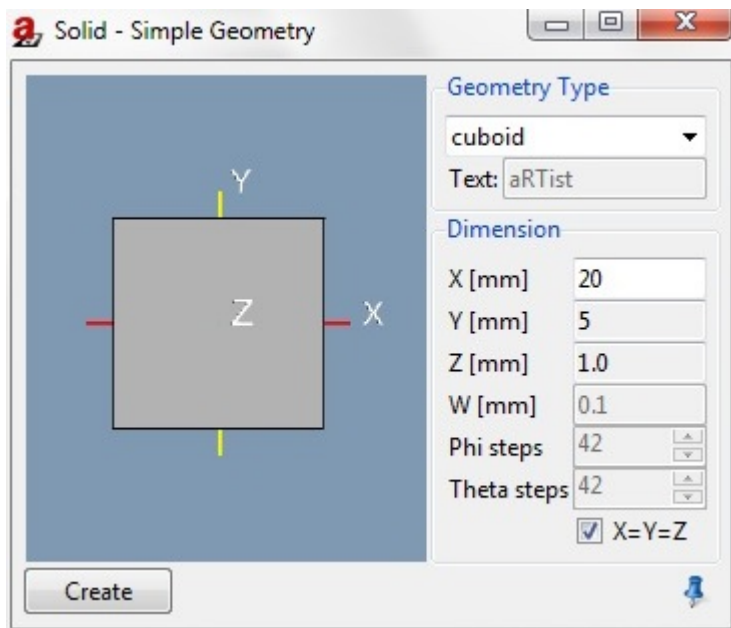- the *Assembly List* of the *Parameter Panel*

Fig. 66: Solid

Tomosynth

The TomoSynth module allows you to simulate tomosynthesis within *aRT*ist (Figure 67). It consists of two pages: the *Laminography* page offers settings for tube and detector position and output options. The *Advanced* page offers settings that are mainly directed towards experts.
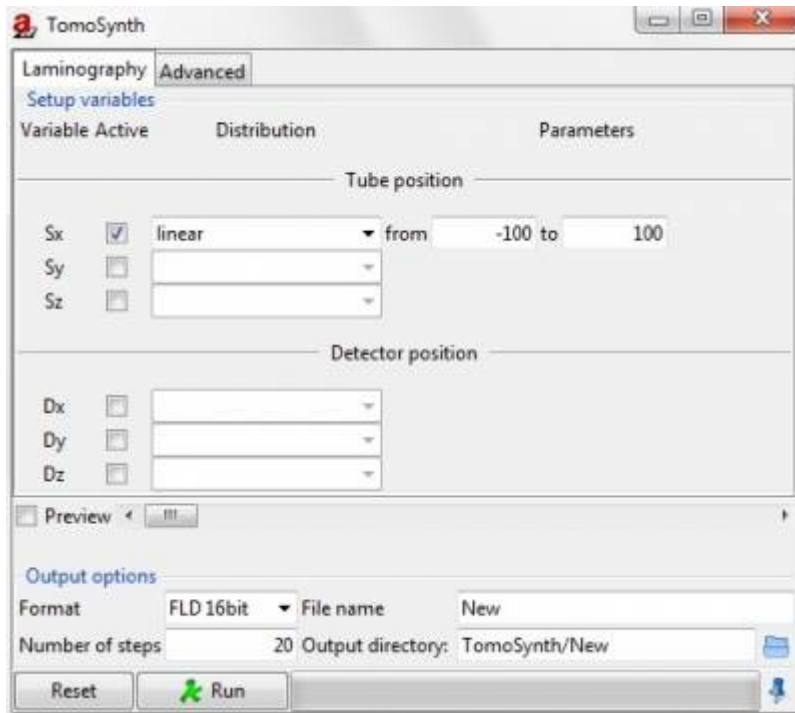
Laminography

Fig. 67: TomoSynth: Laminography

Setup variables

Tube (S) and detector (D) can be moved on the three axes *X, Y* and *Z*. Set the parameters in the following order:

Variable Active is used to select desired axis/axes via checkbox(es).

Distribution is used to choose a distribution out of the available options. A drop-down menu is activated when the corresponding variable is selected. Deterministic and random distributions are available. A deterministic distribution is a mathematical formula, which describes how to compute the value for a given image number step for a total of Nsteps images. Here, step varies from 0…Nsteps-1. A random distribution assigns a (pseudo-)random number with a given probability density to the associated variable. The pseudorandom numbers are generated by hashing sequence numbers with a fixed random seed and therefore are completely reproducible.

Please note that the sign of the radius (*circle, arc, disc*) defines the orientation of the rotation, where positive radius values indicate a right-handed rotation.

Deterministic distributions:

- *linear* is used to increase/decrease linear from *from* to *to*.

- *logarithmic* is used to increase/decrease logarithmic from *from*.

- *constant* sets a variable to a constant value. Most useful to define the axis of rotation for rotating objects.

- *circle_x* revolves in *X*-direction.

- *circle_y* revolves in *Y*-direction.

- *arc_x* is used to calculate arc in *X*-direction.

- *arc_y* is used to calculate arc in *Y*-direction.

- *disc_x* is used to disc in *X*-direction.

- *disc_y* is used to disc in *Y*-direction.

- *formula_rnd* is used to make a formula with a random variable.

- *formula* is used to enter an arbitrary mathematical formula to define a deterministic distribution. The expression is evaluated using the rules of *expr* of the Tcl scripting language. For a detailed reference, please consult the online documentation for Tcl *here*.

Random distribution:

- *uniform*

- *gaussian*

- *lognormal*

Parameters are used to fill in the values of activated parameters.

Preview ☑ Preview ◄ ▥ is used to see a preview in *ImageViewer* of the tube's and/or detector's movement for the current settings when moving the slider (if checkbox is activated). Beneath the settings for tube and detector, you can find a slide control next to the preview checkbox.

Output options

- choose a *Format* from the following options: *RAW, TIFF, TIFF 16bit* and *FLD 16bit*
- put in a *File name*
- put in the *Number of steps*
- choose *Output directory*

Alternatively, you may use the default settings.

When finished press the *Run* button to start the simulation or *Reset* if you wish to discard the changes you have made.

Advanced

The settings in this tab are mainly directed towards experts (Figure 68).

Fig. 68: TomoSynth: Advanced

Protocol
It is possible to change the settings for each part of the scene.

Variable Active selects desired axis/axes via checkbox(es).

Distribution is used to choose a distribution from the available options. A drop-down menu is activated when the corresponding variable is selected. Please note that the sign of the radius (*circle, arc, disc*) defines the orientation of the rotation, where positive radius values indicate a right handed rotation.
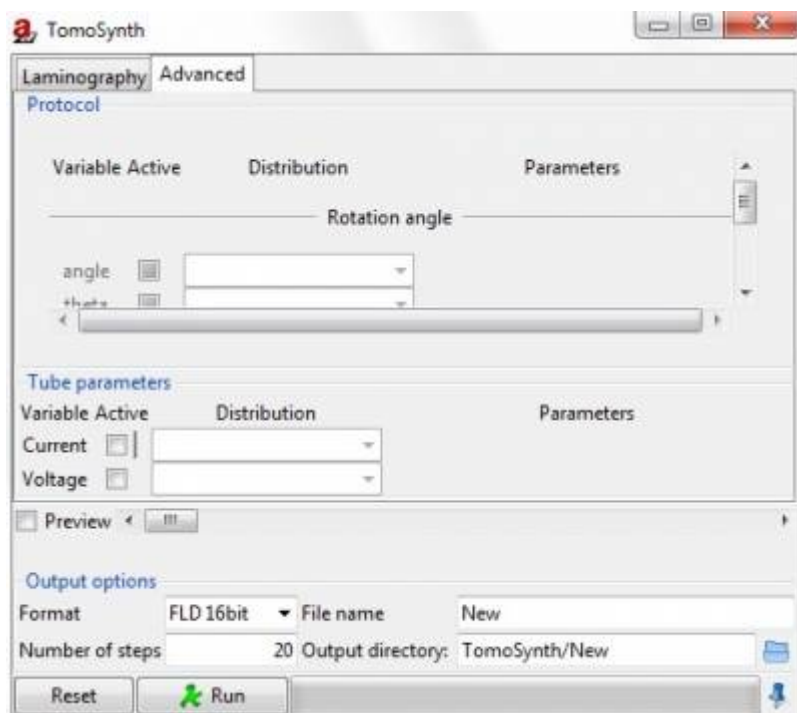
Parameters are used to fill in necessary parameters.

Rotation Angle sets the rotation *angle*, the polar angle *theta*, and the azimuthal angle *phi*.

Position Shift sets the position shift in the three axes: *Δx, Δy* and *Δz*.

Size Scale sets the size scale in the three axes: *Sx, Sy* and *Sz*.

Tube parameters

Current drop-down menu is activated when "*Current*" is selected. Choose a distribution from the available options (*linear, logarithmic, constant, circle_x, circle_y, arc_x, arc_y, uniform, gaussian, lognormal, disc_x, disc_y, formula_rnd* or *formula*).

Voltage drop-down menu is activated when "*Voltage*" is selected. Choose a distribution from the available options (*linear, logarithmic, constant, circle_x, circle_y, arc_x, arc_y, uniform, gaussian, lognormal, disc_x, disc_y, formula_rnd* or *formula*).

Preview is used to see a preview in *ImageViewer*. Beneath the settings for tube and detector you can find a slide control next to the preview checkbox. Activate the checkbox and move the slider to see a preview of the tube's and/or detector's movement for the current settings.

Output options

- choose a *Format* from the following options: *RAW, TIFF, TIFF 16bit* and *FLD 16 bit*
- put in a *File name*
- put in the *Number of steps*
- choose *Output directory*

Tools

In this chapter the following tools of the *Menu Bar* are being explained: *Attenuation Data*, *Detector Properties*, *Materials*, *Spectrum* and *Settings*.

Attenuation Data

The *AttenuationViewer* shows the attenuation-coefficient as a function of the radiation's energy (Figure 69). It consists of different graphs:

- *photo effect*
- *incoherent scattering*
- *coherent scattering*
- *pair production*
- *total*

: The displayed data can be saved in tabulated, or graphical form (see Icon).
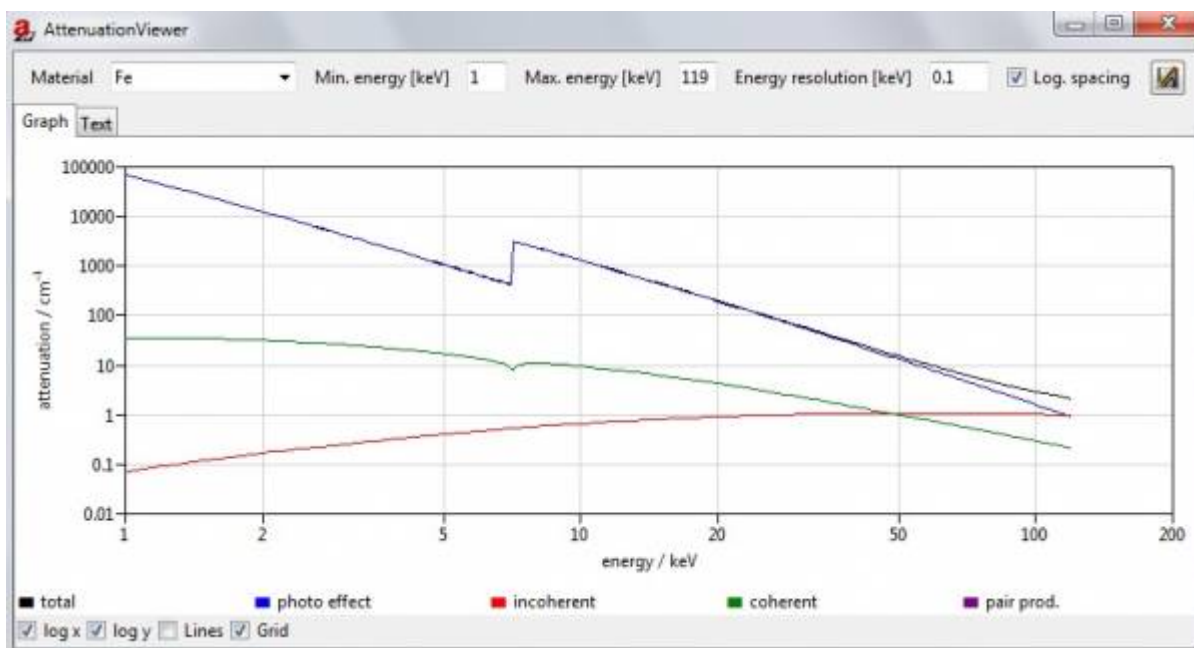

Fig. 69: AttenuationViewer

Detector Properties

The *DetectorViewer* contains plots of the properties of the current detector (Figure 70):

- *Characteristic* is used for grey values, film densities or air dose *mGy* as a function of energy density *J/m²*

- *Noise* is used for SNR as a function of energy density

- *Sensitivity* deposits energy as a function of energy

The type of graph(s) available is dependent on the detector.

On the left-hand side of *DetectorViewer* details of the current detector are displayed:

- *Name*
- *Input unit*
- *Output unit*
- *Pixel count*
- *Pixel size [mm]*
- *Resolution [mm]*
- *Long range unsharpness [mm]*
- *Long range contribution [%]*
- *Minimum output*
- *Maximum output*
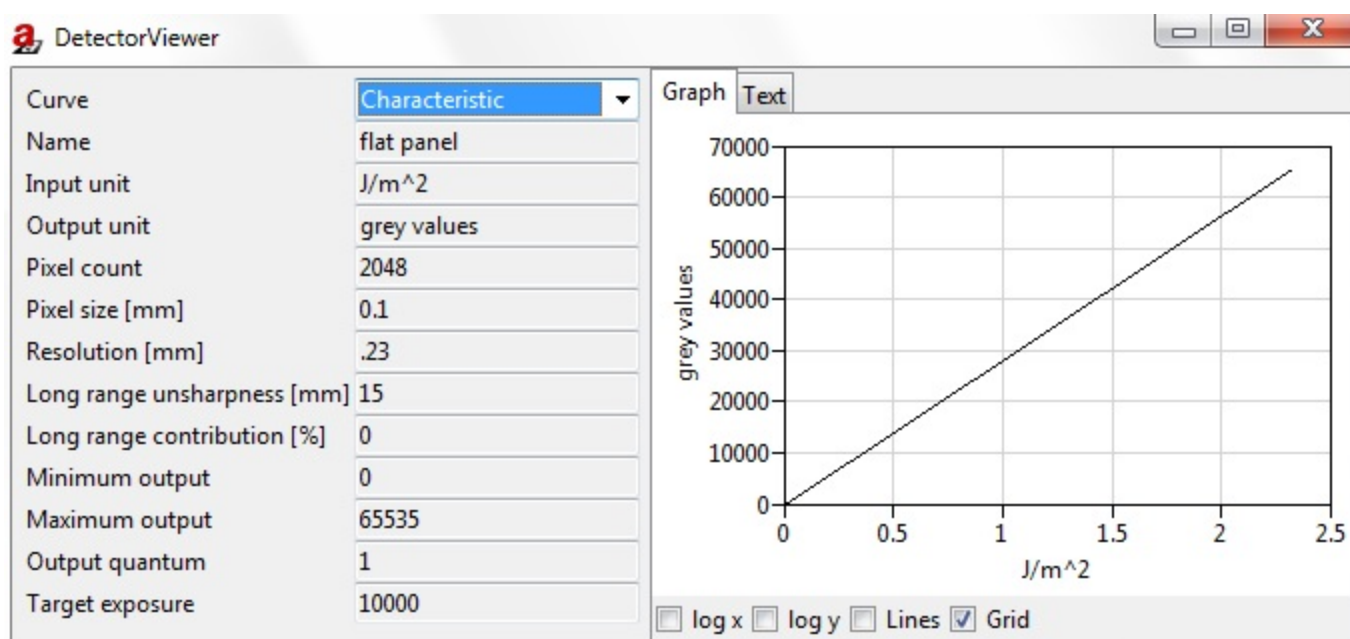- *Output quantum*
- *Target exposure*

Fig. 70: DetectorViewer
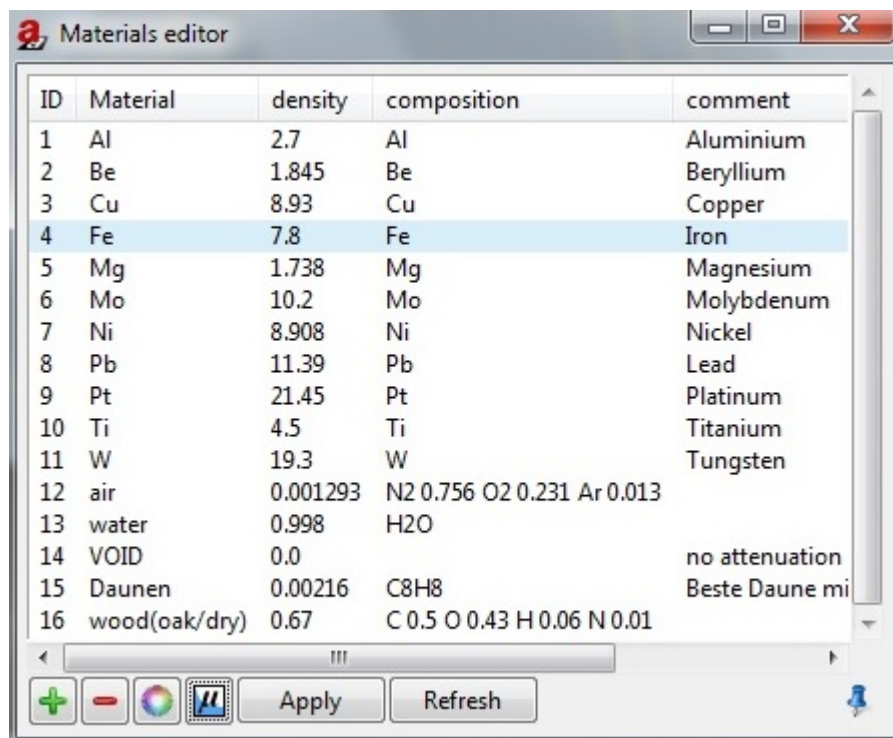
Materials

The *Materials editor* lists all materials that are available within *aRTist* including density and composition (Figure 71). You may dock the editor in the docking area.
It provides the following possibilities to edit materials:

- *Edit* selected material. Click on an entry you wish to edit, for example density or composition. Press enter when finished.
- *Add* is used to append a new material definition.
- *Delete* is used to erase currently selected material.
- *Edit display* is used to revise material display parameters for currently selected material.
- *Attenuation* plots attenuation data for currently selected material.
- *Apply* is used to update material database with table contents.

- *Refresh* is used to reload table contents from material database.

<u>Please note:</u> Densities are given in g/cm³. The composition must be of the form: *H2O 0.96 NaCl 0.04*. The fractional numbers are mass ratios.

Fig. 71: Materials editor

Spectrum

The *Spectrum calculator* offers specific settings for the source (Figure 73). It can also be opened from the *Parameter Panel* → *Source*: . You may dock the Spectrum calculator in the docking area.

Settings

Set

- *Tube* (choose between *Mono* or *General*)
- *voltage [kV]*
- *resolution [keV]*
- *filter material* and *thickness [mm]*, (pick *VOID* as a material if you do not want to use a filter)

Load tube definitions, File type: *.dict*.

| *Save* tube definition (hold or right-click for options) or *delete* tube definition from database.

Tube Settings

If you choose the general tube type an advanced setting will appear. There you can set:

- *Name* | ✚ (add tube definition to database)
- *Maximum voltage [kV]* (optional)
- *Maximum current [mA]* (optional)
- *Maximum power [W]* (optional)
- *Transmission* (enable/disable transmission)
- *Target material* (choose between *Al, Be, Cu, Fe, Mg, Mo, Ni, Pb, Pt, Ti, W, air, water* or *VOID*)
- *Target thickness [mm]*
- *Target angle [deg]*
- *Angle of electron incidence [deg]*
- *Window material* (choose between *Al, Be, Cu, Fe, Mg, Mo, Ni, Pb, Pt, Ti, W, air, water* or *VOID*)
- *Window thickness [mm]*

Under Tube Settings you can set maximum values for the tube and adjust the target material, thickness and angle, the angle of electron incidence and the window material and thickness. Mark the *Show principle* checkbox in the bottom row and a schematic drawing appears which shows the principle of the simulated tube. Press the *Compute* button in the lower part of the window to apply changes when finished.

If the tube voltage is set over 128 keV, a window will appear with the note: *"This spectrum contains (146) entries, the currently recommended value is 128. Reduce?"* You can choose between *Yes* or *No*. → By selecting *Yes*, you can see a reduced spectrum at the [Source](#) → *Source spectrum*. Keep holding the left-click on the mouse or right-click the symbol 📈. There you can open "*Show reduced spectrum*", "*Show reduced spectrum*" or "*Show prewiew spectrum*" to see the different spectrums (Figure [72](#)).
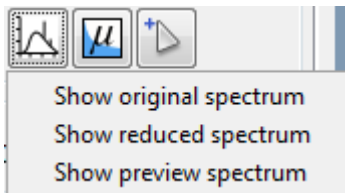
Fig. 72: Spectrum selection

→ By selecting *No*, the spectrum is not reduced and will stay as the original spectrum.
If needed, it is possible to change the number of reduced spectrums bin at [Tools](#) → [Settings](#) → *Advanved*.
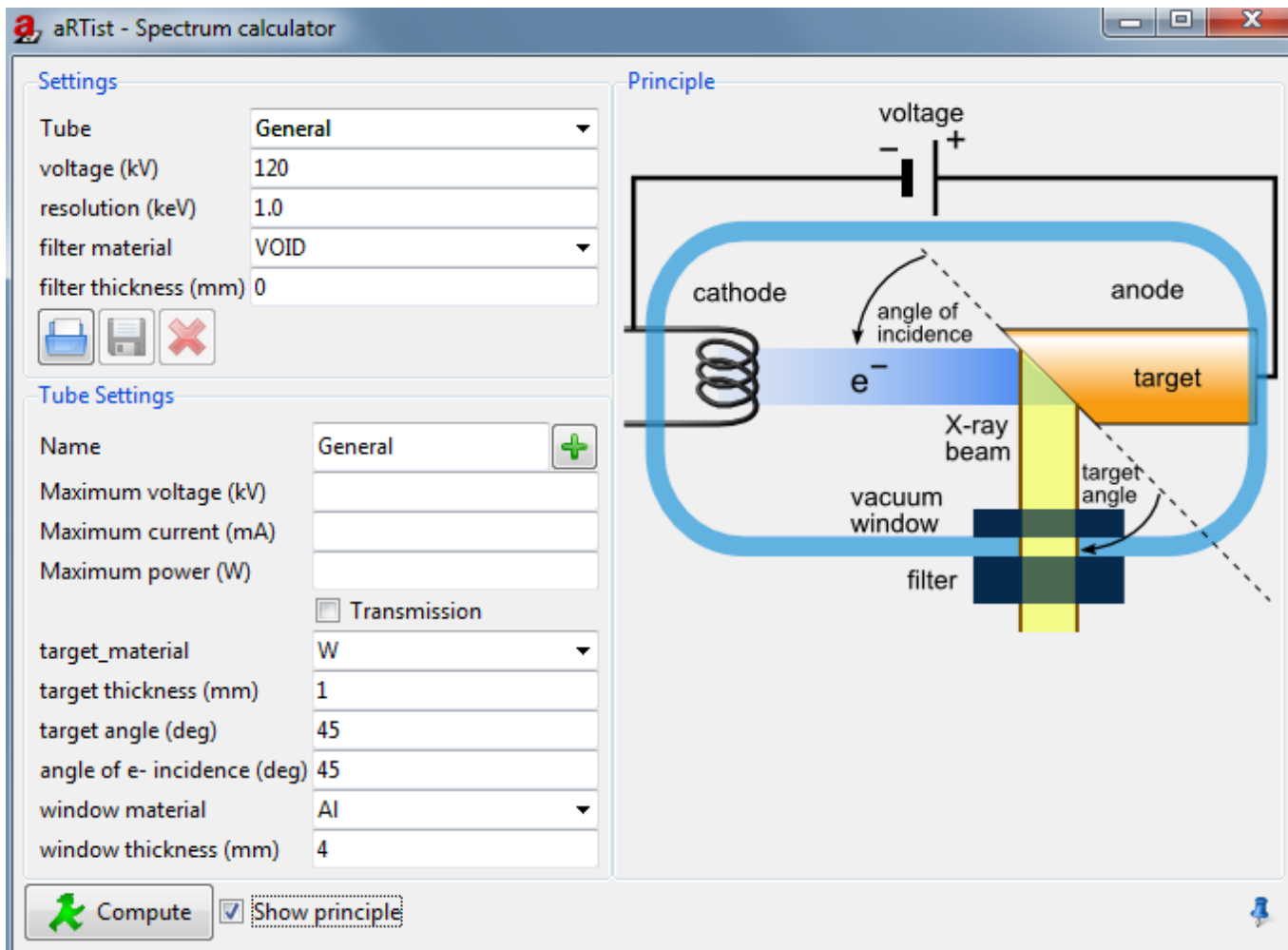
Fig. 73: Spectrum calculator

Settings

Settings

The *Settings* dialog box offers several options to adjust the user interface (Figure).
Set

- *Number of last open files*
- *Default directory*
- *License file*
- *External viewer*
- *Background color*
- *GUI style* *: Choose between *small, normal* or *large*.
- *Dock position*: Choose between *right, left, bottom* or *top*.
- *Display invicible parts*: Choose between *gray, alpha, keep* or *off*.
- *Stereo mode*: Choose between *Anaglyph, CristalEyes, Interlaced* or *Checkerboard*.
- *Anaglyph color mask*: Choose between *red/cyan, red/green, red/blue, yellow/blue* or *green/magenta*.
- *Decimate objects larger than [kB]*

enable/disable

- *Use decimated geometry for preview*
- *Show scene view info in status bar*
- *Show radiograph in 3D view*
- *Prefer GPU computing*
- *Single precision results*

Press *OK* for saving or *Cancel* for quitting these settings. The changes are effective after program restart.
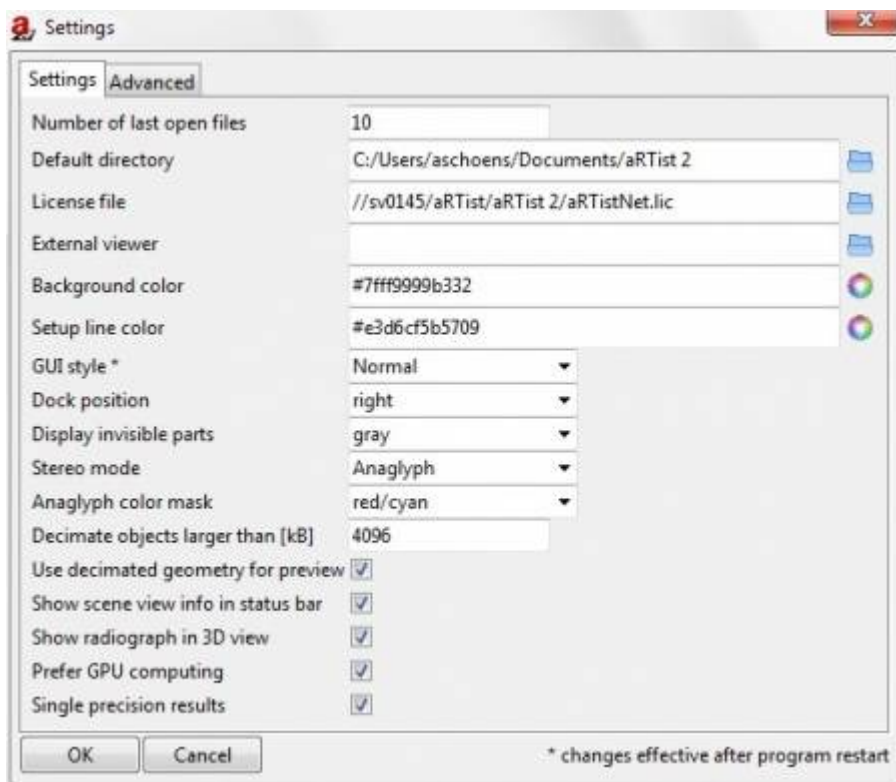
Fig. 74: Settings

Advanced
Advanced settings should normally not be changed (Figure 75, 76 and 77).
*aRTi*st: Set

- *Log level*: Choose between *Info, Error, Warn, Verbose, Debug, Trace* or *Off*. The default setting is *Info*.
- *Compatibility working directory*

Geometry: Set

- *Number of triangles in decimated geometry*

Enable/disable:

- *Double precision coordinates*
- *Triangle strips*

GUI: Enable/disable

- *Pick disabled parts*
- *Show orientation marker*
- *Animate view transitions*
- *Use bounding box level of detail*

Set:

- *Anti-aliasing frames*
- *Multisampling*
- *Max depth peels*
- *Maximum render time [s]*

<u>Engine</u>: Set

- *Number of reduced spectrum bins*
- *Number of preview spectrum bins*

Enable/disable

- *Check for object overlap*
- *Use AVX2*
- *Use FMA*
- *Use AVX*
- *Use SSE2*
- *Use generic vector extensions*
- *Single precision intermediated*
- *Single call GPU calculations*
- *Single pass GPU calculations*
- *Half percision GPU calculations*

<u>Remote Control</u>: Enable/disable

- *Accept connections*

Set

- *Address to listen on*
- *Port to listen on*
- *Maximum increment when port in use*

<u>Scatter</u>: Set

- *Smoothing anisotropy*
- *Smoothing strength*

Press *OK* for saving or *Cancel* for quitting these settings. It is possible to *Reset* the settings on default. The changes are effective after program restart.

Settings

Settings | **Advanced**

Reset

Attention: These settings should not normally be changed!

aRTist

Log level   Info

Compatibility working directory

Geometry

Number of triangles in decimated geometry   100000
Double precision coordinates   ☑
Triangle strips   ☑

GUI

Pick disabled parts   ☐
Show orientation marker   ☐
Animate view transitions   ☑
Use bounding box level of detail   ☐

OK   Cancel   * changes effective after program restart

Fig. 75: Settings, Advanced I

Settings

Settings | **Advanced**

Reset

Attention: These settings should not normally be changed!

Use bounding box level of detail   ☐
Anti-aliasing   ☑
Depth peeling   ☐
Anti-aliasing frames   8
Multisampling   4
Max depth peels   32
Maximum render time [s]   10

Engine

Number of reduced spectrum bins   128
Number of preview spectrum bins   8
Check for object overlap   ☑
Use AVX2   ☑
Use FMA   ☑
Use AVX   ☑
Use SSE2   ☑

OK   Cancel   * changes effective after program restart
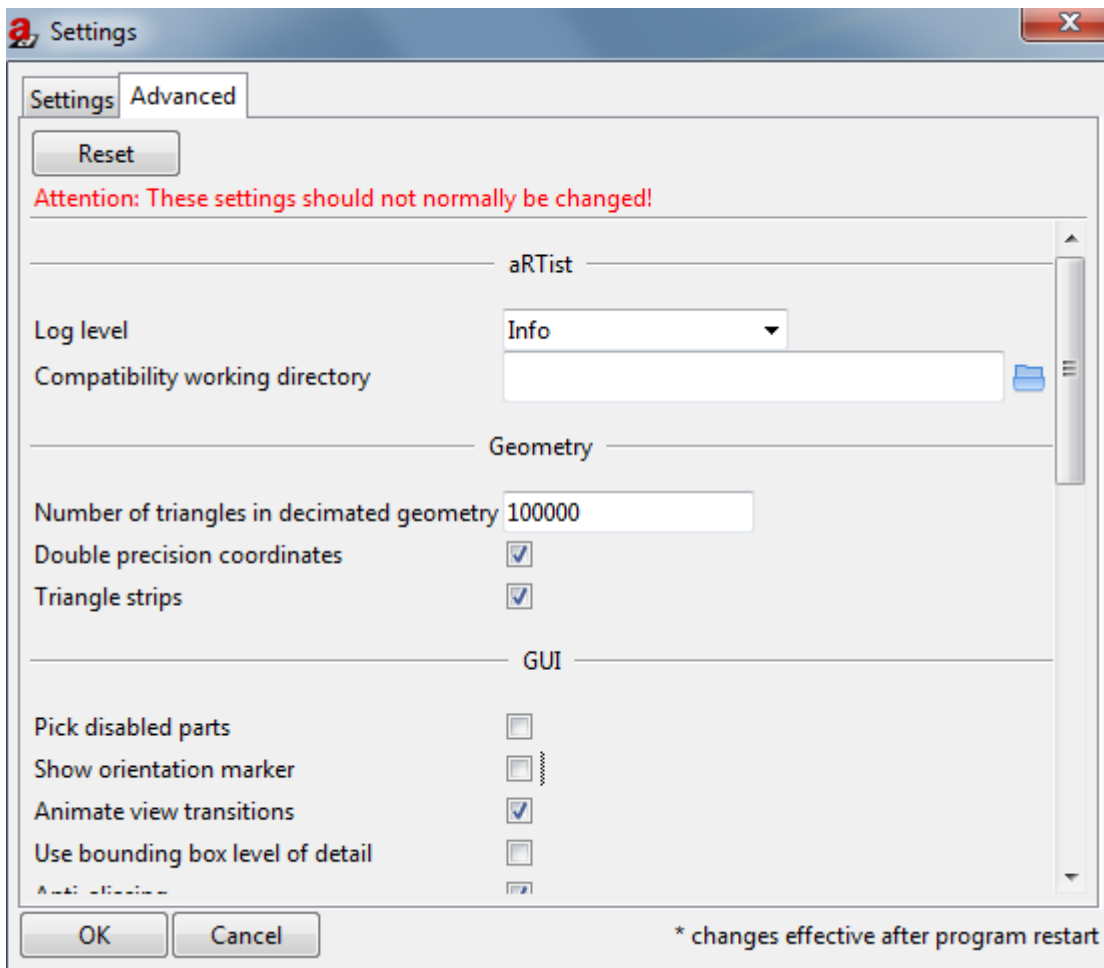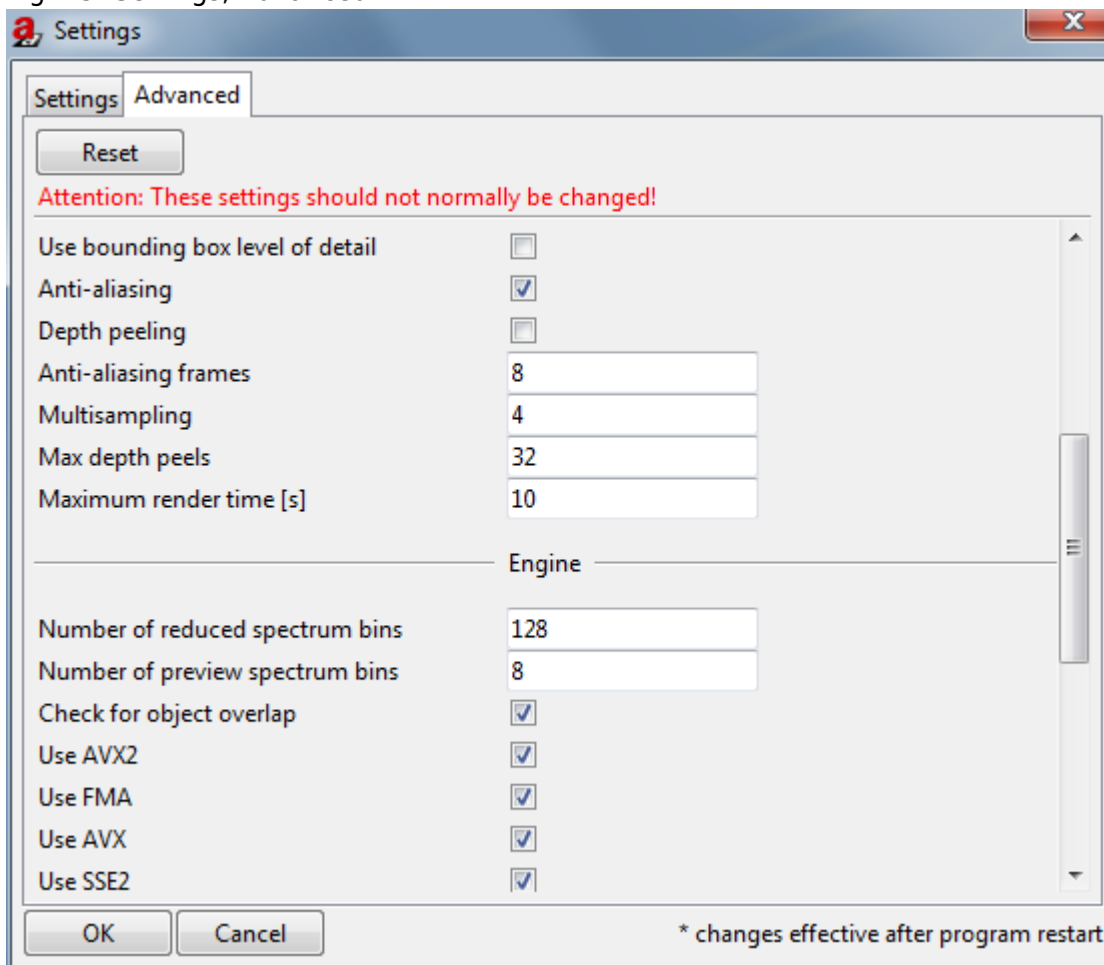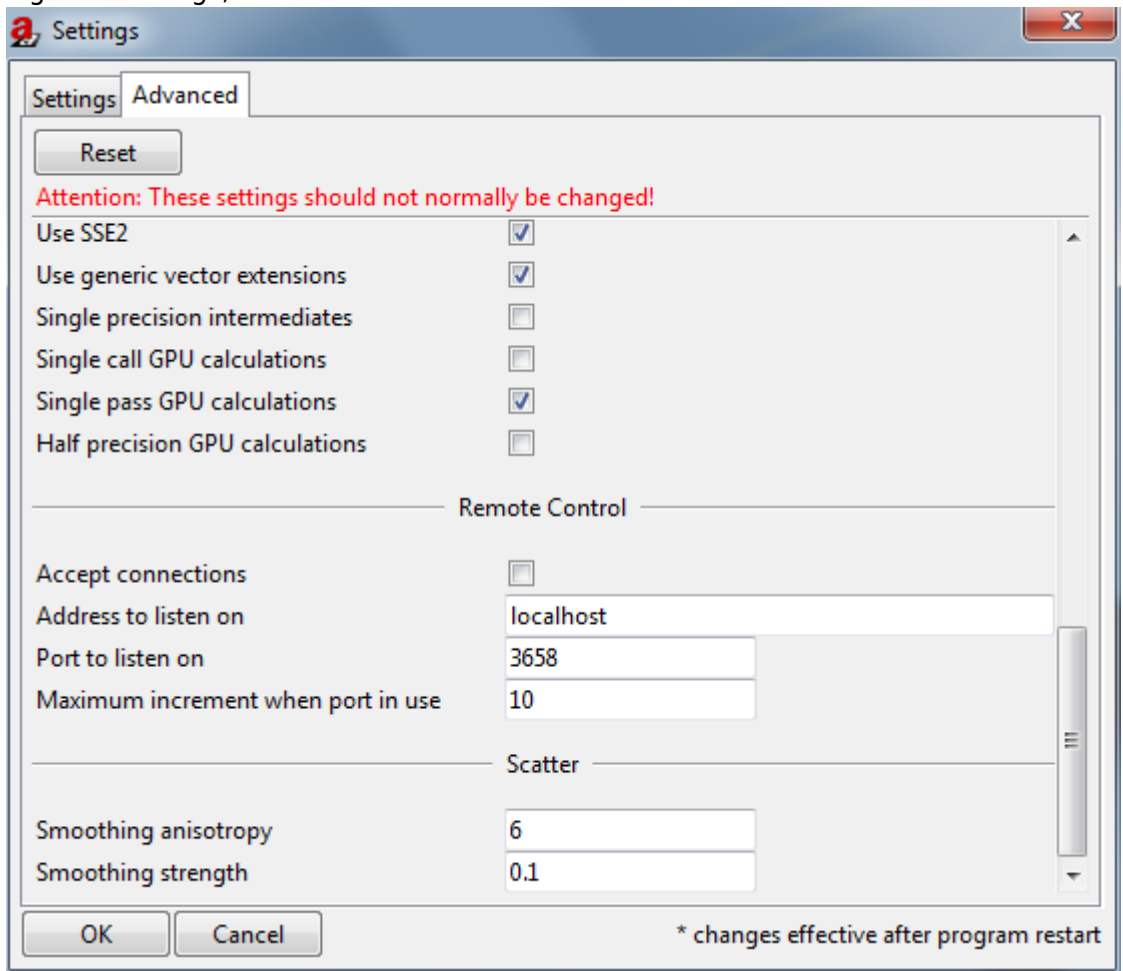
Fig. 76: Settings, Advanced II

Fig. 77: Settings, Advanced III

Geometry Manipulation

In the following you will find a more in-depth explanation for "*Geometry Manipulations*" (*Union, Intersection, Difference, Arrange* and *Pick Destination*) from the [Menu Bar](#) → [Geometry](#) drop-down menu. With the contained commands, which are the red edged ones in Figure [78](#), individual parts in the assembly can be created and arranged.
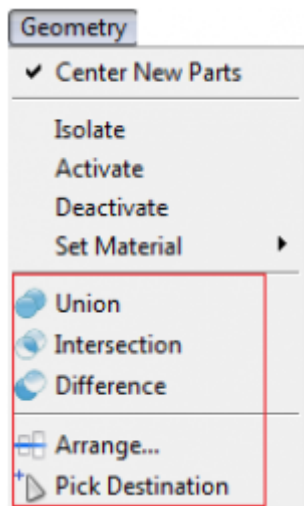

Fig. 78: Geometry Manipulation, red edged

Union/Intersection/Difference: These commands can be used to create more complicated shapes than simple geometric parts like cuboids or cylinders. First, generate the parts you need with the help of the [Solid](#) module. This module can also be opened with its tool bar icon . Translate, scale or rotate the parts as needed so that they overlap. Then select parts by using the Ctrl key on your keyboard and click on them with the left mouse. The first selected part can be recognized by its yellow corners. All subsequent parts will be marked with white corners. If the parts consist of different materials, the new created part will take over the same material as the first selected part.
The effect of each of the three commands is shown in the following pictures:

Union: This command joins two or more parts as depicted in Figure [79](#). The order of selection is not important.
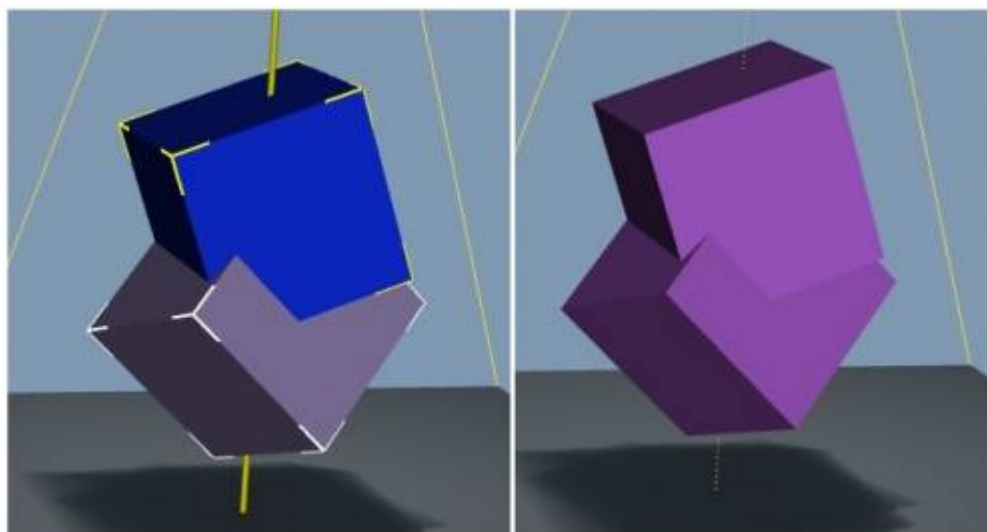

Fig. 79: Union (right) of two cubic parts (left)

Intersection: This command generates a part from the overlapping areas. Everything else is removed as shown in Figure 80. Thereby the order of selection is not important.
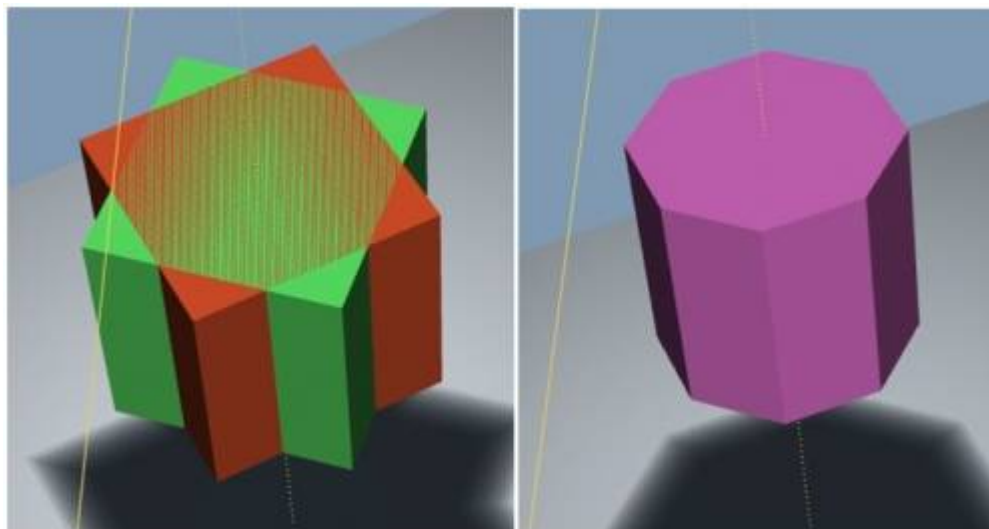

Fig. 80: Intersection (right) of two cubic parts (left)

Difference: This command generates a new part from the difference of at least two other parts. In this case, the order of selection will be important for the result as shown in Figure 81 and Figure 82. The first selected part (yellow corner) is thereby the minuend and all other selected parts (white corners) are the subtrahends.
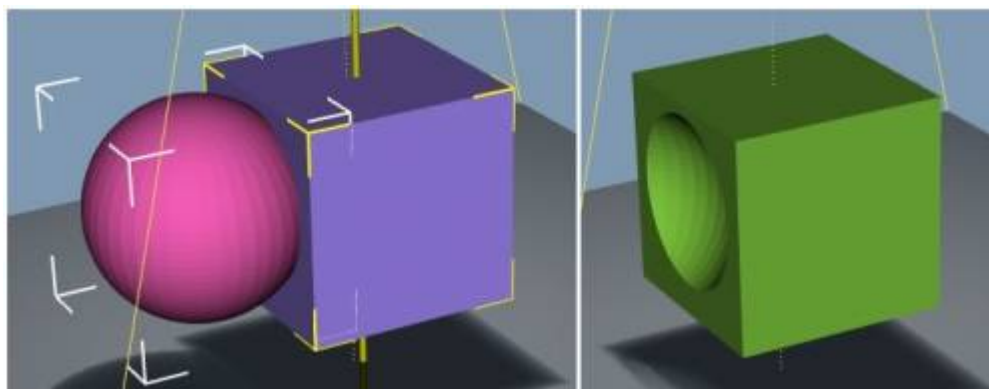
Fig. 81: Difference (right) of cuboid minus ellipse (left)
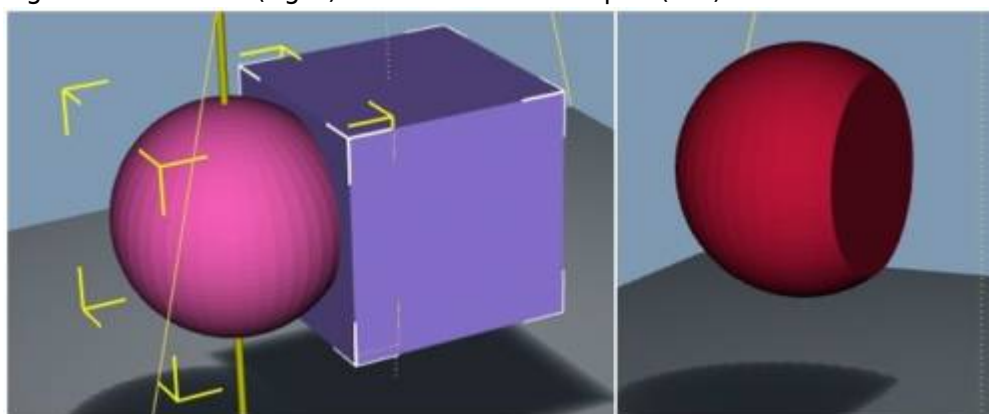

Fig. 82: Difference (right) of ellipse minus cuboid (left)

Arrange: This option allows you to arrange parts of your assembly. Select the parts you want to arrange (use the Ctrl key on your keyboard) and open the *Arrange* dialog box with the tool bar icon or with the *Arrange* command in the *Geometry* menu. There are four options: *none*, *-*, *center*, *+* available

for each of the three axes (*X, Y, Z*). Choose one or more of these options and press either the *Apply* or the *OK* button.

- *Apply* lets you see the changes you have made and undo them if needed.
- *OK* applies the changes and closes the dialog box.
- *Cancel* closes the dialog box without changing the assembly.

Please note: The parts will be aligned with the first selected part which can be recognized by its yellow corners.

The following pictures show the effect of the *Arrange* function on the three parts shown in Figure 83.
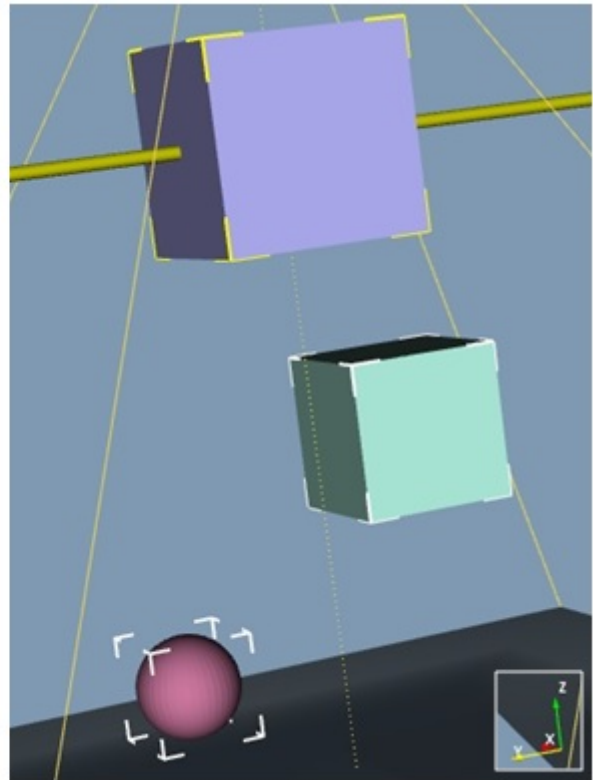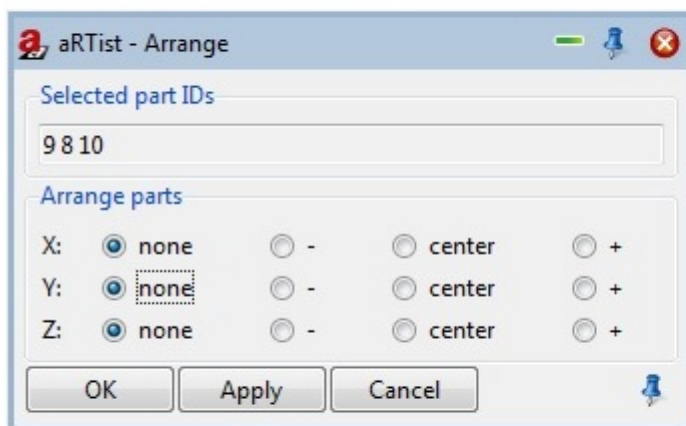
Fig. 83: Arrange

Y-axis

Figure 84 shows the three parts arranged on the Y-axis as an example. **-** and **+** align them with the edges of the light violet cuboid. **Center** centers the parts on the Y-axis.
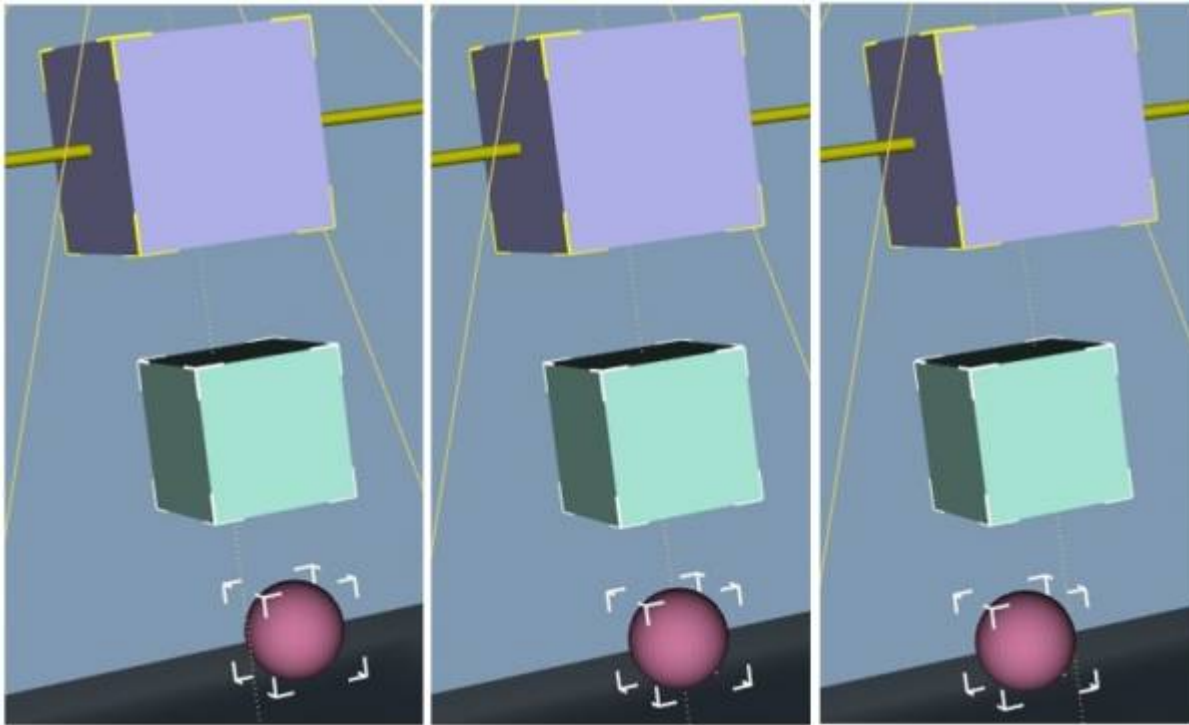
Fig. 84: FLTR: -, center, +

Pick Destination: This command moves parts from one point in the scene view to another point. Mark one or more parts from your assembly and select the command. A grey cross will appear while hovering over the scene view with the mouse cursor. Click on desired destination to relocate selected part(s).

Change Log

- **aRTist 2.12.6**
    - CHG: McRay/TraceView: use binary files to store traces by default
    - CHG: Spectrum calculator: improved tube handling
    - CHG: Spectrum calculator: fix read-in from project file of possible second and third filters
    - CHG: minor fixes

- **aRTist 2.12.5**
    - CHG: Table2Lists: change separators to ", ;\t"
    - CHG: Materials: change comment of default materials to indicate "preset of aRTist"
    - CHG: CtScan 2.7: rename scatter modes to "as configured" and "force McRay"
   - CHG: Library: Dragon-CT project with flat field correction enabled

- **aRTist 2.12.4**
    - FIX: BAMCT writer: fix nw180 in header
    - CHG: Detector MTF improved (experimental)
    - CHG: Module manager: add delete button for user installed modules

- **aRTist 2.12.3**
    - New: Online guide at help menu
    - New: Detector MTF (experimental)
    - New: CtScan postprocessing (experimental)
    - CHG: Spectrum calculator: two additional filters
    - CHG: minor fixes

- **aRTist 2.12.1**
    - Fix: Licenser: checking if network folder is writable
    - CHG: McRay: save scene/spectrum/geometry inside the task configuration

- **aRTist 2.12**
    - FIX: Detector unsharpness matches MTF-10 basic spatial resolution (Duplex Wire iSRb, ASTM E 2002).
    - CHG: Licenser: grand a goodwill period, if license file is temporary unavailable
    - CHG: minor GUI fixes
    - switch version control from Svn to Git

- **aRTist 2.10.4**
    - NEW: Settings/Advanct option: default material
    - CHG: DetectorCalc: Save settings in detector file.
    - CHG: Spectrum picker: make incident spectra optional

- **aRTist 2.10.2/3**
    - CHG: Improoved license dialog.
    - CHG: Licenser: Lite mode, if out of network licenses

- **aRTist 2.10.1**
    - NEW: TomoSynth: new distribution type "list"
    - CHG: parts of material "None" will be ignored by the tracer (scene visualization only)
    - CHG: revised dialog for license information and activation
    - CHG: indroducing version numbers for modules
    - CHG: minor fixes

- **aRTist 2.10**
  - CHG: McRay updated
  - ...
  - CHG: updated copy protection, new license files required

- **aRTist 2.8.3**
  - NEW: module DetectorCalc
  - CHG: minor fixes

- **aRTist 2.8.2**
  - CHG: DigRad and DetectorCalc, extend noise curve
  - CHG: McRay, use full spectrum instead of reduced
  - CHG: McRay, fix/improve opening angle calculation
  - CHG: Scattering: improve reduced detector resolution for McRay
  - CHG: minor fixes and GUI improvements

- **aRTist 2.8.1**
  - CHG: BAM logo
  - CHG: minor GUI improvements

- **aRTist 2.8.0**
  - CHG: McRay, improve smoothing of scatter image
  - CHG: McRay, minor model improvements
  - NEW: Solid, add creation of (step) wedges
  - CHG: improve PDF export of graphs
  - NEW: TomoSynth/SimuPOD, allow (user) control over number of random samples
  - NEW: Image, allow reading 3D FLDs
  - CHG: FileIO, attempt material selection when loading known IQI
  - NEW: allow loading TCL scripts (sourced in namespace External)
  - CHG: update Tcl/Tk to 8.6.2
  - CHG: make orientation marker visibility configurable
  - CHG: improve preview geometries
  - CHG: add multisampling/anti-aliasing options to settings dialog (advanced tab)
  - CHG: minor GUI improvements

- **aRTist 2.6.3**
  - CHG: XSource, fix/improve creation of focal spot profiles
  - CHG: CtScan, enable GPU reconstruction by default
  - BUG: Solid, clean generated polygon data
  - BUG: minor bug fixes

- **aRTist 2.6.2**
  - Special Version for MINTec

- **aRTist 2.6.1**
  - Special Version for Saclay Fall School
  - BUG: program crash

- **aRTist 2.6.0**
  - date: 2014-06-19
  - CHG: restructure menu bar and other GUI revisions
  - CHG: ray-cast engine updated
  - NEW: indicate modifications in title bar
  - NEW: experimental OS X version

- CHG: XRayTools updated
- CHG: McRay updated
- CHG: PODTool updated (optional)
- CHG: show unreduced spectra
- CHG: update ISee!-demo to v1.11.1
- BUG: minor bug fixes

- **aRTist 2.4.2**
  - BUG: fix normalization of source grid with focal spot image

- **aRTist 2.4.1**
  - CHG: new license files

- **aRTist 2.4.0**
  - date: 2013-07-01
  - NEW: include 64bit version
  - NEW: crash recovery
  - BUG: fix a memory leak in CT scan with TIFF output
  - NEW: use AVX2/FMA/AVX processor extension if available
  - NEW: better handling of unsaved changes
  - CHG: div. improvements in Monte-Carlo scatter module
  - CHG: improve/extend POD calculations

- **aRTist 2.2.0**
  - date: 2012-09-06
  - New: McRay scatter mode
  - CHG: CtScan with new Feldkamp implementation
  - NEW: CtScan with volume viewer
  - CHG: GUI improvements
  - NEW: more IQI geometries

- **aRTist 2.0.0**
  - date: 2012-02-23
  - CHG: major code refactoring
  - CHG: new Bremsstrahlung spectra model (XRayTools)
  - CHG: reimplementation of the ray casting engine
  - CHG: GUI redesign
  - NEW: 3D view mouse interaction
  - NEW: live preview