
GAMOS 5.2.0 User's Guide

Release 5.2.0

GAMOS Collaboration

January 20, 2018

1	Introduction	3
1.1	About this document	3
1.2	Introduction to GAMOS	3
1.3	Structure of GAMOS	4
1.4	The plug-in concept	4
2	Getting started	5
2.1	Getting the code and installing it	5
2.1.1	Installing in Linux and Mac OS	5
2.1.2	Installing in Windows	8
2.2	Running an example in Linux or Mac OS	8
2.3	Running an example in Windows	9
2.3.1	GAMOS Graphical User Interface	9
2.4	Compiling GAMOS	10
2.4.1	Compiling your new code	10
3	Geometry	13
3.1	Building your geometry with a text file	13
3.1.1	Description of geometry text file format	13
3.1.2	Dumping your Geant4 geometry in text file format	37
3.1.3	Adding new tags to your input text file	37
3.1.4	Parallel geometries	39
3.2	Building complex geometries in a simple way	40
3.3	Building a simple geometry with one material	40
3.4	Building your geometry with C++ code	40
3.5	Phantom geometries	40
3.5.1	Simple phantom geometries	40
3.5.2	Setting off visualization of phantom geometries	41
3.5.3	Reading DICOM files	41
3.6	Movements	41
3.7	Geometry utilities	43
3.8	Magnetic and electric fields	44
4	Visualization	47
4.1	3D Geant4 visualization	47
4.2	2D visualization	47
5	Generator	49
5.1	Using GAMOS generator	49
5.1.1	Introduction	49
5.1.2	Particle sources	49

5.1.3	Distributions	51
5.2	Reading your generator particles from a text file	56
5.3	Reading your generator particles from a binary file	57
5.4	Event generator changing energy and material	58
5.5	Event generator histograms	58
5.6	Biassing generator distributions	58
5.7	Building your generator with C++	59
5.8	Using ions	59
6	Physics	61
6.1	GAMOS electromagnetic physics lists	61
6.1.1	Basic electromagnetic physics list	61
6.1.2	GAMOS electromagnetic extended physics list	63
6.1.3	Standard electromagnetic physics list options	63
6.1.4	Multiple scattering models	63
6.1.5	Bremsstrahlung angular distributions	64
6.2	GAMOS hadrontherapy physics list	64
6.3	Microdosimetry physics list	65
6.4	Other physics lists	65
6.5	Optical photons	66
6.5.1	Using optical photons as primary generator	68
6.6	X-ray refraction	68
6.7	Coulomb scattering	68
6.8	Atomic deexcitation processes	68
6.9	Decay process	69
6.10	Radioactive decay process	69
6.11	Cerenkov process	69
6.12	Coulomb scattering	70
6.13	Nuclear processes of electromagnetic particles	70
6.14	Replacing process models	70
6.14.1	Replacing a set process models	70
6.14.2	Replacing one process model	70
6.15	Removing a process from a physics list	71
6.16	Production cuts	72
6.16.1	Production cuts by region	72
6.16.2	Energy cuts to range cuts conversion	73
6.16.3	Minimum and maximum production cuts	73
6.16.4	Apply cuts for all processes	73
6.17	User limits	74
6.18	Automatic optimisation of cuts	74
6.18.1	Range rejection	75
6.19	Building your physics list with C++ code	76
7	User Actions	77
7.1	Adding a filter	77
7.2	Adding a classifier	77
7.3	User action name	78
7.4	Creating your GAMOS user action	78
8	Sensitive Detector and Hits	79
8.1	Sensitive detectors	79
8.1.1	Attaching a sensitive detector to a volume	79
8.1.2	Building your sensitive detector with C++ code	80
8.2	Hits	80

8.3	Hits digitization and reconstruction	81
8.3.1	Hits digitization	81
8.3.2	Hits and digits reconstruction	81
8.3.3	Examples of reconstructed hit builders	82
8.4	Detector effects	83
8.4.1	Energy resolution	83
8.4.2	Time resolution	83
8.4.3	Detector measuring time	84
8.4.4	Detector dead time	84
8.4.5	Minimum hit energy	85
8.5	Identifying each sensitive detector copy	86
8.6	Storing and retrieving hits	86
8.6.1	File format	87
8.6.2	Storing reconstructed hits	87
8.7	Hits histograms	88
9	Scoring	91
9.1	Creating a scorer	91
9.2	Scorer classes	92
9.2.1	Scoring in voxelised phantoms	95
9.3	Filter classes	96
9.4	Scorer printers	96
9.5	Classifiers	97
9.6	Multiplying by data	97
9.7	Multiplying by distribution	97
9.8	Convergence testing	97
9.9	Point detector scorer	99
9.9.1	Theoretical basis	99
9.9.2	GAMOS implementation	100
9.9.3	Variance reduction techniques	103
9.9.4	Sum scoring results and plot them	105
10	Variance reduction techniques	107
10.1	Introduction	107
10.2	Importance sampling	107
10.3	Geometrical biasing	108
10.4	Biasing operations	108
10.4.1	Cross section biasing	109
10.4.2	Force collision	109
10.4.3	Uniform bremsstrahlung splitting	109
10.4.4	Directional bremsstrahlung splitting	110
10.4.5	Equal weight particle splitting	110
11	Histogramming	111
11.1	Histogram formats	111
11.2	Histograms in CSV format	111
11.3	Changing histogram minimum, maximum and number of bins	112
11.4	Histograms name separator	112
11.5	Output files name	112
11.6	Analysing your histograms with ROOT	113
11.6.1	Printing the histograms in graphics files	113
11.6.2	Comparing histograms in two files	113
11.7	Creating your own histogram	114
12	Analysis (extracting data)	115

12.1	Introduction: GAMOS data	115
12.2	Data users	116
12.2.1	Behaviour as a function of information object	117
12.2.2	Behaviour as a function of output format	117
12.2.3	Selection of data list for a data user	119
12.2.4	Saving GAMOS data in a ROOT TTree	122
12.2.5	Filter from data	123
12.2.6	Classifier by data	123
12.2.7	Primitive scorer from data	124
12.3	List of available data	124
12.3.1	Position	124
12.3.2	Direction	126
12.3.3	Momentum	128
12.3.4	Energy	129
12.3.5	Geometrical objects	129
12.3.6	Material variables	130
12.3.7	Particle and process	130
12.3.8	Secondary tracks	131
12.3.9	Ancestor data	132
12.3.10	Others	132
12.4	Output file names	132
12.5	Merging results from different jobs	133
12.5.1	Example of Analysing text output files	134
13	Filters	137
13.1	Introduction	137
13.2	Simple filters	138
13.3	Volume filters	139
13.4	Filters of filters	141
13.5	Applying filters to a user action	142
13.6	Checking filters at a user action	142
13.7	Filtering steps in the future	142
14	Classifiers	145
14.1	Introduction	145
14.2	Setting indices to classifiers	146
14.3	Classifying on secondary data	147
15	Distributions	149
15.1	Introduction	149
15.2	Creating a distribution	149
15.3	Assigning a GAMOS data	149
15.4	Reading values from a file	150
15.5	Numeric distributions	150
15.6	String distributions	151
15.6.1	String data distribution	151
15.6.2	Geometrical biasing distribution	151
15.7	Ratio of distributions	152
16	Utility user actions	153
16.1	Counting the number of tracks and events	153
16.2	Counting the processes	153
16.3	Shower shape studies	155
16.4	Killing all tracks	156
16.5	Table of tracks and steps	157

16.6	Material budget studies	157
16.7	Detailed report of where CPU time is spent	157
16.8	Changing the weight using a distribution	158
16.9	Copying the weight to the secondary particles	158
16.10	Stop run after a certain CPU time	159
16.11	Visualising only a set of events	159
16.12	Setting the correlation between the gammas of a radioactive decay chain	159
16.13	Printing electromagnetic parameters	159
17	Managing the verbosity	161
17.1	GAMOS verbosity managers	161
17.1.1	Controlling GAMOS verbosity by event	162
17.2	Using a GAMOS verbosity manager in your code	162
17.3	Creating your own verbosity manager	163
17.4	Controlling the Geant4 verbosity by event and track	163
17.5	Dumping the standard output and error in log files	164
18	Detector applications	165
18.1	Identifying Compton interactions	165
18.1.1	Compton studies histograms	166
18.1.2	Histograms of data about the interactions and the reconstructed hits	167
18.1.3	Classification of good and bad identification histograms as a function of variable	168
18.2	Histograms of gammas at sensitive detectors	169
18.3	Automatic determination of production cuts for a detector	170
18.4	Automatic determination of user limits for a detector	171
19	PET application	173
19.1	PET geometry	173
19.2	PET event classification	174
19.3	PET histograms: event classification	175
19.4	PET output for reconstruction	176
19.4.1	List-mode binary file	176
19.4.2	Projection data file	176
19.5	PET histograms: positrons	177
19.6	Detector histograms: distance between two gammas	177
20	SPECT application	179
20.1	SPECT event classification	179
20.2	SPECT histograms: event classification	180
20.3	SPECT output for reconstruction	181
21	Compton camera application	183
21.1	Compton camera geometry	183
21.2	Compton camera Event Classification	184
21.3	Compton camera histograms: event classification	186
21.4	Compton camera output for reconstruction	186
22	Image reconstruction utilities	189
22.1	List-mode to projection data: <i>lm2pd</i>	189
22.2	Summing projection data files: <i>sumProjdata</i>	189
22.3	Analytic image reconstruction: <i>ssrb_fbp</i>	190
22.4	Visualization tools	191
22.5	Stochastic Image Ensemble method	191
22.5.1	Introduction	191
22.5.2	Getting Started	192

22.5.3	Program Parameters and Flags	194
22.5.4	Analysis	196
23	Radiotherapy application	197
23.1	Geometrical modules	197
23.1.1	JAWS module	197
23.1.2	MLC module	200
23.2	Using phase spaces	207
23.2.1	Writing phase spaces	207
23.2.2	Phase space text file	209
23.2.3	Phase space histograms	209
23.2.4	Reading phase spaces	210
23.2.5	Adding extra information to a phase space	211
23.2.6	Reusing a particle at a phase space without filling the phase space file	212
23.3	Optimisation of a linac simulation	213
23.3.1	Cuts optimisation	213
23.3.2	Electromagnetic parameters optimisation	213
23.3.3	Particle splitting	213
23.3.4	Killing particles at big X/Y	214
23.4	Scoring dose in phantom	214
23.4.1	Saving scores and scores squared in binary file	215
23.4.2	Saving scores and score errors in text file	215
23.4.3	Saving scores in histograms	216
23.5	Analysis utilities	217
23.5.1	Summing phase space files	217
23.5.2	Making histograms out of a phase space file	218
23.5.3	Merging 'sqdose' files	219
23.5.4	Merging '3ddose' files	219
23.5.5	Making histograms out of a 'sqdose' file	220
23.5.6	Automatic determination of production cuts for an accelerator simulation	221
23.5.7	Automatic determination of production cuts for a dose in a phantom simulation	222
23.5.8	Automatic determination of user limits for an accelerator simulation	222
23.5.9	Automatic determination of user limits for a dose in phantom simulation	223
24	Shielding application	225
24.1	Studying penetration	225
24.2	Activation studies	226
24.2.1	Sum activation results and plot them	227
24.3	Correcting decay time to take into account isotope activity	228
24.4	Print channel by channel cross sections	228
24.5	Counting hadronic cross sections	229
24.6	Make histograms of secondary particles from the neutron_hp or particle_hp databases	230
24.7	Print yields of production of secondary particles from charged particles traversing a thich material	231
24.8	Analysis utilities	232
24.8.1	Summing scoring results	232
24.8.2	Summing activation results	232
25	Managing DICOM files	233
25.1	Managing DICOM CT geometry files	233
25.1.1	Converting DICOM CT geometry files to GAMOS simulation files	233
25.1.2	CT geometry file format	235
25.1.3	Reading a DICOM CT geometry file in a GAMOS job	236
25.2	Managing DICOM CT structure files	237
25.2.1	Converting CT structure files	237

25.2.2	Using files with CT structures in your GAMOS job	238
25.2.3	Filtering on DICOM structures	238
25.3	Managing DICOM PET image files	239
25.3.1	Converting DICOM PET files to GAMOS simulation files	239
25.3.2	Reading a DICOM PET image file in a GAMOS job	239
25.4	Managing RT plan data	239
25.4.1	Transforming DICOM RT plan data to GAMOS ASCII format	239
25.4.2	Using DICOM RT plan data in a GAMOS job	241
25.5	Managing RTIon plan data	241
25.5.1	Transforming DICOM RTIon plan data to GAMOS ASCII format	241
25.5.2	Using DICOM RTIon plan data in a GAMOS job	242
25.6	Visualizing geometry superimposed to DICOM images	242
25.7	Partial phantom geometries	243
25.8	Visualizing all types of DICOM files	244
25.8.1	Visualizing CT images	244
25.8.2	Visualizing PET images	244
25.8.3	Visualizing RT structures	244
25.8.4	Visualizing Doses	245
25.8.5	Drawing options	245
25.9	Modifying the material indices of voxels inside a CT structure	246
25.10	Use simple phantom geometries	246
26	Optimising the CPU time of your application	249
26.1	Knowing where the time is spent	249
26.2	Production cuts	249
26.3	Killing particles of small energy	250
26.4	Killing particles	250
26.5	Optimising the particle generation	250
26.6	Limiting the number of user actions	250
26.7	Using variance reduction techniques	250
27	Appendix A: Miscellaneous	251
27.1	Using parameters	251
27.1.1	Checking the usage of parameters	251
27.2	Managing the input data files	252
27.3	Random number seeds	252
27.3.1	Setting the initial random number seeds	253
27.3.2	Restoring the initial random number seeds	253
27.3.3	Changing the random engine	253
27.4	Sending several jobs in the same machine	253
27.5	Identifying touchables	255
27.6	Using asterisks to get volume, particle and material names	256
27.7	Using particle names	256
28	Appendix B: Extending GAMOS functionality with C++ utilities	259
28.1	Converting a Geant4 example into a GAMOS example	259
28.2	Creating your plug-in	260
28.3	Using a parameter in your C++ code	261
28.4	Event classification by interaction types	261
28.5	Structure of GAMOS	262
29	Indices and tables	265
	Bibliography	267

Contents:

INTRODUCTION

About this document

This manual refers to GAMOS version 5.2.0. It is written in DocBook and it is maintained at the following address

http://fismed.ciemat.es/GAMOS/GAMOS_doc/GAMOS.5.2.0/GamosUsersGuide_V5.2.0.pdf

We will use through this manual many terms common to the Monte Carlo simulation terminology and specifically to the Geant4 [*biblio.geant4*] terminology. If you are new to it, please read before, for example, the Geant4 documentation [*biblio.g4doc*]. We have tried though to make this manual self-consistent,

If you find that some of the instructions given here do not give the expected result, please use the *GAMOS bug report system*

<http://telemaco.ciemat.es/bugzilla>

and detail the problem, the GAMOS version and provide as much information as possible. We will also warmly welcome any kind of comment or suggestion you would like to send us about this guide, or about the GAMOS functionalities or user interface.

If you have some questions about something you do not understand or want to ask for some new functionality in GAMOS, please use the *GAMOS Discussion Forum*

http://groups.google.com/group/gamos_users

Introduction to GAMOS

The acronym GAMOS stands for “*Geant4-based Architecture for Medicine-Oriented Simulations*”. It is therefore a Monte Carlo simulation software and it is based on the Geant4 toolkit

[*biblio.geant4*]. The objective of GAMOS is to provide a software framework that serves the unexperienced user to simulate his/her project without having to code in C++ and with a minimal knowledge of Geant4, and at the same time, let an advanced user add new functionalities and easily integrate it with the rest of GAMOS functionality.

We have also tried to provide you with several tools that help you understand in detail your simulation (controlling the verbosity, making histograms about many variables, scoring different quantities, etc.), as well as other tools to help you in optimising your simulation. GAMOS is composed of a core software that covers the main functionality of a Geant4 simulation and a set of applications for specific domains.

Structure of GAMOS

If you look into GAMOS.5.2.0 directory you can find several directories. The directories *tmp*, *lib* and *bin* are internal directories created at GAMOS compilation. The other directories are the following:

- *source*: this is the directory where the GAMOS C++ code lies. You will probably not have to care about this, unless you are an advanced user and need to develop new code.
- *examples*: some first examples. We recommend you that after the GAMOS installation you run the example *examples/test/test.in*
- *tutorials*: you can find nine step-by-step tutorials: *Primer*, *Histogram and Scorers*, *PET*, *SPECT*, *Compton Camera*, *Radiotherapy and Shielding*, *Gamma spectrometry*, *plug-in's*. They include several exercises with increasing difficulty, and the exercise outputs as well as the exercise solutions are provided. We recommend you to follow one or several of these tutorials to become acquainted with GAMOS. The *Primer* may be a good place to start, as it starts with a few basic examples and shows how to visualize geometries and histograms.
- *analysis*: some utilities that may serve you to analyse your output.
- *data*: directory where GAMOS algorithms look for data files (after current directory).

The plug-in concept

To provide the user with a big flexibility in choosing different simulation components (geometry, physics, user actions, histograms, ...) and combining them to his/her will in a simple way, GAMOS is based on the plug-in concept. This means that the "main" program runs without predefined components and the user tells it which components are being loaded at run time (without needing to recompile) by simply listing them in a text input file. This mechanism also lets the user define a new component that was not foreseen by GAMOS and easily tell GAMOS to use it together with any other of his/her own components or GAMOS components.

A common example to better understand the plug-in concept is the plug-in's that are installed on your computer when you open some Internet page with your web browser. Your web browser can use these plug-in's to get an extra functionality (viewing videos, animated figures, ...) without your having to recompile it and even if the web browser designers had never before heard of the new plug-in.

For each of the simulation component types we will describe in the corresponding section which is the command to select it and how to transform a new user component into a GAMOS plug-in.

For the implementation of plug-in's GAMOS has chosen the CERN package ROOT.

GETTING STARTED

This chapter explains the practical details to obtain the GAMOS code, compile and run it.

Getting the code and installing it

GAMOS has been tested in over ten Linux and MacOS distributions and compilers, and in Windows XP and Windows 7 with Visual Studio 2010. Each new release is tested with over sixty tests in at least three different operating systems.

You can download GAMOS from <http://fismed.ciemat.es/GAMOS>

GAMOS depends on Geant4 and on ROOT [*biblio.root*]. To download and install everything you can follow the instruction in the 'Code download' area.

Installing in Linux and Mac OS

As explained in the web page you need to get first the installation scripts and uncompress them in the *scripts* directory (you may do it automatically by downloading the scripts installation utility from the web page).

After that you just need to type the command

```
.installGamos.sh MY_INSTALLATION_DIR
```

where *MY_INSTALLATION_DIR* is the directory where you want to install GAMOS. The directory name has to be an absolute path, i.e. starting with */*. *\$HOME* is OK because it means */home/....*

This command will download the GAMOS code as well as Geant4 and ROOT packages, and will compile them all. Be sure that you have installed in your system the X11 libraries in the directory */usr/X11R6/lib* or */usr/local/lib* or */usr/lib* (check for the following libraries: *libXmu.so*, *libXt.so*, *libXext.so*, *libX11.so*, *libICE.so*).

You may type for example 'locate libXmu.so' to know where they are in they are not in the mentioned directories

as well as the OpenGL libraries (check for the following libraries: *libGL.so*, *libGLU.so*). If the OpenGL libraries are not found, GAMOS will ask you if you want to continue without them. In case of positive answer, GAMOS will be installed without OpenGLX visualisation (but still with VRML visualisation). If the X11 libraries are not found, GAMOS will ask you if you want to continue without them. In case of positive answer, GAMOS will be installed without OpenGLX visualisation (but still with VRML visualisation) and without ROOT histograms. ROOT has some extra checks for libraries, namely *libXpm.so*, *libXft.so* and *libXi.so*, so you also have to install them.

After all the code is compiled, follow the instructions on how to run an example.

If the installation fails, you can find the cause of it in the file *install.log* at the directory where you tried to do the installation. Look for the first occurrence of the characters *error* (with a blank space first, as several class name have these characters). It is likely that the error is due to the fact that you are using a ROOT version not compatible with your system. If this is the case, try to find a new version number at the ROOT web page <http://root.cern.ch> and change

it at the file *installGamosBaseDefs.sh* that you will find at the *scripts* directory. Use an older version if you think your operative system is old and a new version if you think it is old. If you still are not able to install successfully GAMOS, please open a report at the GAMOS Users' Forum, https://groups.google.com/forum/#!forum/gamos_users, and attach the *install.log* file (compress it first).

Tips for installation in Ubuntu

If you are using the Ubuntu Linux distribution you have to take into account that the default installation does not include the software for C++ development. Therefore, if you do not have this software in your installation you should download the following packages (this works for Ubuntu 16.04, other releases may include a different set of packages):

- `sudo apt-get -y install g++`
- `sudo apt-get -y install dpkg-dev`
- `sudo apt-get -y install libx11-dev`
- `sudo apt-get -y install libxpm-dev`
- `sudo apt-get -y install libxft-dev`
- `sudo apt-get -y install libxext-dev`
- `sudo apt-get -y install freeglut3-dev`
- `sudo apt-get -y install libxmu-dev`
- `sudo apt-get -y install libxi-dev`
- `sudo apt-get -y install cmake`
- `sudo apt-get -y install libtiff-dev`

Tips for installation in Fedora Core

If you are using the Fedora Core Linux distribution you have to select the software development tools at your installation. If you do not have this software in your installation you should download the following packages (this works for Fedora Core 25, other releases may include a different set of packages):

- `sudo dnf install -y gcc-c++`
- `sudo dnf install -y expat-devel`
- `sudo dnf install -y libX11-devel`
- `sudo dnf install -y libXpm-devel`
- `sudo dnf install -y libXft-devel`
- `sudo dnf install -y libXext-devel`
- `sudo dnf install -y libXmu-devel`
- `sudo dnf install -y libXi-devel`
- `sudo dnf install -y freeglut-devel`
- `sudo dnf install -y glibc-devel`
- `sudo dnf install -y zlib-devel`
- `sudo dnf install -y pcre-devel`
- `sudo dnf install -y cmake`

Tips for installation in CentOS

If you are using the CentOS Linux distribution you have to take into account that the default installation does not include the software for C++ development. Therefore, if you do not have this software in your installation you should download the following packages (this works for CentOS 7, other releases may include a different set of packages):

- `sudo yum install -y gcc-c++`
- `sudo yum install -y expat-devel`
- `sudo yum install -y libX11-devel`
- `sudo yum install -y libXpm-devel`
- `sudo yum install -y libXft-devel`
- `sudo yum install -y libXext-devel`
- `sudo yum install -y libXmu-devel`
- `sudo yum install -y libXi-devel`
- `sudo yum install -y freeglut-devel`
- `sudo yum install -y glibc-devel`
- `sudo yum install -y zlib-devel`
- `sudo yum install -y pcre-devel`
- `sudo yum install -y cmake`

Downloading the files separately and installing afterwards

If you have poor connection it may be better to download the GAMOS installation files one by one first and then running the installation process. You have to download the following files:

- `GAMOS.5.2.0.tgz`
- `geant4.10.02.gamos.tar.gz`
- `geant4.10.02.gamos.data.tar.gz`
- `root_v5.34.36.source.tar.gz`

Put all the files in a directory, and then type

```
./installGamos.sh MY_INSTALLATION_DIR MY_DOWNLOAD_DIR
```

where *MY_INSTALLATION_DIR* is the directory where you want to install GAMOS and *MY_DOWNLOAD_DIR* is the directory where you copied the installation files. Both directory names have to be absolute paths, i.e. starting with *./*. *\$HOME* is OK because it means */home/....*

Copying GAMOS to a different place

If you have GAMOS installed at a given directory in a given computer and you want to copy it to another directory or to another computer, you can skip the downloading phase and go directly to the compilation (you should not try to use the original GAMOS distribution unless you are sure that the new computer has exactly the same operative system and the directory is the same as in the old computer). To reinstall without downloading use the command:

```
./reInstallGamos.sh MY_INSTALLATION_DIR
```

where *MY_INSTALLATION_DIR* is the directory where you have copied GAMOS. The directory name has to be an absolute path, i.e. starting with /. *\$HOME* is OK because it means */home/....*

Installing in Windows

GAMOS 5.2.0 is still not available in Windows, please use GAMOS.5.0.0 instead or use the VMWARE virtual machine installation.

The first thing to do to install in Windows is selecting a directory, which we will call *MY_INSTALLATION_DIR*, and which can be for example *c:\gamos*. Before installing GAMOS you have to install ROOT. Get the executable file from the GAMOS web page and install it. GAMOS has to know automatically where ROOT is, so be careful not to use the *Standard* installation, but the *Custom* one, and change the default directory, to the directory *MY_INSTALLATION_DIR\external\root\5.34.10\root*.

To install GAMOS you have to download the GAMOS.5.2.0.zip file, copy it to your *MY_INSTALLATION_DIR* and uncompress it. If your windows system does not have an unzip software and you do not have one, you may download a free one from <http://7zip.com>. After it is decompressed you just have to open the file *MY_INSTALLATION_DIR\GAMOS.5.2.0\GAMOSGUI.exe*. The first time it is opened it will configure GAMOS and create a *GAMOSGUI.ini* file. To run GAMOS you just have to select your input file clicking on the *Open Input File* and run it clicking on the *Run* button. All the output files will be written in the same directory where the input file is.

Checking your windows installation

As explained above, it is mandatory that the directory structure is correct, so that the GAMOS executable can find all the dynamic libraries. You should check that ROOT has been installed in the directory *MY_DIRECTORY/external/root/5.34.10/root* while GAMOS has been installed at *MY_DIRECTORY\GAMOS.5.2.0\GAMOSGUI.exe*, where *MY_DIRECTORY* is the directory you chose for GAMOS.

If the directory structure is OK, the first time you execute *GAMOSGUI.exe*, it does not find any file named *GAMOSGUI.ini* and then a *command prompt window* (a black window) pops up. This window should stay for about 1 minute or more, and you should see a list of messages telling you that the GAMOS libraries are being processed. If the black window disappears quickly is that the library processing is giving some error message. In this case you probably are not able to read the messages fast enough before the window disappears, so you should try to process the libraries in other way:

- Open a *command prompt window*. In windows XP.
- Type *c:*, or the disk unit where the GAMOS directories lie
- Type *cd MY_DIRECTORY* where *MY_DIRECTORY* is the directory where *GAMOSGUI.exe* lies
- Type *cd config*
- Type *genmap.win.bat > genmap.log*
- Open the *genmap.log*, for example with *more genmap.log*, or send this file to the GAMOS developers if you need help

Running an example in Linux or Mac OS

If you have done a standard installation, you will have your code compiled and ready to run.

Before running any example you have to set some configuration variables, mainly where you have installed GAMOS and the depending libraries. This is all done in the file

MY_GAMOS_DIR/config/configamos.sh

By *MY_GAMOS_DIR* we mean the installation directory you have chosen, i.e. *MY_INSTALLATION_DIR*, plus the directory where the GAMOS version is installed, i.e. **MY_INSTALLATION_DIR/GAMOS.5.2.0**

or

```
MY_GAMOS_DIR/config/configamos.csh
```

Therefore before running you have to source this file:

```
source MY_GAMOS_DIR/config/configamos.sh
```

or

```
source MY_GAMOS_DIR/config/configamos.csh
```

depending on your shell flavour. *Remember to type this command every time you start a new session to run GAMOS.*

To run your application inside GAMOS you do not have to write a “main” program, as GAMOS provides a unique “main” that serves to run any application. When you run the GAMOS “main” it will load and call the components you want (geometry, physics, generator, hits building, histograms, ...) by simply defining them in the *input command file*. Therefore to run your application simply type

```
gamos MY_INPUT_FILE
```

where *MY_INPUT_FILE* is a typical Geant4 macro file that includes Geant4 and GAMOS commands.

The minimum set of commands that you need are those to select a geometry, a physics list and a generator, to initialize Geant4 and to run N events. In this case your input file may look like this one:

```
/gamos/setParam GmGeometryFromText:FileName test.geom
/gamos/geometry GmGeometryFromText
/gamos/physicsList GmEMPhysics
/gamos/generator GmGenerator
/gamos/generator/addSingleParticleSource MY_SOURCE e- 1*MeV
/run/initialize
/run/beamOn 10
```

This will create a simple geometry, the one described in the file *test.geom* lying in the current directory or in the *MY_GAMOS_DIR/data* directory, set the physics as the low energy electromagnetic Geant4 physics and run 10 events with an electron of 1 MeV as primary particle.

You can then add any of the command described in this document, or any Geant4 command or any command you created yourself.

Beware that Geant4 is a state machine, and the list of available commands depends on the current state. The main state change is triggered by the */run/initialize* command, which changes the state from *G4State_PreInit* to *G4State_Idle*. You may get a full list of the available commands at any moment with the command */control/manual*.

Running an example in Windows

To run in Windows we recommend you to use the Graphical User Interface, that you can find at *MY_INSTALLATION_DIR/GAMOS.5.2.0/GAMOSGUI.exe*.

GAMOS Graphical User Interface

The first you should do is open an input file and then run it. Or alternatively you may create a new one. The list of buttons of this GUI is the following:

- *Open File*: Open an existing input file

- *Run*: run GAMOS with the input file in the screen (it is saved before run)
- *Open Log*: open the log file of a GAMOS run (*gamos.log* in the same directory that the current input file)
- *Open Error Log*: open the error log file of a GAMOS run (*gamos_error.log* in the same directory that the current input file)
- *New File*: create a new input file
- *Save File*: save current input file using current name
- *Save File As*: save current input file changing name
- *Select Font*: select the font for input and log files
- *Help*: open the pdf file of GAMOS User's Guide
- *Exit*: exit GUI

Compiling GAMOS

If you installed GAMOS as explained in the previous section, the compilation will be done automatically. Then you may run your application in GAMOS by writing your user commands without any need of compiling ever more, and therefore you do not need to read this section.

Only if you want to extend the GAMOS functionality by providing new code, you will have to follow the instructions in this section.

GAMOS uses the GNU make tool to manage the compilation and generation of executables. It uses a set of configuration files based on the Geant4 ones. Therefore, if you are familiar to Geant4, you will find no difficulty in compiling GAMOS.

After untarring the installation file, you will have a directory called *MY_GAMOS_DIR* (substitute it by whatever name you used). This is the directory where the GAMOS code is, the rest are the external libraries used by GAMOS and the configuration utilities.

Before compiling, you have to define a few variables, mainly the location of the different external packages. All this is done by sourcing the file

```
source MY_GAMOS_DIR/config/confgamos.sh
```

or

```
source MY_GAMOS_DIR/config/confgamos.csh
```

To compile any directory of GAMOS, and all the directories below, you just have to go to that directory and type *make*. This will compile all the *.cc* files found in the *src* directory, build the library and the plug-in's, and in the case of the directory *GamosCore/GamosApplication* it will also create the *gamos* executable. You may need to type the Linux command *rehash* to refresh your environmental variables in case there was no executable file before starting the compilation.

Compiling your new code

If you have created a new directory with your C++ code you have to compile it following the Geant4 way. The implementation files should have the suffix *.cc* and should be in a subdirectory called *src*. For the declaration files, you have a greater freedom; the Geant4 way is that they have the suffix *.hh* and lie in a subdirectory called *include*, but you can do it your own way (and after that, you have to be consistent in the GNUmakefile, as explained below).

You then have to build a *GNUmakefile*, that will steer the compilation and the building of the library and the plug-in's, when you type the command *make*. For building it you may follow the examples in the *GamosCore/GamosXXX* directories. We take as example the file *GamosCore/GamosGeometry/GNUmakefile*:

```
name := GamosGeometry
G4TARGET := $(name)
G4EXLIB := true
.PHONY: all
all: lib plugin
include $(GAMOSINSTALL)/config/binmake.gmk
include $(GAMOSINSTALL)/config/general.gmk
EXTRALIBS += -lGamosBase_Base -lGamosUtils -lGamosUserActionMgr
```

Let's go one by one through the lines: In the first one you define the name of your library:

```
name := GamosGeometry
```

The following two lines are used internally by the GAMOS scripts and are mandatory:

```
G4TARGET := $(name)
G4EXLIB := true
```

Then you define what you want to do when you type *make*:

```
.PHONY: all
all: lib plugin
```

There are several possibilities:

- *lib* Compile and build the library.
- *plugin* Build the plug-in. Use it if and only if you are creating a new plug-in.
- *plugin_check* Check that you are linking with all the libraries that will be needed. This check is not mandatory, but if you do not do it and you are missing some library, when you run you will get an error message.
- *bin* Build the executable. You will probably never need this, as GAMOS is based on dynamic loading and plug-in's, so that there is only one GAMOS executable, which is built at installation.

The following two lines are needed for configuration

```
include $(GAMOSINSTALL)/config/binmake.gmk
include $(GAMOSINSTALL)/config/general.gmk
```

If you edit the file *general.gmk*, you will see that it is just including a different file for each package. Therefore, instead of using it, you may include all or only a subset of them if you do not need them all.

Finally, you define the GAMOS libraries that will be linked to yours, i.e. the libraries of each of the files that you have included in your code

For the external packages the set of libraries is fixed and defined in the configuration files

```
EXTRALIBS += -lGamosBase -lGamosUtils
-lGamosFactories -lGamosUserActionMgr
```

If you have doubts about which GAMOS libraries to include, you may include them all, by including *include \$(GAMOSINSTALL)/config/gamos_libraries.gmk*.

If you have built several levels of directories you may want to have the possibility of typing *make* at the top most directory to trigger the compilation of all the directories. To do this you just have to add a *GNUmakefile* at the top level directory similar to the one at *\$(GAMOSINSTALL)/source/GamosCore/GNUmakefile*, that we reproduce here:

```
include $(GAMOSINSTALL)/config/architecture.gmk
SUBDIRS = GamosUtils GamosBase GamosUserActionMgr GamosData
GamosAnalysis GamosGeometry GamosMovement GamosPhysics
GamosGenerator GamosSD GamosUtilsUA GamosReadDICOM
GamosScoring GamosApplication
include $(GAMOSINSTALL)/config/globlib.gmk
```

You only have to change the line starting by *SUBDIRS* =, to list the name of your subdirectories.

GEOMETRY

You can describe your detector in three different ways: by defining your setup in a text file, by using one of the geometry examples provided or in the standard Geant4 way, by writing your C++ class inheriting from `G4VUserDetectorConstruction`.

Building your geometry with a text file

You can define your geometry in a simple text file as described in the following subsection.

This format is in the official Geant4 release since geant4.9.1

You can also use as example the one at

MY_GAMOS_DIR/data/test.geom

Once your file is ready, you have to tell GAMOS to use your geometry definition, first telling the name of your file with the command

```
/gamos/setParam GmGeometryFromText:FileName MY_FILENAME
```

The default path to look for this file is defined by the variable `GAMOS_SEARCH_PATH`. For details see section on *Managing the input data files*

and then telling GAMOS to use the constructor of geometry from text file

```
/gamos/geometry GmGeometryFromText
```

Building complex geometries in a simple way

There are several examples of geometries for common medical devices, for example the ones you find in the PET directory and the one to build simple voxelised phantoms. They have been designed to be used for describing different devices by simply changing the configuration data. For more details, please see the corresponding sections of this manual.

Building a simple geometry with one material

This geometry can serve for example when you want to get the cross section of a set of materials, so you do not really need to define any geometry, but just a fake one containing the materials you need so that you can use the Geant4 utilities to get the cross sections.

To use this, you have to set the geometry to :

```
/gamos/geometry GmGeometryUseMaterials
```

The file name where you can define the materials (unless you want to use the Geant4 predefined ones) can be set with the parameter:

```
/gamos/setParam GmGeometryUseMaterials:FileName FILE_NAME
```

Then you have to set a list of materials with the parameter:

```
/gamos/setParam GmGeometryUseMaterials:Materials MATE_1 MATE_2 ..
```

The geometry to be built will be a list of volumes, one with each material.

You can see an example on the use of this utility at *tutorials/ShieldingTutorial/exercise5*.

Building your geometry with C++ code

You can build your geometry by writing your C++ class inheriting from `G4VUserDetectorConstruction` (see example in [\[biblio.g4doc-geometry\]](#)). After that you have to transform it into a GAMOS plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the `GmGeometryFactory`, or see the example *examples/N02*.

Phantom geometries

Simple phantom geometries

The user may build simple regular phantom geometries without the need of writing a DICOM file by using

```
/gamos/geometry GmSimplePhantomGeometry
```

The number of voxels is defined with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:N Voxels NVOXEL_X NVOXEL_Y NVOXEL_Z
```

The minimum and maximum extensions in the three axes are defined with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:PhantomDims MIN_X MAX_X MIN_Y MAX_Y MIN_Z MAX_Z
```

Then you can divide the phantom in different regions along the Z axis with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:MaterialZVoxels NZ_1 NZ_2 ...
```

where NZ_i is the number of voxels along Z of the i region.

Then you can assign the material and material densities of each Z region with the parameters

```
/gamos/setParam GmSimplePhantomGeometry:MaterialNames MATERIAL_1 MATERIAL_2 ...
```

```
/gamos/setParam GmSimplePhantomGeometry:MaterialDensities DENSITY_1 DENSITY_2 ...
```

Setting off visualization of phantom geometries

As the phantom geometry is not built with the text file format, the tag `:VIS`, that serves to set off the visualization, cannot be used for the phantom and the phantom container. Instead you may use the following parameters:

```
/gamos/setParam GmReadPhantomGeometry:Phantom:VisOff TRUE
```

```
/gamos/setParam GmReadPhantomGeometry:PhantomContainer:VisOff TRUE
```


Reading DICOM files

Please see chapter [Managing DICOM files <DICOM_chapter>]_ for all details about managing DICOM files.

Movements

Thanks to the functionality of Geant4, it is possible to displace or rotate a volume during a run. To do this in GAMOS you have first to select the volume you want to move and if you want to move it after a certain number of events or after a certain time is elapsed (the time will be checked after each event, therefore the time interval will only be approximated). You also have to choose how much you want to move it and the axis (the axis of displacement or the axis around which happens the rotation). You have then to set the interval of events or time between movements. After you can define an offset so that the first interval does not start at 0. Finally you can choose that your movement is done forever (i.e. until the number of events in the run are exhausted) or it is only done N times.

The commands to tell GAMOS to make a movement are:

```
/gamos/movement/moveEachTime
```

if you want that the movement happens after a certain interval of time, and

```
/gamos/movement/moveEachNEvents
```

if you want that the movement happens after a certain number of events.

In both cases the command has to be followed by these arguments:

```
MOVEMENT_TYPE VOLUME_NAME VALUE AXIS_X AXIS_Y AXIS_Z TIME_INTERVAL/NEVENT_INTERVAL
```

where *MOVEMENT_TYPE* can be *displace* or *rotate*

where the first word has to be either *displace* or *rotate*

VOLUME_NAME must be one of the Geant4 volumes of your geometry, *VALUE* is the amount by which you want to displace or rotate (default Geant4 units are assumed, i.e. 'mm' and 'rad'; you can change them in the usual way, e.g., '*cm', '*deg'), *AXIS_X* *AXIS_Y* *AXIS_Z* are the three coordinates of the axes, *TIME_INTERVAL/NEVENTS_INTERVAL* is the interval after which the movements will happen. You may optionally use two extra parameters *OFFSET* (0 if not set) to change the value for the first movement, and **NUMBER_OF_INTERVALS** (infinite by default) to set the number of times the movement will happen..

If you want several movements in your run, you can use these commands as many times as you want. Then after each event GAMOS will check which of the movements must be done. If you want for example to move the same volume with different types of movements one after the other, you can use the 'number_of_intervals' and 'offset' arguments to do it.

The movements are managed in GAMOS by a user event action, called *GmMovementEventAction*, that checks at the beginning of event if the movement must be done. Therefore you cannot forget to activate this action with the GAMOS command:

```
/gamos/userAction GmMovementEventAction
```

If you forget it, the above commands will not exist and you will get a Geant4 exception.

There is an example of GAMOS movements that you can run in the directory *MY_GAMOS_DIR/examples/test*. Just run

```
gamos testMovement.in
```

and you will see an OpenGL view of a moving box.

You may implement any kind of movement in GAMOS by describing the movements in a file. To do it you just have to use the command:

/gamos/movement/moveFromFile FILE_NAME

after instantiating the user action

GmMovementEventAction

This file must have the following format:

A first line with the following parameters:

- Name of the volume to be moved
- Type of movement. It can be *Time* or *NEvents* (see above)
- Interval of events or time between movements
- Offset (number of events before movement starts)
- Number of times movement is done
- Number of movements that are describe in the following lines

The offset and number of intervals is optional, as above

For each movement a line has to be written with the following parameters

- Displacement value
- Displacement axis X
- Displacement axis Y
- Displacement axis Z
- Rotation value
- Rotation axis X
- Rotation axis Y
- Rotation axis Z

The above format can be repeated as many times as desired, for the same volume or other.

The following example file describes two movements: a first simple one, occurring five times each ten events, that displaces the volume *mybox* 50 mm along the X axis and rotates it 10 degrees around the Y axis, and a double movement, occurring 2 times each ten events starting from the event 50, that displaces the same volume 50 mm along the Y axis and 100 mm along the Z axis and rotates it 2 deg around the Z axis.

```
mybox NEvents 10 0 5 1
50.*mm 1. 0. 0. 10.*deg 0. 1. 0.
mybox NEvents 10 50 2 2
50.*mm 0. 1. 0. 2*deg 0. 0. 1.
100.*mm 0. 0. 1. 0*deg 0. 0. 0.
```

Geometry utilities

The command:

/gamos/geometry/printVolumeTree VERBOSE_LEVEL

serves to print the hierarchical tree of geometry volumes, starting from the world volume. The **VERBOSE_LEVEL** controls the amount of information given (each level print also the information from previous level). It is a two digit number, each of the two controls a different verbosity. The second digit (the one of the units) controls the information about solids and materials:

- `>=1` Prints logical volume name
- `>=2` Prints solid name, material name, cubic volume and volume mass
- `>=3` Prints solid parameters and visualisation attributes

The first digit (the one of the tens) controls the information about physical volumes:

- `>=1` Prints physical volume name, copy number and parent volume name
- `>=2` Print physical volume position and rotation
- `>=3` Print replica or parameterisation data if the physical volume is of this type

The command:

```
/gamos/geometry/printTouchableS NAME_1 NAME_2
```

Print the list of all touchables whose name includes the words `NAME_1`, `NAME_2`, ..., and for each one: touchable name, solid type, material name, global position and rotation and local position and rotation

The command:

```
/gamos/geometry/printMaterials VERBOSE_LEVEL
```

Print the list of all materials. The `*VERBOSE_LEVEL*` controls the amount of information given (each level print also the information from previous level). It is a one digit number:

- `>=1` Prints elements and isotopes of each material
- `>=2` Prints density effect data

There is a set of geometry utilities that are meant to help the user that is writing some C++ code to for example debug the geometry, get a touchable or a volume by name, etc. They are all in the `GmGeometryUtils` class, which is a singleton. To use them in your C++ code you can do it like in the following example:

```
GmGeometryUtils::GetInstance()->DumpG4LVList();
```

We list here the available methods, with an explanation of their functionality:

- `void DumpSummary(std::ostream& out = G4cout)`: Dumps a summary of the geometry, i.e. number of solids, logical volumes, physical volumes, touchables and materials
- `void DumpG4LVList(std::ostream& out = G4cout)`: Dumps list of logical volumes
- `void DumpG4LVTree(std::ostream& out = G4cout)`: Dumps the hierarchy of logical volumes
- `void DumpG4PVLVTree(std::ostream& out = G4cout)`: Dumps in the following order: #. a logical volume with details #. list of physical volumes that are daughters of this
 - logical volume with details
 - list of logical volumes daughters of this logical volume and for each go to 1
- `void DumpMaterialList(std::ostream& out = G4cout)`: Dumps list of materials
- `void DumpSolid(G4VSolid sol, size_t leafDepth, std::ostream& out = G4cout)*`: Dumps a solid with its attributes
- `void DumpLV(G4LogicalVolume lv, size_t leafDepth, std::ostream& out = G4cout)*`: Dumps a logical volume with its attributes
- `void DumpPV(G4VPhysicalVolume pv, size_t leafDepth, std::ostream& out = G4cout)*`: Dumps a physical volume with its attributes
- `G4LogicalVolume GetTopLV()*`: Gets a pointer to the logical volume on top of the geometry hierarchy
- `G4VPhysicalVolume GetTopPV()*`: Gets a pointer to the physical volume on top of the geometry hierarchy

- *G4Material* GetMaterial(const G4String& name, bool exists = 1)*: Gets the material with the given name
- *std::vector<G4LogicalVolume>* GetLogicalVolumes(const G4String& name, bool exists = 1)*: Gets the list of logical volumes with the given name
- *std::vector<G4VPhysicalVolume>* GetPhysicalVolumes(const G4String& name, bool exists = 1)*: Gets the list of physical volumes with the given name
- *std::vector<GmTouchable>* GetTouchables(const G4String& name, bool exists = 1)*: Gets the list of touchables with the given name.
- *std::set<G4String>* GetAllSDTypes(): Gets all distinct sensitive detector types

Magnetic and electric fields

To set a uniform magnetic field you can use the command

```
/gamos/field/setMagField FIELD_X FIELD_Y FIELD_Z
```

where *FIELD_X FIELD_Y FIELD_Z* are the field values along the three axes.

To set a uniform electric field you can use the command

```
/gamos/field/setElecField FIELD_X FIELD_Y FIELD_Z
```

where *FIELD_X FIELD_Y FIELD_Z* are the field values along the three axes.

To set a uniform electromagnetic field you can use the command

```
/gamos/field/setEMField MAG_FIELD_X MAG_FIELD_Y MAG_FIELD_Z ELEC_FIELD_X ELEC_FIELD_Y ELEC_FIELD_Z
```

where *MAG_FIELD_X MAG_FIELD_Y MAG_FIELD_Z* are the magnetic field values along the three axes and *ELEC_FIELD_X ELEC_FIELD_Y ELEC_FIELD_Z* are the electric field values along the three axes

Remember that in Geant4 internal units 1 Tesla is equal to 0.001; therefore if you do not use any unit it will be understood as 1, that is 1000 Teslas.

You can set a uniform magnetic field uniquely to a list of volumes. To do it use the command

```
/gamos/magneticField/setLocalField FIELD_X FIELD_Y FIELD_Z VOLUME_1 VOLUME_2 ... VOLUME_N
```

where *FIELD_X FIELD_Y FIELD_Z* are the field values along the three axes. And *VOLUME_1 VOLUME_2 ... VOLUME_N* are the list of volumes to which the magnetic field is attached.

If you need other types of fields different than uniform, use the *MagFieldManager* utility.

This is an utility that allows to define arbitrary magnetic fields and control their tracking precision parameters and also calculates the magnetic fields for different structures (circular coil, solenoid, ...) and stores them in a file for later used.

To use if you have to set the GAMOS parameter:

```
/gamos/setParam MagneticFieldModelManager:On 1
```

You can find all the relevant documentation in the file:

```
GAMOS.5.2.0/source/MagFieldManager/doc/user_manual_bfield_manager.pdf
```

In GAMOS it is already installed, so you may skip the installation instructions.

VISUALIZATION

3D Geant4 visualization

Several Geant4 drivers are available to visualize the geometry and tracks: *ASCII*, *DAWN*, *HepRep*, *RayTracer*, *OpenGL*, *VRML* and *Qt*. Most of these drivers require further software to be preinstalled on your system, for what we have not included them by default. They can be installed and used in GAMOS following the procedure to install them in Geant4 (see Geant4 Guide for Application Developers).

Three of the drivers that do not need any more software are VRML, OpenGL and ASCII. VRML can write a file in a format that can be viewed in a variety of commodity browsers, including some Web browsers. OpenGL provides fast graphics for interactive visualization. The ASCII driver dumps the geometry volumes as a list on the screen.

To visualize the geometry and tracks using these drivers it is only needed to add a few lines in your GAMOS script. An example of these lines can be found in the files in the examples directory: *visVRML2ILE.in*, *visOGLIX.in* and *visASCII.in*. Instead of copying the lines in these files in your script you can add or example the line

```
/control/execute PATH_TO_MY_GAMOS_DIRECTORY/examples/visVRML2FILE.in
```

The commands in *visOGLIX.in* will visualize the geometry interactively and superimpose the tracks created by the events as these are run. After the job finishes the drawing will end. The commands in *visVRML2FILE.in* file will write a file containing the geometry and a second one containing the geometry and tracks. The name of these files are *g4_MN.wrl*, where *MN* is a number that starts at *00* and increases if the file already exists; this means the first two files will be *g4_00.wrl* and *g4_01.wrl*, the second ones *g4_02.wrl* and *g4_03.wrl*, etc. VRML is the format supported by the World Web Consortium, and therefore are many available browsers that support its visualization. For example you can use *view3dscene* in Linux or *explorer3D* in Windows.

2D visualization

Often a 2D visualization serves to better understand some detail of the geometry better than a 3D visualization. The 2D visualization utility of GAMOS serves also to visualize some volumes that the Geant4 3D visualization is not able to see, specially complicated volume shapes, like a polygone or polycone or some boolean solid. Another common problem not detected by 3D visualization happens when there is a volume overlap, or a volume that is placed as daughter of a volume when it is indeed in another one (e.g. a crystal is placed on the world when it is contained in a detector); in this case the 3D visualization draws the volume while it will not be seen by the tracks.

The GAMOS 2D visualization is based on sending tracks. When a track crosses the boundary between two volumes, it draws a point, so that if tracks are sent close enough the boundaries will be seen as tracks. The user can choose to draw the geometry in any of the three planes: XY, XZ or YZ. Several options control the use of this utility:

```
/gamos/setParam GmGeneratorScanVis:ScanParams PLANE DISTANCE_BETWEEN_PLANES DIS-  
TANCE_BETWEEN_TRACKS LINES_ANGLE_1 LINES_ANGLE_2 .. LINES_ANGLE_N
```

Where *PLANE* can be *XY*, *XZ* or *YZ*. This command selects to draw planes separated by *DISTANCE_BETWEEN_PLANES* and in each plane the lines to find the volume boundaries will be send separated by *DISTANCE_BETWEEN_TRACKS* covering the full plane in *N* sets at angles *LINES_ANGLE_1* *LINES_ANGLE_2* .. *LINES_ANGLE_N*.

If the plane selected is for example *XY*, you have them to define the limits of the picture with the paramters:

```
/gamos/setParam GmGeneratorScanVis:ScanWorldMinX MIN_X /gamos/setParam GmGeneratorScan-  
Vis:ScanWorldMaxX MAX_X /gamos/setParam GmGeneratorScanVis:ScanWorldMinY MIN_Y /gamos/setParam  
GmGeneratorScanVis:ScanWorldMaxY MAX_X
```

and you have also to define how many *XY* planes will be drawn by defining the limits in *Z*:

```
/gamos/setParam GmGeneratorScanVis:ScanWorldMinZ MIN_Z /gamos/setParam GmGeneratorScan-  
Vis:ScanWorldMaxZ MAX_Z
```

Some options are to define the number of pixels of the figure (what will allow a finer visualization of close lines:

```
/gamos/setParam GmGeneratorScanVis:NPixels NPIXELS
```

And you can choose to print superimposed to each picture the names of the volumes that correspond to each boundary, a different colour for each boundary:

```
/gamos/setParam GmGeneratorScanVis:DrawBorderNames 1
```

Finally you have to set the primary generator:

```
/gamos/generator GmGeneratorScanVis
```

An example of commands are the following ones:

```
*/gamos/setParam GmGeneratorScanVis:DrawBorderNames 1 /gamos/setParam GmGeneratorScanVis:NPixels 1000  
/gamos/setParam GmGeneratorScanVis:ScanWorldMinX -110 /gamos/setParam GmGeneratorScan-  
Vis:ScanWorldMaxX 0 /gamos/setParam GmGeneratorScanVis:ScanWorldMinY -10 /gamos/setParam GmGen-  
eratorScanVis:ScanWorldMaxY 10 /gamos/setParam GmGeneratorScanVis:ScanWorldMinZ -420 /gamos/setParam  
GmGeneratorScanVis:ScanWorldMaxZ -300 /gamos/setParam GmGeneratorScanVis:ScanParams YZ 25. 0.01  
10.*deg 90.*deg
```

```
/gamos/generator GmGeneratorScanVis*
```

GENERATOR

GAMOS offers several primary generators: the general one, which allows you to select among many particles and generator distributions, one that reads the primary particles from a text file and another one that reads them from a binary file. Alternatively you may create your generator in the standard Geant4 way, by writing your C++ class inheriting from *G4VUserPrimaryGeneratorAction*.

Using GAMOS generator

Introduction

The GAMOS generator provides several time, energy, position and direction distributions that the user may combine to his/her will. The user can select to generate as primary particles one or several single particles together with one or several isotopes and set any of the available time, energy, position or direction distributions for each one of the single particles or isotopes.

We describe below the commands to select the single particles, the commands to select the isotopes and then the commands for selecting the time, energy, position and direction distributions.

The first command you have to use is the one that tells GAMOS that you want to use the GAMOS generator:

```
/gamos/generator GmGenerator
```

This command, apart from instantiating the GAMOS generator manager, also instantiates the messenger. Therefore, if you forget to write this command first, the other generator commands described below will not exist and Geant4 will throw an exception.

After this command you have to add one or several particle sources (there is no default primary particle source, therefore if you do not choose one, GAMOS will throw an exception). You may combine several particle sources in each event by repeatedly using the commands described below. Each type of particle source has default distributions of time, energy, position and direction, that you may change with the commands described below.

An important point not to forget is that when you define a particle source you have to give it a name. This name serves to distinguish it when you want later to change its time, energy, position or direction distribution.

Particle sources

Single particle source

To add a single particle source you have to use the command

```
/gamos/generator/addSingleParticleSource SOURCE_NAME PARTICLE_NAME ENERGY
```

SOURCE_NAME is the name of this source, that you have to use if you want later to change its time, energy, position or direction distribution.

PARTICLE_NAME can be any of the Geant4 particles

For a list of the names of the available particles use the Geant4 command */run/particle/dumpList* or see Appendix.

ENERGY is the initial energy of the particle.

If you don't change any property the particle will be generated at time 0., position (0,0,0) and random direction.

If you are using optical photons as primary generator particles, you may set the polarization using the parameter:

/gamos/setParam SOURCE_NAME:Polarization POLARIZ_X POLARIZ_Y POLARIZ_Z

Isotope source

GAMOS implements an isotope generator, that simulates the activity of different isotopes that decay in one or several photons, electrons or positrons. First a file is read with the description of the isotope decays in a format as the one that can be found at *MY_GAMOS_DIR/data/isotopes.dat*, part of which we reproduce here

```
:ISOTOPE Na22
215.5  0.905  e+
1275.0  0.9995  gamma
:ISOTOPE F18
249.8  0.967  e+
```

For each isotope there must be a first line starting by *:ISOTOPE* and followed by the isotope name. Then there is a line for each of the possible isotope decays with three columns describing the decay particle energy, the probability of the decay and the particle type. This file contains the most common isotopes in medical physics. If you want to use another isotope you can add it following the format described above.

If you selected this generator, each event will be generated with one of several primary particles in the decay list of each of the selected isotopes. The presence of each decay particle will occur following its corresponding probability.

To choose an isotope as particle source you have to use the command:

/gamos/generator/addIsotopeSource SOURCE_NAME ISOTOPE_NAME ACTIVITY

SOURCE_NAME is the name of this source, that you have to use if you want later to change its time, energy, position or direction distribution.

ISOTOPE_NAME is one of the isotopes read from the input file,

ACTIVITY is the activity you want to set for that isotope

The available units in Geant4 are becquerel and curie.

You may choose several isotopes by repeatedly writing this command.

This command triggers the reading of the file named *“isotopes.dat”* if it has not been read yet. You may change the name of this file with the command

/gamos/setParam Generator:Isotope:FileName MY_FILENAME

To learn how to change the directory list where GAMOS looks for this file, please read the section *“Managing the input data files”*.

If you don't change any property the particle will be generated with a time distribution of type *“decay”* (see below), energy distribution of type *“constant isotope decay”* (see below), position at (0,0,0) and random direction.

Double back to back particle source

This is a special source that generates two identical particles back to back, i.e. with the same position, energy and time but opposite directions. To select it you have to use the command

```
/gamos/generator/addDoubleBackToBackParticleSource SOURCE_NAME PARTICLE_NAME ENERGY
```

SOURCE_NAME is the name of this source, that you have to use if you want later to change its time, energy, position or direction distribution.

PARTICLE_NAME can be any of the Geant4 particles

For a list of the names of the available particles use the Geant4 command */run/particle/dumpList* or see Appendix.

ENERGY is the initial energy of the particle.

If you don't change any property the particle will be generated at time 0., position (0,0,0) and random direction.

Distributions

Time distributions

- *Constant time* */gamos/generator/timeDist SOURCE_NAME GmGenerDistTimeConstant TIME* All the primary particles will be generated at time 0. If you want to set it at a different time, you can add the extra parameter *TIME*.
- *Time changing at constant interval* */gamos/generator/timeDist SOURCE_NAME GmGenerDistTimeConstantChange TIME_INTERVAL TIME_OFFSET* The time will increase for each event with a constant value. The interval is defined as an extra parameter. If desired a second parameter can be added to set the offset (the time of the first event); if the offset is not set, it is set to 0.
- *Decay time* */gamos/generator/timeDist SOURCE_NAME GmGenerDistTimeDecay ACTIVITY LIFETIME* The primary particles will be generated with a time following a typical decay distribution, with the activity of the particle source. To be concrete the time is sampled with a Poisson distribution that is obtained as follows: $rnd_poiss = -(1.0 / (activity/second)) * \log(RandFlat::shoot());$ the activity is given by the parameter *ACTIVITY*. If the second parameter is set (is is not mandatory) the activity is scaled with the lifetime chosen.

Energy distributions

- *Constant energy* */gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyConstant ENERGY* All the primary particles will be generated with energy given by the parameter *ENERGY*.
- *Constant decay energy* */gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyConstantIsotopeDecay* All the primary particles will be generated with energy given by the energy of the isotope decay selected by the isotope source, as read from the file "*isotopes.dat*".
- *Random flat energy* */gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyRandomFlat MIN_ENERGY MAX_ENERGY* The primary particles will be generated with an energy given by a random distribution between the minimum and maximum energy.
- *Beta decay energy* */gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyBetaDecay* The energy will be sampled following the energy distribution of the decay of the isotope This distribution can not be used for single particle sources. The energy distribution will be read from a file called "*EnergyDist.*+*source_particle_name*+*BetaMinus.dat*" or "*EnergyDist.*+*source_particle_name*+*BetaPlus.dat*". The data is taken from the <http://nucleardata.nuclear.lu.se/toi/>; goto '*Nuclide search*' and save the table '*Beta Spectrum*'. There are several examples at *MY_GAMOS_DIR/data/EnergyDist.XXX.dat*.

- *Gaussian* /gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyGaussian MEAN SIGMA Primary particles will be generated with an energy given by the gaussian distribution of mean MEAN and sigma SIGMA.
- *Energy probabilities from file* /gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyFromFile FILE_NAME CALCULATION_TYPE UNIT This distribution permits to use an arbitrary energy distribution. The energies and probabilities are read from the file named FILENAME. This file contains a list of lines with two words each: ENERGY PROBABILITY The energies are given in MeV (the default Geant4 unit), if no unit is provided.

This data can be interpreted in four different ways:

- If the second parameter is *fixed* the energies used are only those listed. For example if you want a quarter of your primary particles with energy 0.5 MeV and three quarters with energy 1. MeV, you can write the following file:

```
0.5 0.5
1. 0.5
```

- If the second parameter is *histogram* (the default value if the second argument is not provided) the data read will be interpreted as that of a constant-bin histogram: the energies are the values of the centre of each histogram bin and the energies will be randomly distributed (with the corresponding probability) in the histogram bin. In this case if the difference between data points is not constant, an exception will be thrown. For example if you want a quarter of your primary particles uniformly distributed between 0 and 1 MeV and three quarters between 1 and 2 MeV, you can write the following file:

```
0.5 0.25
1.5 0.75
```

- If the second parameter is *interpolate* the data read will be interpreted as that of an histogram, with constant or non-constant bin. As the bins may be not equal, the behaviour has to be different than for the 'histogram' case above: one cannot use the bin centres and calculate the bin limits, but the bin limits have to be explicitly provided. In other words, the energies given will be the limits of the bins and the probability read together with an energy is the probability between the energy and the next energy. We recommend then that the last energy has probability 0. For example if you want a quarter of your primary particles uniformly distributed between 0 and 1 MeV and three quarters between 1 and 2 MeV, you can write the following file:

```
0. 0.25
1. 0.75
2. 0.
```

- If the second parameter is *interpolate_log* the data read will be interpreted as that of an histogram, with constant or non-constant bin. The behaviour is the same as for the 'interpolate' case, but the energies are used logarithmically, i.e. the energy will be constantly distributed inside an energy bin taking the logarithm of the energies. For example if you want a quarter of your primary particles logarithmically uniformly distributed between 0 and 1 MeV and three quarters between 1 and 10 MeV, you can write the following file:

```
0. 0.25
1. 0.75
10. 0.
```

- *Isolethargical* /gamos/generator/energyDist SOURCE_NAME GmGenerDistEnergyIsolethargical ENERGY_MIN ENERGY_MAX The energy will be sampled following a distribution that is flat in the logarithm of the energy

The third argument serves to define which is the unit of the energies in the file. If not provided, it will be taken as MeV (= 1.).

Position distributions

- *Position at a point* `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionPoint POS_X POS_Y POS_Z` All primary particles are generated at the same position. If no extra argument is given the point is (0.,0.,0.).
- *Position in a Geant4 volume* `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInG4Volumes LV_NAME1 LV_NAME2 ...` The position is distributed randomly inside one or several volumes of the Geant4 geometry. The user must add as extra parameters the list of volume names. The volumes can be logical volumes, physical volumes or touchables For understanding the notation to identify touchables in GAMOS, see *Identifying touchables* section.. This distribution can only be used if the volumes are G4Box, G4Orb, G4Sphere, G4Ellipsoid, G4Tubs, G4Cons or G4Polycone Other volume types may be added at user request. If you want to use it for any other volume shape, you have to use the distribution `By default the positions are distributed in the volumes selected without taking care if they contain other volumes inside. If you want to consider only the space that is not filled by daughter volumes you have to add the parameter: /gamos/setParam GmVGenerDistPositionVolumesAndSurfaces:OnlyVolume 1` `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInG4VolumesGeneral TOUCHABLE_NAME1 TOUCHABLE_NAME2 ...` This distribution works with any solid shape, including boolean solids, but it is in general quite slower than the previous one (it creates a random position in the whole world volume and then looks if it is in one of the selected volumes, which can be quite slow if the volume dimensions are small). It is important to realize, that this distribution uses touchable names, not volume names; this means that you have to put for examples `world#1:/MyVolA#2`, instead of `MyVolA`
- *Position in a user defined volume* `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInUserVolumes POS_X POS_Y POS_Z ANG_X ANG_Y ANG_Z SOLID_TYPE SOLID_DIMENSIONS` The particles are randomly distributed inside a volume defined by the user (it does not need to be a real volume in the geometry). The user must provide the definition of the volume as extra parameters. `SOLID_TYPE` can be `Orb`, `Sphere`, `Ellipsoid`, `Tubs`, `Box`. `SOLID_DIMENSIONS` are the solid dimensions. For the order and meaning of the solid dimensions, please look at the corresponding Geant4 solid. By default the volume is placed at position (0.,0.,0). If you want it placed at a different position you have to add three optional extra parameters `POS_X POS_Y POS_Z`. By default the volume is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) `ANG_X ANG_Y ANG_Z`. Those angles are interpreted rotating the volume first around the X axis, then around the Y axis and finally around the Z axis (around the (0,0,0) point, after the displacement is done). By default the positions are distributed in the volumes selected without taking care if they contain other volumes inside. If you want to consider only the space that is not filled by daughter volumes you have to add the parameter: `/gamos/setParam GmVGenerDistPositionVolumesAndSurfaces:OnlyVolume 1`
- *Position in a Geant4 volume surface* `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInG4Surfaces LV_NAME1 LV_NAME2 ...` The position is distributed randomly in the surface of one or several volumes of the Geant4 geometry. The user must add a number of extra parameters with the list of volume names. The volumes can be physical volumes or touchables For understanding the notation to identify touchables in GAMOS, see *Identifying touchables* section.. This distribution can only be used if the volumes are `G4Box`, `G4Orb`, `G4Sphere`, `G4Tubs` or `G4Cons` Other volume types may be added at user request.
- *Position in a user defined volume surface* `/gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInUserSurfaces POS_X POS_Y POS_Z ANG_X ANG_Y ANG_Z SOLID_TYPE SOLID_DIMENSIONS` The particles are randomly distributed in a volume surface of a volume defined by the user. The user must provide the definition of the volume as extra parameters. `SOLID_TYPE` can be `Box`, `Orb`, `Sphere`, `Tubs`, `Cons`. `SOLID_DIMENSIONS` are the solid dimensions. For the order and meaning of the solid dimensions, please look at the corresponding Geant4 solid. By default the volume is placed at position (0.,0.,0). If you want it placed at a different position you have to add three optional extra parameters `POS_X POS_Y POS_Z`. By default the volume is not rotated. If you want it rotated you have to add three optional extra parameters (and always add

the three positions, even if they are (0.,0.,0.)) *ANG_X ANG_Y ANG_Z*. Those angles are interpreted rotating the volume first around the X axis, then around the Y axis and finally around the Z axis (around the (0,0,0) point, after the displacement is done).

- Adding an extra volume If you have defined a distribution of type *GmGenerDistPositionInG4Volumes*, *GmGenerDistPositionInUserVolumes*, *GmGenerDistPositionInG4Surfaces* or *GmGenerDistPositionInUserSurfaces* you can add more volumes with the following user command. */gamos/generator/GmPositionVolumesAndSurfaces/addVolumeOrSurface SOURCE_NAME LV_NAME1 LV_NAME2* for the Geant4 volumes or */gamos/generator/GmPositionVolumesAndSurfaces/addVolumeOrSurface SOURCE_NAME POS_X POS_Y POS_Z ANG_X ANG_Y ANG_Z SOLID_TYPE SOLID_DIMENSIONS* for the user-defined volumes. The primary particles will be distributed equally in the volumes proportionally to their volume or surface.
- *Position in steps along a line* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionLineSteps POS_X POS_Y POS_Z DIR_X DIR_Y DIR_Z STEP* The position is distributed uniformly along a line starting at position *POS_X POS_Y POS_Z* and with direction given by the three director cosines *DIR_X DIR_Y DIR_Z*. Each event is generated in a different point along the line, starting at the initial position, and in steps given by *STEP*.
- *Position in a square* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionSquare HALF_WIDTH POS_X POS_Y POS_Z DIR_X DIR_Y DIR_Z* The position is randomly distributed in a 2D square in the XY plane of half width *HALF_WIDTH* at position (0.,0.,0.). If you want it placed at a different position you have to add three optional extra parameters *POS_X POS_Y POS_Z*. By default the square is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) *DIR_X DIR_Y DIR_Z*. Those are the director cosines of the Z axis of the square (the axis perpendicular to the 2D surface).
- *Position in a rectangle* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionRectangle HALF_WIDTH_X HALF_WIDTH_Y POS_X POS_Y POS_Z DIR_X DIR_Y DIR_Z* The position is randomly distributed in a 2D rectangle in the XY plane of half widths *HALF_WIDTH_X* in X and *HALF_WIDTH_Y* in Y at position (0.,0.,0.). If you want it placed at a different position you have to add three optional extra parameters *POS_X POS_Y POS_Z*. By default the rectangle is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) ******. Those are the director cosines of the Z axis of the rectangle (the axis perpendicular to the 2D surface).
- *Position in a disc* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionDisc RADIUS POS_X POS_Y POS_Z DIR_X DIR_Y DIR_Z* The position is randomly distributed in a disc in the XY plane of radius *RADIUS* at position (0.,0.,0.). If you want it placed at a different position you have to add three optional extra parameters *POS_X POS_Y POS_Z*. By default the cylinder is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) *DIR_X DIR_Y DIR_Z*. Those are the director cosines of the Z axis of the cylinder (the axis perpendicular to the 2D surface).
- *Position in a disc with gaussian distribution* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionDiscGaussian SIGMA POS_X POS_Y POS_Z DIR_X DIR_Y DIR_Z* The position is distributed in a disc in the XY plane with the radius in a gaussian distribution of sigma *SIGMA* and random in phi, at position (0.,0.,0.). If you want it placed at a different position you have to add three optional extra parameters *POS_X POS_Y POS_Z*. By default the cylinder is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) *DIR_X DIR_Y DIR_Z*. Those are the director cosines of the Z axis of the cylinder (the axis perpendicular to the 2D surface).
- *Position randomly in the voxels of a phantom* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionVoxelPhantomMaterials MATERIAL1 MATERIAL2 ...* The position is randomly distributed in the voxels of a phantom with material equal to one of the materials in the list of parameters. There must be at least one volume defined by a parameterisation of type *G4PhantomParameterisation*.
- *Position in the voxels of a phantom file* */gamos/generator/positionDist SOURCE_NAME GmGenerDistPositionInVoxelsFromFile FILE_NAME POS_X POS_Y POS_Z ROT_X ROT_Y ROT_Z* The voxels are read from a file with a format similar to the Geant4 DICOM files (the main difference being that there are no materials): the first

line contains the number of voxels in X, Y and Z; the second one the lower and upper limits in X, then in Y and then in Z; then for each voxel it comes a number representing the relative probability to contain the position. By adding three extra parameters you may displace the phantom. By adding three more you may rotate it; the parameters correspond to the rotations around the X, the Y and the Z axis, in this order.

Direction distributions

- *Random distribution* `/gamos/generator/directionDist SOURCE_NAME GmGenerDistDirectionRandom` The primary particles will be generated in a random distribution so that each solid angle receives the same number of particles.
- *Constant distribution* `/gamos/generator/directionDist SOURCE_NAME GmGenerDistDirectionConst` The primary particles will be generated all in the same direction, given by the extra parameters `DIR_X DIR_Y DIR_Z`.
- *Cone distribution* `/gamos/generator/directionDist SOURCE_NAME GmGenerDistDirectionCone` The primary particles will be generated in a random distribution around a cone, given by the extra parameters `DIR_X DIR_Y DIR_Z OPENING_ANGLE`, so that each solid angle receives the same number of particles.
- *Gaussian along an axis* `/gamos/generator/directionDist SOURCE_NAME GmGenerDistDirectionGaussian` The primary particles will be generated close a line, where the two perpendicular directions are sampled with a Gaussian distribution. The parameters are `DIR_X DIR_Y DIR_Z SIGMA_Y SIGMA_Z`. The direction is first selected along the X axis, the Y and Z are them sampled with two Gaussian distributions centered at 0. and with widths given by `SIGMA_Y SIGMA_Z` and them the direction is rotated with the angles given by `DIR_X DIR_Y DIR_Z`.

Position and direction distributions

It is also possible to create distributions where several of the four variables (time, energy, position and direction) are generated at the same time, so that they are related. You have nevertheless to keep in mind that the order of calling will be time, position, energy and direction distributions.

In case you need a different order, for example if the position is determined by the value of the direction, you can calculate both the direction and the position in the `GeneratePosition` method, return only the position and keep the direction in a class data so that it can be returned in the `GenerateDirection` method.

- *Position in volume surface, pointing towards centre* `GmGenerDistPositionDirectionInVolumeSurface SOLID_TYPE SOLID_DIMENSIONS POS_X POS_Y POS_Z ANG_X ANG_Y ANG_Z` The position is distributed in the surface of a volume defined by the user. The user must provide the definition of the volume as extra parameters. `SOLID_TYPE` can be of type `Box`. `SOLID_DIMENSIONS` are the solid dimensions. By default the volume is placed at position (0.,0.,0). If you want it placed at a different position you have to add three optional extra parameters `POS_X POS_Y POS_Z` By default the volume is not rotated. If you want it rotated you have to add three optional extra parameters (and always add the three positions, even if they are (0.,0.,0.)) `ANG_X ANG_Y ANG_Z`. Those angles are interpreted as rotating the volume first around the X axis, then around the Y axis and finally around the Z axis (around the (0,0,0) point, after the displacement is done). The direction is taken as the line that goes from the position to the point (0.,0.,0.).
- *Position in a point, pointing towards a box* `GmGenerDistPositionDirectionInVolumeSurface POINT_X POINT_Y POINT_Y TOUCHABLE_NAME_1 TOUCHABLE_NAME_2 ...` All particles are generated at a fixed point, pointing towards a random point of the touchables (which must be of type `G4Box`).

As this distribution is of position and direction types at the same time, you have to use the two commands

```
/gamos/generator/positionDist
```

```
/gamos/generator/directionDist
```

You can see editing the file

GamosCore/GamosGenerator/src/GmGenerDistPositionDirectionInVolumeSurface.cc that internally the method *GenerateDirection* knows the position by interrogating the source.

Creating your own distribution

If you want to use a time, energy, position or direction distribution that is not foreseen in GAMOS, you can easily create your own one. Let's see as example the creation of a time distribution randomly distributed between two values.

You have to create your class inheriting from the class *GmVGenerDistTime* (see for example the class *GmGenerDistTimeConstant*)

```
virtual G4double GenerateTime(const GmParticleSource* source);
```

that will return the time value for each event. If you want the user to be able to input some parameters of your class from the command line, like the minimum and maximum time, you have to implement the method

```
virtual void SetParams(const std::vector <
G4String > & params);
```

This method will be called automatically passing the extra parameters in the command line selecting your distribution.

Last, you have to transform your distribution into a plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmGenerDistTimeFactory*, or see the plug-in tutorial.

Then you can choose that any of the particle sources use your new distribution in any GAMOS job by adding the command line

```
/gamos/generator/directionDist SOURCE_NAME MyTimeDist
```

where *MyTimeDist* is the name you chose when defining your plug-in (which could be different from the name of the class itself).

Reading your generator particles from a text file

You can also define your event primary particles with a text file. The format of the input file is the following:

Each line corresponds to a primary particle, with the following variables:

- EventID
- Particle
- PosX
- PosY
- PosZ
- MomX
- MomY
- MomZ
- Time
- Weight

The lines will be read one by one and all the particles that have the same event ID will be simulated together in an event.

To select this generator you have to use the command

/gamos/generator GmGeneratorFromTextFile

By default the file to be read is called ‘‘generator.txt’’, you can see an example at MY_GAMOS_DIR/data/generator.txt. You can change its name with the command

/gamos/setParam GmGeneratorFromTextFile:FileName MY_FILENAME

To change the search path please read the section ‘‘Managing the input data files’’.

A file of this type may be created with the user action *GmTrackDataTextFileUA* using the following list of data:

EventID Particle InitialPosX InitialPosY InitialPosZ InitialMomX InitialMomY InitialMomZ InitialTime InitialWeight

You may skip the first events, by setting the parameter:

/gamos/setParam GmGeneratorFromTextFile:NEventsSkip NEVENTS

Reading your generator particles from a binary file

You can also define your event primary particles with a binary file, with the following data per particle:

- EventID
- Particle
- PosX
- PosY
- PosZ
- MomX
- MomY
- MomZ
- Time
- Weight

To select this generator you have to use the command

/gamos/generator GmGeneratorFromBinFile

By default the file to be read is called ‘‘generator.bin’’.

You can change its name with the command

/gamos/setParam GmGeneratorFromBinFile:FileName MY_FILENAME

To change the search path please read the section ‘‘Managing the input data files’’.

A file of this type may be created with the user action *GmTrackDataBinFileUA* using the following list of data:

EventID Particle InitialPosX InitialPosY InitialPosZ InitialMomX InitialMomY InitialMomZ InitialTime InitialWeight

You may skip the first events, by setting the parameter:

/gamos/setParam GmGeneratorFromBinFile:NEventsSkip NEVENTS

Event generator changing energy and material

As you can find in some of the tutorial exercises this generator serves to scan different energies and different materials and obtain for example the cross sections at each point, the activation, or whatever quantity you want.

To select it you have to use the command:

```
/gamos/generator GmGeneratorChangeEnergyAndMaterial
```

The first thing to choose is the energies. To do it a minimum a maximum energy have to be defined as well as a number of steps:

```
/gamos/setParam GmGeneratorChangeEnergyAndMaterial:minE VALUE
```

```
/gamos/setParam GmGeneratorChangeEnergyAndMaterial:maxE VALUE
```

```
/gamos/setParam GmGeneratorChangeEnergyAndMaterial:nstepsE VALUE
```

Then it must be selected if the steps are done linearly or logarithmically (default)

```
/gamos/setParam GmGeneratorChangeEnergyAndMaterial:logE I/O
```

The materials used will be those found from the geometry constructed. You may define a dummy geometry, as it will not be used anyhow. For each material at a time a world made of one single volume of the material will be used. You may nevertheless select only a subgroup of the materials in the geometry with the parameter:

```
/gamos/setParam GmGeneratorChangeEnergyAndMaterial:VolumesToChange VOLUME_1 VOLUME_2 ...
```

Event generator histograms

These are histograms of event generator particles that may serve to check that you have actually generated what you meant.

The names of all these histograms start with *GmGenerHistosUA*: and they are all dumped into the file *gener.root/csv*. The following histograms are produced:

- Kinetic energy of primary particles ("KinEnergy")
- X position ("Position X")
- Y position ("Position Y")
- Z position ("Position Z")
- Theta angle ("Angle theta")
- Phi angle ("Angle phi")
- Difference between consecutive event times (taken as time of the first primary particle) ("Time between source decays (ns)")

To activate this user action use the command:

```
/gamos/userAction GmGenerHistosUA
```

Biasing generator distributions

If you are using the GAMOS generator, *GmGenerator*, it is possible to bias a primary generator distribution multiplying each resulting value by a probability of occurrence. To do this you have first to define a numeric distribution (see section on *Distributions*), where you assign the probabilities for each resulting value of the generator distribution.

These probabilities are normalized so that the highest is assigned a value of 1. If the generator variable takes a value to which it corresponds a probability value X that is less than 1, then Russian roulette is played with a survival probability of X ; if the value is not accepted, a new value is sample from the generator distribution, if it is accepted, the weight of the particle that is going to use this generator distribution is multiplied by $1/X$.

Several generator distribution variables can be biased, namely: *PosX*, *PosY*, *PosZ*, *PosPerp*, *PosR*, *PosPhi*, *PosTheta*, *DirTheta*, *DirPhi*, *Energy*, *Time*. To bias a distribution you have to use the command:

```
/gamos/generator/addBiasDistribution SOURCE_NAME GENERATOR_DIST_TYPE BIAS_DIST_NAME
```

where *SOURCE_NAME* is the name of a source declared previously, *GENERATOR_DIST_TYPE* is one of the generator distribution variables listed above and *BIAS_DIST_NAME* is the name of the distribution you have previously created (see section on *Distributions*).

A word of caution is due here: if you use low probabilities, the Russian roulette will often fail and then the efficiency of your simulation may be sensibly decreased. In this case you may think on using a generator distribution that includes the bias in an efficient way or create a new one (there is an example on creating a new generator distribution in the *Plugin tutorial*; you may ask for help in the *GAMOS Discussion Forum* if you do not feel confident on doing it). An example of this can be using *GmGenerDistPositionDiscGaussian* instead of *GmGenerDistPositionDisc* and biasing with a Gaussian distribution the *PosPerp* variable.

The word *bias* may be misleading, as indeed the result is not biased. Thanks to the multiplication of the particle's weight by $1/X$ mentioned above the result will not change within statistical fluctuations if you are using some variable that takes into account weights. This is not the case for example if you are using sensitive detectors and hits, so we do not recommend its use for those applications.

It may happen that the distribution has some bins with very low probabilities, so that the search for a valid value can be too long. In this case you will get a warning so that you may reconsider if your biasing is indeed gaining CPU or not. The default value for the number of loops in searching for a bias value is 10000, if you want to change it, you may use the parameter:

```
/gamos/setParam SOURCE_NAME:MaxBiasIterations VALUE
```

Building your generator with C++

You can build your generator by writing your C++ class inheriting from *G4VUserPrimaryGeneratorAction* (see example in [\[biblio.g4doc-generator\]](#)).

After that you have to transform it into a GAMOS plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmGeneratorFactory*, or see the example *examples/N02*.

Using ions

Geant4 uses any kind of ion, whether in its ground state or in an excited state. Ions are named with the element symbol, followed by the atomic mass number and the excitation energy between brackets, e.g. *Fe56[0.0]*, *Na22[0.0]*, *Co57[136.5]* (note that the ground state is named as *0.0*).

There two ways two use ions as primary particles in GAMOS generator. The first one is defining an isotope source, as described above. In this case the isotope itself will not be created, but only its decay products, as defined in the *isotopes.lis* file. In this way you have a direct access to the position, direction, energy or time distributions of these particles, what may increase your simulation efficiency. The alternative way is to define a single particle source and use an ion name as particle. Geant4 will create this ion and will use its radioactive decay utility to decay it (and follow the decay chain until stable ions are produced).

Do not forget that if you want to use ions, you have to define a physics list that creates the physics for them. This means that you cannot use for example *GmEMPhysics*, but you may use *GmExtendedEMPhysics* or any other physics

where ions are created; and if your physics list does not include by default radioactive decay (like in the case of *GmExtendedEMPhysics*) you have to add this physics:

/gamos/physics/addPhysics radioactiveDecay

You can use the GAMOS physics list, use one of the Geant4 physics lists or write your own one, following the standard Geant4 way, i.e. by writing your C++ class inheriting from `G4VUserPhysicsList`.

GAMOS electromagnetic physics lists

Basic electromagnetic physics list

The GAMOS basic electromagnetic physics list defines electromagnetic particles: photons, electrons, positrons and optical photons. To use this physics list you have to use the command:

```
/gamos/physicsList GmEMPhysics
```

This physics list instantiates the most common processes for gammas, electrons and positrons, and also for optical photons. For gammas and electrons uses the low energy (based on Livermore data) models, while for positrons uses the standard models (no low energy model exists). The detail of the processes and models used, plus the options of the models for which the defaults values are changed is the following (see Geant4 documentation for a better understanding of the physics of each process and model):

- *gamma*:
 - photo-electric process: *G4PhotoElectricEffect* using model *G4LivermorePhotoElectricModel* below 1 GeV and *G4PEEffectFluoModel* above.
 - Compton scattering process: *G4ComptonScattering* using model *G4LivermoreComptonModel* below 1 GeV and *G4KleinNishinaCompton* above.
 - Gamma conversion process: *G4GammaConversion* using model *G4LivermoreGammaConversionModel* above 1 GeV and *G4BetheHeitlerModel* above.
 - Rayleigh scattering process: *G4RayleighScattering* using model *G4LivermoreRayleighModel* below 1 GeV and none above.
- *electron*:
 - ionisation process: *G4eIonisation* using *G4LivermoreIonisationModel* with *G4UniversalFluctuation* below 1 GeV and *G4MollerBhabhaModel* above. The *RoverRange* and the *finalRangeRequested* of the step function are set to 0.2 and 0.1 mm.
 - bremsstrahlung process: *G4eBremsstrahlung* using *GmLivermoreBremsstrahlungModel* below 1 GeV and *G4eBremsstrahlungRelModel* above.
- *positron*:
 - ionisation process*: *G4eIonisation* using *G4MollerBhabhaModel*.

- bremsstrahlung process: *G4eBremsstrahlung* using model *G4SeltzerBergerModel* below 1 GeV and *G4eBremsstrahlungRelModel* above.
- multiple scattering process: *G4VMultipleScattering* using model *G4UrbanMscModel*.
- annihilation process: *GmAcollinearEplusAnnihilation* using the *G4eeToTwoGammaModel*, but to properly treat the acollinearity of the two gammas emitted the distribution of the angular difference is given by Colombino et al. [*biblio.Colombino*]. The default Geant4 behaviour can be used if the parameter: */gamos/setParam GmPhysicsPositronStandard:ePlusAcollinearAnnihilation 0*:

- *opticalphoton*:

- scintillation: this process is activated for all particles, except short-lived ones. Process class is *G4Scintillation* with yield factor 1. the option to track secondaries before the primary track ends is activated by default. It may be changed with the parameter: */gamos/setParam GmPhysicsOpticalPhoton:TrackSecondariesFirst 0*
- optical absorption: *G4OpAbsorption*.
- optical rayleigh: *G4OpRayleigh*
- optical boundary process: *G4OpBoundaryProcess* with *unified* model.

The default physics models can be overridden by choosing among the three categories that Geant4 proposes for electromagnetic interactions: standard, low energy or Penelope models. This can be done with the command

/gamos/GmPhysics/replacePhysics PHYSICS_MODEL_NAME

where *PHYSICS_MODEL_NAME* is one of the names in the list below. The default options (those explained above) are underlined>.

The following physics models are available:

- *gammas*:

- **gamma-lowener**: low energy, based on Livermore Evaluated Particle Data Library
- **gamma-standard**: standard electromagnetic processes. photo-electric uses *G4PEEffectFluoModel*, Compton scattering uses *G4KleinNishinaCompton*, gamma conversion uses *G4BetheHeitlerModel*. No Rayleigh scattering.
- **gamma-penelope**: processes a' la Penelope [*biblio.penelope*]. Below 1 GeV photo-electric uses *G4PenelopePhotoElectricModel*, Compton scattering uses *G4PenelopeComptonModel*, gamma conversion uses *G4PenelopeGammaConversionModel*, above 1 GeV the standard models are used. Rayleigh scattering uses *G4PenelopeRayleighModel*.

- *electrons*

- **electron-lowener**: low energy, based on Livermore Evaluated Particle Data Library.
- **electron-standard**: standard electromagnetic processes. Ionisation uses *G4MollerBhabhaModel*, bremsstrahlung uses *G4SeltzerBergerModel* below 1 GeV and *G4eBremsstrahlungRelModel* above, multiple scattering uses *G4UrbanMscModel*.
- **electron-penelope**: processes a' la Penelope. Below 1 GeV ionisation uses *G4PenelopeIonisationModel* and bremsstrahlung uses *G4PenelopeIonisationModel* and bremsstrahlung, above 1 GeV the standard models are used. Multiple scattering uses *G4UrbanMscModel*.

- *positrons*

- **positron-standard**: standard electromagnetic processes
- **positron-penelope**: processes a' la Penelope
- **positron-penelope**: processes a' la Penelope

For details on the physics implemented in each of these physics processes and models, please read the Geant4 physics manual [*biblio.g4doc-physics*].

This physics list also switches on the atomic deexcitation, see section below.

GAMOS electromagnetic extended physics list

This physics list can be selected with the command

```
/gamos/physicsList GmEMExtendedPhysics
```

It creates all GEANT4 particles, using the constructors *G4BosonConstructor*, *G4LeptonConstructor*, *G4MesonConstructor*, *G4BaryonConstructor*, *G4IonConstructor*, *G4ShortLivedConstructor*. For these particles the physics implemented is the following (see GEANT4 Physics Reference manual for a detailed explanation of the physics options).

- *mu+*, *mu-* *G4MuMultipleScattering*, with *G4WentzelVIModel*, *G4MuIonisation* with *SetStepFunction(0.2, 50*um)*, *G4MuBremsstrahlung*, *G4MuPairProduction* and *G4CoulombScattering*
- **alpha*, *He3***G4hMultipleScattering*, *G4ionIonisation* with *SetStepFunction(0.2, 50*um)* and *G4NuclearStopping*
- **GenericIon***G4hMultipleScattering*, *G4ionIonisation* with model *G4IonParametrisedLossModel* and *SetStepFunction(0.1, 10*um)m*, and *G4NuclearStopping*
- **pi+*, *pi-*, *kaon+*, *kaon-***G4hMultipleScattering*, *G4hIonisation* with *SetStepFunction(0.2, 50*um)*, *G4hBremsstrahlung* and *G4hPairProduction*. The minimum energy for the validity of *G4hBremsstrahlung* is set to 10 keV, instead of the default value 1 GeV.
- **Other charged particles***G4hMultipleScattering* and *G4hIonisation*

This physics lists inherits from *GmEMPhysics*, so that the command */gamos/GmEMPhysics/replacePhysics* can be used to change the physics models of the electromagnetic particles.

Standard electromagnetic physics list options

Geant4 offers several options of standard electromagnetic physics, which are used extensively in the Geant4 examples. Those options are named *G4EmStandardPhysics*, *G4EmStandardPhysics_option1*, *G4EmStandardPhysics_option2*, *G4EmStandardPhysics_option3*, and *G4EmStandardPhysics_option4*. See http://www.geant4.org/geant4/collaboration/working_groups/electromagnetic/physlist.shtml for the documentation on these options.

To use these options can be done selecting the physics list:

```
/gamos/physicsList GmEMStandardPhysics
```

By default the first option is selected, to change it the following command has to be used:

```
/gamos/EMStandardPhysics/replacePhysics OPTION
```

where *OPTION* may be *emstandard_opt0*, *emstandard_opt1*, *emstandard_opt2* or *emstandard_opt3* (default).

Multiple scattering models

If you use any of the three GAMOS physics list described above you may use any of the multiple scattering models available in the current version of Geant4. This may be done with the following parameter:

```
/gamos/setParam GmMultipleScattering:Model:PARTICLE_NAME MODEL
```

where *PARTICLE_NAME* can be *e-*, *e+*, *mu* or *hadron* and *MODEL* can be *Urban*, *WentzelVI* or *GoudsmitSaunderson* (see the GEANT4 User's Guide for an explanation of each model). The default model for all particles is *Urban*. If your problem is in the micrometer or nanometer range this model may give incorrect results and we recommend you to use the *GoudsmitSaunderson* one.

Bremsstrahlung angular distributions

The Geant4 low energy electromagnetic model of bremsstrahlung offers three different angular distributions, namely Tsai, 2BN and 2BS (see Geant4 Physics Reference Manual for an explanation of the difference between the three models). If you are using any of the three GAMOS physics lists above you may select among one of the three with the following parameter:

```
/gamos/setParam GmPhysicsElectron:Bremsstrahlung:AngularDistribution MODEL
```

where *MODEL* can be *tsai* (which is the default model), *2bn* or *2bs*.

You should nevertheless take into account that the *2bn* distribution has an intrinsic kinematical limit of 1 MeV, so you should not set the upper energy limit to a higher value if you want to use it.

In the latest Geant4 release, it is also possible to change the bremsstrahlung angular distributions for standard processes. To do it use the same parameter as above. In this case the model is applied for all energies.

GAMOS hadrontherapy physics list

The GAMOS hadrontherapy physics list is based on the hadron-therapy Geant4 advanced example.

This physics list can be selected with the command

```
/gamos/physicsList HadronTherapyPhysics
```

The default physics options are those include in the GEANT4 physics lists *G4EmStandardPhysics_option3*, *G4DecayPhysics* and *G4RadioactiveDecayPhysics*.

This default physics can be changed with the command

```
/HT/Physics/replacePhysics PHYSICS_LIST
```

where *PHYSICS_LIST* can be

- *standard_opt3*: G4EmStandardPhysics_option3(
- *LowE_Livermore*: G4EmLivermorePhysics
- *LowE_Penelope*:G4EmPenelopePhysics
- *local_ion_ion_inelastic*: LocalIonIonInelasticPhysics
- *QGSP_BIC_EMY*: QGSP_BIC_EMY

This list has several commands to select cuts:

```
/HT/Physics/setCuts VALUE UNIT : set the cuts for all volumes and particles
```

```
/HT/Physics/setGCut VALUE UNIT : set the cuts for gammas for all volumes
```

```
/HT/Physics/setECut VALUE UNIT : set the cuts for electrons for all volumes
```

```
/HT/Physics/setPCut VALUE UNIT : set the cuts for positrons for all volumes
```

Microdosimetry physics list

The DNA physics list defines the physics processes and models to simulate interactions of electrons of very low energy (down to 7 eV) in water. It is based on the physics list that can be found in the Geant4 wample *examples/extended/medical/dna/microdosimetry/*.

This physics list can be selected with the command

```
/gamos/physicsList GmDNAPhysics
```

More details of this physics list can be found at <http://geant4-dna.org/>

Other physics lists

You can use easily in GAMOS any of the Geant4 physics lists [*biblio.g4physics*]. All you have to do to use a Geant4 physics list is to add in the file *GAMOS_DIR/source/GamosCore/GamosPhysics/GamosOtherPhysicsList/src/module.cc* (or in any other in GAMOS code or created by you, see Appendix on *Creating your plug-in*) two lines like these ones:

```
#include "QGSP.hh"
DEFINE_GAMOS_PHYSICS (QGSP) ;
```

Compile it and then you can use it in your command file with the line

```
/gamos/physicsList QGSP
```

As you can see looking at this file, all the physics list in the GEANT4 source code are already included and therefore can be selected with a user command

There is also in GAMOS a *GmDummyPhysicsList* that defines all the particles, but only the process *G4Transportation*.

All the GEANT4 physics list that can be found in the directory *source/physics_lists*. These are:

- FTF_BIC
- FTFP_BERT
- FTFP_BERT_HP
- FTFP_BERT_TRV
- FTFP_INCLXX
- FTFP_INCLXX_HP
- LBE
- QGSP_BERT
- QGSP_BERT_HP
- QGSP_BIC
- QGSP_BIC_HP
- QGSP_BIC_AllHP
- QGSP_FTFP_BERT
- QGSP_INCLXX
- QGSP_INCLXX_HP
- QGS_BIC

- Shielding

There is a couple of physics lists that use the new `particle_hp` GEANT4 package, which simulated charged particle inelastic interactions using evaluated particle data bases. Please go to the web page http://telemaco.ciemat.es/GAMOS/particle_hp for details on using this package. The physics list `GmQGSP_BIC_PHP` uses the same physics as `GmQGSP_BIC_HP`, but uses `particle_hp` for proton inelastic interactions. The physics list `GmQGSP_BIC_NHP` uses the same physics as `GmQGSP_BIC_HP`, but uses `particle_hp` for all neutron interactions.

Optical photons

You may add the physics of optical photons to any physics list including the command:

```
/gamos/physics/addPhysics opticalphoton
```

This will activate the scintillation process for all the particles and the `G4OpAbsorption`, `G4OpRayleigh` and `G4OpBoundaryProcess` processes for optical photons. To avoid blowing up the memory by the thousands of optical photons that may be created at one step, by default the secondary optical photons are tracked at the moment they are created; this means that the primary particle is stopped and later it is restarted. You may deactivate this option with the parameter

```
/gamos/setParam GmPhysicsOpticalPhoton:TrackSecondariesFirst 0
```

The yield factor, i.e the number of optical photons created per MeV, is set by default to 1, you may change it with the parameter

```
/gamos/setParam GmPhysicsOpticalPhoton:YieldFactor FACTOR
```

To define the optical properties of the medium, you must create a `G4MaterialPropertiesTable` which is linked to the `G4Material` in question (without it no optical photon will be created). All the properties can be managed in the geometry text file, using tags similar to those of the geometry text file utility. For details on the meaning of these parameters please refer to the Geant4 documentation. We describe here these tags with the parameters that should accompany them.

To define a new `G4MaterialPropertiesTable` you have to use the tag:

```
:MATE_PROPERTIES_TABLE
```

- Table name

Then you can add different properties to the table. There are two kinds of properties: those that are defined by a single number (called constant properties in Geant4 notation) and those that are defined by a list of numbers, one associated to an energy.

To define a constant property the following tag must be used:

```
:MATEPT_ADD_CONST_PROPERTY
```

- Material properties table name
- Property name
- Property value

To define non constant properties, you have to define first the list of energies that you will use to assign the different properties to the table:

```
:MATEPT_ADD_ENERGIES
```

- Material properties table name
- Energy 1
- Energy 2

- ...
- Energy N

Then to define a property:

:MATEPT_ADD_PROPERTY

- Material properties table name
- Property name
- Value 1
- Value 2
- ...
- Value N

The material properties table can be attached to a material with the tag (use it several times to attach it to several materials):

:MATEPT_ATTACH_TO_MATERIAL

- Material properties table name
- Material name

You can create an optical surface with the tag:

:OPTICAL SURFACE

- Name
- Boundary process model. It can be - UNIFIED - GLISUR - LUT
- Boundary process model. It can be - UNIFIED - GLISUR - LUT
- Surface finish type. It can be - polishedfrontpainted - polishedbackpainted - ground - groundfrontpainted - groundbackpainted - polishedlumirrorair - polishedlumirrorglue - polishedair - polishedteflonair - polishedtioair - polishedtyvekair - polishedvm2000air - polishedvm2000glue - etchedlumirrorair - etchedlumirrorglue - etchedair - etchedteflonair - etchedtioair - etchedtyvekair - etchedvm2000air - etchedvm2000glue - groundlumirrorair - groundlumirrorglue - groundair - groundteflonair - groundtioair - groundtyvekair - groundvm2000air - groundvm2000glue
- Surface type. It can be - dielectric_metal - dielectric_dielectric - dielectric_LUT

Optionally the polish value and the sigma alpha value can be set by adding two extra arguments:

The material properties table can be attached to an optical surface with the tag (use it several times to attach it to several optical surfaces):

:MATEPT_ATTACH_TO_OPTICAL SURFACE

- Material properties table name
- Optical surface name

You can create a logical border surface with the tag:

:LOGICAL_BORDER SURFACE

- Material properties table name
- First physical volume name
- Second physical volume name

- Optical surface name

You can create a logical skin surface with the tag:

:LOGICAL_SKIN SURFACE

- Material properties table name
- Logical volume name
- Optical surface name

Using optical photons as primary generator

If you are using optical photons as primary generator particles, you may set the polarization using the parameter (see chapter on *Generator*):

/gamos/setParam SOURCE_NAME:Polarization POLARIZ_X POLARIZ_Y POLARIZ_Z

X-ray refraction

GAMOS offers the novelty of an X-ray refraction process, which implements a Snell's law refraction and is based on the GEANT4 refraction process for optical photons. You may instantiate it with the command:

/gamos/physics/addPhysics xray-refraction

always after the */run/initialize* command. Is it also necessary to set the refraction indices for each material and for each energy. This can be done by using the command (you may use one command per material)

/gamos/geometry/setRefractionIndex MATERIAL_NAME ENERGY_1 REFRACTION_INDEX_1 ENERGY_2 REFRACTION_INDEX_2 ...

the value for a given energy of a given material will be interpolated among these set of values. It is important that you give a refraction index for all possible energies in your problem, if not you will get a warning that the error is out of range.

If you use this code you should quote the following reference in your results:

Zhentian Wang,Zhifeng Huang,Li Zhang,Zhiqiang Chen,Kejun Kang. Implement X-ray refraction effect in Geant4 for phase contrast imaging. IEEE Nuclear Science Symposium Conference Record,2009,1082-3654.

Coulomb scattering

You may activate Coulomb scattering for electrons and positrons by using the command:

/gamos/physics/addPhysics CoulombScattering

Atomic deexcitation processes

The atomic deexcitation, i.e. the production of characteristic X-rays and Auger electrons, when the atoms are excited after an ionisation or photo electric process, is activated by default if one of the two GAMOS electromagnetic physics list is selected. To deactivate it you can use the parameter:

/gamos/setParam GmEMPhysicsList:AtomicDeexcitation 0

The default behaviour activates atomic deexcitation only for the default GEANT4 region, *DefaultRegionForTheWorld*, which is created for volumes not associated to any region (see section on *Production cuts by region*). To activate it for list of regions you can use the parameter:

```
/gamos/setParam AtomicDeexcitation:Regions REGION_1 REGION_2 ...
```

You may also want to change the production cuts, that is the minimum value of the energy of the characteristic X-rays or Auger electrons. This can be done independently for the secondary particles generated by ionisation and photo electric effect. The parameters to do it for ionisation are:

By default production of characteristic X-rays by fluorescence and Auger electrons is on, while PIXE is off. You can also select which process to activate with the parameters

```
/gamos/setParam AtomicDeexcitation:Fluorescence 1
```

```
/gamos/setParam AtomicDeexcitation:Auger 1
```

```
/gamos/setParam AtomicDeexcitation:PIXE 1
```

It is important to note, that the production of secondary gammas or electrons is affected by the same cuts than the production of bremsstrahlung gammas and ionisation electrons. This means that if these cuts are high, those particles will not be produced; although the atomic deexcitation processes will be active the energy of the secondary particles will be added to the energy deposited. We remind you that the production cuts are set by range in GEANT4, but you can find the range to energy threshold conversion for each material at the beginning of your job.

Decay process

You can activate the decay process for all particles for which it is applicable with the user command:

```
/gamos/physics/addPhysics decay
```

Radioactive decay process

Each time an ion is created by a GAMOS command (to use as primary particle, to create a filter to select it, ...), the radioactive decay process is activated for it. You can avoid this by using the parameter

```
/gamos/setParam Physics:RadioactiveDecay 0
```

But if the ion is created by Geant4 and you want to activate the radioactive decay process to have to use command:

```
/gamos/physics/addPhysics radioactiveDecay
```

This command has to be used after the */run/initialize* command, and activates automatically this process for all ions.

Cerenkov process

You can activate the Cerenkov processes for all charged particles with the user command:

```
/gamos/physics/addPhysics cerenkov
```

Several parameters can be set before this command to control the Cerenkov process:

```
/gamos/setParam GmPhysicsCerenkov:MaxNumPhotonsPerStep VALUE
```

limits the step size by specifying a maximum (average) number of Cerenkov photons created during the step. The default value is 0, i.e. no limit.

/gamos/setParam GmPhysicsCerenkov:TrackSecondariesFirst 1

serves to avoid blowing up the memory by the thousands of photons that may be created at one step. by tracking the secondary photons at the moment they are created; this means that the primary particle is stopped and later it is restarted.

/gamos/setParam GmPhysicsCerenkov:MaxBetaChange VALUE

sets the maximum allowed change in $\beta = v/c$ in % (perCent). The default value is 0, i.e. no limit.

Coulomb scattering

Coulomb scattering is an alternative to multiple scattering where the particles are transported colliding with an atom per step (instead of averaging multiple collisions). It is therefore a precise but slow process.

This process can be activated with the user command:

/gamos/physics/addPhysics CoulombScattering

You also have to remember to delete the multiple scattering process:

/gamos/physics/removeProcessesByName msc

Nuclear processes of electromagnetic particles

You can activate the nuclear process of gammas, electrons or positrons with one of the following user commands:

/gamos/physics/addPhysics gammaNuclear

/gamos/physics/addPhysics electronNuclear

/gamos/physics/addPhysics positronNuclear

Replacing process models

Replacing a set process models

Whatever physics list you have chosen, you can replace part of the electromagnetic models of one of more particles by one of the three models (standard, low energy or Penelope) with the command:

/gamos/physics/replacePhysics MODELS_1 MODELS_2 ...

where *MODELS_i ...* may be: *gamma-standard gamma-lowener gamma-penelope electron-standard electron-lowener electron-penelope positron-standard positron-penelope* (there are no positron low energy models in Geant4).

Replacing one process model

Another command allows to replace only one model of one process, also independent of the physics list you have set:

/gamos/physics/replaceProcessModel PARTICLE_NAME PROCESS_NAME MODEL_NAME ...

The following set of particles, process and model can be used:

- gamma phot standard
- gamma phot lowener

- gamma phot penelope
- gamma compt standard
- gamma compt lowener
- gamma compt penelope
- gamma conv standard
- gamma conv lowener
- gamma conv penelope
- gamma Rayl lowener
- gamma Rayl penelope
- electron eIoni standard
- electron eIoni lowener
- electron eIoni penelope
- electron eBrem standard
- electron eBrem lowener
- electron msc Urban
- electron msc WentzelVI
- electron msc GoudsmitSounderson
- positron eIoni penelope
- positron eIoni standard
- positron eIoni lowener
- positron eBrem penelope
- positron eBrem standard
- positron eBrem lowener
- positron annihil lowener
- positron annihil penelope
- positron msc Urban
- positron msc WentzelVI
- positron msc GoudsmitSounderson

Removing a process from a physics list

Several commands can be used to remove a process that is instantiated by a physics list without having to modify the C++ code and recompile

The process to be removed can be selected by name

```
/gamos/physics/removeProcessesByName PROCESS_NAME_1 PROCESS_NAME_2 ...
```

To find the name that GEANT4 gives to a process, you may use the user action

```
/gamos/userAction GmCountProcessesUA
```

which will give you a list of the process names attached to each particle in your physics list.

If you do not want to select all the processes give a given name, but only if they are attached to a particle, you can use the command

```
/gamos/physics/removeProcessesByParticleAndName PARTICLE_NAME_1 PROCESS_NAME_1 PARTI-  
CLE_NAME_2 PROCESS_NAME_2 ...
```

Alternatively you may select all process of a given type

```
/gamos/physics/removeProcessesByType PROCESS_TYPE_1 PROCESS_TYPE_2 ...
```

where the types are those defined by GEANT4: *Transportation, Electromagnetic, Optical, Hadronic, Photolepton_hadron, Decay, General, Parameterisation, UserDefined or NotDefined*

Production cuts

Several physics processes, namely bremsstrahlung, ionisation from all charged particles and e+e- pair production from muons have very high cross sections at low energies. It is therefore necessary to implement a production cut so that all particles below it are not generated, but their energy is accounted as energy deposited. GEANT4 uses production cuts in range, instead of in energy as used previously by GEANT3 and most Monte Carlo codes. A cut of for example 1. mm for photons means that no photon will be produced if the expected range in the current material is less than 1. mm.

If you use the GAMOS electromagnetic physics list, the default production cut value is 0.1 mm for all processes in all materials. The Geant4 command `/run/setCut 'value' 'unit'` set the cuts for all process to the desired value. But do not forget to use the command `/run/initialize` after setting the cuts, if you want that your change is effective. The Geant4 command `/run/particle/dumpCutValues` dumps the list of materials and for each one the list of cuts for each particle (in GAMOS it is printed by default).

One word of caution is due here: internally Geant4 converts the cuts for range in each material to cuts for energy and uses these for their computations. There is a low energy limit so that if the range cut you use is very small it will be converted to this energy value. The default value in Geant4 is 990 eV. You may change this value with the Geant4 user command:

```
/cuts/setLowEdge ENERGY ENERGY_UNIT
```

Production cuts by region

A region in Geant4 is a set of G4LogicalVolume's that share common properties. You can define a region in the text file where you defined your geometry, by using the tag

```
:REGION REGION_NAME LOGICAL_VOLUME_NAME(s)
```

where REGION_NAME is the name that identifies the region and LOGICAL_VOLUME_NAME(s) is the list of G4LogicalVolume's that belong to the region. For example:

```
:REGION myRegion Crystal Wall
```

Alternatively you can define a new region in your user script through a user command:

```
/gamos/geometry/createRegion REGION_NAME LOGICAL_VOLUME_NAME
```

Do not forget that in Geant4 when a logical volume belongs to a region automatically all its daughters belong to the same region, unless there is another region explicitly defined for some of the daughters.

Also do not forget that regions in Geant4 have to be set in a hierarchical way: if you place a volume A in the world and inside it you place a volume B, you cannot create a new region for B unless you have explicitly created a region for A.

Once you have defined a region, you may set a cut for the particles that traverse that region with the tag

```
:CUTS REGION_NAME gamma_CUT e-_CUT e+_CUT
```

where REGION_NAME is the name of a previously defined region, gamma_CUT e-_CUT e+_CUT are the cuts for gamma, electrons and positrons (the cut for positron is optional; if not set it will take the one for electrons).

Alternatively you can set the cuts in your user script through a user command:

```
/gamos/physics/setCuts REGION_NAME gamma_CUT e-_CUT e+_CUT
```

Energy cuts to range cuts conversion

If you want to know the translation from an energy cut value to a range cut value for a given particle in a given material, you can do with the following instructions. First you have to instantiate the user action

```
/gamos/userAction GmCutsEnergy2RangeUA
```

and then you can use the command

```
/gamos/physics/ECuts2RangeCuts MATERIAL_NAME CUT_VALUE PARTICLE_NAME
```

You may use in the material name an '*' if you want to name several materials at the same time. For the particle name only the following names have a meaning: *gamma*, *e-*, *e+*, *e*, ****. This will produce a table with the conversion for each material and particle similar to the following one:

```
GmCutsEnergy2RangeUA: MATERIAL: G4_AIR PART: gamma
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 286588
GmCutsEnergy2RangeUA: MATERIAL: G4_AIR PART: e-
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 129.155
GmCutsEnergy2RangeUA: MATERIAL: G4_AIR PART: e+
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 131.938
GmCutsEnergy2RangeUA: MATERIAL: G4_WATER PART: gamma
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 334.152
GmCutsEnergy2RangeUA: MATERIAL: G4_WATER PART: e-
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 0.134781
GmCutsEnergy2RangeUA: MATERIAL: G4_WATER PART: e+
ENERGY CUT: 0.1 (MeV) = RANGE CUT: 0.137686
```

Minimum and maximum production cuts

GEANT4 has internal limits in the minimum and maximum energy of a production cuts. These values are by default 990 eV and 100 TeV. You may change these values with the command:

```
/gamos/physics/prodCutsEnergyLimits MIN_ENERGY MAX_ENERGY
```

Apply cuts for all processes

By default the production cuts are only applied to ionisation, bremsstrahlung and to electron-positron production by muons, but they may be applied to any process by using the command:

```
/gamos/physics/applyCutsForAllProcesses
```

You may observe that the behaviour of the two commands is indeed quite different. While the GEANT4 electromagnetic process implements a mechanism to make use of the production cuts, the hadronic processes do not. Therefore the first command is executed at each process, not creating the corresponding secondary particles if their range is small. On the other side, the second command is implemented as a user stacking action, so that the tracks are created but killed before they are started to be tracked.

User limits

The user limits is a mechanism that Geant4 offers to limit the tracking of a particle. There are five types of user limits:

- Limit the step size
- Limit the track length
- Limit the time of flight
- Stop the particle when the kinetic energy is below a limit (and deposit its energy locally)
- Stop the particle when the expected range is below a limit (and deposit its energy locally)

In GAMOS a user can set the user limits through simple user commands and user limits can be set independently for different particle types in the same or different logical volumes. The set of commands to set user limits are

- `/gamos/physics/userLimits/setUserLimits` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MAX_STEP MAX_TRK_LENGTH MAX_TOF MIN_KIN_E MIN_RANGE* where *USER_LIMITS_NAME* is the name of the user limits (every user limits must have a name, so that new logical volumes and particles can be added with user commands later), *MAX_STEP* *MAX_TRK_LENGTH* *MAX_TOF* *MIN_KIN_E* and *MIN_RANGE* are the values of the five user limit types described above. If you want to set only one user limit type you can use one of the following commands:
- `/gamos/physics/userLimits/setMaxStep` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MAX_STEP*
- `/gamos/physics/userLimits/setMaxTrkLen` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MAX_TRK_LENGTH*
- `/gamos/physics/userLimits/setMaxTOF` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MAX_TOF*
- `/gamos/physics/userLimits/setMinEKin` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MIN_KIN_E*
- `/gamos/physics/userLimits/setMinRange` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MIN_RANGE*

There is another command in GAMOS that serves to set the minimum range user limit using a distance value, but internally it is applied as a minimum kinetic energy limit. This permits to use range values but avoids the lengthy process of converting the kinetic energy at each step into range. For this you can use the command:

- `/gamos/physics/userLimits/setMinEKinByRange` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MIN_RANGE*

Once a user limit is set, you may apply it to a new pair of logical volume and particle type with the following command:

- `/gamos/physics/userLimits/addLVAndParticle` *USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME* for the list of particle names in GAMOS, see section *Particle names* .

A last command serves to print the active user limits for each volume and each particle:

```
/gamos/physics/userLimits/print
```

Automatic optimisation of cuts

The production cuts and user limits are powerful methods to tune your simulation so that you can save a lot of CPU time by not tracking the particles that are not going to contribute to your results. Nevertheless the tuning of cuts is usually a long and difficult task. We have developed in GAMOS a method to help the user to obtain the optimal value

of the production cuts and user limits for her/his application in a single job. We describe here the basic idea of the method and then in the corresponding sections the details of its implementation for different cases.

To use this utility you have to define in a clear way which are the results you don't want to change when cuts change, for example

- Number of particles reaching a region
- Dose distribution
- Number/energy/spatial distribution of hits
- Shower shape in a volume
-

For each track that contributes to your result GAMOS stores all its history: for the track itself and each of its ancestors stores energy, range, region, process and particle type. In the case of production cuts this information is stored at particle creation. In the case of user limits this information is stored at each step (for parents only the steps before the creation of the interesting track).

At the end of run you can get for each region/process/particle a list of all the ranges or energies of all the particles created. Then you can easily know if you apply a cut how many particles are below it.

This is valid if you are only interested in counting how many particles you lose. For other cases another approach should be used. For example if you want to check how your dose distribution changes, you can build a set of filters, each one with a set of cut values. These filters do not really cut the particles but only serve to tag a particle if it would have been killed by the set of cuts in the filter. Therefore, for each track that contributes to your results, GAMOS can check if it (or any of its ancestors) would have been killed by each of these sets of cut values. If you build your results N times in a job, each one using only those tracks that pass one filter, you can compare each result to see how it changes with each set of cuts.

Although as mentioned above, the production cuts are only necessary for ionization and bremsstrahlung processes, this method allows to extend the production cuts to other processes. To do this we provide the abovementioned numbers and plots separately for each process so that the cuts can be automatically set in an easy way. If you want to apply the same cuts to all processes you can use the GAMOS command

```
/gamos/GMphysics/applyCutsToAllProcesses
```

that will instantiate an object of type `G4EmProcessOptions` and invoke the method `SetApplyCuts(true)`.

In the PET and Radiotherapy applications chapters you can see the details of how to apply this technique in these fields.

Range rejection

The range rejection technique consists on killing a particle at creation and depositing all its energy locally if it is not going to leave the current volume. To do this in practical terms, the particle is killed if the range is smaller than the distance to the volume boundary (although it would have a chance to exit the volume, this chance is considered negligible).

You can analyze which would be the effect of applying this technique by using the same user action as for the production cuts, e.g.:

```
/gamos/userAction GmProdCutsStudyUA RTCutsStudyFilter
```

It will produce a table with the number of tracks that would be killed by the range rejection and would not reach the target (the tracks themselves or any of their children). As for the production cuts, they are printed by region, by particle and by creator process type. To get a closer inside on this technique several plots are produced (one per each region, each particle and each creator process) representing the logarithm of the difference safety-range vs the logarithm of the range. To distinguish the cases where the range is bigger than the safety (no range rejection) those cases are plotted

in the bins -15 to -5, while the cases where the range is smaller than the safety occupy the bins -5 to 5 (if there is a case, not likely for a radiotherapy simulation) where the $\log_{10}(\text{fabs}(\text{safety-range}))$ is smaller than -5 (of course before the -10 subtraction), it is set to -5 and if is bigger than 5 it is set to 5.

Building your physics list with C++ code

To build your physics list, first write it in the usual Geant4 way, that is, inheriting from `G4VUserPhysicsList` (see example in [biblio.g4doc-physics]). After that you have to transform it into a plug-in. To learn how to do this, see the instructions in the section above.

USER ACTIONS

Geant4 user actions are the way the user can interact with a job at the beginning/end of each run, beginning/end of each event, beginning/end of each track or at each step. The user can write a class, inheriting from one of the Geant4 user action abstract classes, and Geant4 will take care of calling the user code.

The GAMOS user actions classes provide all the functionality of the Geant4 classes, and also allow the user to define several user actions of the same type in the same job and to define a class that inherits from several user action types at the same time. Moreover, as they are plug-in's, the user can activate them by means of a user command.

User actions can be associated to filters or classifiers, as explained below.

Several functionalities are already implemented in GAMOS as user actions (like histograms, event classifiers, ...) and you may use them as examples for creating your own one.

All user actions are instantiated with the same user command:

```
/gamos/userAction USER_ACTION_NAME
```

Adding a filter

One or several filters can be added to a user action by simply adding their names in the user command where an action is selected. See section on *Filters* to get a list of the available filters in GAMOS and how to add parameters to a filter.

If the filter you are using is a step filter and your user action is a stepping user action, the method `SteppingAction` will only be called if all the filters accept the step. If the filter you are using is a track filter it affects the callings to the `PreUserTrackingAction` and `PostUserTrackingAction` for tracking actions and `ClassifyNewTrack` for stacking actions.

For example

```
/gamos/userAction GmTrackDataHistosUA GmGammaFilter
```

will only produce histograms for tracks whose particle is a gamma.

Adding a classifier

One classifier can be added to a user action by simply adding their names in the user command where an action is selected. See section on *Classifiers* to get a list of the available classifiers in GAMOS and how to add parameters to a classifier.

It is up to the concrete user action to use the classifiers or not. See examples in this User's Guide. For example

```
/gamos/userAction GmTrackDataHistosUA GmClassifierByParticle
```

will produce a different set of histograms for each particle type.

User action name

The name of the user action is the name of the class itself plus the name of the filters (in the order which they are given in the user command) plus the name of the classifier, all separated by underscore characters. For example if you typed:

/gamos/userAction GmTrackHistosUA GmGammaFilter GmPrimaryFilter GmClassifierByParticle

the name of the user action will be *GmTrackHistosUA_GmGammaFilter_GmPrimaryFilter_GmClassifierByParticle*. Unless explicitly mentioned in this guide, this name will be the name of the histogram or data file in case the user action is writing a file (plus the corresponding file suffix), unless the file name is changed with a parameter. It is also the name that has to be used for any parameter corresponding to the user action.

Creating your GAMOS user action

If you need a new user action you have to create a class inheriting from one or several of the GAMOS user actions: *GmUserRunAction*, *GmUserEventAction*, *GmUserTrackingAction*, *GmUserSteppingAction*, *GmUserStackingAction*.

Then you implement the same methods as for the Geant4 user actions:

- *GmUserRunAction*: - virtual void BeginOfRunAction(const G4Run* aRun); - virtual void EndOfRunAction(const G4Run* aRun);
- *GmUserEventAction*: - virtual void BeginOfEventAction(const G4Event* anEvent); - virtual void EndOfEventAction(const G4Event* anEvent);
- *GmUserTrackingAction*: - virtual void PreUserTrackingAction(const G4Track* aTrack); - virtual void PostUserTrackingAction(const G4Track* aTrack);
- *GmUserSteppingAction*: - virtual void UserSteppingAction(const G4Step* aStep);
- *GmUserStackingAction*: - virtual G4ClassificationOfNewTrack ClassifyNewTrack(const G4Track* aTrack, G4ClassificationOfNewTrack oldClassificationx); - virtual void NewStage(); - virtual void PrepareNewEvent();

Finally you have to transform your class into a plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmUserActionFactory*, or see the *PlugIn* tutorial.

If you define twice the same user action in your command file, you will get a warning message, but it will be executed twice.

SENSITIVE DETECTOR AND HITS

Sensitive detectors

Attaching a sensitive detector to a volume

The sensitive detector class in Geant4 has the task of creating hits (deposits of energy) each time a track traverses a sensitive volume and loses some energy. You can write your own sensitive detector class, inheriting from `G4VSensitiveDetector`, that produces your own hits, and attach it to any volume in your geometry. However GAMOS provides some utilities to make this easier and without the need of C++ programming or a detailed knowledge of how the sensitive detector and hits work in Geant4.

GAMOS provides several predefined sensitive detectors, that you can find in *GamosCore/GamosSD*:

- *GmSDSimple*. It is a general-purpose class, that produces hits with the position at the centre of the detector. The position can be changed to be at the centre of the detector, the centre of the upper face (positive Z) or the centre of the lower face (negative Z) with the parameter: `/gamos/setParam SD:PosType:**SDTYPE centre/upper/lower`. The identification of each detector unit is done as explained in the sub-chapter *Identifying each sensitive detector copy*.
- *GmSDSimpleExactPos*. It is similar to *GmSDSimple* but the hits position is the centroid of the energy depositions of the different tracks that produced it (weighted by their energy).
- *GmSDOpticalPhoton*. This class inherits from *GmSDSimple*, but only produces hits if the process that defined the step is “*OpAbsorption*”.
- *GmSDSeparateByTime*. It is similar to *GmSDSimple* but it assigns energy depositions to different hits if the time interval is different. Time intervals dimension is given by the measuring time. Three parameters control the behaviour of this sensitive detector: Set a minimal value of the interval dimension: `/gamos/setParam SD-SeparateByTime:MinimalMeasuringTime:*SDTYPE VALUE*` Do not accept hits with time above a value: the value is calculated as the product of the average time given by the particle sources, the number of events and a factor given by: `/gamos/setParam SDSeparateByTime:BigTimeFactor:*SDTYPE VALUE*` To apply this limit, the following parameter has to be set to 1: `/gamos/setParam SDSeparateByTime:DiscardBigTime:*SDTYPE 1*`

There are other classes that serve to make a virtual segmentation, when you have a big sensitive volume that you want to segment in different pieces, although you have not segmented it in your geometry. The only class currently implemented is:

- *GmSDVirtSegmBox*. It divides a box into voxels of equal size in X, Y and Z. Several parameters control its use: Number of divisions in X, Y and Z: `/gamos/setParam SD:VirtSegmBox:NDiv:**SDTYPE NDIV_X NDIV_Y NDIV_Z` where *SDTYPE* is the type of the sensitive detector. Widths of the voxels in X, Y and Z: `/gamos/setParam SD:VirtSegmBox:Width:**SDTYPE WIDTH_X WIDTH_Y WIDTH_Z` It must be taken into account that the product *NDIV * WIDTH* has to be equal to the total length of the mother box, in each of the three coordinates. If this is not the case when a step happens near the positive border it will

be assigned a division number bigger than *NDIV* and an exception will be thrown. For efficiency, a maximum number of voxels in each direction is allowed, you may change it with the parameter: `/gamos/setParam SD:VirtSegmBox:MaxNVoxels:**SDTYPE VALUE` The divisions start at the negative wall of the box that is selected as sensitive volume, that is the offsets are the half lengths of the box. You may change these offsets with the parameter: `/gamos/setParam SD:VirtSegmBox:Offset:**SDTYPE OFFSET_X OFFSET_Y OFFSET_Z` If the offsets are not set, they are calculated at each track step, as it may happen that different volumes are assigned to the same sensitive detector type. If all the volumes assigned to a sensitive detector type have the same solid dimensions, we recommend you that the offsets are only calculated once, what can be done with the parameter: `/gamos/setParam SD:VirtSegmBox:OffsetOnce:**SDTYPE TRUE`

To attach a GAMOS sensitive detector to a logical volume in your geometry, you have to use the command

```
/gamos/SD/assocSD2LogVol SD_CLASS SD_TYPE LOGICAL_VOLUME_NAME
```

The `SD_CLASS` has to be one of the sensitive detector types described above (or any other that you create). The `SD_TYPE` serves to differentiate your different sensitive detectors, so that you can later apply different properties to them (e.g. different energy resolutions)

You may see the PET application for an illustration of this. The `LOGICAL_VOLUME_NAME` is the name of the `G4LogicalVolume` in your geometry that you want to make sensitive. You may repeat this command with different logical volumes.

Building your sensitive detector with C++ code

To build a new sensitive detector you can do it the usual Geant4 way, that is, inheriting from `G4VSensitiveDetector` (see example in [\[biblio.g4doc-physics\]](#)). After that you have to transform it into a plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the `GmSensDetFactory`.

You may also choose to inherit your sensitive detector from the GAMOS class `GmVSD`, so that you can profit easily from its extra functionality. Namely, it will take care of applying the energy and time resolutions by detector type, accumulating the energy of different energy depositions if they happen in the same detector unit, taking into account the measuring time and the dead time and invoking the digitization and reconstruction of hits at the end of the event. There are at least two methods that you have to define in your class:

```
virtual long int GetDetUnitID( G4Step* aStep );
```

Serves to define the logic you want to apply to give a different number to each detector unit (for example to each individual crystal)

```
virtual void CalculateAndSetPosition( GmHit* hit, G4Step* aStep ) = 0;
```

Serves to define the way you want to define the position of the hit (centre of the crystal, barycentre of energy depositions, ...).

Hits

If you have activated any of the GAMOS sensitive detectors, each time a track deposits some energy in any copy of the selected logical volume (you can indeed select several volumes by repeating the command) a `GmHit` will be created. If the energy deposition happens in the same volume copy (a real one, or a virtual one in case of a virtually segmented sensitive detector) than a previous one in the same event, a new hit is not created, but the existing hit is updated, adding to it the new energy deposition.

The `GmHit` stores the following variables:

- *long int theDetUnitID*: Identification of the touchable
- *G4int theEventID*: Event number

- *G4double theEnergy*: Total energy
- *G4double theTimeMin*: Minimum time of energy depositions This is the time considered when you call the method *GetTime()*. You may also invoke explicitly *GetTimeMin()* or *GetTimeMax()*
- *G4double theTimeMax*: Maximum time of energy depositions
- *G4ThreeVector thePosition*: Position (it is defined in the Sensitive detector class; it can be the centre of gravity of the energy depositions, the centre of the volume, ...)
- *std::set< G4int > theTrackIDs*: The list of track numbers
- *std::set < G4int > theOriginalTrackIDs*: The list of original track numbers (a track is called original if it is a gamma, electron or positron and it is a primary particle, or if it is a gamma created in an original positron annihilation or by a radioactive decay of the primary isotope)
- *std::vector < GmEDepo > theEDepos*:* The list of energy depositions. A *GmEDepo* contains the energy and position of each step.
- *G4String theSDType*: The type of sensitive detector

Hits digitization and reconstruction

Hits digitization

The conversion of the hits into digital signals is very dependent on the detector. Therefore GAMOS just provides a general class, *GmVDigitizer*. The user may inherit her/his own digitizer from it and implement the two methods:

```
virtual std::vector < GmDigit* > DigitizeHits(const
std::vector < GmHit* > &);
virtual void CleanDigits();
```

These two methods will be called automatically. The first one at the end of each event, to convert the hits in digits, and the second one at the beginning of each event, to clear the digits of the previous event.

You can then convert your digitizer into a GAMOS plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmDigitizerFactory*. After this, you select it with the command

```
/gamos/SD/digitizer MY_DIGITIZER
```

Hits and digits reconstruction

The digital signals are usually treated so that they become ‘reconstructed hits’, which contain sensible variables, like energy, time, This conversion is also very dependent on the detector, and therefore GAMOS just provides a general class, *GmVRecHitBuilderFromDigits*. There is also another class *GmVRecHitBuilderFromHits*, which serves in case the user wants to transform the hits into reconstructed hits directly.

The user may inherit her/his own reconstructed hit builder from it and implement the two methods:

```
virtual std::vector < GmRecHit* > ReconstructDigits(const
std::vector < GmDigit* > &) = 0;
virtual void CleanRecHits();
```

in the case of *GmVRecHitBuilderFromDigits*, or

```
virtual std::vector < GmRecHit* > ReconstructHits(const
std::vector < GmHit* > &) = 0;
virtual void CleanRecHits();
```

in the case of *GmVRecHitBuilderFromHits*.

These two methods will be called automatically. The first one at the end of each event, to convert the digits or hits into reconstructed hits, and the second one at the beginning of each event, to clear the reconstructed hits of the previous event.

You can then convert your reconstructed hit builder into a GAMOS plug-in. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmRecHitBuilderFactory*. After this, you select it with the command

```
/gamos/SD/recHitBuilder MY_RECHITBUILDER
```

Examples of reconstructed hit builders

If you do not need to simulate in detail the electronics of your detector, you may use one of the simple reconstructed hit builders implemented in GAMOS, which serve to merge hits that are close to each other. The energy of the reconstructed hit is the sum of the hit energies, while the position is the weighted sum of the hit positions (weighted by the energy of each hit).

This can be useful, for example, for recovering the total energy of a photon when it has suffered a Compton scattering near the photoelectric interaction (the user has always the freedom to choose how near the other hits are); or for clustering together all the energy depositions of the particle shower produced by the electron following a photoelectric interaction.

The merging of this is always started by the hit that has bigger energy in each detector type (only hits in the same detector type are merged). Then the hits are looked one by one to check if they are close. Each time a hit is added, its position is recalculated. There are two ways to calculate the reconstructed hit position:

- By default it is defined as the barycentre, i.e. the average of all the hits position, weighted by their energy.
- If the following parameter is set: `/gamos/setParam SD:RecHit:PosAtBarycentre:*SDTYPE 0*` the reconstructed hit position will be taken as that of the hit with highest energy.

The reconstructed hit builders implemented are:

- *GmRecHitBuilderByDistance*. Two hits are merged if they are separated by a distance closer than the parameter `/gamos/setParam SD:GmRecHitBuilderByDistance:HitsDistInRecHit` that by default takes a value of 10 mm.
- *GmRecHitBuilderByBlock*. Two hits are merged if they are in sensitive detectors that belong to the same block, i.e. a volume whose parent volume is the same. To check if two hits belong to the same block, the detector unit ID (i.e. the identification of the touchable where the hits is located), is used. Usually the detector unit ID is built from the volume copy numbers of the volume ancestors (e.g. `volume_copy_number + 100 * parent_volume_copy_number + 100*100*grandparent_volume_copy_number`), and therefore two detector unit IDs are considered to belong to the same block if their division by a number given by the parameter `/gamos/setParam SD:GmRecHitBuilderByBlock:NShift VALUE` gives the same results. This parameter takes a default value of 100. The number of ancestors to be used is given by the parameter: `/gamos/setParam SD:GmRecHitBuilderByBlock:NAncestors VALUE`
- *GmRecHitBuilder1to1*. Two hits are never merged, so that a reconstructed hit is built for each hit.

If you want any of these reconstructed hit builders to be active you can select one of them with the command

```
/gamos/SD/recHitBuilder *RHITBUILDER_NAME*
```

where *RHITBUILDER_NAME* is one of the classes described above.

Detector effects

There are a number of detector effects which can be simulated with a few user commands. It is important to note that these effects are applied to the reconstructed hits, as these hits are meant to simulate the digitization+reconstruction process that happens in the detector. If for any reason you want to see a detector effect on the hits, you should use the *GmRecHitBuilder1to1* reconstructed hit builder.

Energy resolution

If you use one of the GAMOS sensitive detectors or you inherit your own one from *GmVSD* you can smear automatically the energy of the hits for each detector type with a gaussian given by the value of the parameter

```
/gamos/setParam SD:EnergyResol:*SDTYPE VALUE*
```

where *SDTYPE* is the type of the sensitive detector.

A more complicated energy resolution can be implemented, corresponding to a more realistic calorimeter resolution. Apart from the constant term described above, energy independent:

$$\sigma/E = K$$

which may be due to calibration errors, non-uniformities and non-linearities in photomultipliers, proportional counters, ADC's, etc, two other terms can be defined.

The first one is a term where the relative gaussian error is inversely proportional to the square root of the energy, a term that can be attributed to the statistical fluctuations in the energy loss and multiple scattering during the shower development:

$$\sigma/E = K * / \text{sqrt}(E)$$

where K can be set with the parameter

```
/gamos/setParam SD:EnergyResolFluct:*SDTYPE VALUE*
```

where *SDTYPE* is the type of the sensitive detector.

In the second term the relative gaussian error is inversely proportional to the energy, and it can be attributed to instrumental effects, being rather energy-independent (noise, pedestal):

$$\sigma/E = K * / E$$

where K can be set with the parameter

```
/gamos/setParam SD:EnergyResolInstr:*SDTYPE VALUE*
```

where *SDTYPE* is the type of the sensitive detector.

If you have an energy function that is different, you may implement it by creating a new reconstructed hit builder class inheriting from *GmVRecHitBuilder* and overwrite the method:

```
virtual void SmearRecHitsEnergy();
```

Time resolution

The time of the hits can also be smeared for each detector type with a gaussian given by the value of the parameter

```
/gamos/setParam SD:TimeResol:**SDTYPE VALUE
```

The time smeared is *theTimeMin*

If you have a time resolution function that is different, you may implement it by creating a new reconstructed hit builder class inheriting from *GmVRecHitBuilder* and overwrite the method:

```
virtual void SmearRecHitsTime();
```

Detector measuring time

A detector has a finite time resolution, so that it is not able to distinguish hits that come from different events when their time is close.

You can define the value of the measuring time for each detector type with the parameter

```
/gamos/setParam SD:MeasuringTime: *SDTYPE VALUE*
```

where *SDTYPE* is the type of the sensitive detector.

that takes a default value of 0 ns.

GAMOS offers several treatments of the measuring time, so that the user can choose the one that best fits the detector under study. To select the treatment type you have to use the parameter

```
/gamos/setParam SD:MeasuringType: *SDTYPE VALUE*
```

The following measurement types are currently available:

- *Trigger*. When hits are produced in one event, the hit with smallest time triggers, so that all hits of all events with a time later than this hit time and shorter than this hit time + measuring time are considered together. This is the default behaviour. .. COMMENT: <para>If your detector is a PET or Compton camera it is likely that it waits for a time at least as long as the measuring time to start the coincidence sorting. But it may happen that this time is even bigger than the measuring time. If you think you should simulate this effect, you can do it by adding an extra parameter:</para>

```
<para><emphasis role="bold">/gamos/setParam SD:Trigger:CoincidenceDelayTime:<emphasis>SDTYPE</emphasis></er
VALUE</para>
```

Several independent triggers are defined: it may be that each detector volume is associated to a trigger or that detectors that have a common ancestor in the geometry hierarchy are associated to the same trigger. By default the trigger detector units are the same units explained in the section on *Identifying each sensitive detector copy*. But you may change this behaviour by changing how many ancestors levels are used with the following parameter: `/gamos/setParam SD:Trigger:NAncestors: *SDTYPE* VALUE` This may serve you for example to define a unique global trigger, by setting the number of ancestors equal to the maximum number of volumes in the geometrical hierarchy of the detectors. .. COMMENT: A warning is due here: the last events in a job are not properly treated, as it happens similarly in a real detector: it is considered that no more hits are created in a time less than measuring time. Usually it is a very small fraction of the events.

- *Interval*. The triggers happen at a constant interval. Starting at time 0, the hits are merged if their time are between $N \cdot \text{measuringTime}$ and $(N+1) \cdot \text{measuringTime}$.
- *Backwards*. Hits are accumulated if they have a time after the event time minus the measuring time. The event time is computed as the time of creation of the first particle in the event.

Detector dead time

A detector takes a finite time to transform an energy deposit into an electronic signal, and during that time it is *dead* and cannot account for any other energy deposition. The dead time is considered to start after the hit time plus the measuring time; in the case that you set the type of measuring to *Interval*, the dead time starts also at the end of the measuring time, but in this case it is the current triggering time plus the measuring time.

GAMOS holds a list of the dead sensitive detectors, i.e. those that have produced a hit in a time prior than the current time minus the dead time. You can define the value of the dead time for each detector type with the parameter

```
/gamos/setParam SD:DeadTime:*SDTYPE* VALUE
```

that takes a default value of 0 ns.

The dead time affects by default all detectors in a block. This means that if a detector is dead it is assumed that all detectors that are placed in the same ancestor are also dead (the usual behaviour for example in a PET or SPECT detector, where all crystals in a block share the readout). You may change the number of ancestors that become dead by setting the parameter

```
/gamos/setParam SD:DeadTimeDUListByBlock:NAncestors:*SDTYPE N_ANCESTORS*
```

which as just mention takes a default value equal to the value of the parameter *SD:DetUnitID:NAncestors:*SDTYPE**. Take into account that the number of ancestors include the detector itself, so that you can tell GAMOS to consider dead only the crystal itself by setting this parameter to 1.

A warning is due here: the hits that belong to the same block are identified by the detector unit ID, so if you are using a bigger number of ancestors you should take care that the hit detector unit ID takes into account a number of ancestors as least as big as the number used here (see section on *Hits*).

You also have the option to define your detector as paralizable (default) or non-paralizable by setting the parameter

```
/gamos/setParam SD:DeadTimeParalizable:*SDTYPE TRUE/FALSE*
```

In a non-paralizable detector, an event happening during the dead time since the previous event is simply lost, while in a paralizable detector, an event happening during the dead time since the previous one will not just be missed, but will restart the dead time.

Minimum hit energy

You may set a threshold to the minimum hit energy, so that hits with smaller energy will not be used (they may be used in the next event if more energy is added to them and they are inside the time window):

```
/gamos/setParam SD:Hit:MinimumEnergy:*SDTYPE VALUE*
```

To simulate Constant Fraction Discriminator (CFD) thresholds, you may also set a minimum energy and a behaviour to treat this threshold. First to set the parameter:

```
/gamos/setParam SD:RecHit:MinimumEnergy:*SDTYPE VALUE*
```

Then you have to select among the two possible behaviours:

```
/gamos/setParam SD:RecHit:MinimumEnergyBehaviour:*SDTYPE BEHAV_TYPE*
```

where *BEHAV_TYPE* can be:

- *DeleteIfSmall* if one of the hits in the reconstructed hit has smaller energy than the threshold all reconstructed hits are deleted
- *AcceptIfBig* if one of the hits in the reconstructed hits has bigger energy than the threshold all reconstructed hits are accepted, else all are deleted

Remember that there is no default behaviour, so if no behaviour is selected, the minimum energy threshold is not taken into account.

Identifying each sensitive detector copy

To identify each detector unit individually you have to give a different detector unit ID to each copy of your sensitive detectors (each G4VTouchable in Geant4 terminology). GAMOS does this automatically for you. If you use the *GmSDSimple* class, it will give each SD copy a detector unit ID that will be the

*copy number of physical volume + 100 * copy number of parent physical volume + 100*100 copy number of grand-parent physical volume**

You can change the number of ancestor volumes to build the detector unit ID with the parameter

```
/gamos/setParam SD:DetUnitID:NAncestors:*SDTYPE VALUE*
```

that takes a default value of 3. These ancestors include the detector itself, i.e. if set to 1 it is only the detector, without looking at the geometry hierarchy.

The ID is made multiplying the copy number of each ancestor by a fixed value *NShift* to the power of the ancestor level minus 1:

$$ID = copyNo_1 + copyNo_2 * NShift + copyNo_3 * pow(NShift,2)+$$

You can change the value of the *NShift* that multiplies each ancestor copy number using the parameter

```
/gamos/setParam SD:DetUnitID:NShift:*SDTYPE VALUE*
```

that takes a default value of 100.

Storing and retrieving hits

GAMOS provides you with the option of storing in a file the hits produced during a run, and reading them back in another run. Creating hits from track in the sensitive detector or reading them from a file will produce the same *in-memory* representation, therefore you can apply any of the above described effects and produce any histogram in the same way independently on how the hits are produced.

This can serve you for example for doing a study on how much your results change with different energy resolutions: you produce the hits (with zero resolution) and store them, and in another run you can read applying a certain energy resolution; this way you spare the time to recreate them (what usually is several orders of magnitude slower than reading them).

To use this utility you just have to activate the user action

```
/gamos/userAction GmHitsWriteUA
```

The name of the file can be controlled with the parameter

```
/gamos/setParam hits:FileName MY_FILENAME
```

If this parameter is not found the name will be *hits.out*

You can read back the hits by activating the user action

```
/gamos/userAction GmHitsReadUA
```

The name of the file can be controlled with the parameter

```
/gamos/setParam hits:FileName MY_FILENAME
```

If this parameter is not found the name will be *hits.out*

As commented above, you can use all the other options in your script and just read the hits instead of generating them. But probably you do not want that new hits are created when you are reading them from a file; in this case, you should also activate the user action

/gamos/userAction GmKillAllUA

File format

The file to be written can be a text file or a binary file. The format depends on the value of the parameter

/gamos/setParam SD:GmHitsWriteUA:BinFile TRUE/FALSE

The text file contains a line for each hit with the following information:

- Event ID
- Sensitive detector type
- Detector unit ID
- Energy (MeV)
- Minimum time (ns)
- Maximum time (ns)
- Position X (mm)
- Position Y (mm)
- Position Z (mm)
- Number of original tracks One line per original track with track ID
- Number of tracks One line per track with track ID

The binary file contains the same information in the following format

- Event ID : float
- Sensitive detector type : char[10] (only first 10 characters are stored, if type has less than 10 characters blank spaces will be added at the end)
- Detector unit ID : unsigned long long
- Energy (MeV) : float
- Minimum time (ns) : unsigned long long
- Maximum time (ns) : unsigned long long
- Position X (mm) : float
- Position Y (mm) : float
- Position Z (mm) : float
- Number of original tracks : unsigned int One line per original track with track ID : unsigned int
- Number of tracks : unsigned int One line per track with track ID : unsigned int

Storing reconstructed hits

It is also possible to store the reconstructed hits by instantiating the user action:

/gamos/userAction GmRecHitsWriteUA

The parameter

/gamos/setParam SD:GmRecHitsWriteUA:BinFile 0/1

serves to select between binary (default) or text format.

The text file contains a line for each hit with the following information:

- " Gamos RecHit; energy= " Energy (MeV)
- " time_min= " Minimum time (ns)
- " time_max= " Maximum time (ns)
- " posXYZ= " Position_X Position_Y Position_Z (mm)
- " N SimHits= " Number of simulated hits

The binary file contains the same information in the following format

- Energy (MeV) : float
- Minimum time (ns) : unsigned long long
- Maximum time (ns) : unsigned long long
- Position X (mm) : float
- Position Y (mm) : float
- Position Z (mm) : float
- Number of hits : unsigned int

Hits histograms

There are two user actions that provide a number of hits histograms. *GmHitsHistosUA* provides statistics about simulated hits, while *GmRecHitsHistosUA* provides statistics about reconstructed hits.

The class *GmHitsHistosUA* produces a file named *hits.root* with the following histograms:

- Number of hits ("Nhits")
- Number of hits compatible in time ("Nhits good")
- Kinetic energy ("Energy (keV)")
- Width (maximum separation between energy deposits) ("Width R3 (mm)")
- Width Z ("Width Z (mm)")
- Width phi ("Width phi (deg)")
- Number of energy deposits ("N E depos")
- Maximum difference in time between energy deposits ("Time span (ns)")
- Maximum distance between a hit and the rest ("Distance between hits (mm)")
- Position X ("X hit (mm)")
- Position Y ("Y hit (mm)")
- Position Z ("Z hit (mm)")
- Position R2 ($=\sqrt{X*X+Y*Y}$) ("R2 hit (mm)")
- Position phi ("PHI hit (deg)")
- Position theta ("THETA hit (deg)")
- Position R3 ($=\sqrt{X*X+Y*Y+Z*Z}$) ("R3 hit (mm)")

The class *GmRecHitsHistosUA* produces a file named *recHits.root* with the following histograms:

- Number of hits (“N rec hits”)
- Kinetic energy (“Energy (keV)”)
- Width (maximum separation between hits) (“Width R3 (mm)”)
- Width Z (“Width Z (mm)”)
- Width phi (“Width phi (deg)”)
- Number of simulated simulated hits (“N sim hits”)
- Maximum difference in time between simulated hits (“Time span (ns)”)
- Maximum distance between a hit and the rest (“Distance between hits (mm)”)
- Position X (“X hit (mm)”)
- Position Y (“Y hit (mm)”)
- Position Z (“Z hit (mm)”)
- Position R2 ($=\sqrt{X*X+Y*Y}$) (“R2 hit (mm)”)
- Position phi (“PHI hit (deg)”)
- Position theta (“THETA hit (deg)”)
- Position R3 ($=\sqrt{X*X+Y*Y+Z*Z}$) (“R3 hit (mm)”)

SCORING

Creating a scorer

Geant4 provides several classes to score different quantities in the selected volumes. GAMOS provides all the Geant4 functionality through user commands and also some extra functionality that we describe in this section.

The first thing you should do to use GAMOS scoring is to create a multifunctional detector [*biblio.g4doc-scoring*] and associate it with a list of logical volumes, with the user command

```
/gamos/scoring/createMFDetector MFD_NAME LOGICAL_VOLUME_NAME(s)
```

where *MFD_NAME* is the detector name that will be used later. and *LOGICAL_VOLUME_NAME(s)* is a list of logical volumes that you associate to this detector.

Then you should add to the detector one of the GAMOS scorers, or your own ones, with the user command

```
/gamos/scoring/addScorer2MFD SCORER_NAME SCORER_CLASS MFD_NAME SCORER_PARAMETERS
```

SCORER_NAME is a name you give to the scorer to be used later, *SCORER_CLASS* is one of the available scorer classes, *MFD_NAME* is one of the multifunctional detectors created above and *SCORER_PARAMETERS* are the optional parameters a scorer may need (see below for the description of the scorers). You may repeat this command to associate several scorers to the same detector.

To each of the defined scorers you can add a filter, to select for which track conditions the scoring will be done:

```
/gamos/scoring/addFilter2Scorer FILTER_NAME/CLASS SCORER_NAME
```

FILTER_NAME/CLASS is the name of a *GmVFilter* class or the name you gave to a filter built from a filter class by using the command */gamos/filter* (see section on *Filters*) and *SCORER_NAME* is one of the scorers defined above.

You may repeat this command to associate several filters to the same scorer. See section on *Filters* for a description of the available filters and how to create your own one.

By default a different count is scored for each of the copies of the selected volumes with different copy number. This is managed by the scorer classifier *GmScorerClassifierByIAncesor*. The user can attach different classifiers to the different scorers so that the counts are done in different ways:

```
/gamos/scoring/assignClassifier2Scorer CLASSIFIER_NAME/CLASS SCORER_NAME
```

CLASSIFIER_NAME/CLASS is the name of a *GmVClassifier* class or the name you gave to a classifier built from a classifier class by using the command */gamos/classifier* (see section on *Classifiers*) and *SCORER_NAME* is one of the scorers defined above.

Finally you may select the format of the scoring results by associating one of the available scorer printers for each scorer,

```
/gamos/scoring/addPrinter2Scorer PRINTER_NAME SCORER_NAME
```

PRINTER_NAME is the name of a *GmVPSPrinter* class or the name you gave to a printer built from a printer class by using the command */gamos/printer* (see section on *Scorer printers* below) and *SCORER_NAME* is one of the scorers defined above.

If no printer is attached to a scorer, it will use the printer type *GmG4PSPrinterG4cout*.

The scoring is done by default taking into account the track weight, except for the scorers when it is explicitly mentioned (see scorers description). If you do not want to take weights into account you can switch them off with the command

```
/gamos/scoring/useTrackWeight SCORER_NAME FALSE
```

The quantities scored are given per event by default. If you want the score without dividing by the number of events, use the command

```
/gamos/scoring/printByEvent PRINTER_NAME FALSE
```

The error in the scored quantity per voxel is also calculated by default, using the following formula:

$$\text{sqrt}\{ (SumV2*nEvents - SumV*SumV) / (nEvents-1) \} / nEvents$$

where *nEvents* is the total number of events in the run, *SumV* is the sum of the scored quantity value times its weight (i.e. the scored quantity itself) and *SumV2* is the sum of squares of scored quantity value times its weight.

When the scoring is done per event, this sum of squares is done summing first all the values times its weight belonging to all the particles of the same event and then squaring this quantity. In this way the correlations between particles of the same event is properly taken into account. If the scoring is not by event, the error calculation uses the same formula, but the sum of squares is not done summing the contribution of the particles of the same event, but squaring each contribution individually.

Calculating the errors makes it necessary to store the square of the weights, increasing substantially the memory usage and CPU time. If you want to deactivate this option for a scorer, use the command

```
/gamos/scoring/scoreErrors SCORER_NAME FALSE
```

SCORER_NAME is one of the scorers defined above.

As mentioned above the errors that are calculated taking into account the number of events. You have to be careful then if you set to off the option of scoring by event and keep on the option of calculating the errors. In the default GAMOS scorer printers, the errors are printed are relative, i.e. the error divided by the value, so no caution is necessary, but be careful if you define a printer yourself.

Each scorer has a default unit (see section below), but it can be overridden with the command:

```
/gamos/scoring/scorerUnit SCORER_NAME UNIT_NAME UNIT_VALUE
```

Scorer classes

All the available scorers in Geant4 are also available in GAMOS. The classes have been slightly changed to provide the extra functionality.

The scorers can be classified in the following types:

- *Track length scorers - GmG4PSTrackLength*: The track length is defined by the sum of step lengths of the particles inside the cell (i.e., the volume where the scoring happens). A particle weight is not applied by default. There are two extra parameters, that are FALSE by default and can be set TRUE or FALSE: to multiply by the kinetic energy and to divide by the velocity.

If the energy track flux is required then you should set them to TRUE FALSE. Alternatively to measure the flux per unit velocity then you should set them to FALSE TRUE. Finally to measure the flux energy per unit velocity then you should set them to TRUE TRUE.

- *GmG4PSPassageTrackLength*: The passage track length is the same as the track length in *GmG4PSTrackLength*, except that only tracks which pass through the volume are taken into account. This means that newly-generated or stopped tracks inside the cell are excluded from the calculation. A particle weight is not applied by default.
- *Deposited energy scorers - GmG4PSEnergyDeposit*: This scorer stores a sum of particles' energy deposits at each step in the cell. - *GmG4PSDoseDeposit*: In some cases, dose is a more convenient way to evaluate the effect of energy deposit in a cell than simple deposited energy. The dose deposit is defined by the sum of energy deposits at each step in a cell divided by the mass of the cell. The mass is calculated from the density and volume of the cell taken from the methods of *G4VSolid* and *G4LogicalVolume*. - *GmG4PSKerma*: This scorer stores a sum of particles' kerma each step in the cell, that is the energies of the charged secondary particles produced by non-charged particles, divided by the mass of the cell. The following scorers serve to calculate the deposited dose in a volume dividing it virtually in smaller pieces. This means that the energy deposited is not divided by the volume the track is in, but by the division volume as defined by the scorer. As you may find other type of use for these scorers, you can choose also to divided by the total volume defined with the scorer, by setting the parameter: */gamos/setParam SCORER_NAME:UseTotalVolume 1 - GmPSSphericalDoseDeposit*: This scorer is designed to compute the dose deposit and assign a different index to each spherical radius interval, despite no spherical division is defined in the geometry. It creates intervals as a function of spherical radius with the parameters given at score definition: *NSLICES_R MIN_R MAX_R* and calculates the volume of each spherical shell so that dose can be computed. Three extra parameters may be given: *CENTRE_X CENTRE_Y CENTRE_Z*, if the you do not want the sphere to be centred at the origin of coordinates, (0,0,0). - *GmPSCylindricalRZDoseDeposit*: This scorer is designed to compute the dose deposit and assign a different index to each interval in cylindrical coordinates (in 2D radius and Z), despite no cylindrical division is defined in the geometry. It creates intervals as a function of the cylindrical radius and the Z coordinate with the parameters given at score definition: *NSLICES_R MIN_R MAX_R NSLICES_Z MIN_Z MAX_Z* and calculates the volume of each cylindrical section so that dose can be computed. Three extra parameters may be given: *CENTRE_X CENTRE_Y CENTRE_Z*, if you do not want the cylinder to be centred at the origin of coordinates, (0,0,0), and another three: *ZAXIS_X ZAXIS_Y ZAXIS_Z*, if the you do not want the cylinder axis to be along the Z global axis, (0,0,1).

The index of the scorer is build as $NSLICES_R * indexZ + indexR$.

- *GmPSCylindricalRPhiDoseDeposit*: This scorer is designed to compute the dose deposit and assign a different index to each interval in cylindrical coordinates (in 2D radius and phi angle), despite no cylindrical division is defined in the geometry. It creates intervals as a function of the cylindrical radius and the phi angle with the parameters given at score definition: *NSLICES_R MIN_R MAX_R NSLICES_PHI MIN_PHI MAX_PHI* and calculates the volume of each cylindrical section so that dose can be computed. Two more parameters are mandatory, to define the dimension in Z: *MIN_Z MAX_Z*. Three extra parameters are optional: *CENTRE_X CENTRE_Y CENTRE_Z*, if you do not want the cylinder to be centred at the origin of coordinates, (0,0,0), and another three: *ZAXIS_X ZAXIS_Y ZAXIS_Z*, if the you do not want the cylinder axis to be along the Z global axis, (0,0,1). The index of the scorer is build as $NSLICES_R * indexPHI + indexR$.
- *GmPSCylindricalZPhiDoseDeposit*: This scorer is designed to compute the dose deposit and assign a different index to each interval in cylindrical coordinates (in 2D radius and phi angle), despite no cylindrical division is defined in the geometry. It creates intervals as a function of the cylindrical radius and the phi angle with the parameters given at score definition: *NSLICES_Z MIN_Z MAX_Z NSLICES_PHI MIN_PHI MAX_PHI* and calculates the volume of each cylindrical section so that dose can be computed. Two more parameters are mandatory, to define the dimension in radius: *MIN_R MAX_R*. Three extra parameters are optional: *CENTRE_X CENTRE_Y CENTRE_Z*, if you do not want the cylinder to be centred at the origin of coordinates, (0,0,0), and another three: *ZAXIS_X ZAXIS_Y ZAXIS_Z*, if the you do not want the cylinder axis to be along the Z global axis, (0,0,1). The index of the scorer is build as $NSLICES_Z * indexPHI + indexZ$.
- *Current and flux scorers* - There are two different definitions of a particles flow for a given geometry. One is a current and the other is a flux. In our scorers, the current is simply defined as the number of particle's (with

the particle's weight) passing through a certain surface or volume, while the flux takes the particle's injection angle to the geometry into account (dividing the area of the surface that is traversed and the cosine of the angle between the track direction and the surface normal). The current and flux are usually defined at a surface, but volume current and volume flux are also provided. - *GmPSSurfaceFlux*: This is a surface based scorer.

The quantity is the flux, i.e. the number of tracks that reach the surface, divided by the area and the cosine of the angle between the track and the surface normal. Instead of the flux, the user may choose not to divide by the area, by setting the parameter: */gamos/setParam SCORER_NAME:DivideByArea 0* And also not to divide by the cosine of the angle between the track direction and the surface normal, by setting the parameter: */gamos/setParam SCORER_NAME:DivideByAngle 0* It serves to score the flux in the surfaces of a box, tube or sphere. The user can choose the direction of the particle to be scored with the parameter */gamos/setParam SCORER_NAME:Direction DIRECTION* The choices are *In*, *Out* or *InOut*. *In* scores incoming particles to the volume, while *Out* scores only outgoing particles from the volume. *InOut* scores both directions. The user should also define the list of volume surfaces in which the scoring will be done: */gamos/setParam SCORER_NAME:Surfaces SURFACE_1 SURFACE_2 ...* These surfaces depend on the volume solid: *Box - X+*: YZ plane at positive X - *X-*: YZ plane at positive X - *Y+*: XZ plane at positive Y - *Y-*: XZ plane at positive Y - *Z+*: XY plane at positive Z - *Z-*: XY plane at positive Z *Tube - INNER*: inner radius surface - *OUTER*: outer radius surface - *PHI*: if the full phi (360 degrees) is not defined, it is the two surfaces composed of the points of same phi that limit the tube volume - *UP*: the upper surface - *DOWN*: the lower surface *Sphere - INNER*: inner radius surface - *OUTER*: outer radius surface - *THETA*: if the full theta (180 degrees) is not defined, it is the two surfaces composed of the points of same theta that limit the sphere volume - *PHI*: if the full phi (360 degrees) is not defined, it is the two surfaces composed of the points of same phi that limit the sphere volume

- *GmPSSurfaceFlux*: This is a surface based scorer. The quantity is the flux, i.e. the number of tracks that reach the surface, divided by the area and the cosine of the angle between the track and the surface normal.
- *GmG4PSPassageCellCurrent*: Passage current is a volume-based scorer. The current is defined by the number of tracks that pass through the volume. .. COMMENT: %A passage current is defined for a volume.
- *GmG4PSCellFlux*: Cell flux is a volume based flux scorer. The cell flux is defined by a track length (L) of the particle inside a volume divided by the volume (V) of this cell. The track length is calculated by a sum of the step lengths in the volume. The expression for cell flux is given by the sum of $(W*L)/V$, where W is a particle weight, and is multiplied by the track length at each step. This scorer is also called *GmPSVolumeFlux*.
- *GmG4PSPassageCellFlux*: Passage cell flux is a volume based scorer similar to *G4PSCellFlux*. The only difference is that tracks which pass through a cell are taken into account. It means that tracks generated or stopped inside the volume are excluded from the calculation.
- *In/Out behaviour* For the scorers *GmPSSurfaceFlux* and *GmG4PSTrackCounter* you can make the scoring only for tracks that are entering, only for tracks that are exiting or both for tracks that are entering or exiting (default behaviour). To select among these three options you can add an extra parameter when defining the scorer that can be *In*, *Out* or *InOut*
- *Other scorers - GmG4PSMinKinEAtGeneration*: This scorer records the minimum kinetic energy of secondary particles at their production point in the volume in an event. This primitive scorer does not integrate the quantity, but records the minimum quantity. - *GmG4PSNofSecondary*: This class scores the number of secondary particles generated in the volume. A particle weight is not applied by default.

The user can choose if the scoring is done for all types of particles (default) or only for a set of particles, by naming them as extra parameters.

- *GmG4PSNofStep*: This class scores the number of steps in the cell. A particle weight is not applied by default. If an extra parameter is set to TRUE those steps with step length zero will not be taken into account.

- *GmG4PSCellCharge*: This class scores the total charge of particles which have stopped or have been created in the volume, i.e. the tracks that enter count as +1 and the tracks that exit count as -1.
- *GmG4PSTrackCounter*: This class scores the number of tracks in a cell.
- *For Event Biasing* Scoring for event biasing is a very specific use case whereby particle weights and fluxes through importance cells are required. The goals of the scoring technique are to: - appraise particle quantities related to special regions or surfaces, - be applicable to all "cells" (physical volumes or replicas) of a given geometry, - be customizable. .. COMMENT: %Standard scoring is provided for quantities such as tracks entering a cell, average weight of entering tracks, energy of entering tracks, and collisions inside the cell. A number of scorers have been created for this specific application: - *GmG4PSNofCollision*: This scorer records the number of collisions that occur within a scored volume/cell. - *GmG4PSPopulation*: This scores the number of tracks within in a given cell per event. A particle weight is not applied by default. - *GmG4PSTermination*: This scores the number of tracks that are terminated in a given cell per event. A particle weight is not applied by default.
- *From data - GmG4PSData*: This score uses a GAMOS data as quantity to score. See section on *GAMOS data*.

All scorers that must divide by the volume to obtain the relevant quantity use by default the cubic volume of the physical volume in which the particle is being propagated. If you want to use instead the total volume of all copies of the logical volume with the same name as the volume the particle is in, you have to set the parameter:

```
/gamos/setParam SCORER_NAME:IntegrateVolumes 1
```

Scoring in voxelised phantoms

When you are navigating in a voxelised pantom geometry and you are using the regular navigation you have the option of skipping voxel frontiers when the material does not change (see section on *Reading DICOM files*). In this case the scorer *GmG4PSDoseDeposit* will automatically distribute in the voxels traversed the dose corresponding to the energy deposited along each step. A correction is nevertheless needed: when the particle travels it loses energy, and therefore it is not right to distribute the dose proportionally to the length traversed inside each voxel. The needed corrections to the multiple scattering and energy loss when the particle loses energy are done in an iterative way: the increase of multiple scattering increases the path length, what increases the energy lost, what makes the multiple scattering even bigger, what increases the path length and therefore the energy lost. By default the number of iterations is two, but you may change it with the parameter:

```
/gamos/setParam GmEnergySplitter:NIterations NITER
```

Other scorers need the same corrections, namely *GmG4PSEnergyDeposit*, *GmG4PSEnergyLost*, *GmG4PSKerma* and *GmG4PSTrackLength*.

In this case the index used is the copy number of the voxels traversed, what means that the classifier index is not used at all. You may nevertheless force the use of the classifier index with the parameter:

```
/gamos/scoring/printer SCORER_NAME:UseClassifierIndex 1
```

but you have to be conscient that if you do so the score will not be distributed in the voxels traversed, even if the classifier is *GmClassifierBy1Ancestor*:

Other scorers are also affected by the feature of skipping voxel boundaries:

- *GmG4PSNofSecondary*: the score is done at the last voxel traversed, while using the default classifier *GmClassifierBy1Ancestor* would assign it to the one corresponding to the origin of the step, that is the first voxel.
- *GmG4PSNofCollision*: the same as *GmG4PSNofSecondary*.
- *GmG4PSNofStep*: a score is assigned to each one of the voxels traversed.
- *GmG4PSTrackCounter*: for the first step it is always that the track exited; for intermediate voxels it is always that the track entered and exited; for the last voxel it is always that the track entered.

The flux and current scorers are not corrected, as it likely has no sense to calculate the flux entering and exiting voxels. Please contact the GAMOS team if you think you need this feature.

Filter classes

See section on *Filters*.

Scorer printers

A scorer printer serves to select the format of the output of a scorer. These classes are unique to GAMOS, as Geant4 does not provide this functionality. As mentioned above, several printers can be associated to the same scorer.

To associate a printer to a scorer, you can use the command mentioned above:

```
/gamos/scoring/addPrinter2Scorer PRINTER_NAME SCORER_NAME
```

The name of the printer is the name of the printer class (see list below). If you want to change the printer name, or you want to add some parameters to a printer, you can use the command

```
/gamos/scoring/printer PRINTER_NAME PRINTER_CLASS PARAMETERS
```

where *PRINTER_NAME* is the new name you want to give to the printer and *PRINTER_CLASS* is the name of the printer class (see list below).

The general-use scorer printers currently in GAMOS are the following:

- *GmPSPrinterG4cout*: Prints in the standard output the summary of scoring: - "MultiFunctionalDet: " Detector name - "PrimitiveScorer: " Scorer name - "Number of entries: " Number of scores - For each per score one line with:
 - "index: " IndexName " = " Score value " +-(REL) " Relative score error (=error/value) " sumV2= " Square of score values. This last value is needed instead of the error for properly taking into account the correlations when the errors of several files are summed. If you do not want it to be printed you have to set the parameter: */gamos/setParam PRINTER_NAME:PrintSumV2 1*
 - ScorerName " SUM ALL: " Sum of all scores " +-(REL) " Relative error " " Unit name

An example is the following:

```
MultiFunctionalDet: fluxDet
PrimitiveScorer: fluxScorer
Number of entries= 2
index: 1 = 0.0422424 +-(REL) 0.02251 cm-2 sumV2= 108.251
index: 2 = 0.010207 +-(REL) 0.0710814 cm-2 sumV2= 53.6754
fluxScorer SUM ALL: 0.0524494 +-(REL) 0.00119606 cm-2
```

where *MFD_NAME* is the name of multifunctional detector and *SCORER_NAME* is the name of the scorer. The columns after *index* have the following meaning: Index, scorer value, scorer error (relative, i.e. error/value), unit name. At the end it is printed the sum of all scores, with error (non relative).

- *GmPSPrinterTextFile*: Prints in a text file the same information than the *GmPSPrinterG4cout* printer. The name of the file is *PRINTER-NAME.out*. It can be changed with the parameter */gamos/setParam PRINTER-NAME_SCORER-NAME:FileName FILE-NAME*
- *GmPSPrinterBinFile*: Prints in a binary file almost the same information than the *GmPSPrinterG4cout* printer. The format of the data is the following: - Detector name (char[30]) - Scorer name (char[20]) - Unit name (char[10]) - Unit value (float) - Number of entries (unsigned int) For each classifier index: - Index name (char[20]) - Score value (float) - Square of score value (float). This value is needed instead of the error for

properly taking into account the correlations when the errors of several files are summed. The name of the file is *PRINTER_NAME.out*. It can be changed with the parameter */gamos/setParam PRINTER-NAME_SCORER-NAME:FileName FILE-NAME*

- *GmPSPrinterHistos*: Prints scoring data in histograms. The name of the file is *PRINTER_NAME.root*. It can be changed with the parameter */gamos/setParam PRINTER_NAME:FileName FILE_NAME*. The user must define the histogram name, number of bins and axis minimum and maximum, giving them as arguments of the */gamos/scoring/printer* command (this means that the class *GmPSPrinterHistos* cannot be used directly). The scoring data corresponding to index *IDX* will be used to set the content of the histogram bin *IDX*. It should be noted that the first histogram bin is number 1, while it may happen that the first index is not 1. If this is the case, a number (positive or negative) can be added to the index by defining an offset as a parameter: */gamos/setParam PRINTER_NAME:OffsetX OFFSET*. If and only if you are using a *GmCompoundClassifier* built from two classifiers, you may want to produce a 2-dimensional histogram. In this case you should give eight instead of four parameters to the */gamos/scoring/printer* command: the first four should correspond to the data of the first classifier and the second four to the second one. Then three histograms will be filled: the compound index will be split in the two indices (as defined with the *NShift* of the *GmCompoundClassifier*); then an histogram will be filled for the first index, another for the second one and a 2-dimensional histogram combining the two indices. A second offset may be defined for the second index: */gamos/setParam PRINTER_NAME:OffsetY OFFSET*

Other scorer printers are provided for specific applications. See corresponding sections in this guide.

Classifiers

See section on *Filters*.

Multiplying by data

You can multiply the quantity you are scoring by any of the GAMOS data (see section on *GAMOS data*), so that before scoring the quantity value it will be multiplied by the value of that the GAMOS data has in that step. To do it you have to use the parameter:

```
/gamos/setParam SCORER_NAME:MultiplyByData DATA_NAME
```

where *DATA_NAME* is the name of the data you want to use.

Multiplying by distribution

You can multiply the quantity you are scoring by a GAMOS distribution (see section on *GAMOS distributions*), so that before scoring the quantity value it will be multiplied by the value of that the GAMOS distribution has in that step. To do it you have to use the parameter:

```
/gamos/setParam SCORER_NAME:MultiplyByDistribution DISTRIBUTION_NAME
```

where *DISTRIBUTION_NAME* is the name of a distribution you have previously created.

Convergence testing

The fact of having a small relative error in a score does not always guarantee that it has converged to a good result. This may be specially true when there are contributions of very different weights to the scorer, and the high weight scores have not been properly sampled with your statistics. If you suspect of a wrong behaviour you can activate

the convergence test with the parameter (as these tests consume some CPU time and memory, but by default they are deactivated):

```
/gamos/setParam SCORER_NAME:ConvergenceTester CONVERGENCE_NAME
```

where *SCORER_NAME* is the name of the scorer and *CONVERGENCE_NAME* is the name you want to give to the convergence tester, which will be used in the report. If you are using a point detector scorer, you should substitute *SCORER_NAME* by *GmPDS*.

The convergence tests are taken from the Geant4 code, and are inspired in the MCNP tests. The tests are based in the analysis of the sum of scores at the end of each event. The following variables are printed about the sum of scores :

- **EFFICIENCY* = *Proportion of events that have a non zero value.
- *MEAN* = Average value
- *VAR* = variance of values
- *SD* = standard deviation
- *R* = The estimated relative error = $SD / MEAN / \sqrt{\text{number of values}}$
- **SHIFT* = *Shift in the midpoint of the confidence interval to a higher value = $(\text{SUM}(X_i - MEAN) * (X_i - MEAN) * (X_i - MEAN) - (N - \text{Number_of_non_zero_values}) * MEAN * MEAN * MEAN) / (2 * VAR * N)$
- **VOV* = *Variance of variance = $(\text{SUM}(X_i - MEAN) * (X_i - MEAN) * (X_i - MEAN) * (X_i - MEAN) + (N - \text{Number_of_non_zero_values}) * MEAN * MEAN * MEAN * MEAN) / (VAR * VAR) - 1./N$
- **FOM* = *Figure of merit = $1 / (R * R) / \text{CPU_time_of_last_event}$
- *** THE LARGEST VALUE* and where in which event it happened

To get a feeling of how big are the fluctuations the same variables, i.e. mean, variance, R, shift and FOM, are printed again but adding to the values a new one equal to the largest value (so that this value is counted twice). And also the ratio of this *affected* to the original ones.

Then the results of eight convergence tests are shown:

- *MEAN distribution is RANDOM*
- *r follows 1/std::sqrt(N)*
- *r is monotonically decrease*
- *r is less than 0.1*. This value can be changed with the parameter: `/gamos/setParam GmConvergenceTester:Rmin VALUE`
- *VOV follows 1/std::sqrt(N)*
- *VOV is monotonically decrease*
- *FOM distribution is RANDOM*
- *SLOPE is large enough*

Finally it prints the evolution of several variables, i.e. the change of these variables when more events are added. The variables are printed each N/16 events, where N is the total number of events. The variables printed are the following:

- *mean*
- *var*
- *sd*
- *r*
- *vov*

- *shift*
- *e*
- *r2eff*
- *r2int*

Point detector scorer

The point detector scorer covers the problems where the quantity to be calculated is the flux or the dose of neutral particles (neutrons or gammas) in a very small detector (small with respect to the setup dimensions) situated far from the primary particles source. In this kind of problems the fraction of particles that reach the detector is very reduced, and therefore if one wants to calculate the flux or dose by conventional methods the statistics needed would be prohibitive.

The technique implemented in GAMOS is similar to the F5 tally implemented in MCNP. It is based on the following idea: normal tracks are propagated and for each neutron or gamma interaction is calculated the probability that it would be deviated in the direction of the point detector (instead of the real direction towards which it is deviated) and the probability that it reaches the point detector without any further interaction. The first probability is based on a precalculated table of angle probabilities for each interaction type, each material and each energy, multiplied by the solid angle covered by the point detector. The second probability is based of the number of mean free paths that the neutron or gamma would have to traverse along the path to reach the point detector. A similar calculation is done for each neutron or gamma initial step, except that the angle probability it is based on is simplified as a constant distribution in the local reference system of the parent particle (i.e. probability equal to 0.5). Summing up these probabilities for a number of events, usually several orders of magnitude less than with conventional methods, one can get the flux and the energy spectrum at the point detector, and with these magnitude the equivalent doses (H*, Hp(0,15,...)) can be obtained.

In this chapter we describe first the theoretical basis of the method. Then we explain how it is implemented in GAMOS, how to select the different available options, and which are the possible outputs (tables or histograms). In the third part we describe the several variance reduction techniques that can be used to reduce the CPU time, and give some recommendations on how to use the output table and histograms to help in determining the best variance reduction techniques for a given problem.

Theoretical basis

The point detector scorer calculates the flux at a point based on the probability that particles reach it. Suppose we call $p(\mu, \phi)$ the probability that at a source or neutron/gamma interaction the the particle produced or scattered goes into the solid angle $d\Omega$ about the direction (μ, ϕ) , where $\mu = \cos(\theta)$. if R is the distance to the detector from the source or interaction point

$$p(\mu, \phi)d\Omega e^{\int_0^R \Sigma_t(s)ds}$$

yields the probability that the particle reaches the detector point with no further collisions, where

$$e^{\int_0^R \Sigma_t(s)ds}$$

is the attenuation of a beam of monoenergetic particles passing through a material medium, where s is measured along the direction from the source or interaction point to the detector and $\Sigma(s)$ is the macroscopic total cross section at s. If dA is an element of area normal to the line of flight to the detector, $d\Omega = dA/R^2$. As the flux is the number of particles passing through a unit area normal to the line of flight direction, we can write the flux as

$$\Phi = p(\mu, \phi)e^\lambda/R^2$$

If there is azimuthal geometry, then $p(\mu, \phi) = p(\mu)/2\phi$, and we can finally write

The attenuation term, $d\Omega = dA/R^2$. As the flux is the number of particles passing through a unit area normal to the line of flight direction, we can write the flux as

$$\Phi = p(\mu, \phi)e^\lambda/R^2$$

If there is azimuthal geometry, then $p(\mu, \phi) = p(\mu)/2\phi$, and we can finally write

The attenuation term, e^λ is calculated summing up the number of interaction lengths from the source or interaction point until the detector; this calculation depends on the neutron/gamma energy and the materials traversed, and it is computed in GAMOS through the use of geantino

A geantino is a pseudoparticle that has only transportation, but no physical process. .

The R2 term in the denominator causes a singularity: when a source or interaction event occurs near the detector point, R approaches zero and the flux approaches infinity. The technique is still valid and unbiased, but convergence is slower and often impractical. To avoid this singularity, a fictitious sphere of radius R0 surrounding the detector point is defined (which we call the exclusion sphere). If we can assume that the flux is uniformly distributed inside this sphere, then it can be demonstrated that the average flux in the sphere is

$$\Phi(R < R0) = \frac{p(\mu)(1 - e^{-\sum_t R_0})}{\frac{2}{3}\pi R_0^3 \sum_t}$$

The exclusion sphere radius has a default value of 1 mm, which can be changed with the parameter

```
/gamos/setParam GmPDS:ExclusionRadius RADIUS
```

GAMOS implementation

The point detector scorer is selected as other GAMOS scorers, although, as explained below, it does not really behave as the other scorers. To activate you have to select the user action:

```
/gamos/userAction GmPDSUA
```

But before this command you have to set the parameters that drive the behaviour of the point detector scoring. You may first decide if you want to score the flux for neutrons, gammas or for both with the two parameters

```
/gamos/setParam GmPDS:ScoreNeutrons 1/0
```

```
/gamos/setParam GmPDS:ScoreGammas 1/0
```

There are several parameters that you may change distinctly for neutrons or gammas, or that you may want to set the same value for both. All the parameters start with *GmPDS* and then they are followed by *neutron:* or *gamma:*; if the second word is not one of these two, then the parameter serves for both. We will then omit this word in the explanation of the parameters in this section, but remember you can always add it, except in the cases where it is explicitly mentioned that there is a unique parameter for both particles.

Creating angle deviation probability densities

The first thing you should do before running your point detector scoring job, is to create a file that contains the probability density functions of the emission angles for each energy, process and material. Else GAMOS will assume flat probabilities for all angles (what may be enough in your case).

To do this you can run a job that writes in a ROOT file the probability of emission angle for all the materials that constitute your setup. You may follow the example found in *tutorials/ShieldingTutorial/exercise3/exercise3.angle.gg.in*, that we explain here. The class that makes the work is called *GmPDSCreateAngleTablesUA*, but remember that the parameters have to go before the class that uses them, so you should add this line at the end of your script

```
/gamos/userAction GmPDSCreateAngleTablesUA
```

The histogram will only be saved for the secondary particle type defined with the parameter

```
/gamos/setParam USER_ACTION_NAME:SecondaryParticleName NAME
```

The histogram file name is by default *angleDeviation*, but it can be changed by the parameter

```
/gamos/setParam USER_ACTION_NAME:HistoFileName NAME
```

To create the tables for different materials and tables you may use the generator *GmGeneratorChangeEnergyAndMaterials* (see section on *Event generator changing energy and material*).

If you have in your setup several particles that produce neutrons (or gammas) you should repeat the process with different primary particle types. Then you may add all histograms in a file by using the ROOT command

```
hadd MERGED_FILE.root FILE1.root FILE2.root ...
```

The file created in this way should be passed to the point detector scorer.

```
/gamos/setParam GmPDS:AngleDeviationFileName FILE_NAME
```

the default name is *angleDeviation.neutron.root* for neutrons and *angleDeviation.gamma.root* for gammas.

When this file is read it will be checked if the number of events in each histograms is big enough, else the histogram will not be used. You may control the minimum number of events in the histogram with the parameter:

```
/gamos/setParam GmPDS:InteractionAngleManager::MinimumNumberOfEntries VALUE
```

Point detector volume

The detector where the flux will be scored has to be a real volume in your geometry. It should be an small volume, as small as you want, although you may decide to make it the size of the real detector in your experiment. It is important to take into account that despite the flux is calculated as the probability that tracks reach the detector, if a track actually reaches it, it will also be counted. The name of the detector volume is given with the parameter

```
/gamos/setParam GmPDS:DetectorName VOLUME_NAME
```

If you want to score the flux in several points you may place the detector volume at several places.

Energy and dose equivalent bins

The flux will be scored in different energy bins. These energy bins should be defined in a file whose name is

```
/gamos/setParam GmPDS:EnergiesFileName FILE_NAME
```

The format of this file is a set of one-column lines with the energy values.

There are several dose equivalent magnitudes that can be calculated, namely the ambient dose and the personal dose at several angles: 0, 15, 30, 45, 60, 76. To select them you have to set the following parameters (not a different one for neutrons and gammas)

```
/gamos/setParam GmPDS:PrintHstar 1
```

```
/gamos/setParam GmPDS:PrintHp0 1
```

```
/gamos/setParam GmPDS:PrintHp15 1
```

```
/gamos/setParam GmPDS:PrintHp30 1
```

```
/gamos/setParam GmPDS:PrintHp45 1
```

```
/gamos/setParam GmPDS:PrintHp60 1
```

```
/gamos/setParam GmPDS:PrintHp75 1
```

For each magnitude you have to define the conversion factor for each energy bin. These are defined in a file whose name is given by the parameter

```
/gamos/setParam GmPDS:Flux2DoseFileName FILE_NAME
```

This file must contain in each line the energy and the conversion factors in $pSv\ cm^2$ for the magnitudes: H*(10) H_pslab(10,0) H_pslab(10,15) H_pslab(10,30) H_pslab(10,45) H_pslab(10,60) H_pslab(10,75). The default name of this file is *Flux2Dose.neutron.ICRU57.lis* for neutrons and *Flux2Dose.gamma.ICRU57.lis* for gammas. These two files contain the conversion factor given by the ICRU57 report, and can be found in the data directory of the GAMOS release. By default these files will also be used to read the list of energy bins to be used for the flux results.

Scoring with filters

As for any user action, you may add one or more filters so that only track steps that pass them will be scored. The filter is applied to the neutron or gamma that creates the geantino, not to the geantino that reaches the detector.

By default the filter will act on the *G4Track*, but it may act on the *G4Step* if the following parameter is set.

```
/gamos/setParam GmPDS:FiltersOnTrack 0
```

Scoring per category

The scoring of flux and equivalent dose can be split following different criteria.

The first possibility is to score separately the contributions from the real particles that reach the point detector and the indirect contributions calculated with the geantinos. The first one will have in the score name the word *Neutron* or *Gamma*, while the second one will have the word *Geantino*. To activate this option the following parameter should be used

```
/gamos/setParam GmPDS:ScoreSeparatelyTrueAndGeantino 1
```

The second possibility is use a classifier and get a different score for each classifier index. You can add a classifier to *GmPDSUA* as you do for any other user action. By default the classifier will act on the **G4Track*, but it may act on the *G4Step* if the following parameter is set.

```
/gamos/setParam GmPDS:ClassifierOnTrack 0
```

Output format

At the end of the run the results are written in the standard output or in a file given by the parameter

```
/gamos/setParam GmPDS:ResultsFileName FILE_NAME
```

The following table is written for each score or neutrons or gammas (one per point detector copy, one per category as describe above). First a line with the filter names, classifier name, particle type, detector name and copy, score type (named *ALL* if it is not a sub category), the detector position and the number of events in the job. Then follows a line for each energy with the filter names, classifier name, particle type, detector name and copy, score type, energy value, flux value, relative flux error (error divided by value), number of particles contributing to the flux and then, for later statistical processing, sum of flux values squared, and to the third and fourth power. After all energies comes a line with the total sum of flux in all energies, which has the filter names, classifier name, particle type, detector name and copy, score type, the words *FLUX_TOTAL/particle* followed by the total flux value and relative error and the total number of particles contributing to the flux. If the dose equivalent magnitudes are required, they come at the end in a line starting also with the filter names, classifier name, particle type, detector name and copy and score type followed by the magnitude name (*Hstar*, *Hp(10,0)*, *Hp(10,15)*, ...). An example output is the following

```

% % % % SCORE IN POINT DETECTOR FOR set ALL: NeutronInelastic: gamma: PD1: ALL: at (4800,0,0) NEVENTS=
ALL: NeutronInelastic: gamma: PD1: ALL: ENERGY= 0.01 FLUX= 0 +-(REL) 0 N 6 Fwei2 0 Fwei3 0 Fwei4 0
ALL: NeutronInelastic: gamma: PD1: ALL: ENERGY= 0.015 FLUX= 0 +-(REL) 0 N 0 Fwei2 0 Fwei3 0 Fwei4 0
...
ALL: NeutronInelastic: gamma: PD1: ALL: FLUX_TOTAL/particle= 1.33272e-13 +-(REL) 0.97118 N 12
ALL: NeutronInelastic: gamma: PD1: ALL: Hstar= 1.49381e-13 pSv/particle

```

Control Histograms

There are several optional histograms that may help you in better understanding the behaviour of your point detector scoring. By default they are created, but you can switch them off with the parameter

```
/gamos/setParam GmPDS:ControlHistograms 0
```

The histograms are written in a file named *pds*, but this name can be changed with the parameter:

```
/gamos/setParam GmPDS:HistosFileName VALUE
```

All histograms start with the filter names, classifier name and particle type. There is an histogram about the particle interaction.

- *interaction log Energy (MeV)*: Kinetic energy of neutrons/gamma at each step initial position.

A set histograms are about the distance to each detector, so they also have the detector name and copy:

- *interaction dist to detector (mm)*: Distance from each neutron interaction or origin to the detector.
- *interaction dist to detector (mm) weighted by Hstar*: Distance from each neutron interaction or origin to the detector, weighted by Hstar.
- *interaction dist to detector (mm) vs weight*: Distance from each neutron interaction or origin to the detector versus weight when detector is reached.

The rest of histograms are for each detector copy and score type. The variables are calculated when the geantino or neutron/gamma reaches the detector, so they have the name *At detector*, and they also have the score type. There is a set for the geantinos and another set for the neutron/gamma particles. They are the following:

- *log10(energy) (MeV)*: log10 of track kinetic energy times the weight.
- *log10(energy) no weighted (MeV)*: log10 of track kinetic energy.
- *log10(energy) weighted by Hstar*: log10 of track kinetic energy times the weight and Hstar value.
- *log10(weight)*: log10 of track weight.
- *log10(weight) weighted by Hstar*: log10 of track weight times the weight and Hstar value.
- *log10(energy) vs log10(weight)*: log10 of track kinetic energy versus track weight.

Variance reduction techniques

There are three variance reduction techniques available, that will provide a better efficiency, i.e. less CPU time for the same error.

Kill contributions of low weight

The first technique consist on not taking into account the contributions of low weight. It works checking the weight of the geantinos at each step of their propagation (each time they change volume) and playing Russian roulette with them if their way is smaller than the value given by the parameter

/gamos/setParam GmPDS:UseMinimumGeantinoWeight MIN_WEIGHT

which by default takes a value of 1.E-30. The Russian roulette factor is given by the parameter

/gamos/setParam GmPDS:MinimumGeantinoWeightRRFactor VALUE

which by default takes a value of 100. To activate this technique you have to set the parameter

/gamos/setParam GmPDS:UseVRMinimumGeantinoWeight 1

You may use the control histograms, specially the histogram *log10 of geantino kinetic energy versus geantino weight* to determine which are the minimum weight and Russian roulette factor that best match your application.

Kill contributions too far from detector

A more efficient technique consists on not taking into account the contributions of interactions or sources that are too far from the detector, without the need to send a geantino to calculate the probability to reach it. But this technique assumes that the probability is smaller if the point is farther from the detector, what is not always the case, as it may be that a point closer to the detector traverses denser materials. Indeed the contributions are not eliminated, but, to avoid biasing Russian roulette is played. The maximum distance is set with the parameter

/gamos/setParam GmPDS:MaximumDistance DISTANCE

which by default takes a value of 1000 mm. The Russian roulette factor is given by the parameter

/gamos/setParam GmPDS:MaximumDistanceRRFactor VALUE

which by default takes a value of 100. To activate this technique you have to set the parameter

/gamos/setParam GmPDS:UseVRMaximumDistance 1

You may use the control histograms, specially the histogram *Neutron: PD N: interaction dist to detector (mm) vs weight* to determine which are the maximum distance and Russian roulette factor that best match your application.

Split particles of big weight

The idea of this technique is to sample more those cases which bigger probabilities. It may be very useful also to eliminate the seldomly occurring big weight contributions that will make your error big. The user defines a minimum and maximum weight for the particles to be split:

/gamos/setParam GmPDS:MinimumWeightForSplitting MIN_VALUE

which by default takes a value of 1.E-30

/gamos/setParam GmPDS:MaximumWeightForSplitting MAX_VALUE

which by default takes a value of 1.E-5; and the maximum splitting number:

/gamos/setParam GmPDS:MaximumSplitting MAX_SPLIT

which by default takes a value of 100.

Each time a geantino reaches the detector, its weight will be checked. If it is between the minimum and maximum, the splitting number will be calculated making a logarithmic interpolation using 0 at the minimum weight and the maximum splitting number at the maximum weight. If this number, nSplit, is bigger than one, nSplit-1 particles will be created at the position, direction, energy and time of the neutron or gamma that created the geantino.

To activate this technique you have to set the parameter

/gamos/setParam GmPDS:UseVRSplitting 1

You may use the control histograms, specially the histogram *SCORE_NAME At detector : geantino : log10(energy) vs log10(weight)* to determine which are the minimum and maximum weight and splitting number that best match your application.

Sum scoring results and plot them

If you want to sum the scoring results from several jobs, for example, because you have split a long job in several ones, there is an utility to do it. You first have to build a file with the list of files you want to sum, one line per file. Then you have to run the utility *sumScores*. This utility will scan all files and will look for all scorers with the same name (same *MultiFunctionalDet:* and *PrimitiveScorer:*). It merges this scorers calculating the average of each score. It also propagates the error and gives it relative to the value (divided by it); and it also gets the error as the deviation of the values, i.e. as the

VARIANCE REDUCTION TECHNIQUES

Introduction

Variance reduction are the techniques that allow to get the same precision of the estimates of a Monte Carlo simulation reducing the CPU time and without introducing a bias in the result. The usual criteria to measure the gain of a variance reduction is the efficiency, defined as the CPU time divided by the square of the error or the variable of interest:

$$\epsilon = \frac{T}{\sigma}$$

Importance sampling

Importance sampling is a variance reduction techniques that consists in estimating the properties of a particular distribution, while using a different sampling than the distribution of interest. It may help you saving CPU time if you choose to sample more times when the contribution to the distribution is bigger. An example can be a case when you want to estimate the dose of particles that reach a volume and you sample more particles when the distance to reach that volume is smaller; you may do this by duplicating a particle N times (of courser reducing its weight 1/N) when the particle comes closer to the volume, calculating N as an inverse of the distance to the volume.

In GAMOS you may do importance sampling using many different criteria: any of the GAMOS data can be used, as the importance sampling process uses a GAMOS distribution, taking the output value of the distribution as the splitting value that corresponds to each input value. If the splitting value (SPLIT_VALUE) is bigger than 1 it will duplicate the current particle SPLIT_VALUE-1 times (producing SPLIT_VALUE equal particles, with a weight reduced by 1/(SPLIT_VALUE)). If the splitting value is smaller than 1, Russian roulette will be played with the particle with a surviving probability SPLIT_VALUE; if it survives its weight will be incremented by 1/(SPLIT_VALUE).

You may activate the importance sampling for a list of particles with the command:

```
/gamos/physics/VR/importanceSampling NAME DISTRIBUTION_NAME PARTICLE_NAME_1 PARTICLE_NAME_2  
...
```

where the NAME you give to the importance sampling will be later used to set the parameters.

As the number of particle duplications will be set by the classifier index, you may want to set a different index than the default of the classifier. See section on *Classifiers* to learn how to do this.

Several options can be controlled with GAMOS parameters. First you may decide to deactivate the splitting, leaving only Russian roulette, or viceversa, with the parameters:

```
/gamos/setParam IMPORTANCE_SAMPLING_NAME:ApplySplitting 0  
/gamos/setParam IMPORTANCE_SAMPLING_NAME:ApplyRussianRoulette 0
```

If you do not want that a particle already split is split again, you may control how many times a particle is split with the parameter:

```
/gamos/setParam IMPORTANCE_SAMPLING_NAME:MaxSplitTimes VALUE
```

You may also set that the same particle is not split several times, by setting the parameter:

```
/gamos/setParam IMPORTANCE_SAMPLING_NAME:SplitAtSeveralSteps
```

Finally you may add one or more filters, so that particles have to pass them before being split:

```
/gamos/setParam IMPORTANCE_SAMPLING_NAME:Filters FILTER_1 FILTER_2 ...
```

Geometrical biasing

Many types of geometrical biasing can be done with GAMOS using the importance sampling and assigning different distributions with different data. For example you may use a different weight for each physical volume, logical volume or touchable or you can divide your space in bins along X, Y or Z or combinations of these variables. You may also consider the special distribution *GmGeometricalBiasingDistribution* (see chapter on *Distributions*), that calculates the weight as the division of the weights of the entering and exiting volumes.

Biasing operations

GAMOS provides commands to use the Geant4 biasing techniques in an easy way. Moreover several extra biasing techniques are implemented in GAMOS non existing in Geant4. We describe here those techniques and how to use them.

Following the Geant4 notation, the first thing you have to create a biasing operator, with the command:

```
/gamos/physics/biasing/createOperator OPERATOR_NAME OPERATOR_TYPE
```

where *OPERATOR_NAME* is the name you choose for your operator, and *OPERATOR_TYPE* has to be one of the following ones:

- *ChangeCrossSection*:
- *ForceCollision*
- *BremsSplitting*
- *DirBremsSplitting*
- *EWPS*

These biasing operators will only valid for the particle processes selected by the user. Therefore you have to select the particles and processes. This can be done with three different commands:

Select a certain process of a certain particle:

```
/gamos/physics/biasing/addParticleProcesses2Oper PARTICLE_NAME PROCESS_NAME OPERATOR_NAME
```

Select all the processes of a certain particle:

```
/gamos/physics/biasing/addProcesses2Oper PROCESS_NAME OPERATOR_NAME
```

Select a certain process of all particles :

```
/gamos/physics/biasing/addParticle2Oper PARTICLE_NAME OPERATOR_NAME
```

Finally after the */run/initialize* command (when the geometry is already built) you have to attach the biasing operator to one or several logical volumes, so that biasing will only occur when a particle is being tracked in one of those volumes:

```
/gamos/physics/biasing/associateOper2LogVol LOGICAL_VOLUMES OPERATOR_NAME
```

Example:

```
/gamos/physics/biasing/createOperator boper ForceCollision
/gamos/physics/biasing/addParticleProcesses2Oper gamma compt boper
/run/initialize
/gamos/physics/biasing/associateOper2LogVol crystal boper
```

Cross section biasing

The cross section of any process of any particle can be changed by using the biasing operator *ChangeCrossSection*. The cross section will be multiplied by a factor that has to be chosen by the user with the GAMOS parameter

```
/gamos/setParam OPERATOR_NAME:XSfactor VALUE
```

Force collision

You can force a process to happen in a volume by using the biasing operator *ForceCollision*. We copy here the description of this biasing from the Geant4 user's manual.

G4BOptrForceCollision is a biasing operator that implements a force collision scheme quite close to the one provided by MCNP. It handles the scheme through the following sequence:

1. The operator starts by using a G4BOptrCloning cloning operation, making a copy of the primary entering the volume. The primary is given a zero weight.
2. The primary is then transported through to the volume, without interactions. This is done with the operator requesting forced free flight G4BOptrForceFreeFlight operations to all physics processes. The weight is zero to prevent the primary to contribute to scores. This flight purpose is to accumulate the probability to fly through the volume without interaction. When the primary reaches the volume boundary, the first free flight operation restores the primary weight to its initial weight and all operations multiply this weight by their weight for non-interaction flight. The operator then abandons here the primary track, letting it back to normal tracking.
3. The copy of the primary track starts and the track is forced to interact in the volume, using the G4BOptrForceCommonTruncatedExp operation, itself using the total cross-section to compute the forced interaction law (exponential law limited to path length in the volume). One of the physics processes is randomly selected (on the basis of cross-section values) for the interaction.
4. Other processes are receiving a forced free flight operation, from the operator.
5. The copy of the primary is transported up to its interaction point. With these operations configured, the G4BiasingProcessInterface instances have all needed information to automatically compute the weight of the primary track and of its interaction products. As this operation starts on the volume boundary, a single force interaction occurs: if the track survives the interaction (e.g Compton process), as it moved apart the boundary, the operator does not consider it further.

Uniform bremsstrahlung splitting

Bremsstrahlung splitting is a technique aimed to increase the statistics of gammas produced by bremsstrahlung in the region of interest, while reducing the time spent in tracking the electrons. It is a technique very frequently used to simulate radiotherapy linear accelerators. To use this technique you have to select the biasing operator *BremsSplitting*. The behaviour of this operator is controlled with a few parameters:

```
/gamos/setParam OPERATOR_NAME:NSplit VALUE: number of gammas that will be produced at each
bremsstrahlung interaction. The weight of each gamma is set to 1./VALUE.
```

/gamos/setParam OPERATOR_NAME:BiasOnlyOnce 0/1: if set to 1 (default is 1) when the particle that suffers a bremsstrahlung collision is the descendant of a particle that already was splitted (e.g. electron(bremsstrahlung) -> gamma(compton) -> electron) is is not splitted.

/gamos/setParam OPERATOR_NAME:BiasPrimaryOnly 0/1: if set to 1 (default is 0) only primary particles will be splitted.

Directional bremsstrahlung splitting

This technique first defines a rectangle in the XY plane situated at a selected Z coordinate. Then for each of the gammas generated by bremsstrahlung splitting it checks if its directions points towards this rectangle and if not plays Russian roulette with it. The result is that you end with many gammas in the region of interest and only a few gammas out of it. To use this technique you have to select the biasing operator *DirBremsSplitting*. The behaviour of this operator is controlled with a few parameters:

/gamos/setParam OPERATOR_NAME:NSplit VALUE: number of gammas that will be produced at each bremsstrahlung interaction. The weight of each gamma is set to 1./VALUE.

/gamos/setParam OPERATOR_NAME:XDim VALUE: dimension of the rectangle of interest in the X coordinate (rectangle extends from -X to +X)

/gamos/setParam OPERATOR_NAME:YDim VALUE: dimension of the rectangle of interest in the Y coordinate (rectangle extends from -Y to +Y)

/gamos/setParam OPERATOR_NAME:ZPos VALUE: Z position of the XY plane where the rectangle of interest is defined

/gamos/setParam OPERATOR_NAME:NKill VALUE: number used to play Russian roulette (probability is 1./VALUE). By default it is the same as NSplit

/gamos/setParam OPERATOR_NAME:BiasOnlyOnce 0/1: if set to 1 (default is 1) when the particle that suffers a bremsstrahlung collision is the descendant of a particle that already was splitted (e.g. electron(bremsstrahlung) -> gamma(compton) -> electron) is is not splitted.

/gamos/setParam OPERATOR_NAME:BiasPrimaryOnly 0/1: if set to 1 (default is 0) only primary particles will be splitted.

Equal weight particle splitting

When bremsstrahlung splitting is used it often happens that some of the particles that contribute to the magnitude of interest (e.g. dose in a radiotherapy planning) have been splitted and have an small weight, while others have survived Russian roulette and their weight has been set back to 1. The fact of having particles of different weights contributing to a magnitude increases substantially the total error and therefore should be avoided. This technique is based in the one described at [\[biblio.dbs\]](#), and it handles the weights of all secondary particles created so that those that reach the region of interest have the same weight.

To use this technique you have to select the biasing operator *EWPS*. The behaviour of this operator is controlled by the same parameters than the direction bremsstrahlung technique described above.

To assure that all particle weights are treated correctly, it is necessary to include the following processes:

/gamos/physics/biasing/EWPS/addParticleProcesses2Oper e- eBrem OPERATOR_NAME

/gamos/physics/biasing/EWPS/addParticleProcesses2Oper e+ annihil OPERATOR_NAME

/gamos/physics/biasing/EWPS/addParticleProcesses2Oper gamma compt OPERATOR_NAME

/gamos/physics/biasing/EWPS/addParticleProcesses2Oper gamma conv OPERATOR_NAME

/gamos/physics/biasing/EWPS/addParticleProcesses2Oper gamma phot OPERATOR_NAME

HISTOGRAMMING

Histogram formats

GAMOS supports several data analysis formats. The format is selected at run time by the user, so that the same C++ code can be used to write any format. In this GAMOS version there are two formats implemented, ROOT and CSV (Comma Separated Value). For the ROOT format, we refer you to the ROOT documentation [[biblio.root](#)]. The CSV format is explained below.

You can choose which format to use with the command

```
/gamos/analysis/fileFormat FORMAT
```

where *FORMAT* can be *ROOT*, *root*, *CSV*, *csv*

You can also use several formats at the same file, using the command

```
/gamos/analysis/addFileFormat FORMAT
```

Whatever format you choose, you can also get a print out in the screen of the main parameters of each histogram using the command

```
/gamos/analysis/printHisto VERBOSITY_LEVEL
```

where *VERBOSITY_LEVEL* is the level of verbosity:

- 0: Prints number of entries, mean, RMS, underflow and overflow
- 1: Prints number of entries, mean with error, RMS with error, underflow and overflow
- 2: Prints number of entries, mean with error, RMS with error, underflow and overflow. Also number of bins, axis lower and upper edges

Histograms in CSV format

The CSV (Comma Separated Value) format is a simple text file where the values are separated by commas. The utility of this format is that it can be easily read by any analysis package (Excel, Origin, Matlab, ..) and converted to its own format.

The information written in GAMOS is the following:

- *Histograms ID*: The first word is “ID”, then the following info is dumped: his_name,number_of_bins,Xaxis_minimum, Xaxis_maximum, bin_content, bin_error, number_of_entries,mean,mean_error,RMS,RMS_error. The bin_content, bin_error is the list of entries in each bin with its error. It has indeed number_of_binsX+2 pairs value-error, as the first one is the underflow (entries below axis_minimum) and the last one is the overflow (entries above axis maximum).

- *Histograms 2D*: The first word is “2D”, then the following info is dumped: his_name,number_of_binsX, Xaxis_minimum, Xaxis_maximum,number_of_binsY,Yaxis_minimum,Yaxis_maximum, bin_content, bin_error, number_of_entries,mean,RMS. The bin_content, bin_error is the list of entries in each bin with its error. It has indeed (number_of_binsX+2)*(number_of_binsY+2) pairs value-error, as the first row/column is the underflow (entries below axis_minimum) and the last row/column is the overflow (entries above axis maximum).
- *Histograms profile 1D*: The first word is “PROFILE1D”, the rest is the same as the histograms 1D.
- *Histograms profile 2D*: The first word is “PROFILE2D”, the rest is the same as the histograms 2D.

You can see an example of 1D histogram here:

```
"1D", "example", 10, 0, 1000, 0, 3, 3, 1, 3, 5, 6, 12, 15, 16, 2, 66, 0, 460.2, 353.81
```

Changing histogram minimum, maximum and number of bins

You may change minimum, maximum and number of bins of any histogram with the following user commands.

- *1D or 1D profile histograms* /gamos/analysis/histo1NBins HISTO_NAME VALUE /gamos/analysis/histo1Min HISTO_NAME VALUE /gamos/analysis/histo1Max HISTO_NAME VALUE
- *2D or 2D profile histograms* /gamos/analysis/histo2NBinsX HISTO_NAME VALUE /gamos/analysis/histo2MinX HISTO_NAME VALUE /gamos/analysis/histo2MaxX HISTO_NAME VALUE /gamos/analysis/histo2NBinsY HISTO_NAME VALUE /gamos/analysis/histo2MinY HISTO_NAME VALUE /gamos/analysis/histo2MaxY HISTO_NAME VALUE

HISTO_NAME may contain asterisks to include several histograms at the same time. For example:

```
/gamos/analysis/histo1NBins *Energy 300
```

will set to 300 the number of bins of all histograms ending with “Energy”.

The name of each histogram is documented in this guide. If you are not sure about an histogram name, you may run the job with limited statistics and open the file.

Histograms name separator

Histogram names in GAMOS are quite verbose, and often they include a colon as separator, like *Many histograms SD: Only PE: Event Type, neutron - neutron : nCapture : CONCRETE : IE-4 : cos Deviation Angle* or *GmStepDataHistosUA_GmClassifierByParticle:e-:FinalKineticEnergy*. But some ROOT interactive scripts do not allow having a colon in the histogram name. If you use ROOT scripting and you want to change this separator, you may do it with the paramter:

```
/gamos/setParam Histos:Separator SEPARATOR
```

where SEPARATOR is any character or set of characters.

Output files name

The name of the histogram file for each of the GAMOS histogram classes is explained in the corresponding section of this guide. As these classes are user actions, to this file name it is added the name of the filters and classifier, as explained in the section on *Using a common histogram class*. Also there it is explained how to change the name with the parameter

```
/gamos/setParam FILE_NAME:FileName NEW_FILE_NAME
```

If you are running a job and you want to identify all your histogram files with a characteristic prefix or suffix (for example to differentiate them from the files from another job), you may do it by defining the parameters:

```
/gamos/setParam GmAnalysisMgr:FileNamePrefix PREFIX
```

The name *PREFIX* will be added at the beginning of all histogram names.

```
/gamos/setParam GmAnalysisMgr:FileNameSuffix SUFFIX
```

The name *SUFFIX* will be added at the end of all histogram names, before the file type (.root or .csv).

In the case of other output files (or input), like the text or binary files with several kinds of data that are explained in different sections of this guide, you can also add a prefix or suffix with the above commands to all of them, as all the input and output of these classes is controlled in GAMOS through a common base class.

Analysing your histograms with ROOT

You may open a ROOT file by typing in the command line *root* and then open a browser by typing in the ROOT prompt *new TBrowser*. This will open a window where you can see at the left a sub-window with the set of folders. Under the folder titled *PROOF Sessions* you can see a folder with the name of the directory you are in. By clicking on it you may navigate on your folders and open your ROOT file. By double-clicking on your ROOT file when it appears in the right sub-window you will see the list of your histograms. If you do double-click in an histogram, ROOT will visualise it.

Alternatively you may directly open your file when you start a ROOT session by typing *root MYFILE.root*. Then by double-clicking in *ROOT Files* you will see your file and you can open your histograms as below

You may get more instructions on using ROOT in the web page <http://root.cern.ch>

We provide a few utilities to help you in analysing ROOT files. They can be found in the directory *analysis/ROOTUtilities*.

Printing the histograms in graphics files

For a faster visualisation of the histograms in a ROOT file, you may print each of the histogram in a graphics file of type *gif*. Open a ROOT session and then type the command *.x printAll.C++("MYFILE.root")*. There are two optional extra arguments, that can be given after the file name separated by a comma:

- A boolean (0 or 1) to indicate if the Y axis is in logarithmic scale or not
- An string (in between quotes) indicating the type of histogram representation ("lego","cont",...)

For each histogram a graphics file will be written with the name of the histogram preceded by *his* and followed by *.gif*. If you want another format different than *gif*, you may edit the *printAll.C* file and change the histogram name

You may also run the file without opening a ROOT session by typing

```
root -b -p -q .x printAll.C++("\MYFILE.root\")
```

Comparing histograms in two files

This utility serves to compare the histograms of two files that contain the same histograms, for example, to compare the results of two jobs you have run with different options. To do the comparison open a ROOT session and then type the command *.x compareAll.C++("MYFILE1.root","MYFILE2.root")*. There are two optional extra arguments, that can be given after the file names separated by a comma:

- A boolean (0 or 1) to indicate if the histograms are compared normalising one to the other or not

- A boolean (0 or 1) to indicate if the Y axis is in logarithmic scale or not

For each histogram two graphics file will be written: the first one contains the histograms the histogram of the first file on the top left quadrant, the histogram fo the second file on the top right one and in the lower half the histogram of the first file in red and, superimposed, the histogram of the second file in blue; in the second graphics file there is the division of the histogram of the first file and that of the second file. The first file name is built with the name of the histogram preceded by *his* and followed by *.gif*. The second file name is the same as the first plus *_div* before *.gif*. If you want another format different than *gif*, you may edit the *compareAll.C* file and change the histogram name

You may also run the file without opening a ROOT session by typing (beware the inverted slashes)

```
root -b -p -q .x compareAll.C++\(\\"MYFILE.root\")
```

Creating your own histogram

When you write your histogram in C++ you just have to take care of creating and filling it. The class `GmAnalysisMgr` will take care of automatically writing it in the file format you chose.

To use `GmAnalysisMgr` you have to instantiate it in your histogram class, passing to it the name of your file (you may use the same name for several of your histogram classes or different ones):

```
GmAnalysisMgr myAnaMgr = GmAnalysisMgr::GetInstance(“MY_FILE_NAME”)*
```

There are four types of histograms currently supported by GAMOS: 1-dimensional, 2-dimensional, profile 1-dimensional and profile 2-dimensional

A profile histogram sets the value of a bin as the average of all entries in that bin, and the error as the RMS of these entries. To create your histogram and register it to GAMOS you have to create it with a line like:

```
myAnaMgr-> CreateHisto1D(HNAM,NBINS,MAXBIN,MINBIN,HIS_NUMBER);
```

```
myAnaMgr-> CreateHisto2D(HNAM,NBINSX,MAXBINX,MINBINX, NBINSY,MAXBINY,MINBINY,HIS_NUMBER);
```

```
myAnaMgr-> CreateHistoProfile1D(HNAM,NBINS,MAXBIN,MINBIN,HIS_NUMBER);
```

```
myAnaMgr-> CreateHistoProfile2D(HNAM,NBINSX,MAXBINX,MINBINX, NBINSY,MAXBINY,MINBINY,HIS_NUMBER);
```

The last argument is the histogram number, that can be later used to retrieve this histogram from any method in any class. If you don't set it, GAMOS will assign it automatically starting from 1.

Once a histogram is registered you can get a pointer to it by asking `GmAnalysisMgr` for a histogram by its number or its name:

```
myAnaMgr-> GetHisto1(HIS_NUMBER)-> Fill(value)
```

```
myAnaMgr-> GetHisto2(HIS_NUMBER)-> Fill(value)
```

```
myAnaMgr-> GetHistoProfile1(HIS_NUMBER)-> Fill(value)
```

```
myAnaMgr-> GetHistoProfile2(HIS_NUMBER)-> Fill(value)
```

or similarly if you want to retrieve it by the histogram name.

ANALYSIS (EXTRACTING DATA)

Introduction: GAMOS data

There is a big variety of data that can be extracted out of a simulation (position, energy lost, event ID, particle name, angle between primary and secondary, ...). We have transformed each of these data into a plug-in, so that it can be used with a user command to fill an histogram, be dumped into a text or binary file, filter on an interval of its values, make a classification as a function of the data value intervals or use them for importance sampling.

A GAMOS data plug-in may return a different value depending on when it is called. The following options are possible:

- At each track step
- At the beginning of a track
- At the end of a track
- At the beginning of a event
- At the end of a event
- At the beginning of a run
- At the end of a run
- When a secondary track is created (this case is used for relating the variables of an initial track and the secondary tracks it creates)

Each of the GAMOS data does not have to implement the methods to return a value in all cases, as in some cases it has no meaning. Examples can be the position for a run, the final energy for a secondary track (which has not been started to be tracked) or the track ID for an event. If a data is used when the case has no meaning, it will return an exception.

For many types of data one can make a distinction between the *Initial* and the *Final* values. This again has a different meaning depending on when it is called:

- *Track step*: *Initial* refers to the data at PreStepPoint (i.e. at the first point of the step). *Final* refers to the data at PostStepPoint (i.e. at the second point of the step)
- *Track*: *Initial* refers to the data at vertex. *Final* refers to the data at the current point
- *Event*: *Initial* refers to the data of the first particle in the event. *Final* has no meaning
- *Run*: *Initial* has no meaning. *Final* has no meaning
- *Secondary tracks*: *Initial* refers to the data when the particle is created (not being tracked yet). *Final* has no meaning

The data of these type have in their name the word *Initial* or *Final*.

Other data are of type *Change*, because they return the data at the *Final* point minus the data at the *Initial* point. Of course, these data can only have sense for *Step* and *Track*. The data of these type have in their name the word *Change*.

Still other data are of type *Accumulated*, because they return the data accumulated along the steps. The data of these type have in their name the word *Accumulated*. Their behaviour depends on when they are called:

- *Track step*: the value of the step itself.
- *Track*: the value summed up of all the step of the track.
- *Event*: the value summed up of all the tracks of the event.
- *Run*: the value summed up of all the events of the run.
- *Secondary tracks*: it has no meaning.

Be aware of the behaviour of the filters when you use data of type *Accumulated* with a user action. If an step does not pass a filter, the data will not be accumulated, and when the value is printed at the end of the track, event or run it may give a wrong result. You may want to use the option to switch off the filters at stepping with the parameter:

```
/gamos/setParam USER_ACTION_NAME:ApplyFiltersToStepping false
```

See section on *Filters* for more details on this option.

Another source of classification of data is their C++ type: double, int or string. The behaviour of double and integer data is very similar, the main difference being that in a binary file they are written in different format, and occupy a different number of bytes. Each string data has a variable that defines the number of characters that will be used when writing a binary file. The string data cannot be used to fill an histogram.

It is possible to use at the same time several data to fill 2D, 1D profile or 2D profile histograms. To do it the data has to be separated by *.vs.* if it is for a 2D histogram, *.prof.* if it is for a 1D profile histogram and both *.vs.* and *.prof.* if it is for a 2D profile histogram. See section on *Data users* for examples on this.

Data can also be used in mathematical expressions, for example $\log_{10}(\text{InitialKineticEnergy})$, $\sqrt{2 * \text{FinalPosX} * \text{FinalPosY}}$, ...

Each data has as a property the histogram number of bins (that for all is 100), the minimum and maximum. In case it is a composed data, i.e. using mathematical expressions, the minimum and maximum will be calculated with the same formula used for the data. If you want to avoid this calculation the following parameter must be set:

```
/gamos/setParam GmVData:ComposeHistoLimits 0
```

These values can be changed as for any histogram (see section on Histograms).

Data users

The data users are user actions that use the data to fill histograms or to dump them in text or binary files or dump them in the screen. There is one user action for each of the five types of information objects (step, track, event, run and secondary tracks), and one user action for each of the four actions, making in total twenty user actions. There is also a user action to dump data in a ROOT Tree; its behaviour is explained in a dedicated section below. We will describe in this section the available data users and how they behave with the different data types mentioned above.

The data can also be used to filter on an interval of values, in classifiers to give a different classification index in different user-defined intervals and in scorers to score the data values. These three usages are explained in the corresponding sections.

The behaviour of each of the data users depends on one side on the type of information object they manage and on the other side on the type of output provided.

Behaviour as a function of information object

The behaviour of each of the data users for each of the information objects is the following:

- The *Step* data users implement a *UserSteppingAction* method, which extracts the data at each track step, from the *G4Step* object.
- The *Track* data users implement a *PostUserTrackingAction* method, which extracts the data at the end of the current track step, from the *G4Track* object (equal to the current track status). For the data of type *Initial* the data is extracted from the vertex information of the *G4Track* object. For the data of type *Accumulated* these data users implement a *UserSteppingAction* method, which accumulates the data at each track step, and a *PreUserTrackingAction*, which initializes the data to 0; the *PostUserTrackingAction* uses the accumulated data, instead of extracting the data from the *G4Track* object.
- The *SecondaryTrack* data users implement a *UserSteppingAction* method, which extracts the data at each track step, from the *G4Track* object of all secondary tracks created in the step. When setting filters or classifiers to one of these data users you have to remember that they are applied by default to the primary tracks; if you want to applied them to the secondary tracks you have to explicitly do it (see sections on Filters and Classifiers)
- The *Stack* data users implement a *ClassifyNewTrack* method, which extracts the data when the tracks are taken from the stack, from the *G4Track* object.
- The *Event* data users implement a *EndOfEventAction* method, which extracts the data at the end of the event, from the *G4Event* object. For the data of type *Initial* the data is extracted from the information of the first primary particle of the first primary vertex. For the data of type *Accumulated* these data users implement a *UserSteppingAction* method, which accumulates the data at each track step, and a *BeginOfEventAction*, which initializes the data to 0; the *EndOfEventAction* uses the accumulated data, instead of extracting the data from the *G4Event* object.
- The *Run* data users implement a *EndOfRunAction* method, which extracts the data at the end of the run, from the *G4Run* object. For the data of type *Accumulated* these data users implement a *UserSteppingAction* method, which accumulates the data at each track step, and a *BeginOfRunAction*, which initializes the data to 0; the *EndOfRunAction* uses the accumulated data, instead of extracting the data from the *G4Run* object.

Note: the data of type *Initial* is printed (or used to fill an histogram) at the end of track, event or run, with the aim of having together all the information relevant to an object. This behaviour can be changed, so that this data is print at the start with the parameter

```
/gamos/setParam DATA_USER_NAME:UseAtInitial 1
```

Behaviour as a function of output format

The behaviour of each of the data users for each of the output type provided is the following:

-
- *Histograms*: the histogram data users (i.e. *GmStepDataHistosUA*, *GmTrackDataHistosUA*, *GmSecondaryTrackDataHistosUA*, *GmStackDataHistosUA*, *GmEventDataHistosUA*, *GmRunDataHistosUA*,) fill histograms with the data values. By default 1-dimensional histograms are filled but if several data are used at the same type other histogram types are possible: if a 2-dimensional histogram is required the names of the two data have to be separated by *.vs.*; if a 1-dimensional profile histogram is required the names of the two data have to be separated by *.prof.*; if a 2-dimensional profile histogram is required the names of the three data have to be separated by *.vs.* and *.prof.*. For example: *FinalKineticEnergy.vs.InitialKineticEnergy* will fill a 2D histogram. *FinalKineticEnergy.prof.InitialKineticEnergy* will fill a 1D profile histogram. *FinalKineticEnergy.vs.InitialKineticEnergy.prof.InitialPositionZ* will fill a 2D profile histogram. The name of the histogram file is given by the name of the data user plus the filters and classifiers plus the histogram type (i.e. *.root* or *.csv*). It can be changed with the parameter: */gamos/setParam DATA_USER_NAME:FileName NEW_FILE_NAME* For example: */gamos/userAction GmStepDataHistosUA GmGammaFilter* will produce

a file named *GmStepDataHistosUA_GmGammaFilter.root*, whose name can be changed with the command: `/gamos/setParam GmStepDataHistosUA_GmGammaFilter:FileName NEW_FILE_NAME` to produce a file named *NEW_FILE_NAME.root*

-
- *Text files:* These data users (i.e. *GmStepDataTextFileUA*, *GmTrackDataTextFileUA*, *GmSecondaryTrackDataTextFileUA*, *GmStackDataTextFileUA*, *GmEventDataTextFileUA*, *GmRunDataTextFileUA*) dump the data values into text files. A line is written for each call to the data user object (i.e. each *G4Step*, *G4Track*, ...). The text file is indeed a CSV (Comma Separated Value) file; this means that values are separated by commas and string values are surrounded by double quotes. Optionally a header line can be written containing the names and order of the data to be written. With the aim of clearly distinguishing it, the first word of the header is always *HEADER:*. The second word is the number of words, and then the data names. To select this option the following parameter has to be used: `/gamos/setParam FILE_NAME:WriteHeaderData 1` The name of the text file is given by the name of the data user plus the filters and classifiers plus (i.e. *.out*). It can be changed with the parameter: `/gamos/setParam DATA_USER_NAME:FileName NEW_FILE_NAME` For example: `/gamos/userAction GmStepDataTextFileUA GmGammaFilter` will produce a file named *GmStepDataTextFileUA_GmGammaFilter.out*, whose name can be changed with the command: `/gamos/setParam GmStepDataTextFileUA_GmGammaFilter:FileName NEW_FILE_NAME` to produce a file named *NEW_FILE_NAME.out*
-
- *Binary files:* These data users (i.e. *GmStepDataBinFileUA*, *GmTrackDataBinFileUA*, *GmSecondaryTrackDataBinFileUA*, *GmStackDataBinFileUA*, *GmEventDataBinFileUA*, *GmRunDataBinFileUA*,) dump the data values into binary files. Data of integer type are written as *int*, occupying four bytes. Data of double type are written as *float* (for space savings), occupying four bytes (NOTE: in case you want to write all your data as double, you just have to replace *float* by *double* in the four *WriteBin(...)* methods in the file *GamosCore/GamosData/Data/src/GmVData.cc*, and recompile GAMOS). Data of string type are written as a list of *char*; each string data has a default number of characters, that can be overridden with the parameter: `/gamos/setParam DATA_NAME:NumberOfChars NCHARS` To allow checking the format when the file is read back, specially useful if the file is read in a different computer system than the one where it was written, or by another program than GAMOS, a check line is read. This check line consists on first an *L* or *B* character, depending on whether the computer system uses little endian or big endian; second the integer *1234567890* is written, occupying four bytes; third the float *3.40282e+38* is written, occupying four bytes, last the double *1.79769e+308* is written, occupying eight bytes. Optionally four other headers can be written after the first check line. If any header is being written before then four integer numbers are always written, whose values are 0 or 1 depending if the corresponding header is written. The first optional header serves to check that your software system is able to read the file, which you may have written using another system. It writes first a *char* equal to *L* or *B* depending if the system uses little endian or big endian convention. Then it writes as an *int* the number *1234567890*, as a *float* the number *3.40282e+38* and as a *double* the number *1.79769e+308*. You should check that you read back those numbers when reading your file. To write this header the following parameter has to be used: `/gamos/setParam FILE_NAME:WriteHeaderCheck 1` The second optional header contains the names and order of the data to be written; the first word is an integer with the number of data; then for each data it is written an integer with the number of characters of the data name, the data name itself with this number of characters, the number of characters of the data C++ type, and finally the C++ type itself with this number of characters (which can be *int*, *float* or *char*). To write this header the following parameter has to be used: `/gamos/setParam FILE_NAME:WriteHeaderData 1` The third optional header is a float containing the number of events used in the job (see for example the section on *Radiotherapy phase space reading* or *using files as generator* for the exact meaning of the number of events in these cases). To write this header the following parameter has to be used: `/gamos/setParam FILE_NAME:WriteHeaderNEvents 1` The fourth optional header is an integer containing the number of calls to the data user object. To write this header the following parameter has to be used: `/gamos/setParam FILE_NAME:WriteHeaderNCalls 1` The name of the binary file is given by the name of the data user plus the filters and classifiers plus (i.e. *.out*). It can be changed with the parameter: `/gamos/setParam DATA_USER_NAME:FileName NEW_FILE_NAME` For example: `/gamos/userAction GmStepDataBinFileUA GmGammaFilter` will produce

a file named *GmStepDataBinFileUA_GmGammaFilter.out*, whose name can be changed with the command: */gamos/setParam GmStepDataBinFileUA_GmGammaFilter:FileName NEW_FILE_NAME* to produce a file named *NEW_FILE_NAME.out*

-
- *Cout*: These data users (i.e. *GmStepDataCoutUA*, *GmTrackDataCoutUA*, *GmSecondaryTrackDataCoutUA*, *GmStackDataCoutUA*, *GmEventDataCoutUA*, *GmRunDataCoutUA*) dump the data values into the standard output (they may be helpful to debug an application, together with the command */tracking/verbose 1*. Similarly to the text file data users, a line is written for each call to the data user object (i.e. each *G4Step*, *G4Track*, ...). The text file is indeed a CSV (Comma Separated Value) file; this means that values are separated by commas and string values are surrounded by double quotes.

Selection of data list for a data user

Each data user has a default list of data, that we will enumerate below. This list can be changed with the parameter:

```
/gamos/setParam DATA_USER_NAME:DataList DATA_1 DATA_2 ...
```

For example, if we use:

```
/gamos/userAction GmStepDataBinFileUA GmGammaFilter
```

the data list may be changed with the parameter:

```
/gamos/setParam          GmStepDataBinFileUA_GmGammaFilter:DataList          TrackID          FinalPosX
sqrt(2*FinalPosX*FinalPosY) InitialKineticEnergy
```

Default data list for each data user

The default data list of each of the fifteen data users are:

- *GmStepDataHistosUA*:
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *AccumulatedEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmStepDataTextFileUA*:
- *EventID*
- *TrackID*
- *Particle*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*

- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *AccumulateEnergyLost*
- *AccumulateEnergyDeposited*
- *GmStepDataBinFileUA:*
- *EventID*
- *TrackID*
- *Particle*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *AccumulateEnergyLost*
- *AccumulateEnergyDeposited*
- *GmTrackDataHistosUA:*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *AccumulatedEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmTrackDataTextFileUA:*
- *EventID*
- *TrackID*
- *Particle*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*

- *AccumulateEnergyLost*
- *AccumulateEnergyDeposited*
- *GmTrackDataBinFileUA:*
- *EventID*
- *TrackID*
- *Particle*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *AccumulateEnergyLost*
- *AccumulateEnergyDeposited*
- *GmEventDataHistosUA:*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmEventDataTextFileUA:*
- *EventID*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmEventDataBinFileUA:*
- *EventID*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmRunDataHistosUA:*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmRunDataTextFileUA:*
- *RunID*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *GmRunDataBinFileUA:*
- *RunID*
- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*

- *GmSecondaryTrackDataHistosUA:*
- *TrackID*
- *InitialPrimMinusSecoKineticEnergy*
- *FinalPrimMinusSecoKineticEnergy*
- *SecoDividedInitialPrimKineticEnergy*
- *SecoKineticEnergy*
- *PrimSecoAngleChange*
- *InitialPrimKineticEnergy*
- *FinalPrimKineticEnergy*
- *GmSecondaryTrackDataTextFileUA:*
- *EventID*
- *TrackID*
- *InitialPrimMinusSecoKineticEnergy*
- *FinalPrimMinusSecoKineticEnergy*
- *SecoDividedInitialPrimKineticEnergy*
- *SecoKineticEnergy*
- *PrimSecoAngleChange*
- *InitialPrimKineticEnergy*
- *FinalPrimKineticEnergy*
- *GmSecondaryTrackDataBinFileUA:*
- *TrackID*
- *InitialPrimMinusSecoKineticEnergy*
- *FinalPrimMinusSecoKineticEnergy*
- *SecoDividedInitialPrimKineticEnergy*
- *SecoKineticEnergy*
- *PrimSecoAngleChange*
- *InitialPrimKineticEnergy*
- *FinalPrimKineticEnergy*

Saving GAMOS data in a ROOT TTree

The *GmDataTTreeUA* data user sorts GAMOS data in a ROOT TTree structure and save the new TTree in a ROOT TFile. The *GmDataTTreeUA* is invoked with the usual command:

```
/gamos/userAction GmDataTTreeUA
```

By default the TTree is created with the *GmDataTTree* name and saved in the *GmDataTTree.root* file. The following command can be used to modify both both TFile and TTree names:

```
/gamos/setParam DATA_USER_NAME:TreeFileName NEW_NAME
```


Four categories of GAMOS data can be saved in the TTree: Event data, Track data, Secondary Track data, and Step data. GAMOS users can select one list of data for each one of the four categories. If data with no meaning for a given category are included in the correspondent list (e.g. adding trackID data to the Event data list), an exception will be returned. By default, the four data lists are empty and nothing is saved in the TTree unless otherwise specified. The four data lists are selected as follows:

```
/gamos/setParam DATA_USER_NAME:EventDataList EVENT_DATA_1 EVENT_DATA_2 ...
```

```
/gamos/setParam DATA_USER_NAME:TrackDataList TRACK_DATA_1 TRACK_DATA_2 ...
```

```
/gamos/setParam DATA_USER_NAME:SecondaryTrackDataList SECO_DATA_1 SECO_DATA_2 ...
```

```
/gamos/setParam DATA_USER_NAME:StepDataList STEP_DATA_1 STEP_DATA_2 ...
```

DATA_USER_NAME is the name of the user action, i.e. the word *GmDataTreeUA* plus the name of filters and classifier

The TTree is filled at the end of each GAMOS event and contains a number of entries equal to the number of simulated events. Each of the selected GAMOS data is stored in an independent TBranch inside the TTree.

Data are saved as integers, doubles, or strings. For each TTree entry, each of the selected GAMOS data is stored in a `std::vector`, apart from Event data that correspond to single values. The size of a `std::vector` varies in accordance with the corresponding category: track data vectors contain a number of elements equal to the number of tracks for a given event; step and secondary track data vectors equal to the number of steps for the same event.

Filter from data

It is possible to use any GAMOS data to filter on its value (see chapter on Filters). To use it you have to use the command to define a filter, using the filter named *GmNumericDataFilter* if the data is of numeric type (int or float) and *GmStringDataFilter* if the data is of string type and passing the following arguments:

GmNumericDataFilter: In this case after the filter name you have to give the the minimum and maximum value of the data. For example:

```
/gamos/filter dataF GmNumericDataFilter InitialKineticEnergy 0. 0.3
```

GmStringDataFilter: In this case after the filter name, you have to give the list of accepted values. For example:

```
/gamos/filter dataF GmStringDataFilter InitialLogicalVolume detector world
```

Classifier by data

It is possible to use any GAMOS data to classify on its value (see chapter on Classifiers). To use it you have to use the command to define a classifier, using the classifier named *GmClassifierByNumericData* if the data is of numeric type (int or float) and *GmClassifierByStringData* if the data is of string type and passing the following arguments:

- *Data is integer or float*: In this case after the classifier name you have to give the number of intervals, the minimum data limit and the maximum data limit. For example to define 20 energy intervals between 0 and 10 : `/gamos/classifier dataC GmClassifierByData InitialKineticEnergy 20 0. 10.` GAMOS will check if the data value is inside the interval defined by the minimum and maximum. If it is smaller, it will assign an index 0, if it is bigger it will assign an index INT_MAX (2^{32}). In both cases a warning will be printed. If you do not want to allow this behaviour, but you want that the job is stopped if the data is out of limits, you have to set the parameter: `/gamos/setParam CLASSIFIER_NAME:AllowOutOfLimits 0`
- *Data is string*: In this case no argument is needed after the classifier name. For example: `/gamos/classifier dataC GmClassifierByData InitialLogicalVolume` If you want to assign different indices than 0 to N, you may use the command : `/gamos/classifier/setIndices VALUE_1 INDEX_1 VALUE_2 INDEX_2` See section on *Setting indices to classifiers* for further details on this feature.

Primitive scorer from data

It is possible to use any GAMOS data, if its type is interger or float, to build an scorer on its value (see chapter on Scorers). To use it you have to use the command to define a scorer, using the scorer named *GmG4PSData* and passing to it the data name and the minimum and maximum value of the data. For example:

```
/gamos/scoring/createMFDdetector contDet container
```

```
/gamos/scoring/addScorer2MFD dataScorer GmG4PSData contDet InitialKineticEnergy
```

List of available data

There are over one hundred types of data currently available in GAMOS. We list here them all, mentioning for each one if it is of type *double* (default, not needed to be indicated), *int* or *string*, if it is of type *Accumulated* (*Initial*, *Final* or *Change* is in the data name), the minimum and maximum values if an histogram is filled with it, and if it is not available for *Step*, *Track*, *Event*, *Run*, or *Secondary track*.

For each data we explain how its value is obtained for each of these five cases, if the value is not evident (i.e. the behaviour of *FinalKineticEnergy* should be evident for what we have explained above if called for a *Step* or a *Track*).

We may also get a list of available data for a GAMOS release by typing in your terminal window (after GAMOS has been configured)

```
SealPluginDump | grep GmData
```

For a better description of the available data we have classified them in several groups

Position

The position is a 3-dimensional vector, so the following data can be obtained from it:

- *X*: projection on X axis
- *Y*: projection on Y axis
- *Z*: projection on Z axis
- *Mag*: magnitude
- *Perp*: perpendicular projection (i.e. $\sqrt{X*X+Y*Y}$)
- *Phi*: phi angle (in degrees)
- *Theta*: theta angle (in degrees)

All these data can be of type *Initial*, *Final* or *Change*. The *Initial* data are available for all information objects except *Run*, while the *Final* and *Change* data are available only for *Step* and *Track*. There are other data that calculate the local position, i.e. the global position transformed into the local frame of the volume the particle is in. The *InitialLocalPos* data for a *Step* object calculates the transformation in the volume the track step has just traversed, while for a *Track* object calculates the transformation in the volume where the track was initiated. The *FinalLocalPos* data for a *Step* object calculate the transformation in the volume the track step is going to enter, the same as for a *Track* object. You may be interested in getting the transformation in the volume the track step has just traversed, but using the position and the end of the step. In this case you can use another set of data, called *LocalInPre*. For a *Track* object these data use the current position of the track, but the volume where the track was initiated. If you want to get the vertex position for a step (for a track is obtained as *InitialPos*) you may use the data *Vertex*; this data are useful for *StackData* data users, as the initial or final positions are not calculated at stacking action. Another set of data, *RandomPos* select randomly a point in the line joining the *PreStepPoint* and *PostStepPoint*.

The histogram limits for X , Y and Z are -1000 (mm) to 1000 (mm), for Mag and $Perp$ are 0 to 1000 (mm), for Phi are 0 to 360 (deg) and for $Theta$ are 0 to 180 (deg). The position data are:

- *InitialPosX*
- *InitialPosY*
- *InitialPosZ*
- *InitialPosMag*
- *InitialPosPerp*
- *InitialPosPhi*
- *InitialPosTheta*
- *FinalPosX*
- *FinalPosY*
- *FinalPosZ*
- *FinalPosMag*
- *FinalPosPerp*
- *FinalPosPhi*
- *FinalPosTheta*
- *InitialLocalPosX*
- *InitialLocalPosY*
- *InitialLocalPosZ*
- *InitialLocalPosMag*
- *InitialLocalPosPerp*
- *InitialLocalPosPhi*
- *InitialLocalPosTheta*
- *FinalLocalPosX*
- *FinalLocalPosY*
- *FinalLocalPosZ*
- *FinalLocalPosMag*
- *FinalLocalPosPerp*
- *FinalLocalPosPhi*
- *FinalLocalPosTheta*
- *FinalLocalInPrePosX*
- *FinalLocalInPrePosY*
- *FinalLocalInPrePosZ*
- *FinalLocalInPrePosMag*
- *FinalLocalInPrePosPerp*
- *FinalLocalInPrePosPhi*

- *FinalLocalInPrePosTheta*
- *RandomPosX*
- *RandomPosY*
- *RandomPosZ*
- *RandomPosMag*
- *RandomPosPerp*
- *RandomPosPhi*
- *RandomPosTheta*
- *VertexPosX*
- *VertexPosY*
- *VertexPosZ*
- *VertexPosMag*
- *VertexPosPerp*
- *VertexPosPhi*
- *VertexPosTheta*
- *PosChangeX*
- *PosChangeY*
- *PosChangeZ*
- *PosChangeMag*
- *PosChangePerp*
- *PosChangePhi*
- *PosChangeTheta*

The *Pos* positions are calculated in the global reference frame, the *LocalPos* are calculated in the reference frame of the volume the step or track is in.

Direction

The direction data are very similar to the position data, so that all what was said for position data is also valid for them. The main different is the histogram limits, which are for *X*, *Y* and *Z* are -1 (*MeV*) to 1 (*MeV*), for *Mag* and *Perp* are 0 (*MeV*) to 1 (*MeV*), for *Phi* are 0 (*deg*) to 360 (*deg*) and for *Theta* are 0 (*MeV*) to 180 (*deg*). Another difference is that the *InitialDirMag* and *FinalDirMag* do not exists, because their value would always be one. The direction data are:

- *InitialDirX*
- *InitialDirY*
- *InitialDirZ*
- *InitialDirPerp*
- *InitialDirPhi*
- *InitialDirTheta*
- *FinalDirX*

- *FinalDirY*
- *FinalDirZ*
- *FinalDirPerp*
- *FinalDirPhi*
- *FinalDirTheta*
- *InitialLocalDirX*
- *InitialLocalDirY*
- *InitialLocalDirZ*
- *InitialLocalDirPerp*
- *InitialLocalDirPhi*
- *InitialLocalDirTheta*
- *FinalLocalDirX*
- *FinalLocalDirY*
- *FinalLocalDirZ*
- *FinalLocalDirPerp*
- *FinalLocalDirPhi*
- *FinalLocalDirTheta*
- *FinalLocalInPreDirX*
- *FinalLocalInPreDirY*
- *FinalLocalInPreDirZ*
- *FinalLocalInPreDirPerp*
- *FinalLocalInPreDirPhi*
- *FinalLocalInPreDirTheta*
- *VertexDirX*
- *VertexDirY*
- *VertexDirZ*
- *VertexDirMag*
- *VertexDirPerp*
- *VertexDirPhi*
- *VertexDirTheta*
- *DirChangeX*
- *DirChangeY*
- *DirChangeZ*
- *DirChangeMag*
- *DirChangePerp*
- *DirChangePhi*

- *DirChangeTheta*
- *AngleChange* This data represents the angle between the initial and the final directions (i.e. $\text{acos}(\text{InitialDir} \cdot \text{FinalDir})$). In contrast, the *DirChangeXXX* are calculated subtraction the final minus the initial direction, what gives a 3-dimensional vector, from which the X, Y, Z, Mag, ... magnitudes are obtained

Momentum

The momentum data are very similar to the position data, so that all what was said for position data is also valid for them. The main different is the histogram limits, which are for X, Y and Z are 0 to 1 (MeV), for Mag and Perp are 0 to 1 (MeV), for Phi are 0 to 360 (deg) and for Theta are 0 to 180 (deg). The momentum data are:

- *InitialMomX*
- *InitialMomY*
- *InitialMomZ*
- *InitialMomMag*
- *InitialMomPerp*
- *InitialMomPhi*
- *InitialMomTheta*
- *FinalMomX*
- *FinalMomY*
- *FinalMomZ*
- *FinalMomMag*
- *FinalMomPerp*
- *FinalMomPhi*
- *FinalMomTheta*
- *VertexMomX*
- *VertexMomY*
- *VertexMomZ*
- *VertexMomMag*
- *VertexMomPerp*
- *VertexMomPhi*
- *VertexMomTheta*
- *MomChangeX*
- *MomChangeY*
- *MomChangeZ*
- *MomChangeMag*
- *MomChangePerp*
- *MomChangePhi*
- *MomChangeTheta*

Energy

The energy data are a miscellaneous set of data that extract some information about energy. They have in common that the histogram limits are 0 (MeV) to 1 (MeV). They are the following:

- *InitialKineticEnergy*
- *FinalKineticEnergy*
- *KineticEnergyChange*
- *InitialTotalEnergy*
- *FinalTotalEnergy*
- *VertexKineticEnergy*
- *InitialDEDX* This is not exactly the energy lost divided by the path length, as one may think from its name, but the dE/dX, i.e. the fraction of energy lost per distance travelled as calculated by the GEANT4 utilities with the final kinetic energy in the volume material the PreStepPoint (or track vertex) is in .
- *FinalDEDX* Similar as above.

Plus some data of type *Accumulated*:

- *AccumulateEnergyLost*
- *AccumulatedEnergyDeposited*
- *AccumulatedDose*: This is the energy deposited divided by the volume mass
- *AccumulatedKerma*: This is the sum of energies of the charged secondary particles produced by non-charged particles, divided by the volume mass
- *InitialGammaEquivalentDose* Calculates the equivalent dose multiplying the volume flux (step length divided by cubic volume) by flux to equivalent dose conversion factor read from an input file. These factors are by default the ICRU report 57, and they are read from the file *MY_GAMOS_DIR/data/Flux2Dose.gamma.ICRU57.lis*. You may change the input file with the parameter: */gamos/setParam GmDataInitialGammaEquivalentDose:Flux2DoseFileName NEW_FILE_NAME* You may select to calculate the personale equivalent dose, *AccumulatedKerma*: You may select the magnitude to use, *H*(10)* (default), *Hp(10,0)*, *Hp(10,15)*, *Hp(10,30)*, *Hp(10,45)*, *Hp(10,60)*, *Hp(10,75)* with the parameter: */gamos/setParam GmDataInitialGammaEquivalentDose:EquivDoseType TYPE* where *TYPE* may be *Hstar*, *Hp0*, *Hp15*, *Hp30*, *Hp45*, *Hp60*, *H75*
- *FinalGammaEquivalentDose* Similar as above
- *InitialNeutronEquivalentDose* Calculates the equivalent dose multiplying the volume flux (step length divided by cubic volume) by flux to equivalent dose conversion factor read from an input file. These factors are by default the ICRU report 57, and they are read from the file *MY_GAMOS_DIR/data/Flux2Dose.neutron.ICRU57.lis*. You may select the magnitude to use, *H*(10)* (default), *Hp(10,0)*, *Hp(10,15)*, *Hp(10,30)*, *Hp(10,45)*, *Hp(10,60)*, *Hp(10,75)* with the parameter: */gamos/setParam GmDataInitialNeutronEquivalentDose:EquivDoseType TYPE* where *TYPE* may be *Hstar*, *Hp0*, *Hp15*, *Hp30*, *Hp45*, *Hp60*, *H75*
- *FinalNeutronEquivalentDose* Similar as above

Geometrical objects

These are string data that extract the name of different geometrical objects, and also the type of solids (Box, Tubs, Polycone, ...). All of them have 25 characters when written in a binary file. There is also an integer data for the copy number of physical volumes. They are the following:

- *InitialSolid*

- *FinalSolid*
- *InitialLogicalVolume*
- *FinalLogicalVolume*
- *InitialPhysicalVolume*
- *FinalPhysicalVolume*
- *InitialTouchable*
- *FinalTouchable*
- *InitialRegion*
- *FinalRegion*
- *InitialMaterial*
- *FinalMaterial*
- *InitialSolidType*
- *FinalSolidType*
- *InitialPVCopyNumber*
- *FinalPVCopyNumber*

Material variables

These are numeric data that are related to the material. They are the following:

- *InitialDensity*
- *FinalDensity*
- *InitialPressure*
- *FinalPressure*
- *InitialTemperature*
- *FinalTemperature*
- *InitialRadLength*
- *FinalRadLength*
- *InitialNuclearInterLength*
- *FinalNuclearInterLength*
- *InitialElectronDensity*
- *FinalElectronDensity*

Particle and process

These are string data that extract the name of the particle and the processes, and also data about the particle properties. They are only available for *Step*, *Track* and *Secondary Track*. All of them have 25 characters when written in a binary file. They are the following:

- *Particle*

- *InitialProcess*. Name of process that defined the point PreStepPoint. Only available for *Step*
- *FinalProcess*. Name of process that defined the point PostStepPoint. Only available for *Step*
- *CreatorProcess*. Name of process that created the particle track. Not available for *Event* nor *Run*
- *ParticlePDGEncoding*. Particle code following the Particle Data Group numbering.
- *ParticleCharge*.
- *ParticleMass*.
- *ParticleLifeTime*.
- *ParticleWidth*.
- *ParticleStable*. 1 if stable, 0 if unstable.
- *ParticleType*.
- *ParticleSubType*. The particles types and subtypes are listed below:

GEANT4 particle types and subtypes

G4Particle Type Subtype
 gamma gamma photon e+ lepton e- lepton e proton baryon nucleon neutron baryon nucleon
 deuteron nucleus static triton nucleus static He3 nucleus static alpha nucleus static GenericIon nucleus generic mu+
 lepton mu mu- lepton mu tau+ lepton tau tau- lepton tau nu_e lepton e nu_mu lepton mu nu_tau lepton tau anti_nu_e
 lepton e anti_nu_mu lepton mu anti_nu_tau lepton tau anti_neutron baryon nucleon anti_proton baryon nucleon pi+
 meson pi pi- meson pi pi0 meson pi B+ meson B B- meson B B0 meson B Bs0 meson Bs D+ meson D D- meson
 D D0 meson D Ds+ meson Ds Ds- meson Ds J/psi meson J/psi anti_B0 meson B anti_Bs0 meson Bs anti_D0 meson
 D anti_kaon0 meson kaon eta meson eta eta_prime meson eta_prime kaon+ meson kaon kaon- meson kaon kaon0
 meson kaon kaon0L meson kaon kaon0S meson kaon B+ meson B B- meson B B0 meson B Bs0 meson Bs D+ meson
 D D- meson D D0 meson D Ds+ meson Ds Ds- meson Ds J/psi meson J/psi anti_B0 meson B anti_Bs0 meson Bs
 anti_D0 meson D anti_kaon0 meson kaon eta meson eta eta_prime meson eta_prime kaon+ meson kaon kaon- meson
 kaon kaon0 meson kaon kaon0L meson kaon kaon0S meson kaon anti_lambda baryon lambda anti_lambda_c+ baryon
 lambda_c anti_omega- baryon omega anti_omega_c0 baryon omega_c anti_sigma+ baryon sigma anti_sigma- baryon
 sigma anti_sigma0 baryon sigma anti_sigma_c+ baryon sigma_c anti_sigma_c++ baryon sigma_c anti_sigma_c0
 baryon sigma_c anti_xi- baryon xi anti_xi0 baryon xi anti_xi_c+ baryon xi_c anti_xi_c0 baryon xi_c lambda baryon
 lambda lambda_c+ baryon lambda_c omega- baryon omega omega_c0 baryon omega_c opticalphoton opticalphoton
 photon sigma+ baryon sigma sigma- baryon sigma sigma0 baryon sigma sigma_c+ baryon sigma_c sigma_c++ baryon
 sigma_c sigma_c0 baryon sigma_c xi- baryon xi xi0 baryon xi xi_c+ baryon xi_c xi_c0 baryon xi_c geantino geantino
 geantino chargedgeantino geantino geantino

Secondary tracks

These are data that give information about the secondary particles or the primary particle when secondary particles are created. They are only available for *Secondary Track*. They are the following:

- *PrimParticle*. Primary particle name
- *FinalPrimMinusSecoKineticEnergy*. Primary particle PostStepPoint minus secondary kinetic energies
- *SecoDividedInitialPrimKineticEnergy*. Fraction of kinetic energy of primary at PreStepPoint taken by a secondary particle
- *SecoKineticEnergy*. Secondary particle kinetic energy
- *SecoDirTheta*. Secondary direction angle theta
- *FinalPrimSecoAngle*. Angle between secondary particle and Primary particle at PostStepPoint

- *InitialPrimKineticEnergy*. Primary particle PreStepPoint kinetic energy when a secondary particle is emitted
- *FinalPrimKineticEnergy*. Primary particle PostStepPoint kinetic energy when a secondary particle is emitted
- *VertexPrimKineticEnergy*. Primary particle vertex kinetic energy when a secondary particle is emitted

Ancestor data

It is also possible to get data from an ancestor step (a previous step of the current track or of any of the tracks that is an ancestor of the current track. It is just enough to add the name *AncestorStep* in the name of the data.

Others

These are miscellaneous data:

- *TrackID*: ID of track. It is of integer type. Only available for *Step*, *Track* and *Secondary Track*
- *ParentTrackID*: ID of parent track. It is of integer type. Only available for *Step*, *Track* and *Secondary Track*
- *EventID*: ID of event. It is of integer type. Only available for *Step*, *Track* and *Event*.
- *RunID*: ID of run. It is of integer type. Not available for *Secondary Track*.
- *StepNumber*: number of step. It is of integer type. Only available for *Step* and *Track*
- *InitialWeight*: initial track weight. Not available for *Event* nor *Run*
- *FinalWeight*: final track weight. Not available for *Secondary Track* nor *Event* nor *Run*
- *AccumulatedLength*: Accumulated step length. Not available for *Secondary Track*
- *TrackLength*: step length for *Step* and track length for *Track*. Gives same result than *AccumulatedLength* but is faster
- *NofSecondaries*: number of secondary tracks produced. Only available for *Step* and *Track*
- *SumSecoKineticEnergy*: sum of the kinetic energies of all secondary particles produced. Only available for *Step* and *Track*
- *InitialTime*: initial global time
- *FinalTime*: final global time
- *InitialLocalTime*: initial local time
- *FinalLocalTime*: initial local time
- *InitialProperTime*: initial proper time
- *FinalProperTime*: final proper time
- *TimeChange*: Global time change. Only available for *Step*
- *NofHits*: number of hits. Only available for tracks and events

Output file names

Each user action has a default name for the output file it produces (histogram, text or binary file), as it is explained in the corresponding section of this guide. As these classes are user actions, they can be used with filters and classifiers and in this cases, to the default file name it is added the name of the filters and classifier. This default file name can be changed with the parameter

```
/gamos/setParam FILE_NAME:FileName NEW_FILE_NAME
```

If you are running a job and you want to identify all your output files with a characteristic prefix or suffix (for example to differentiate them from the files from another job), you may do it by defining the parameters:

```
/gamos/setParam GmAnalysisMgr:FileNamePrefix PREFIX
```

the name *PREFIX* will be added at the beginning of all output file names.

```
/gamos/setParam GmAnalysisMgr:FileNameSuffix SUFFIX
```

the name *SUFFIX* will be added at the end of all output file names, before the file type (.root or .csv).

In the case of other output files (or input), like the text or binary files with several kinds of data that are explained in different sections of this guide, you can also add a prefix or suffix with the above commands to all of them, as all the input and output of these classes is controlled in GAMOS through a common base class.

Merging results from different jobs

When running an application you may get a table of the different types of results. It is usual that you run different jobs with the same setup to accumulate statistics or with different setups to study the dependency of your results with some parameters. In the analysis directory of your GAMOS distribution there are specific utilities for specific outputs, for example, *RTPhaseSpace/sumPS*, *RTDose/sumSqdose*, *RTDose/sum3ddose*, *Shielding/sumScores*, *Shielding/sumActivation*, *NuclMed/PET/sumProjdata*; they are described in the corresponding section of this manual. In this section we will describe an utility to analyse the output tables produced from various runs (extract, sum or compare results) and also how to merge and analyse the output files of good events

If you have run one or several jobs and produced a table with the statistics of good events, we describe here an utility that allows to process these results and present them in a more convenient way. This utility analyses the files containing the output tables and allows you to extract the desired numbers out of them and present them together in a single table, with the possibility to add the numbers from different tables. We will describe the steps you should follow to do this and illustrate them with an example.

The first thing you should do is to write a file that describe what you want to do with the output files. In the first section of this file you should list the names of the files to be used. For each file there should be a line containing two words: the first must be *:FILE_TXT* and the second the file name. For example if you have run three jobs with three different energy resolution values, you may put in your file

```
:FILE_TXT out.pet.5
:FILE_TXT out.pet.10
:FILE_TXT out.pet.15
```

After this you should describe the actions to be taken. Several actions can be done, for each one you have to write a line in which you describe which lines are going to be processed, which words in these lines and which action is done (print value, sum values from different files, ..)

In this action line you should first describe the criteria to identify which lines in each file are to be processed. This can be done by setting which words in the line are going to be looked for. Instead of setting the full word one can set the starting characters (prefix) or the ending characters (suffix). To set the list of prefixes you add the words:

:PRE or *:PREFIX* followed by a list of pairs: word position numbers and word characters. For example

```
:PRE 1 Events 2 PET
```

will look for the lines in which the first word starts by *Events* and the second word starts by *PET*.

In a similar way it can be done with the suffixes:

:SUF or *:SUFFIX* followed by a list of pairs: word position numbers and word characters.

If there are several lines in a file satisfying the prefix and suffix conditions, you may set that only one line or a few of them are processed, by using the word `:L` or `:LINE` followed by the list of numbers corresponding to the time order the line appears. For example:

```
:PRE 1 Events 2 PET :L 1
```

will only process the first line found.

Then you have to define which are the words in the line that are going to be processed by using the word `:W` or `:WORD` followed by the list of numbers of the words in the lines. For example:

```
:PRE 1 Events 2 PET :L 1 :W 3 4
```

will process the third and fourth words of the lines selected.

After you have to define the kind of treatment to be done to the words, by using the word `:T` or `:TREATMENT` followed by the treatment type. For example:

```
:PRE 1 Events 2 PET :L 1 :W 3 4 :T PT
```

The treatment type can be one of the following ones:

- *P* or *PRINT*: print the words found at each line of each file, each one in a line
- *PT* or *PRINT_TOGETHER*: print the words found at each line of each file, putting in one line the words extracted from each file (i.e. all the words found corresponding to the same instruction line together)
- *S* or *SUM*: print the sum of the values of the words found in any line of any file
- *M* or *MEAN*: print the mean of the values of the words found in any line of any file

Finally you may write a message that will be printed together with the results, that may serve you to identify the output. For example:

```
:PRE 1 Events 2 PET :L 1 :W 3 4 :T PT :M PET_EVENTS
```

Example of Analysing text output files

As an example of this utility, we will analyse three PET output files and extract some information from the table of PET events classification. Let's take an output file with a table similar to this one:

```
-----
Events PET           : 862    8.62 %
----- Good PET -----
ALL                  : 93    0.93 %
(  1) PET 2 recHits close to vtx : 0    0 %
(  2) PET 2 recHits far from vtx  : 93   0.93 %
( 11) PET 3->2 recHit close to vtx : 0    0 %
( 12) PET 3->2 recHit far from vtx : 0    0 %
----- Random Coincidences -----
ALL                  : 769   7.69 %
( 101) PET 2 recHits close to vtx  : 0    0 %
( 102) PET 2 recHits far from vtx  : 769  7.69 %
( 111) PET 3->2 recHit close to vtx : 0    0 %
( 112) PET 3->2 recHit far from vtx : 0    0 %
...

```

We have three files that we have run with different random seeds and we want to extract the following information:

- Sum of total number of PET events in the three files
- Mean number of good PET events in each of the three files

- List of numbers of random coincidences PET events in each of the three files

The file to obtain these results, that we name *anaout.is*: may look like this:

```
:FILE_TXT out.exercise3c.1
:FILE_TXT out.exercise3c.2
:FILE_TXT out.exercise3c.3
:PRE 1 Events 2 PET :W 4 :T S :M "Events"
:PRE 1 ALL :L 1 :W 3 :T M :M "Good PET Events"
:PRE 1 ALL :L 2 :W 3 :T PT :M "Mean Random coincidences PET Events"
```

And if you run the command:

analysOutput anaout.lis

Then you may get a result like this:

```
Events 2596
Good PET Events 92.6667
Mean Random coincidences PET Events      769      765      763
```


Introduction

A filter is a class that receives a *G4Step* or a *G4Track* and accepts it or not depending on some given criteria. A GAMOS filter has therefore two main methods

- *AcceptStep(const G4Step)**: receives a *G4Step* pointer and decides to return true or false
- *AcceptTrack(const G4Track)**: receives a *G4Track* pointer and decides to return true or false

A filter may implement the two methods or only one of them. If the *AcceptStep* method is not implemented by a filter, the *AcceptStep* method from the base class is invoked and it calls the *AcceptTrack* method passing to it the *G4Track* corresponding to the *G4Step*. If the *AcceptTrack* method is not implemented by a filter, the method from the base class returns true.

Filters can act on user actions or scorers. If one or several filters are set to act on a user action, the *PreUserTrackingAction*, *PostUserTrackingAction* and *ClassifyNewTrack* methods will only be invoked if the *AcceptTrack* method of every filter returns true, and *UserSteppingAction* method will only be invoked if the *AcceptStep* method of every filter returns true. In the case of filters applied to scorers, they only act if the scorer is called, i.e. if the step happens in the selected volume. For details on filters acting on scorers see the section on *Scorers*.

The use of user actions and scorers together with filters is a powerful means to obtain very detailed information on the simulation through simple user commands. See the tutorial on *Histograms and scorers* for examples on this.

Several filters need some extra parameters (see list of filters below) that control their behaviour. To use these filters they have to be declared first with the following command

```
/gamos/filter FILTER_NAME FILTER_CLASS PARAMETER_1 PARAMETER_2 ...
```

where *FILTER_NAME* is the new name you want to give to a filter to attach it to a user action of a scorer, *FILTER_CLASS* is the name of the filter class, and *PARAMETER_1 PARAMETER_2 ...* are the values of the parameters that the filter needs. Those filters that do not need any parameter, can be assigned directly to a user action or a scorer without giving them a new name: the name will be the one of the filter class.

To attach one or more filters to a user action you put them after the user action name in the command line that selects it

```
/gamos/userAction USER_ACTION FILTER_NAME
```

or you can use filters to act on a scorer with the command

```
/gamos/scoring/addFilter2Scorer FILTER_NAME SCORER_NAME
```

Filter are plug-in's, so that a user can create her/his own filter and select it with a user command. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmFilterFactory*, or the *Histograms and scorers* tutorial.

Simple filters

The list of available filters can be obtained by typing *SealPluginDump* on your terminal window and look for the list of classes after *Category GmFilterFactory*: (they all contain the word *Filter*). The list of simple filters in the current GAMOS version is the following:

- *GmGammaFilter*: accepts a track if the particle is a gamma
- *GmElectronFilter*: accepts a track if the particle is an electron
- *GmPositronFilter*: accepts a track if the particle is a positron
- *GmElectronOrPositronFilter*: accepts a track if the particle is an electron or positron
- *GmEMParticleFilter*: accepts a track if the particle is of electromagnetic type (gamma, electron or positron)
- *GmProtonFilter*: accepts a track if the particle is a proton
- *GmNeutronFilter*: accepts a track if the particle is a neutron
- *GmParticleFilter*: accepts a track if the particle is in the list of particles given as extra parameters (see the list of particle names in the section *Using particle names*).
- *GmChargedFilter*: accepts a track if the particle is charged.
- *GmNeutralFilter*: accepts a track if the particle is neutral
- *GmPrimaryFilter*: accepts a track if it is a primary (it does not come from another track)
- *GmSecondaryFilter*: accepts a track if it is a secondary (it comes from another track)
- *GmKineticEnergyFilter*: accepts a track if its kinetic energy is between the two values given as extra parameters (including the two values). For steps it considers the energy at the beginning, that is the *G4PreStepPoint* energy
- *GmPostKineticEnergyFilter*: accepts a track if its kinetic energy is between the two values given as extra parameters(including the two values). For steps it considers the energy at the end, that is the *G4PostStepPoint* energy
- *GmVertexKineticEnergyFilter*: accepts a track if its vertex kinetic energy (the energy at creation) is between the two values given as extra parameters(including the two values)
- *GmEnergyChangeFilter*: accepts a track if the energy lost is between the values given by the two extra parameters (including the two values)
- *GmDepositedEnergyFilter*: accepts a track if the deposited energy is between the values given by the two extra parameters (including the two values)
- *GmInitialRangeFilter*: accepts a track if its range at creation is between the two values given as extra parameters (including the two values)
- *GmRangeFilter*: accepts a track if the range is between the values given by the two extra parameters (including the two values)
- *GmStepNumberFilter*: accepts a track if the step number is between the values given by the two extra parameters. It does not implement the *AcceptTrack* method
- *GmPositionXFilter*: accepts a track if the final X position is between the values given by the two extra parameters.
- *GmPositionPreXFilter*: accepts a track if the initial X position is between the values given by the two extra parameters.
- *GmPositionYFilter*: accepts a track if the final Y position is between the values given by the two extra parameters.

- *GmPositionPreYFilter*: accepts a track if the initial Y position is between the values given by the two extra parameters.
- *GmPositionZFilter*: accepts a track if the final Z position is between the values given by the two extra parameters.
- *GmPositionPreZFilter*: accepts a track if the initial Z position is between the values given by the two extra parameters.
- *GmDirectionPhiFilter*: accepts a track if the final phi direction is between the values given by the two extra parameters.
- *GmDirectionPrePhiFilter*: accepts a track if the initial phi direction is between the values given by the two extra parameters.
- *GmDirectionThetaFilter*: accepts a track if the final phi direction is between the values given by the two extra parameters.
- *GmDirectionPreThetaFilter*: accepts a track if the initial phi direction is between the values given by the two extra parameters.
- *GmProcessFilter*: accepts a step if the process name that defined it is in the list given as extra parameters. It does not implement the `AcceptTrack` method
- *GmParticleProcessFilter*: accepts a track if the particle and the process that defined the step are in the list given as extra parameters. The parameters must be provided as a list of pairs particle name - process name. It does not implement the `AcceptTrack` method
- *GmCreatorProcessFilter*: accepts a track if the process that created the track is in the list given as extra parameters. It does not implement the `AcceptTrack` method
- *GmInteractionFilter*: accepts a step if the process that defined it is an interaction. i.e. if it is of one of the `G4ProcessType`'s: `fElectromagnetic`, `fOptical`, `fHadronic`, `fPhotolepton_hadron`, `fDecay`, `fGeneral` or `fParameterisation`. It does not implement the `AcceptTrack` method
- *GmMaterialFilter*: accepts a step if the material that it is in (the initial step point) is in the list given as extra parameters. It does not implement the `AcceptTrack` method
- *GmFilterFromClassifier*: accepts a track if the classifier given as first parameter returns a value equal to the second parameter. It does not implement the `AcceptTrack` method
- `% PETCutsStudyFilter`: `% RTCutsStudyFilter`: `% RTFilterByEILong` `% \item{\bf GmMinRangeCutOutsideVoxelFilter}`: `% \item{\bf GmOtherSeedFilter}`: `% \item{\bf GmProdCutAllProcOutsideVoxelFilter}`: `% \item{\bf GmProdCutOutsideVoxelFilter}`:

Volume filters

These are a set of filters that accept tracks under one of the following conditions

- *In*: particle is in a volume
- *Enter*: particle is entering a volume. *AcceptTrack* method returns always false
- *Exit*: particle is exiting a volume. *AcceptTrack* method returns always false, except in the case where one of the selected volumes is the world and track is exiting it
- *Traverse*: particle traverses a volume, it does neither enter nor exits it
- *Start*: particle is starting in a volume. *AcceptTrack* method may only return true if it is the first step
- *End*: particle is ending in a volume. *AcceptTrack* method may only return true if the track is ending

The volume names are given as extra parameters to the filter. The types of volume are the following ones

- *LogicalVolume*: a *G4LogicalVolume* object
- *PhysicalVolume*: a *G4VPhysicalVolume* object. The volume name and copy number are set separated with a ':' character, e.g. *volA:1* (see section on *Identifying touchables*)
- *PhysicalVolumeReplicated*: for optimisation reasons the *PhysicalVolume* filters use the pointers to the physical volumes to find them. But in the case of replicated physical volumes (i.e. divisions, replicas and parameterisations), there is a unique *PhysicalVolume* pointer, while the different copies are managed internally by Geant4. Therefore you cannot select only a few of the copies of a physical volume with the above filters. For these cases you should use the *PhysicalVolumeReplicated* filters.
- *Touchable*: a *G4Touchable* object. The volume name and copy number are set separated with a ':' character, the ancestors are separated by a '/' character, e.g. *volB:3/volA:1* (see section on *Identifying touchables*)
- *Region*: a *G4Region* object
- *LogicalVolumeChildren*: a *G4LogicalVolume* object or any of the *G4LogicalVolume* children of it
- *PhysicalVolumeChildren*: a *G4VPhysicalVolume* object or any of the *G4VPhysicalVolume* children of it
- *PhysicalVolumeReplicatedChildren*: please read the discussion about *PhysicalVolumeReplicated*
- *RegionChildren*: a *G4Touchable* object or any of the *G4Touchable* children of it
- *TouchableChildren*: a *G4Region* object or any of the *G4Region* children of it

The name of these filters is constructed combining the geometry condition and the volume type, e.g. *GmInPhysicalVolumeFilter*, *GmTraverseTouchableFilter*, *GmEndRegionFilter*, *GmTraverseLogicalVolumeChildrenFilter*, *GmExitRegionChildrenFilter*.

There is a special case when parallel worlds are used: the scorers see the parallel world volumes, but the user actions do not. This means that you cannot use the above filters on a parallel world volume if you are using the filter for a user action, while you can if you use it for a scorer. If you want to do it you have to use the special parallel filters, namely

- EndParallelLogicalVolumeFilter
- EndParallelPhysicalVolumeFilter
- EndParallelPhysicalVolumeReplicatedFilter
- EndParallelRegionFilter
- EnterParallelLogicalVolumeFilter
- EnterParallelPhysicalVolumeFilter
- EnterParallelPhysicalVolumeReplicatedFilter
- EnterParallelRegionFilter
- ExitParallelLogicalVolumeFilter
- ExitParallelPhysicalVolumeFilter
- ExitParallelPhysicalVolumeReplicatedFilter
- ExitParallelRegionFilter
- InParallelLogicalVolumeFilter
- InParallelPhysicalVolumeFilter
- InParallelPhysicalVolumeReplicatedFilter
- InParallelRegionFilter
- StartParallelLogicalVolumeFilter

- StartParallelPhysicalVolumeFilter
- StartParallelPhysicalVolumeReplicatedFilter
- StartParallelRegionFilter
- TraverseParallelLogicalVolumeFilter
- TraverseParallelPhysicalVolumeFilter
- TraverseParallelPhysicalVolumeReplicatedFilter
- TraverseParallelRegionFilter

And it also means that you cannot use the above filters to act on a mass world whose position coincides with the one of a parallel world in the case of scorers, because the scorer filter will see the parallel world volume instead of the mass one. If you need to filter on mass volumes for a scorer a few volume filters are implemented, namely *GmInMassLogicalVolumeFilter*, *GmInMassPhysicalVolumeFilter*, *GmInMassPhysicalVolumeReplicatedFilter* and *GmInMassRegionFilter*.

When you use for example *GmExitLogicalVolumeFilter* the step is accepted when it exits the selected volume, without looking if the next volume it is entering is another copy of the selected volume. If what you want is that the steps are only accepted when they exit the selected volume and enter a different one, you may use the a filter that checks that if the PreStepPoint and the PostStepPoint are in different logical volumes:

- GmDifferentLogicalVolumeFilter

Although you may use the inverse of this filter to check if the two volumes are the same, GAMOS provides explicitly this filter:

- GmSameLogicalVolumeFilter

Filters of filters

There are another set of filters that receive as parameter one or several filters and act on them. The list of composed filters in the current GAMOS version is the following:

- *GmORFilter*: returns true if one of the filters returns true
- *GmXORFilter*: returns true if one and only one of the filters returns true
- *GmANDFilter*: returns true if every filter returns true
- *GmParentFilter*: apply a filter to the parent track instead of the current track
- *GmHistoryFilter*: returns true if all the filters in one of the previous steps, or the beginning of track, have returned true (i.e. it does not check again the current step if it was accepted in a previous one or begin of track)
- *GmHistoryAllFilter*: returns true if all the filters in all previous steps, and the beginning of track have returned true (i.e. it does not check again the current step if it was rejected in a previous one or begin of track)
- *GmHistoryOrAncestorsFilter*: behaves similarly as the *GmHistoryFilter* but also returns true if the condition is fulfilled by any step or track of the ancestors of the current track
- *GmHistoryAncestorsAllFilter*: behaves similarly as the *GmHistoryAllFilter* but also returns false if the condition is not fulfilled by any step or track of the ancestors of the current track
- *GmAncestorsFilter*: behaves similarly as the *GmHistoryOrAncestorsFilter* but it does not check if a previous step has passed the filter
- *GmOnAllSecondariesFilter*: makes the list of filters act of the secondary tracks created in the step, or in the whole track; returns true if all secondary tracks created accept all the filters.

- *GmOn1SecondaryFilter*: makes the list of filters act of the secondary tracks created in the step, or in the whole track; returns true if any of the secondary tracks created accept all the filters.
- *GmOnSecondaryFilter*: this filter is designed to work with *GmSecondaryTrackData* users. The filter will be applied not the current step or track but to the secondary tracks created.
- *GmInverseFilter*: returns the opposite that the filter it receives as only parameter

Applying filters to a user action

It may happen that you do not want that a filter is applied to a user action in all circumstances (i.e. each begin of track, each track step and each end of track). An example can be a user action that is using a “accumulating” data (see section on*GAMOS data*): at *PreUserTrackingAction* the variable, for example energy deposited, is initialized to 0, at *SteppingAction* the new energy deposit is added and at *PostUserTrackingAction* the variable is printed; but if you use in this case a filter that checks that the track is in a certain volume, it may happen that the *PreUserTrackingAction* is not invoked because the track at that moment was not in the volume and then the variable will be wrong. To prevent these cases there is the option of not applying the filters in a given circumstance by setting the parameters:

```
/gamos/setParam USER_ACTION_NAME:ApplyFiltersToStepping false
```

```
/gamos/setParam USER_ACTION_NAME:ApplyFiltersToPreTracking false
```

```
/gamos/setParam USER_ACTION_NAME:ApplyFiltersToPostTracking false
```

```
/gamos/setParam USER_ACTION_NAME:ApplyFiltersToStacking false
```

where *USER_ACTION_NAME* is the name of the user action where you want the filter to take effect, which includes the name of the filters and classifiers (see section on*User action names*).

Checking filters at a user action

The default behaviour of a user action is that their filters are invoked one after the other and all of them have to return OK before the user action method is invoked. For efficiency reasons if a filter is not passed, the other filters will not even be asked. This behaviour may represent a problem in some cases. For example if you implement two history filters and the first one is not accepted for a step, the second one will not be invoked; but history filters need to be invoked at each step, because they have to check if any of the steps in a track history passed the filter. To prevent these cases there is the option of not applying the filters in a given circumstance by setting the parameters:

```
/gamos/setParam USER_ACTION_NAME:CheckAllFiltersAtStepping false
```

```
/gamos/setParam USER_ACTION_NAME:CheckAllFiltersAtPreTracking false
```

```
/gamos/setParam USER_ACTION_NAME:CheckAllFiltersAtPostTracking false
```

```
/gamos/setParam USER_ACTION_NAME:CheckAllFiltersAtStacking false
```

where *USER_ACTION_NAME* is the name of the user action where you want this to take effect, which includes the name of the filters and classifiers (see section on*User action names*).

Filtering steps in the future

It is possible to filter an step with a future condition. For example you may want to score the energy of the steps that are in a volume only if the track in a future step reaches another volume. This feature needs a special mechanism, as the steps cannot be saved until knowing if a future step will fulfill the second condition, and when this happens the steps to be saved have been deleted (Geant4 only saves one step at a time).

To use this mechanism a *GmFutureFilter* has to be defined, giving as arguments two filters: first the one that should fulfill the steps to be used (for scoring, for saving their information, filling histograms, ...) and second the one that should fulfill the steps that trigger the use of the first steps.

```
/gamos/filter myFutureFilter GmFutureFilter FILTER_PAST FILTER_FUTURE
```

If a future filter is used with a user action, it must be the only filter. If you want to add more normal filter together with the future filter you should combine them with *GmANDFilter* and put them as *FILTER_PAST* of the future filter.

GmFutureFilter only acts in one track, this means that the steps to be used as well as the future steps belong to the same track. If you want to set a condition in the future also to the children tracks of the steps to be used (e.g. score the energy of the steps that are in a volume only if the track or any of the secondary tracks created at this step or any future step of this track, including the children of these secondary tracks, reach another given volume, you have to use *GmFutureWithChildrenFilter*.

Future filters only act on steps, so if you use them for example together with a *GmTrackDataHistosUA* they will always return true, i.e. they will have no effect.

In the case of scorers the use of future filters has to take into account the fact that scorers are only invoked when a step happens in the detector volume that has attached the scorer. This could create a problem if the *FILTER_FUTURE* implies a geometrical condition, like for example *if steps happens in a given volume*. In this case the scorer will not be invoked and this filter will never be checked. The solution for this is that you attach a detector to the volume of the *FILTER_FUTURE* and then add in the *FILTER_PAST* filter asking for steps to be only in the volume(s) where you want to score.

CLASSIFIERS

Introduction

A classifier is a class that contains a method that receives a *G4Step* or a *G4Track* and returns a different index (an integer) depending on some given criteria. In other words it classifies the step or track and returns the index of its classification. These classes are unique to GAMOS, as Geant4 does not provide this functionality.

Classifiers can act on user actions or scorers. If one or several classifiers are set to act on a user action, it is up to the concrete user action to determine which use it makes of them, or to ignore them. The most common use of classifiers by user actions is to produce a different histogram or table for each classification index. For details on classifiers acting on scorers see the section on *Scorers*.

The use of user actions and scorers together with classifiers is a powerful means to obtain very detailed information on the simulation through simple user commands. See the tutorial in section *Histograms and scorers* for more details on this.

Several classifiers need some extra parameters (see list of classifiers below) that control their behaviour. To use these classifiers they have to be declared first with the following command

```
/gamos/classifier CLASSIFIER_NAME CLASSIFIER_CLASS PARAMETER_1 PARAMETER_2 ...
```

where *CLASSIFIER_NAME* is the new name you want to give to a classifier to attach it to a user action or to a scorer, *CLASSIFIER_CLASS* is the name of the classifier class, and *PARAMETER_1 PARAMETER_2 ...* are the values of the parameters that the classifier needs. Those classifiers that do not need any parameter, can be assigned directly to a user action or a scorer without giving them a new name: the name will be the one of the classifier class.

To attach one classifier to a user action you put its name after the user action name in the command line that selects it

```
/gamos/userAction USER_ACTION CLASSIFIER_NAME
```

or you can use a classifier to act on a scorer with the command

```
/gamos/scoring/assignClassifier2Scorer CLASSIFIER_NAME SCORER_NAME
```

If you want to use more than one classifier for a user action or a scorer, you have to use the classifier *GmCompoundClassifier* (see below).

Each classifier has a method, *GetIndexName(G4int index)*, that returns a different name for each index value. If a classifier does not implement this method, the one in the base class returns the index number converted to a string. This method is used by user actions and scorers to add the index number to the name of the histograms, tables or scores.

Classifiers are plug-in's, so that a user can create her/his own classifier and select it with a user command. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmClassifierFactory*, or the *Histograms and scorers* tutorial.

The following classifiers are currently implemented in GAMOS:

- *GmPSClassifierByIAncessor*: It assigns a different index to different copy numbers of a volume. It has an extra parameter that sets the level of the ancestor; if it is $N = 0$, it will use the copy numbers of the volume itself, if it is $N > 0$, it will look for the copy numbers of the N -th ancestor
- *GmPSClassifierByAncestors*: It assigns a different index to different copy numbers of the sensitive volume. It has two extra parameters that set the number of ancestor levels (*NAncessor*) and the maximum number of copies in a level (*NShift*). The index is built as a sum of *NShift:sup: 'N' * copyNumber_of_Nancestor* from $N=0$ to $N=NAncessor-1$
- *GmClassifierByLogicalVolume*: It assigns a different index to different logical volumes
- *GmClassifierByPhysicalVolume*: It assigns a different index to different physical volumes
- *GmClassifierByPhysicalVolumeReplicated*: See discussion about *PhysicalVolumeReplicated* filters
- *GmClassifierByRegion*: It assigns a different index to different regions
- *GmClassifierByMaterial*: It assigns a different index to different volume materials
- *GmClassifierByKineticEnergy*: The user must define a minimum, a maximum and a width of the energy intervals. It creates kinetic energy (at *PreStepPoint*) intervals with these values and assigns a different index to different intervals
- *GmClassifierByProcess*: It assigns a different index to different process names that define the *G4Step*
- *GmClassifierByParticleProcess*: It assigns a different index to different particle-process name pairs that define the *G4Step*. This means that the ionisation for electrons and positrons will produce a different index, despite the process being called the same for both particles
- *GmClassifierByCreatorProcess*: It assigns a different index to different track creator process names
- *GmClassifierByParticle*: It assigns a different index to different particle types
- *GmClassifierByPrimaryParticle*: It assigns a different index following the particle type for the primary that originated the current particle, or the particle itself if it is a primary
- *GmClassifierBySecondariesList*: It builds a list of the secondary particle names in alphabetical order, without repeating names, and assigns a different index to each list
- *GmClassifierBySecondariesListExclude*: It builds a list of the secondary particle names in alphabetical order, without repeating names, and assigns a different index to each list. When the classifier is defined, you can add as parameter a list of particles and they will be excluded from the list
- *GmClassifierByHadronicReaction*: It builds the name of the hadronic reaction, and assigns a different index to each reaction name. The name is built taking into account the target nucleus, the projectile and the emitted particles, without taking into account the gammas, except in the case of neutron capture. Examples of reaction names are: $\text{Be9}(n,p)\text{Li9}$, $\text{Fe56}(p,pn)\text{Fe55}$, $\text{Pb208}(a,p)\text{Tl211}$
- *GmCompoundClassifier*: This classifier receives a list of classifiers and builds an index as $\text{Classifier}_1 * \text{NShift} + \text{Classifier}_2 * \text{NShift} * \text{NShift} + \dots$ Where *NShift* is defined by the parameter `/gamos/setParam GmCompoundClassifier:NShift NSHIFT`, that by default takes a value of 100. Be aware that the classifier index is stored as a 32-bit integer, so be careful that the index is not too big (bigger than 2^{32}).

Filters are plug-in's, so that a user can create her/his own filter and select it with a user command. To learn how to do this, see the instructions in the section *Creating your plug-in*, using the *GmClassifierFactory*.

Setting indices to classifiers

In most classifiers by default the indices are built automatically as the objects of classification appear and they start with 0 and grow one by one. You may want to change this behaviour (it may be specially useful if you are using importance sampling). To do it you should use the command:


```
/gamos/classifier/setIndices VALUE_1 INDEX_1 VALUE_2 INDEX_2
```

where *VALUE_{ii}* are the names of the objects of classifications (logical volumes, process names, ...), and *INDEX_{ii}* are the new indices. In the case of classifier *GmClassifierByParticleProcess* you need two values for each index: particle name and process name.

You should be careful to list all the values that may appear in the classifier. If not, GAMOS will automatically assign an index to the new values, starting with the number following the maximum index defined with the user command. The exceptions are the classifiers *GmClassifierBy1Ancestor* and *GmClassifierByAncestors*, for which all values should be given.

Classifying on secondary data

As mentioned above the classifiers take the data to be classified from a *G4Step* or a *G4Track*. But when a classifier is used by *GmSecondaryTrackHistosUA* (see chapter on *Analysis* it may be useful that the classification is based on the data of the secondary particles created in the step. To get this behaviour it should be used the paramter:

```
/gamos/setParam CLASSIFIER_NAME:OnSecondary 1
```


DISTRIBUTIONS

Introduction

The GAMOS distribution classes represent one-dimensional functions, which assign an output value for each input variable of the a given variable. In other words, they represent one-dimensional functions:

$$y = f(x)$$

In fact the correspondence between x and y values is read from a file, so that any arbitrary function shape can be easily defined.

There are currently three users of GAMOS distributions. We briefly describe them here and give further details in the corresponding section:

- To bias a primary generator distribution. In this case the output values represent the probabilities of occurrence of the variables (e.g. position coordinate X, direction angle theta, energy, time).
- To define the weights in importance sampling. A GAMOS data should be defined so that the input values are those of the data at each track step.
- To define the index corresponding to each value for the classifier *GmClassifierByDistribution*. A GAMOS data should be defined so that the input values are those of the data at each track or track step.

Creating a distribution

A distribution has to be created with the command

```
/gamos/distribution DISTRIBUTION_NAME DISTRIBUTION_CLASS
```

where *DISTRIBUTION_NAME* is the name you give to the distribution, which will be used to assign it to an object and to define its parameters; and *DISTRIBUTION_CLASS* is the class name (see below for the list available distribution classes).

Assigning a GAMOS data

Except in the use of a GAMOS distribution is for primary generator biasing, you have to assign a GAMOS data. This data is the variable of the distribution, i.e. it will be used to extract the input value for each track or track step. To assign a data the following parameter should be used:

```
/gamos/setParam DISTRIBUTION_NAME:Data DATA_NAME
```

Reading values from a file

Many of the distributions described below need to read the data from a file. The parameter

```
/gamos/setParam DISTRIBUTION_NAME:FileName FILE_NAME
```

defines the name of the file that contains the distribution data. This file is a two column set of values, where the first column gives the input values (x) and the second the output values (y). After being read the input values are ordered in increasing order. To calculate the output value corresponding to an input value, each distribution uses its own algorithm, see below. If the input value is smaller than the minimum value in the file or bigger than the maximum value, an exception will be thrown, as the value cannot be calculated.

It is also possible to read the data from a histogram in ROOT or CSV format. In this case an extra parameter has to be provided to set the name of the histogram:

```
/gamos/setParam DISTRIBUTION_NAME:HistoName HISTO_NAME
```

The input values are taken as the centres of the histogram bins. The minimum and maximum values read are extended to the lower and upper edge of the histogram axis, i.e. minus and plus half a bin. If the input value is between this minimum and the lowest bin centre, the output is interpolated using the lowest and second lowest bins. Similarly for values between the maximum and the highest bin centre.

The type of the file will be automatically determined from the file name: if it ends by *.csv* it will be taken as a CSV format histogram file, if it ends by *.root* it will be taken as a ROOT format histogram file, else it will be taken as a text file. If you do not want to follow this convention then you have to set the file type explicitly by using the parameters:

```
/gamos/setParam DISTRIBUTION_NAME:FileNameCSV FILE_NAME
```

```
/gamos/setParam DISTRIBUTION_NAME:FileNameROOT FILE_NAME
```

Numeric distributions

A numeric distribution is a distribution where the input and output values are numbers. There are several classes of numeric distributions:

- *GmGaussianDistribution*. The sigma and constant value of the Gaussian distribution has to be given as two extra parameters when the distribution is created. By default the Gaussian distribution is centred at 0, if you want to centre it at other value, you have to give a third parameter when the distribution is created. To assign the data, the following parameter must be used: `/gamos/setParam DISTRIBUTION_NAME:Data DATA_I_NAME`
- *GmPolynomialDistribution*. It can serve to define any polynomial distribution: $f(x) = a_0 + a_1 * x + a_2 * pow(x,2) + \dots$. The parameters a_i are given as extra parameters when the distribution is created. The number of extra parameters defines the order of the polynomial: `POLYNOMIAL_ORDER = NUMBER_OF_PARAMETERS - 1`. To assign the data, the following parameter must be used: `/gamos/setParam DISTRIBUTION_NAME:Data DATA_I_NAME`

The other numeric distributions are general ones: they read the input - output relationship from a file, so that any distribution shape can be defined. The distribution finds among the list of read input values (after ordering them) the two that are closest (i.e. one bigger and one smaller); then it interpolates the two output values corresponding to these two input values. The difference between the distributions described below lies in the way this interpolation is done.

GAMOS will check if the data value is inside the interval defined by the minimum and maximum of the distribution. If it is smaller, it will assign an index 0, if it is bigger it will assign an index INT_MAX (2³²). In both cases a warning will be printed. If you do not want to allow this behaviour, but you want that the job is stopped if the data is out of limits, you have to set the parameter:

```
/gamos/setParam DISTRIBUTION_NAME:AllowOutOfLimits 0
```

- *GmNumericDistributionLinLin*. Interpolates linearly in the input and linearly in the output: $f(x) = f(x_S) + (x-x_S)/(x_B-x_S)*(f(x_B)-f(x_S))$ (where x_S and x_B are the closest smaller and bigger values of x read from the file).
- *GmNumericDistributionLogLin*. Interpolates logarithmically in the input and linearly in the output: $f(x) = f(x_S) + (\log(x)-\log(x_S))/(\log(x_B)-\log(x_S))*(f(x_B)-f(x_S))$ (where x_S and x_B are the closest smaller and bigger values of x read from the file).
- *GmNumericDistributionLinLog*. Interpolates linearly in the input and logarithmically in the output: $f(x) = f(x_S) + \exp((x-x_S)/(x_B-x_S)*(\log(f(x_B))-log(f(x_S))))$ (where x_S and x_B are the closest smaller and bigger values of x read from the file).
- *GmNumericDistributionLogLog*. Interpolates logarithmically in the input and logarithmically in the output: $f(x) = f(x_S) + \exp((\log(x)-\log(x_S))/(\log(x_B)-\log(x_S))*(\log(f(x_B))-log(f(x_S))))$ (where x_S and x_B are the closest smaller and bigger values of x read from the file).
- *GmNumericDistributionLower*. Takes the value corresponding to the closest smaller value: $f(x) = f(x_S)$.
- *GmNumericDistributionUpper*. Takes the value corresponding to the closest bigger value: $f(x) = f(x_B)$.

String distributions

A string distribution is a distribution where the input values are strings (e.g. particle names, process names, volume names) and the output values are numbers. The values must be defined in a text file similar to the one for numeric distributions: two columns where the first column is a list of names and the second column the values that correspond to each name.

String data distribution

The *GmStringDistribution* is a general string distribution. It takes the value from the data associated to it, which has to be of type string (see chapter on *Analysis*)

Geometrical biasing distribution

The *GmGeometricalBiasingDistribution* is a special distribution that serves to do geometrical biasing. For each track step it checks if the step is at a volume boundary and if so it calculates the output value as the division between the value at the end and the value at the beginning of the step (the *PostStepPoint* and *PreStepPoint*), else it returns 1. The file is supposed to contain in the first column the list of volume names and in the second the list of weights. Two data has to be assigned to it, what can be done with the parameters:

```
/gamos/setParam DISTRIBUTION_NAME:DataPre DATA_NAME
```

```
/gamos/setParam DISTRIBUTION_NAME:DataPost DATA_NAME
```

These data should be one of the volume data: *FinalLogicalVolume* and *InitialLogicalVolume* *FinalPhysicalVolume* and *InitialPhysicalVolume*, *FinalTouchable* and *InitialTouchable*, etc. But this is not enforced, so that you may find another use for this distribution.

If your geometry is complicated it may happen that at some place there is a frontier between two volumes with too different weights, so that if a track traverses this frontier it gets a too high index. To avoid this you may set a maximum value of the index with the parameter:

```
/gamos/setParam DISTRIBUTION_NAME:MaxValue VALUE
```

Another trick to give more flexibility to your biasing is to avoid that a tracks receives a weight given by this distribution several times

Ratio of distributions

You may define a distribution whose value is defined as the ratio of two distribution values. The distribution class is named *GmDistributionRatio*. The two distributions has to be defined with the parameters:

/gamos/setParam DISTRIBUTION_NAME:DistributionPre DISTRIBUTION

/gamos/setParam DISTRIBUTION_NAME:DistributionPost DISTRIBUTION

UTILITY USER ACTIONS

These are a miscellaneous set of user actions that add some functionality. As for any user action, filters can be assigned to them to select for which type of tracks they will be activated and in some cases classifiers may be used to classify the output tables.

Counting the number of tracks and events

This utility prints a line every N events with the event number, the number of tracks in this event and the accumulated number of tracks in all events:

```
%%% EVENT 0 NTRACKS 4 TOTAL NTRACKS 4
%%% EVENT 1000 NTRACKS 6 TOTAL NTRACKS 4663
%%% EVENT 2000 NTRACKS 4 TOTAL NTRACKS 9440
```

Its main use is to inform the user of the progress of the job in interactive running. To activate this utility use the command:

```
/gamos/userAction GmCountTracksUA
```

The user can control the interval of events as well as the first event to start printing with the parameters:

```
/gamos/setParam USER_ACTION_NAME:EachNEvent NEV
```

```
/gamos/setParam USER_ACTION_NAME:FirstEvent NEV
```

These parameters have a default value of 10 and 0 respectively.

This utility distinguishes for the ionisation and bremsstrahlung processes those cases when a secondary particle is emitted and those when the step is limited to assure a correct energy loss and multiple scattering but no secondary particle is emitted (it adds *_NoSeco* at the end of the process name).

Counting the processes

To activate this utility use the command:

```
/gamos/userAction GmCountProcessesUA
```

It prints four tables:

- At the beginning of run all the active processes for each particle type:

```
PROC_LIST e+ : Transportation
PROC_LIST e+ : annihil
PROC_LIST e+ : eBrem
```

```
PROC_LIST e+ : eBrem_NoSeco
PROC_LIST e+ : eIoni
PROC_LIST e+ : eIoni_NoSeco
PROC_LIST e+ : msc
PROC_LIST e- : Transportation
PROC_LIST e- : eBrem
PROC_LIST e- : eBrem_NoSeco
PROC_LIST e- : eIoni
PROC_LIST e- : eIoni_NoSeco
PROC_LIST e- : msc
PROC_LIST gamma : Rayl
PROC_LIST gamma : Transportation
PROC_LIST gamma : compt
PROC_LIST gamma : conv
PROC_LIST gamma : phot
```

This list may not be printed unless you set the parameter. */gamos/setParam USER_ACTION_NAME:PrintProcList 1*

- At the end of run how many times a process determined the step for each particle type:

```
PROC_COUNT e+ : Transportation = 1870
PROC_COUNT e+ : annihil = 728
PROC_COUNT e+ : eBrem = 86
PROC_COUNT e+ : eBrem_NoSeco = 5
PROC_COUNT e+ : eIoni = 861
PROC_COUNT e+ : eIoni_NoSeco = 864
PROC_COUNT e+ : msc = 5531
PROC_COUNT e+ : ALL = 9945
PROC_COUNT e- : Transportation = 1977
PROC_COUNT e- : eBrem = 142
PROC_COUNT e- : eBrem_NoSeco = 10
PROC_COUNT e- : eIoni = 1736
PROC_COUNT e- : eIoni_NoSeco = 9812
PROC_COUNT e- : msc = 25744
PROC_COUNT e- : ALL = 39421
PROC_COUNT gamma : Rayl = 4
PROC_COUNT gamma : Transportation = 4657
PROC_COUNT gamma : compt = 53
PROC_COUNT gamma : phot = 125
PROC_COUNT gamma : ALL = 4839
```

- At the end of run how many times a process was the creator of a particle for each particle type:

```
PROC_CREATOR_COUNT e+ : Primary = 1000
PROC_CREATOR_COUNT e- : Primary = 1000
PROC_CREATOR_COUNT e- : compt = 53
PROC_CREATOR_COUNT e- : eIoni = 2597
PROC_CREATOR_COUNT e- : phot = 125
PROC_CREATOR_COUNT gamma : annihil = 1456
PROC_CREATOR_COUNT gamma : eBrem = 228
```

You may choose to print the creator model name together with the creator process name (separated by a “_”), by setting the parameter

/gamos/setParam USER_ACTION_NAME:PrintCreatorModel TRUE

- At the end of run how many particles of each type were created:


```
PART_LIST: e+ = 1000
PART_LIST: e- = 1485
PART_LIST: gamma = 2285
```

The *eIoni_NoSeco* and *eBrem_NoSeco* refer to the cases when ionisation or bremsstrahlung processes limit the step but no secondary particle is produced. The reason for this limitation is to guarantee that the step are not too big so that physics precision may be spoiled. You may have a look for example at <http://fismed.ciemat.es/GAMOS/RToptim/ParameterHelp.html> for a description of the Geant4 electromagnetic physics parameters.

This user action may be combined with filters and classifiers.

Shower shape studies

GAMOS offers an utility to help in doing shower shape studies. To use it is enough to active the user action

```
/gamos/userAction GmShowerShapeUA
```

Each time a new track step satisfies the conditions of the filters associated to this action a new shower is started, and the user action accumulates all the energy deposits of all the children created from this track step. This means taht each track step that passes the filter initiates a new shower, unless the track is a children of a track which has already initiated a shower. But you should not forget the logic of the user actions and filters: the user action *GmShowerShapeUA* will only be invoked for the steps that pass the associated filters; i.e. if one of the children track steps does not pass the filter, it will not be included in the shower. If this behaviour does not provide you with the expected results, you may consider using the filter *GmAncestorsFilter* (see section of *Filters*).

As we just mentioned each step of a track may initiate a new shower, but if you want to include all the steps of the same track (and their children) in a unique shower, you have to set the parameter:

```
/gamos/setParam USER_ACTION_NAME:IncludeOtherStepsOfFirstTrack TRUE
```

At the end of each event, the showers are analysed and several variables are calculated for each of them:

- *Total energy*: sum of the deposited energies of all steps belonging to the shower
- *Shower direction*: two options are provided to define the shower direction. The first one (default) is to take the direction of the particle that initiated the shower; this is the direction of th PreStepPoint of the first track step. The second option is to calculate a shower ‘average’ direction, formed joining the point where the track is initiated and an ‘average’ shower point. This ‘average’ point is calculating as the sum of all the shower step points weighted with the energy deposited at each step. To use this second definition instead of the first one, you have to set the parameter: `/gamos/setParam USER_ACTION_NAME:ShowerDirection Shower`
- *Step point longitudinal length*: distance from an step position to the initial shower point along the shower direction
- *Step point transversal length*: minimum distance from an step position to the line built from the initial shower point and the shower direction

From the above two definitions several shower variables are calculated:

- *Maximum longitudinal length*: maximum of the step point longitudinal lengths
- *Maximum transversal length*: maximum of the step point transversal lengths
- *Average longitudinal length*: average of the step point longitudinal lengths, weighted with the energy deposits
- *Average transversal length*: average of the step point transversal lengths, weighted with the energy deposits

The deposition of energy in the step is actually done in small quantities approximately uniformly distributed along the step. As it would be too CPU tiem consuming to simulate eacho of these interactions, we just consider a deposition

point, making a linear interpolation between the two steps edges; in other words, we use as energy deposit point the average of the PreStepPoint and PostStepPoint. If you want to use instead the PreStepPoint or PostStepPoint, you have to use the parameter:

```
/gamos/setParam USER_ACTION_NAME:StepPointToUse TYPE
```

where *TYPE* can be *Pre*, *Post* or *Linear* (the default choice). Other algorithms may be implemented in the future at user request.

With the above-mentioned histograms variables a file named *shower.root/csv* is written with the following histograms (the second name is the type of histogram for histogram limits setting (see section on *Using a common histogram class*):

- Total energy (" Total Energy")("E")
- Maximum longitudinal length (" Maximum R transv")("Pos")
- Maximum transversal length (" Maximum R transv")("Pos")
- Average longitudinal length (" Maximum R transv")("Pos")
- Average transversal length (" Maximum R transv")("Pos")
- Shower direction theta angle (" Shower direction theta")("Angle")
- Shower direction phi angle (" Shower direction phi")("Angle")

The sum of energy deposits inside a certain transversal distance (a radius) is calculated and an histogram is filled. By default the list of radii is: 0.1, 0.3, 0.5, 1., 5. mm and infinite (DBL_MAX). You may change this list with the parameter:

```
/gamos/setParam USER_ACTION_NAME:Radii RADIUS_1 RADIUS_2 ...
```

With these variables two sets of histograms are filled:

- Energy inside radius ("Energy inside radius")("E")
- Energy inside radius divided by total shower energy (" Relative energy inside radius")("E")

Killing all tracks

The action

```
/gamos/userAction GmKillAtStackingActionUA
```

serves to kill all particles at the stacking action G4ClassificationOfNewTrack method, i.e. before they start being tracked. You may use it in combination with one or several filters to kill only the particles that are accepted by them. For example,

```
/gamos/userAction GmKillAtStackingActionUA GmElectronFilter
```

will only kill electrons.

If you want to apply the killing at stepping action, probably because you want to apply some step filter, you can use the user action:

```
/gamos/userAction GmKillAtSteppingActionUA
```

It is also possible to kill at the beginning of tracking action (when the track starts). The difference with respect to killing at stacking action is that at least a first step is done for each track, and you can get some information of each track. To do this, you can use the user actions:

```
/gamos/userAction GmKillAtTrackingActionUA
```

Table of tracks and steps

You may get a table of the number of tracks and steps by instantiating the user action

```
/gamos/userAction GmCountTracksAndStepsUA
```

It will produce a table with the number of tracks and steps in the whole run. You may get more details by using it with filters and classifiers. For example the command:

```
/gamos/userAction GmCountTracksAndStepsUA GmClassifierByParticle
```

will produce a table similar to this one

```
%%% COUNT_TRACKS_AND_STEPS: GmClassifierByParticle
%%% COUNT_TRACKS: gamma = 100
%%% COUNT_TRACKS: e- = 8
%%% COUNT_TRACKS: e+ = 1
%%% COUNT_TRACKS: ALL = 109
%%% COUNT_NSTEPS: gamma = 399
%%% COUNT_NSTEPS: e- = 29
%%% COUNT_NSTEPS: e+ = 4
%%% COUNT_STEPS: ALL = 432
```

Material budget studies

The action

```
/gamos/userAction GmMaterialBudgetUA
```

serves to make an study of the material budget of your geometry. We compute the material budget of a geometry along a line by integrating the radiation length of each volume traversed multiplied by the length of the line segment in that volume. This user action may be useful for checking your geometry.

The usual way to use this user action is to send parallel geantinos (the Geant4 particle that does not interact, only traverses the geometry) starting in a plane perpendicular to the geantino direction and computing the material budget for each of them

The user action *GmMaterialBudgetUA* computes the material budget along each track and writes in the file *matbud.root* / *csv* the following histograms:

- 1D Profile histogram of material budget in position X (" Position X")
- 1D Profile histogram of material budget in position Y (" Position Y")
- 1D Profile histogram of material budget in position Z (" Position Z")
- 2D Profile histogram of material budget in positions X and Y (" Position XY")
- 2D Profile histogram of material budget in positions X and Z (" Position XZ")
- 2D Profile histogram of material budget in positions Y and Z (" Position YZ")

Detailed report of where CPU time is spent

You may get a detailed report of where the CPU time is spent by instantiating the user action

```
/gamos/userAction GmTimeStudyUA CLASSIFIER_1 CLASSIFIER_2 ...
```

By selecting different classifiers you can get a report of the time spent by each particle, in each logical volume, in each energy bin, etc. (see section on *Classifiers*).

The table will have a format similar to the following one:

```

##### TIMING RESULTS for timer GmTimeStudyUA_ClassifierByParticleAndKinE
GmTimeStudyUA_GmClassifierByParticle_GmClassifierByKineticEnergy
e+/0.0001-0.001:  User=0  Real=0  Sys=0.01
e+/0.001-0.01:   User=0.07 Real=0.16 Sys=0
e+/0.01-0.1:    User=1.3  Real=1.48 Sys=0.04
e+/0.1-1:       User=74.66 Real=78.74 Sys=1.17
e+/1e-05-0.0001: User=0  Real=0  Sys=0
e+/1e-06-1e-05:  User=0  Real=0  Sys=0
e-/0.0001-0.001: User=0.53 Real=0.37 Sys=0.02
e-/0.001-0.01:   User=12.42 Real=13.58 Sys=0.29
e-/0.01-0.1:    User=5.98 Real=6.42 Sys=0.16
e-/0.1-1:       User=7.78 Real=8.8  Sys=0.22
e-/1-10:        User=0  Real=0  Sys=0
e-/1e-05-0.0001: User=0  Real=0  Sys=0
e-/1e-06-1e-05:  User=0  Real=0  Sys=0
gamma/0.001-0.01: User=1.29 Real=1.49 Sys=0.02
gamma/0.01-0.1:   User=0.63 Real=0.58 Sys=0.01
gamma/0.1-1:     User=27.95 Real=29.7 Sys=0.48
gamma/1-10:      User=0.38 Real=0.36 Sys=0
    
```

which can be obtained with the commands

```

/gamos/classifier ClassifierByParticleAndKinE GmCompoundClassifier GmClassifierByParticle GmClassifierByKineticEnergy
    
```

```

/gamos/userAction GmTimeStudyUA ClassifierByParticleAndKinE
    
```

The time in each category is the time counted at each step. Exactly it is the counting from the beginning to the end of the method *G4SteppingManager::Stepping()*. To do this without modifying the Geant4 class, the class *GmTimeStudyMgr* inherits from *G4VSteppingVerbose* and it is set as the stepping verbose class, substituting the class *G4SteppingVerbose* class, that is the one that controls the verbosity of the command */tracking/verbose*. This means that this command will have no effect and if you want to use it you should do with the parameter

```

/gamos/setParam GmTimeStudyUA:G4VerboseLevel VERB
    
```

You may observe that the time summed over all the categories is smaller than the run time given by the command */run/verbose 1*. This is because the time is only the time spent at the method mentioned above, which does not take into account the initialisation and termination times of each track, event and run.

Changing the weight using a distribution

The command */gamos/userAction GmChangeWeightUA* changes the weight at each step or track following a GAMOS distribution. You have to set the distribution with the parameter:

```

/gamos/setParam USER_ACTION_NAME:Distribution DIST_NAME
    
```

Copying the weight to the secondary particles

Geant4 usually copies the weight of a track to the secondary tracks that it creates. But in the process is RadioactiveDecay, this does not work. Therefore if you are using radioactive decay and need this feature you have to activate the user action :

/gamos/userAction GmCopyWeightToSecondaryUA

Stop run after a certain CPU time

There may be some times when you want to limit the CPU time so that the job stops even if all the events demanded have not run. To do this you can use the user action */gamos/userAction GmStopRunAfterTimeUA*. The time is limited with the parameter:

/gamos/setParam USER_ACTION_NAME:Time TIME

In fact, the time is only checked after each event, so you may find that the time it actually stops is bigger than what you demanded.

Visualising only a set of events

This utility is designed for the case when you want to visualise only a few events but not the rest (e.g. because the interesting event happens after many others and storing them all occupies a lot of space in the visualization file). To do this you have to select the user action:

/gamos/userAction GmVisStoreTrajUA

You can set the list of events that you want to visualise with the parameter

/gamos/setParam USER_ACTION_NAME:EventLis EVENT_1 EVENT_2 ...

And starting by one event in the list you can select how many events are to be visualised with the parameter:

/gamos/setParam USER_ACTION_NAME:NEvents NEVENTS

Setting the correlation between the gammas of a radioactive decay chain

The user action:

/gamos/userAction GmRadDecayGammaAngleCorrelationUA

is a prototype class to allow the user to set the correlation between the gammas produced in a decay chain. As of now it only sets the correlations for the two gammas, of energies 1.1732 and 1.3325 MeV of the decay chain of the Cobalt 60 radioisotope, and the correlation is a random one. If you are interested in this feature please contact the GAMOS team.

Printing electromagnetic parameters

The user action:

/gamos/userAction GmPrintEMParamsUA

serves to print the current electromagnetic parameters. See the Geant4 documentation to understand the meaning of each one.

MANAGING THE VERBOSITY

GAMOS verbosity managers

While GAMOS is running you can control the amount of information you get on the screen (and in the log files) with the GAMOS verbosity management. There are six levels of verbosity (each level includes the verbosity of the previous levels):

- *Silent* (= -1): no output is printed (only when there is an exception and the job stops, you will get the details of why it happened)
- *Error* (= 0): only error messages are printed
- *Warning* (= 1): only error and warning messages are printed
- *Info* (= 2): you get some detailed information of what is happening. Mainly messages at each run and each event
- *Debug* (= 3): you get a quite detailed information of what is happening. Mainly messages at each track and each step
- *Test* (= 4): this level is only meant for testing your code the first time you write it

On top of this, the verbosity in GAMOS is classified in different types, so that you can set different verbosity options for different parts of the code. The list of parts of the code that have its independent verbosity in the current GAMOS version is the following:

- *GmBaseVerbosity*: controls the verbosity of the base classes (filters, classifiers, input/output management, ...)
- *GmGeomVerbosity*: controls the verbosity of the geometry classes
- *GmGenerVerbosity*: controls the verbosity of the GAMOS generator
- *GmPhysicsVerbosity*: controls the verbosity of the physics classes
- *GmSDVerbosity*: controls the verbosity of the sensitive detectors, hits, digits and reconstructed hits
- *GmUAVerbosity*: controls the verbosity of the classes for user action management
- *GmScoringVerbosity*: controls the verbosity of the scoring classes
- *GmReadDICOMVerbosity*: controls the verbosity of the classes to read DICOM files
- *GmDataVerbosity*: controls the verbosity of the GAMOS data
- *GmDistributionVerbosity*: controls the verbosity of the GAMOS distributions
- *GmAnalysisVerbosity*: controls the verbosity of the analysis classes
- *GmUtilsUAVerbosity*: controls the verbosity of the utility user actions

- *DetVerbosity*: controls the verbosity of the Detector packages (the base classes for PET, SPECT and Compton Camera applications)
- *PETVerbosity*: controls the verbosity of the classes in the PET package. If set, it sets automatically the *DetVerbosity*
- *SPECTVerbosity*: controls the verbosity of the classes in the SPECT package. If set, it sets automatically the *DetVerbosity*
- *CCVerbosity*: controls the verbosity of the classes in the Compton Camera. If set, it sets automatically the *DetVerbosity*
- *RTVerbosity*: controls the verbosity of the classes in the RadioTherapy package
- *ShieldingVerbosity*: controls the verbosity of the classes in the RadioTherapy package

You can set the verbosity of each of the GAMOS verbosity types with a simple command on your command input file, for example:

```
/gamos/verbosity VERB_CLASS VERB_LEVEL
```

where *VERB_CLASS* is one of the verbosity name in the list above, and *VERB_LEVEL* can be any of the six values described above (in non-capital letters). Instead of the names, you may use the numbers that appear besides them.

By default the values of all GAMOS verbosity types are *warning*.

Controlling GAMOS verbosity by event

You may control the verbosity of each of the GAMOS verbosity managers event by event, that is, activate it for a certain interval of events and deactivate for another interval. To do this you have first to activate the user action:

```
/gamos/userAction GmGamosVerboseByEventUA
```

Then you have to set the event intervals with the command:

```
**/gamos/verbosity/byEvent VERB_CLASS VERB_LEVEL FIRST_EVENT LAST_EVENT
```

where *VERB_CLASS* is one of the verbosity name in the previous section, *VERB_LEVEL* can be any of the six values described in the previous section, and *FIRST_EVENT* *LAST_EVENT* are the first and last event affected by the verbosity level.

You may use as many times as desired this command, so that any number of event intervals may be defined for any of the GAMOS verbosity types.

The use of this command overwrites the command `/gamos/verbosity` so that all the events not included in an interval (including the *LAST_EVENT*) have verbosity *silent*.

Using a GAMOS verbosity manager in your code

If you write some new code, for example a new generator distribution, you may use one of the GAMOS verbosity managers following the instructions below.

Each of the GAMOS verbosity managers instantiates an object of the type *GmVerbosity*. If you want that your code is only printed when the corresponding verbosity level is set, you have to write this verbosity with the value of the level in parenthesis. For example, if you write one new generator position distribution and you want that a message is printed when somebody chooses it in the input file, you can write a message like the following one in the constructor of your class:

```
G4cout << GenerVerb(infoVerb)
<< \``MyPositionGeneratorDistribution created'` << G4endl;
```


This message will only be printed if the generator verbosity is set to *info* or a level above (*debug* or *test*).

If you want for example that your distribution prints a message with the calculated position at each event, you may write

```
G4cout << GenerVerb(debugVerb)
<< ``MyPositionGeneratorDistribution position = ''
<< position << G4endl;
```

This message will only be printed if the generator verbosity is set to *debug* or a level above (*test*).

As you may have deduced the rules for using each of the GAMOS verbosity managers are that the name of the verbosity is the same as the name of the verbosity manager simplified: no *Gm* at the beginning and *Verb* instead of *Verbosity* (e.g. from *GmAnaVerbosity*, you use *AnaVerb*). And the name of the level in C++ code is the same as the one in the input command file adding *Verb* (e.g. for *warning*, you use *warningVerb*).

Creating your own verbosity manager

You can create your own verbosity manager for the code you use taking as example one of the GAMOS verbosity managers (for example the class *GmGenerVerbosityMgr* in the package *GamosCore/GamosGenerator*).

First create a class inheriting from *GmVerbosityMgr* and fill it as follows:

- In the include file (i.e. the one with suffix *.hh*) of this class define an object of type *GmVerbosity* as extern

```
extern GmVerbosity MyVerb;
```

- In the method void *SetFilterLevel(int fl)* call the same method of your *GmVerbosity* object

```
MyVerb.SetFilterLevel( fl );
```

- In the method void *GetFilterLevel(int fl)* call the same method of your *GmVerbosity* object

```
MyVerb.GetFilterLevel( fl );
```

Finally you have to transform your class into a plug-in:

```
DEFINE_GAMOS_VERBOSITY(MyVerbosityMgr);
```

Controlling the Geant4 verbosity by event and track

If you want to print the detailed step information provided by Geant4 for a given interval of events or tracks, but you do not want that it is printed for all, you can use the user action

```
/gamos/userAction GmTrackingVerboseUA
```

You have to define the minimum and maximum events for which you want the verbosity on, and you can also set it ON only each N events:

```
/gamos/setParam GmTrackingVerboseUA:EventMin VALUE
```

```
/gamos/setParam GmTrackingVerboseUA:EventMax VALUE
```

```
/gamos/setParam GmTrackingVerboseUA:EventStep VALUE
```

If these parameters are not set, the verbosity will be ON for all events.

You can also define for which tracks interval in the selected events the verbosity will be ON, and set it ON only each N tracks:

/gamos/setParam GmTrackingVerboseUA:TrackMin VALUE

/gamos/setParam GmTrackingVerboseUA:TrackMax VALUE

/gamos/setParam GmTrackingVerboseUA:TrackStep VALUE

If these parameters are not set, the verbosity will be ON for all tracks.

Finally you may select the Geant4 verbosity level (by default 1) with the parameter

/gamos/setParam GmTrackingVerboseUA:VerboseLevel VALUE

Dumping the standard output and error in log files

By default the standard output (what is printed by G4cout or std::cout) is saved in a file called *gamos.log*, while the standard error (what is printed by G4cerr or std::cerr) is saved in a file called *gamos_error.log*.

The user may change the name of the output file with the command:

/gamos/log/setCoutFile FILE_NAME

and the name of the error file can be changed with the command:

/gamos/log/setCerrFile FILE_NAME

To avoid filling any log file the following user command must be used:

/gamos/log/writeFiles FALSE

DETECTOR APPLICATIONS

There are three detector applications in GAMOS, two related to Nuclear Medicine, i.e. *PET*, *SPECT* and another one that, while also used in Nuclear Medicine, it has also an extensive use in other fields, *Compton camera*. We describe in this chapter the utilities that are common to all detector applications, while we leave those that are specific to each of the in the corresponding chapter.

Identifying Compton interactions

It is frequent that a gamma that enters a detector suffers one or more Compton interactions before the photoelectric one, and then it may leave several hits in different detector elements. GAMOS offers several algorithms to identify which of the hits corresponds to the first gamma interaction, so that the line to the origin of the positron annihilation can be properly reconstructed. This is specially useful for detectors that have 3D identification capabilities, like solid state or liquid xenon detectors. For those without these capabilities the only algorithm of choice is likely the one that is based on the energy of the hits.

The following algorithms are currently available

- *Det1stHitByEnergy*: Classifies the reconstructed hits in order of decreasing energy and selects the first one as the one corresponding to the 1st Compton interaction. A parameter can be used to select instead the second, third, ..., highest energy hit: */gamos/setParam Det1stHitByEnergy:Order ORDER*
- *Det1stHitByXYPos*: If the detector has cylindrical symmetry and the gamma travels preferentially increasing the position in the XY plane (i.e. the cylinder radius) you may use this algorithm, which classifies the reconstructed hits in order of increasing position in the XY plane (i.e. $\sqrt{x^2+y^2}$) and selects the first one as the one corresponding to the 1st Compton interaction. A parameter can be used to select instead the second, third, ..., highest energy hit: */gamos/setParam Det1stHitByXYPos:Order ORDER*
- *Det1stHitByXPos*: If the detector is a block and the gamma travels preferentially increasing the position along the positive or negative X axis you may use this algorithm, which classifies the reconstructed hits in order of increasing absolute value of X position and selects the first one as the one corresponding to the 1st Compton interaction. A parameter can be used to select instead the second, third, ..., highest energy hit: */gamos/setParam Det1stHitByXPos:Order ORDER*
- *Det1stHitByXYZPos*: Even if the detector has some symmetry, it may happen that the best variable is the 3D position (i.e. the spherical radius). You may then use this algorithm, which classifies the reconstructed hits in order of increasing 3D position (i.e. $\sqrt{x^2+y^2+z^2}$) and selects the first one as the one corresponding to the 1st Compton interaction. A parameter can be used to select instead the second, third, ..., highest energy hit: */gamos/setParam Det1stHitByXYZPos:Order ORDER*
- *Det1stHitByDistanceToOther*: This algorithm is a generalization of the two above: it calculate the minimum distance from each reconstructed hit to any of the reconstructed hit that belongs to the other sets (a PET event has two sets, corresponding to the two gammas from positron annihilation) and classifies the reconstructed hits in order of increasing value of this minimum distance. It then the first one as the one corresponding to

the 1st Compton interaction. A parameter can be used to select instead the second, third, ..., highest energy hit: `/gamos/setParam Det1stHitByDistanceToOther:Order ORDER` This algorithm cannot be used for SPECT detectors

- *Det1stHitByComptonCone*: The idea of this algorithm is to profit from the fixed relationship between the initial and final energy and the deviation angle in Compton interactions. It selects a pair of reconstructed hits and between the two chooses one as corresponding to the first interaction. Assuming that the gamma suffered the first interaction with energy equal to the electron mass; for SPECT detectors you may change this value with the parameter: `/gamos/setParam DetRecHitCone:InitialHitEnergy ENERGY` It then calculates the deviation angle using the initial energy and final energy (= initial - hit energy). Using the line joining the two reconstructed hits and this angle it can build the cone of possible directions of the gamma before the first interaction. If the hit selected as first is indeed the one corresponding to the first gamma interaction, this cone points ideally to the origin of the gamma; as the origin of the gamma cannot be determined in a real detector, it is assumed that there is another gamma from the positron annihilation with the same direction and opposite sense and that it has left a hit in the opposite side of the detector before any other interaction. As it cannot be known which of the hits in the other side of the detector corresponds to the first interaction, the algorithm computes the distance from the cone surface to each of these reconstructed hits (only those that do not belong to the hit set being analysed are considered); the hit with smallest distance is selected, and this minimal distance is stored. The whole algorithm is repeated selecting as first the other hit of the pair of reconstructed hits, and then making other pairs of hits with the rest of hits in the set. By looking at all the minimal distances stored, it selects as good combination the one with the smallest distance. For PET detectors when the algorithm is applied to the second set of hits, the position of the first set is used, instead of looking at the position of all hits in the other side of the detector.

These algorithms act only on those event that have been accepted by the PET/SPECT/ComptonCamera classifier. To use them the first step is to merge the reconstructed hits that are supposed to correspond to the interaction of the same gamma into a single one. This is done by setting the distance between hits to be merged with the parameter (substitute PET by SPECT or ComptonCamera):

```
/gamos/setParam PET:EvtClass:ComptonRecHitDist DISTANCE
```

which by default takes a value of 0. By default the position of the set of hits is the one of the hit with highest energy (i.e. the *Det1stHitByEnergy* algorithm is used). Other algorithms may be selected instead with the parameter:

```
/gamos/setParam PET:EvtClass:1stHitAlgorithm ALGORITHM
```

where *ALGORITHM* is one of those enumerated above.

For the case of PET and Compton camera events it is possible to use a different algorithm for the first and second set of hits, what may be specially useful if the Compton cone algorithm is used, but also in other cases. To do it two different parameters should be set:

```
/gamos/setParam PET:EvtClass:1stHitAlgorithmFirst ALGORITHM
```

```
/gamos/setParam PET:EvtClass:1stHitAlgorithmSecond ALGORITHM
```

Compton studies histograms

Several histograms can be filled to help you in determining which is the most efficient algorithm to identify the hit corresponding to the first gamma interaction. To produce them you have to set the parameter (substitute PET by SPECT or CC):

```
/gamos/setParam PET:EvtClass:ComptonStudyHistos 1
```

which by default takes a value of 0.

The histograms count the proportion of times that the algorithm used selected correctly the hit closest to the first gamma interaction and how are the hits are from the gamma interactions (what can serve you to have an estimate of the precision you may reach).

The logic of this class is the following: the interactions of the original gammas are stored (we understand by an original gamma the one that is created as primary particle, comes from the annihilation of a positron that is a primary particle or, if a radioactive ion was the primary particle, the gamma originated by the ion decay or by the annihilation of the positron that was created by the ion decay). Each gamma interaction is associated to the closest reconstructed hit. If the first interaction identification algorithm chooses as first hit the one that is associated to the first gamma interaction, it is considered that the choice is correct.

The following histograms are filled:

- *N gamma interactions*: Number of original gamma interactions in one event
- *N rec hits*: Number of reconstructed hits in one event
- *N gamma interactions - N rec hits*: Number of original gamma interactions minus number of reconstructed hits in one event

Histograms of data about the interactions and the reconstructed hits

Another set of histograms may help to evaluate if the energy is a good criteria to identify the reconstructed hit that corresponds to the first gamma interaction. The energy lost at each first interaction of an “original gamma” is plotted, and also the energy of the second one, the third one, To avoid reserving space for an unlimited number of interactions, the third and later interactions are grouped in a unique histogram. You may change the interaction number to start this grouping with the parameter:

`/gamos/setParam DetCAIgoEnergy:HistoGroupingNumber NUMBER`

Also histograms of the difference between the first interaction and each of the other ones, the second and each of the other ones, etc. are done. And two-dimensional histograms of the energy of the first vs the rest, the second vs the rest, etc. The name of the histograms are (assuming the grouping number is three):

- *DetCompton:Algo:Interaction: Energy* Energy of gamma interactions
- *DetCompton:Algo:Interaction: Energy: 1st* Energy of first gamma interaction
- *DetCompton:Algo:Interaction: Energy: 2nd* Energy of second gamma interaction
- *DetCompton:Algo:Interaction: Energy: 3rd+* Energy of third and higher gamma interactions
- *DetCompton:Algo:Interaction: Energy: 1st - others* Energy of first gamma interaction minus energy of other interaction
- *DetCompton:Algo:Interaction: Energy: 2nd - others* Energy of second gamma interaction minus energy of other interaction
- *DetCompton:Algo:Interaction: Energy: 3rd+ - others* Energy of third and higher gamma interactions minus energy of other interaction
- *DetCompton:Algo:Interaction: Energy: 1st - others* Energy of first gamma interaction vs energy of other interaction
- *DetCompton:Algo:Interaction: Energy: 2nd - others* Energy of second gamma interaction vs energy of other interaction
- *DetCompton:Algo:Interaction: Energy: 3rd+ - others* Energy of third and higher gamma interactions vs energy of other interaction

By default all these histograms have 100 bins between 0. and 1.

The same set of histograms can be done for the position instead of energy. Several position variables can be selected: X, Y, Z, XY ($\sqrt{X^2+Y^2}$), XZ, YZ, XYZ.

To fill these histograms the following parameter should be used:

`/gamos/setParam DetComptonStudyHistosUA:AlgorithmVariables VAR1 VAR2 ... VARN`

where the variables can be *Energy*, *XPos*, *YPos*, *ZPos*, *XYPos*, *XZPos*, *YZPos* or *XYZPos*.

At the same time that the histograms about gamma interaction data, they are filled also histograms about the reconstructed hits that are associated to each interaction are filled (so the first hit is the one closest to the first interaction, etc.) The only difference is that instead of the word *Interaction* in their name, they have the word *RecHit*.

The classes that manage this histograms are plug-ins, so that other variables could easily be created.

Classification of good and bad identification histograms as a function of variable

It may happen that different algorithms to identify the first gamma interaction may be optimal for different types of reconstructed hit set. For example for sets with only two hits the Compton cone algorithm may be optimal, while for sets with more hits it may give worse results; or the minimal position algorithm may be optimal when hits have small energy, but if there is one with high energy (and therefore high deviation angle) it may give worse results. To help you in doing this optimization several histograms about the number of good and bad first gamma interaction identification can be done for different variables, e.g. for different number of reconstructed hits in the set, for different maximum hit energy bins, for different position variables. The histograms are about the distance between the reconstructed hit sets (using the position of the hit identified as corresponding to the first interaction) and the interactions. For example, for the “number of reconstructed hits” variable the following histograms are filled:

- *DetCompton:Classif: NRecHit: INTERVAL_LOWER_EDGE-INTERVAL_UPPER_EDGE: Dist RecHitSet - Associated Interaction* Distance between a reconstructed hit set and the interaction to which it is associated
- *DetCompton:Classif: NRecHit: INTERVAL_LOWER_EDGE-INTERVAL_UPPER_EDGE: Dist RecHitSet - 1st Interaction* Distance between a reconstructed hit set and the first interaction
- *DetCompton:Classif: NRecHit: INTERVAL_LOWER_EDGE-INTERVAL_UPPER_EDGE: Dist RecHitSet - 1st Interaction, Wrong assoc.* Distance between a reconstructed hit set and the first interaction, only for those sets with wrong association (other hit is closest to the first interaction than the one identified by the algorithm)
- *DetCompton:Classif: NRecHit: INTERVAL_LOWER_EDGE-INTERVAL_UPPER_EDGE: Dist RecHitSet - 1st Interaction, Good assoc.* Distance between a reconstructed hit set and the first interaction, only for those sets with good association

These four histograms are filled for each interval of the variable, being *INTERVAL_LOWER_EDGE* the lower edge of each interval and *INTERVAL_UPPER_EDGE* the upper edge of each interval. The intervals are set by the parameters:

`/gamos/setParam DetCClassifEnergy:Min MIN_VALUE`

`/gamos/setParam DetCClassifEnergy:Max MAX_VALUE`

`/gamos/setParam DetCClassifEnergy:Step STEP_VALUE`

The intervals will be built between *MIN_VALUE* and *MAX_VALUE* each *STEP_VALUE*. Alternatively you can define the N+1 limits of the N intervals with the parameter:

`/gamos/setParam DetCClassifEnergy:IntervalLimits VALUE_1 VALUE_2 ... VALUE_N+1`

The same set of histograms can be done for the maximum, minimum or average reconstructed hit energy, and minimum position *X*, *Y*, *Z*, *XY* ($\sqrt{X^2+Y^2}$), *XZ*, *YZ*, *XYZ*.

To fill these histograms the following parameter should be used:

`/gamos/setParam DetComptonStudyHistosUA:ClassificationVariables VAR1 VAR2 ... VARN`

where the variables can be *EnergyMin*, *EnergyMax*, *EnergyAverage*, *XPosMin*, *YPosMin*, *ZPosMin*, *XYPosMin*, *XZPosMin*, *YZPosMin* or *XYZPosMin*.

Another variable, but in this case it is always filled is the variable *ALL*, which indeed is not a variable and it makes no classification, but fill all reconstructed hit in a unique set of histograms.

The classes that manage this histograms are plug-ins, so that other variables could easily be created.

Histograms of gammas at sensitive detectors

We describe here the *GmHistosGammaAtSD*, an utility that prints information about the interaction of ‘original’ gammas in the sensitive detector. It is a user action and therefore it can be activated with the command

```
/gamos/userAction GmHistosGammaAtSD
```

At the end of run a table like the following one is printed:

```

$$$$$$$$$ Classification of Gamma Interactions in SD $$$$$$$$
$$GC: nEvents      : 1000000
$$GC: n gamma in SD : 516170 : 25.8085 %
$$GC:  n PE       : 321588 : 62.30273 %
$$GC:   PE 0 COMP : 157614 : 49.011157 %
$$GC:   PE 1 COMP : 111304 : 34.610744 %
$$GC:   PE >1 COMP : 52670  : 16.378099 %
$$GC:  n COMP     : 222590 : 43.12339 %
$$GC:   1 COMP    : 159193 : 71.518487 %
$$GC:   >1 COMP   : 63397  : 28.481513 %
$$GC: nGamma_COMP/event : 0.585931

```

- *nEvents* : total number of events.
- *n gamma in SD* : number of ‘original’ gammas reaching one sensitive detector.
- *n PE* : number of ‘original’ gammas with photoelectric interaction in SD
- *PE 0 COMP* : number of ‘original’ gammas with photoelectric interaction and no Compton interactions in SD (percentage relates to ‘n PE’).
- *PE 1 COMP* : number of ‘original’ gammas with photoelectric interaction and one Compton interaction in SD (percentage relates to ‘n PE’).
- *PE >1 COMP* : number of ‘original’ gammas with photoelectric interaction and more than one Compton interaction in SD (percentage relates to ‘n PE’).
- *n COMP* : number of ‘original’ gammas with Compton interactions in SD.
- *1 COMP* : number of ‘original’ gammas with only one Compton interaction in SD (percentage relates to ‘n COMP’).
- *>1 COMP* : number of ‘original’ gammas with more than one Compton interactions in SD (percentage relates to ‘n COMP’).
- *nCOMP/event* : number of Compton interactions in SD per ‘original’ gamma.

This class also makes several histograms, all of them have the words *Gamma At SD*: included in their names. And all are written to the file `gammaSD.root/csv`. The following histograms are defined:

- Event Type / 100, i.e. no Rayleigh counting (“Event Type”)
- Number of photoelectric interactions per ‘original’ gamma (“N PhotoElec”)
- Number of Compton interactions per ‘original’ gamma (“N Compton”)
- Number of Rayleigh interactions per ‘original’ gamma (“N Rayleigh”)
- Number of photoelectric interactions vs Number of Compton+Rayleigh interactions per ‘original’ gamma (“N PhotoElec vs Compton+Rayleigh”)
- Energy of gamma upon entering SD (“Energy at entering SD (keV)”)

- Energy lost in the photoelectric interactions (“Energy lost PhotoElec (keV)”);
- Difference in position from the gamma entering SD to the point where a photoelectric interaction happened (“Diff pos when PhotoElec (mm)”)
- Difference in direction from the gamma entering SD to the point where a photoelectric interaction happened (“Diff dir when PhotoElec (mm)”)
- Difference in kinetic energy from the gamma entering SD to the point where a photoelectric interaction happened (“Diff energy when PhotoElec (mm)”)
- Energy lost in the Compton interactions (“Energy lost Compton (keV)”);
- Angle variation in the Compton interactions (“Angle variation Compton (mrad)”)
- Energy lost in the Rayleigh interactions (must be 0.)(“Energy lost Rayleigh (eV)”)
- Angle variation in the Rayleigh interactions (“Angle variation Rayleigh (mrad)”)

All these histograms are repeated for the different types of events (a prefix in the name marks the event type the histogram refers to):

- “ALL: ”: every event
- “No PE: ” events where no photoelectric interaction occurred
- “Only PE: ” events where only photoelectric interaction occurred
- “PE + 1 Compt: ” events where one photoelectric interaction plus only one Compton interaction occurred
- “PE + >1 Compt: ” events where one photoelectric interaction plus more than one Compton interaction occurred

Automatic determination of production cuts for a detector

The method used in GAMOS to determine the best production cuts is based on what we can call an ‘inverse reasoning’. We count each particle that reaches the sensitive detector and we calculate first the range of the particle in the region where it is created. Then we can know that if we put a range cut in that region smaller than the calculated range, that particle would not reach our target plane. We also compute the range of the mother particle in the region where it was created and the same consecutively for all the ancestors. We know then that if we set in any of the regions where each of the ancestor particles are created a cut smaller than the corresponding range, we would stop the chain of particles and therefore we would have no particle in the target plane. After running a good number of tracks we can know for each particle type and for each region which is the biggest range we can put if we do not want to lose any particle. Indeed we may allow to lose a small amount of particles if this speeds up our simulation. To know easily which is the biggest cut you can use to lose less than a given percentage of particles, GAMOS provides a set of plots (one per each particle type and per each region) and a simple script to get automatically the cut values.

One warning is due here: as mentioned above when a track reaches the target, its range fills a histogram, but also the range of all the ancestors of this track. It may happen then that when you set a certain cut and the abovementioned script gives you how many tracks would be killed, more than one killed track correspond to the same track reaching the target (i.e., with a cut you kill the track that reaches the target and the parent track). Therefore you might have an overcounting of the number of tracks killed by a cut. To avoid this the total number of tracks (the last lines of output) is not computed as the sum of tracks in the region. This number uses a histogram that contains only one entry per track reaching the target, the one corresponding to the track with the smallest range. If you want to set a different cut for each region and are worried for this double counting, you may have a look at the histogram named “trackInfos per Track in target”, that plots per each track reaching the target how many track informations are kept in the histograms. Another useful histogram for this case may be the 2D histogram “trackInfo Region vs trackInfo Region”, that plots all the region number of all the pairs of track informations that correspond to the same track reaching the target (you can get a list of which region number corresponds to which region at the end of the standard output file).

To use this utility in GAMOS it is only needed to add this command in your script:


```
/gamos/userAction GmProdCutsStudyUA DetCutsStudyFilter
```

that will use as target condition that a track enters a sensitive detector

This command will produce at the end of run a table and a histogram file with the needed information. The table will contain the minimum range that can be applied for each region/particle/process not to lose any track reaching the target, and it will look like this

```
%%%% PRODUCTION CUTS STUDY RESULTS
GmProdCutsStudyUA: REGION= DefaultRegionForTheWorld PARTICLE=
gamma PROCESS= ALL MIN RANGE= 353161.38
GmProdCutsStudyUA: REGION= DefaultRegionForTheWorld PARTICLE=
gamma PROCESS= eBrem MIN RANGE= 353161.38
```

To get the cuts values for not losing a given percentage of particles in the target plane you can execute the ROOT script that can be found at GamosCore/GamosPhysics/Cuts/getProdCutsEffect.C :

```
*root -b -p -q .x getProdCutsEffect.C++\\(\\ "prodcuts.root\\",percentage\\) *
```

and look at the last lines of output, those that contain the word 'FINAL', like the following ones

```
PARTICLE= e+ FINAL= 17 / 19
PARTICLE= e- FINAL= 72 / 34185
PARTICLE= gamma FINAL= 72 / 34184
```

Automatic determination of user limits for a detector

The method used in GAMOS to determine the minimum range user limits is similar to the one used to determine the best production cuts. The main difference is that when a track reaches the target we do not have to look at the range it had when created, but at the range it had in every step. This is because even if we want the minimum step, the track may have crossed several regions and the smallest range may not correspond to the last step. What we do nevertheless is to consider only the last step when there are a set of contiguous steps in the same region. Also for the ancestor tracks we have to store the information of each step, starting of course with the one when the track that reached the target (or its n-th ancestor if we are looking at the (n+1)-th ancestor) was created.

The same warning as for the production cuts should be mentioned here, but in this case it is more than a mere warning: when a track reaches the target, we accumulate one-track information of the last step in each region, for each of the ancestor tracks. Therefore it is very likely that there are more than one track information per track reaching the target, and therefore there will be overcounting of the number of tracks killed by a cut. As for the production cuts you should keep an eye on this.

To use this utility in GAMOS it is only needed to add the command in your script:

```
/gamos/userAction GmMinRangeLimitsStudyUA DetCutsStudyFilter
```

what will produce at the end of run a table and a histogram file with the needed information.

```
root -b -p -q .x getMinRangeCutsEffect.C++\\(\\ "prodcuts.root\\",percentage\\) \\|& tee out
```

and look at the last lines of output, those that contain the word 'FINAL', like the following ones

```
PARTICLE= e+ FINAL= 17 / 19
PARTICLE= e- FINAL= 72 / 34185
PARTICLE= gamma FINAL= 72 / 34184
```


PET APPLICATION

Before reading this chapter we recommend you to read the chapter on Detector, that contains the utilities that are common to all detector applications.

The PET application contains two directories. The first one, PetGeometry, contains an utility to build a simple PET ring detector by just defining a few parameters. The second one contains the PET event classifier, the Compton identification algorithms and a few histogram classes.

Many of the utilities for PET detectors are related to the sensitive detectors that they contain, so please read the Sensitive Detectors chapter if you have not done it yet.

PET geometry

The directory `source/Detector/PET/GeometryData` contains several examples of building simple PET ring detectors with text geometry files by just defining the following parameters:

- `:P NCRYSTALS_transaxial VALUE` : Number of crystals per block in the transaxial direction
- `:P NCRYSTALS_axial VALUE` : Number of crystals per block in the axial direction
- `:P NBLOCKS VALUE` : Number of blocks of crystals per ring
- `:P NRINGS VALUE` : Number of rings of blocks
- `:P CRYSTAL_size_transaxial VALUE` : Crystal size, trans-axial
- `:P CRYSTAL_size_axial VALUE` : Crystal size, axial
- `:P CRYSTAL_size_radial VALUE` : Crystal size, radial
- `:P DIAMETER VALUE` : Detector ring diameter
- `:P CRYSTAL_MATE VALUE` : Name of crystal material
- `:P WORLD_Z VALUE` : World Z dimension (the X and Y are calculated as slightly bigger than the detector)

There are several examples of simplified commercial PET detectors in the files with suffix `.geom` in that directory. To use this utility you just have to choose as your geometry the `GmGeometryFromText` one:

```
/gamos/geometry GmGeometryFromText
```

NOTE: This module is just thought for simple PET geometries. If you want to do more complicated geometries, we recommend you to describe them with a text file (see section *Building your geometry with a text file*).

PET event classification

The class `PETEventClassifierUA` in the directory `NuclearMedicine/PET` classifies the events as PET by looking at the reconstructed hits. It is a GAMOS user action, so you can activate it with the command

```
/gamos/userAction PETEventClassifierUA
```

First it counts how many reconstructed hits have 511 keV

Indeed the exact mass of the electron is always used within a precision given by the parameter

```
/gamos/setParam PET:EvtClass:511EPrec ENERGY_FRACTION
```

That is the energies accepted will be those between $511 \cdot (1 - \text{ENERGY_FRACTION})$ and $511 \cdot (1 + \text{ENERGY_FRACTION})$. Alternatively you may use the following two parameters

```
/gamos/setParam PET:EvtClass:511EPrecMin ENERGY_MIN
```

```
/gamos/setParam PET:EvtClass:511EPrecMax ENERGY_MAX
```

where `ENERGY_MIN` is the minimum energy, that by default takes a value of $0.7 \cdot 511$ keV, and `ENERGY_MAX` is the maximum energy, that by default takes a value of $1.3 \cdot 511$ keV.

To recover hits when one of several Compton interactions have occurred you may switch the merging of hits that are close into a single one. You may set the distance to merge hits with the parameter

```
/gamos/setParam PET:EvtClass:ComptonRecHitDist DIST
```

`DIST` takes by default a value of 0, what means that no Compton hits merging will be done. The merging of hits first identifies the hit with biggest energy and merges with it those hits that are closer than the given distance. The same algorithm is repeated with the hits not associated until no hit is left. If hits merging is chosen you may choose among several algorithms to try to identify the hit that corresponds to the first gamma interaction (and this will be the position of the whole set). See section *Identifying Compton interactions* to learn how to do it. If no algorithm is selected the position will be the one of biggest energy, or second biggest, or n-th biggest, where the order is given by the parameter

```
/gamos/setParam PET:1stHitByEnergy:Order ORDER
```

`ORDER` takes by default a value of 1, that is, the position is that of the hit with biggest energy (see section on *Identifying Compton interactions*).

If two hits are found it will be checked that their relative time difference is less than the value given by the parameter

```
/gamos/setParam PET:EvtClass:CoincidenceTime COINCIDENCE_TIME
```

which by default takes a value almost infinite (it is assumed that one of the two started the trigger and the other must be in the coincidence time opened at that moment).

If two 511-keV hits are finally found, the event is classified as a good PET event. Then the sub-classification code enters in the game:

- *More than two 511-keV hits:* If more than two hits are found, the two that are closer to 511 keV will be taken and the event will receive a subclassification type of 3->2.
- *%If the distance to the event vertex is too big, we can check if still the distance to the phantom is too big (TO BE DONE)*
- *Random coincidence:* It is checked that each of the two 511-keV hits is built only from tracks from the same 'original' gamma 'original' gammas are gammas that are primary particles or that are directly created by the annihilation of positron that is a primary particle, and that the two hits come from the same event
- *Scattered:* The event is classified as scattered if any of the 511-keV gammas has suffered an interaction in the list of volumes defined by the parameter `/gamos/setParam DetCountScatteringUA:VolumeNames VOLUME_1 VOLUME_2 ...` and this interaction is of one of the process types defined by the parameter `/gamos/setParam DetCountScatteringUA:ProcessNames PROCESS_1 PROCESS_2 ...` (by default Compton are Rayleigh interactions)

are counted); and it has lost in the volumes more energy than the parameter `/gamos/setParam DetCountScatteringUA:EnergyMin ENER_MIN` (by default the minimal energy is 0.)

- *Check PET line distance:* a line joining the position of the two reconstructed hits is built and the distance of closest approach (DCA) to the origin of the positron is calculated; the events are classified as 'near' of 'far' if the DCA is smaller or bigger than the parameter `/gamos/setParam PET:EvtClass:LineDistToVtx DISTANCE` where *DISTANCE* has a default value of 10*mm.

The ClassifyPET method returns an integer with several digits containing the event classification:

- 0 if it is not PET, 1 if it is PET and PET line is close to the event vertex, 2 if it is PET and PET line is far from event vertex
- 10*1 if the search for 511-keV reconstructed hits found more than 2
- 100*1 if the event is a random coincidence event
- 1000*1 if the event is a scattered event

At the end of the run a table is printed with the number of events in each of the combinations of the sub-classification types.

PET histograms: event classification

These histograms are related to the event classification explained above. They are produced if the event classification user action is selected. The name of all these histograms starts with `"PETEvtClass: "` and all are written in the file `pet.root/csv`.

The following histograms are written:

- Classification index of event ("PET classification"). This index is the one described in the precedent section.
- % For a more compact presentation of the results, those events with an index digit 1,2 are summed up in the bin with index digit 5 (i.e. 0,1 and 2 in bin 5, 10, 11 and 12 in bin 15, 1, 11 and 21 in bin 51, etc.)
- Number of 511-keV reconstructed hits before cleaning if there are more than two and searching for Compton hits, i.e., hits produced merging two crystals (see above) ("N 511 recHits initial")
- The energy of the 511-keV reconstructed hits ("PET RecHit energy (keV)")
- The energy of the 511-keV reconstructed hits that are rejected because there are more than two ("PETEvtClass: Extra PET RecHit energy (keV)")
- Distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits (=DCA) ("PET dist line - vertex (mm)")
- Z coordinate if distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits (=DCA) ("PET dist line - vertex Z (mm)")
- Phi coordinate if distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits (=DCA) ("PET dist line - vertex PHI (mm)")
- Radial position in the XY plane of the primary positrons ("e+ position (X,Y) (mm)")
- Radial position in the XY plane of the primary positrons, only for those events that are good PET events ("e+ position (X,Y)_PET Events (mm)")

There are also a histogram of the DCA and the reconstructed hit energies for each of the sixteen subclassification types (combinations of the 4 subindices) and also for the all the events that are far from vertex, all the events where there were more than two hits, all the events with random coincidences and all the scattered events.

PET output for reconstruction

List-mode binary file

If the event is classified as a PET one, it is dumped in a binary file given by the name

```
/gamos/setParam pet:FileName MY_FILENAME
```

that takes the name *pet.out* by default. The variables written are given by the structure

```
struct PetOutput
{
char name[8];
float xVtx, yVtx, zVtx, x1, y1, z1, x2, y2, z2;
}
```

where *name* is *PET*, *xVtx*, *yVtx*, *zVtx* are the coordinates of the event vertex, *x1*, *y1*, *z1* are the coordinates of the first reconstructed hit, *x2*, *y2*, *z2* are the coordinates of the second vertex.

The same data that is written to the file can be written in the standard output if the parameter

```
/gamos/setParam PET:EvtClass:DumpEventListMode TRUE
```

is set to true. The positions in the standard output (not in the file) will be written in cylindrical coordinates by default. If you want them in cartesian coordinates you should set to true the parameter

```
/gamos/setParam pet:DumpCartesian TRUE
```

Projection data file

PET events can also be stored as projection data (sinograms) by selecting the following parameter

```
/gamos/setParam PET:EvtClass:DumpProjData 1
```

Projection data or sinogram is a common way to organize PET events and represents the standard input for image reconstruction algorithms. By default, GAMOS generates a file compatible with its own image reconstructor (see “Image reconstruction utilities” section), data are written in pairs, including a header text file (.hv) and a binary data file (.v). Nevertheless, the user can also select an output file compatible with the STIR package [[biblio.ImageJ](#)], a Open Source software for use in tomographic imaging. In this case, data files are also written in pairs: a header text file (.hs) and a binary data file (.s); but following the STIR Interfile format. The type of output format (0 for GAMOS ad-hoc or 1 for STIR Interfile) is controlled by the “OutFormat” parameter:

```
/gamos/setParam PET:ProjData:OutFormat OUT_TYPE
```

In order to generate the sinograms, the user must specify the dimensions of image axial and transaxial field of view (FOV). These values are given in millimetres by

```
/gamos/setParam PET:ProjData:DistAxial D_AXIAL
```

```
/gamos/setParam PET:ProjData:DiameterTranFOV D_TRANSAXIAL
```

In addition, projection data are discretized in axial, tangential and angular dimensions. The user must indicate the number of axial planes (N_PLANES, by default 40), tangential bins (N_BINS, by default 81) and angular views (N_ANGULAR, by default 80) using these parameters:

```
/gamos/setParam PET:ProjData:Nplanes N_PLANES
```

```
/gamos/setParam PET:ProjData:Nbin N_BINS
```

```
/gamos/setParam PET:ProjData:NangViews N_ANGULAR
```

Note that once axial FOV is fixed, the plane width is given by $D_AXIAL/(N_PLANES-1)$; which represents the double of the slice thickness in the reconstructed image, by default. In the same way, the image transaxial pixel size will have a value of $D_TRANSAXIAL/(N_BINS-1)$.

In 3-dimensional PET acquisition the “maximum ring difference” is another fundamental parameter which limits the axial distance between the two planes (rings) of a detected PET coincidence. It is used to discard lines of response which are too axially tilted, since these events can deteriorate the image quality with some reconstruction methods. By default (value -1) all axial planes are included; the number of planes which defines the “maximum ring difference” can be changed with

```
/gamos/setParam PET:ProjData:MaxRingDiff MY_MRD
```

The projection data file name (without extension) is given by

```
/gamos/setParam PET:ProjData:Filename MY_PROJDATA_FILE
```

that takes the name *default_sino3D* by default.

PET histograms: positrons

These histograms are related to the original positron and the two secondary gammas created at its annihilation. They are produced if the user action *PETHistosPositron* is activated with the command

```
/gamos/userAction PETHistosPositron
```

The name of all these histograms starts with “*PETPositron*” and all are written in the file *pet.root/csv*.

The histograms are the following:

- Positron energy at creation (“e+ initial energy (keV)”)
- Positron range (“e+ range (mm)”)
- Positron energy at annihilation (“e+ energy at annihilation (keV)”)
- Positron energy at creation vs range (“e+ initial energy (keV) vs range (mm)”)
- Positron energy at annihilation vs secondary gammas energy (“e+e- gammas vs e+ energy (keV)”)
- Positron energy at annihilation vs sum of secondary gammas energy - 1.022 MeV - positron energy at annihilation (“total gamma energy vs e+ energy (keV)”). This is the energy deposited locally at annihilation.

Detector histograms: distance between two gammas

These histograms are related to the distance between the line joining two gammas originated at the positron annihilation and the vertex position, at the moment of the gamma creation and when they hit the sensitive detector. Also other histograms about these two gammas are provided for further information. They are produced if the user action *DetHistosGammaDist* is activated with the command

```
/gamos/userAction DetHistosGammaDist.
```

The name of all these histograms starts with “*NHGammaDist*: “ and all are written in the file *NHGammaDist.root/csv/txt*.

The histograms are the following:

- Angle between the direction of the two gammas when they are created (“angle between gammas at vertex (deg)”)
- Angle between the direction of the two gammas when they enter the sensitive detector (“angle between gammas at entering SD (deg)”)

- Distance of closest approach between vertex and the line joining the vertex points of the two gammas (“DCA orig from Gamma vertex (mm)”)
- Distance of closest approach between vertex and the line joining the points where the gammas enter the sensitive detectors (“DCA orig from Gamma entering SD (mm)”)
- Z position of vertex of gamma if it reaches the sensitive detector (“Orig Pos Z if Gamma reaches SD (mm)”)
- R position of vertex of gamma if it reaches the sensitive detector (“Orig Pos R if Gamma reaches SD (mm)”)

SPECT APPLICATION

Before reading this chapter we recommend you to read the chapter on Detector, that contains the utilities that are common to all detector applications.

The SPECT example is quite similar to the PET one; we will describe here the main characteristics that differentiate it and refer to the PET example for those characteristics they share.

Many of the utilities for SPECT detectors are related to the sensitive detectors that they contain, so please read the Sensitive Detectors chapter if you have not done it yet.

SPECT event classification

The class `SPECTEventClassifierUA` in the directory `SPECT` classifies the events as SPECT by looking at the reconstructed hits. It is a GAMOS user action, so you can activate it with the command

```
/gamos/userAction SPECTEventClassifierUA
```

The first thing you have to define is the original gamma energy, what you can do with the parameter:

```
/gamos/setParam SPECT:EvtClass:GammaEnergy ENERGY
```

Another thing you have to define is where is your collimator. You can define it by giving the collimator volume name:

```
/gamos/setParam SPECT:EvtClass:CollimatorVolume VOLUME_NAME
```

Then the user action counts how many reconstructed hits have this energy within a precision given by the two parameters

```
/gamos/setParam SPECT:EvtClass:EPrecMin ENERGY_MIN
```

```
/gamos/setParam SPECT:EvtClass:EPrecMax ENERGY_MAX
```

where `ENERGY_MIN` is the minimum energy, that by default takes a value of $0.7 * \text{GammaEnergy}$, and `ENERGY_MAX` is the maximum energy, that by default takes a value of $1.3 * \text{GammaEnergy}$.

To recover hits when one of several Compton interactions have occurred you may switch the merging of hits that are close into one. You may set the distance to merge hits with the parameter

```
/gamos/setParam SPECT:EvtClass:ComptonRecHitDist DIST
```

`DIST` takes by default a value of 0, what means that no Compton hits merging will be done. If you want hits merging you may select as position of the combined hits the one of the biggest energy, or the second biggest, or the n-th biggest, where the order is given by the parameter

```
/gamos/setParam SPECT:EvtClass:SelectPosOrder ORDER
```

`ORDER` takes by default a value of 1, that is, the position is that of the hit with biggest energy.

If one good hit is finally found, the event is classified as a good SPECT event. Then the sub-classification code enters in the game:

- *More than one hit:* If more than one hit is found, the one that is closer to the GammaEnergy will be taken and the event will receive a subclassification type of 2->1.
- *%If the distance to the event vertex is too big, we can check if still the distance to the phantom is too big (TO BE DONE)*
- *Random coincidence:* It is checked that the hit is built only from tracks from the same 'original' gamma 'original' gammas are gammas that are primary particles or that are directly created by the annihilation of positron that is a primary particle
- *Scattered:* The event is classified as scattered if any of the original gamma has suffered an interaction in the list of volumes defined by the parameter `/gamos/setParam DetCountScatteringUA:VolumeNames VOLUME_1 VOLUME_2 ...` and this interaction is of one of the process types defined by the parameter `/gamos/setParam DetCountScatteringUA:ProcessNames PROCESS_1 PROCESS_2 ...` (by default Compton are Rayleigh interactions are counted); and it has lost in the volumes more energy than the parameter `/gamos/setParam DetCountScatteringUA:EnergyMin ENER_MIN` (by default the minimal energy is 0.)
- *Check SPECT line distance:* a line joining the position of the hit with the centre of the collimator volume is built and the distance of closest approach (DCA) to the origin of the positron is calculated; the events are classified as 'near' or 'far' if the DCA is smaller or bigger than the parameter `/gamos/setParam SPECT:EvtClass:LineDistToVtx DISTANCE` where *DISTANCE* has a default value of 10*mm.
- *Gamma traversed collimator:* it checks if the original gamma (or one of them if there are several) that left the hits traversed the volume(s) defined as collimator with the parameter: `/gamos/setParam SPECT:EvtClass:CollimatorVolume VOLUME_1 VOLUME_2 ...` where *DISTANCE* has a default value of 10*mm.

The event will be rejected if the number of hits found is bigger than the value given by the parameter

`/gamos/setParam SPECT:EvtClass:RejectIfNRecHits VALUE`

The ClassifySPECT method returns an integer with several digits containing the event classification:

- 0 if it is not SPECT, 1 if it is SPECT and SPECT line is close to the event vertex, 2 if it is SPECT and SPECT line is far from event vertex
- 10*1 if the search for 511-keV reconstructed hits found more than 2
- 100*1 if the event is a random coincidence event
- 1000*1 if the event is a scattered event
- 10000*1 if there are several collimators and the collimator traversed by the original gamma is the wrong one, i.e. not the one closest to the hit
- 20000*1 if the original gamma did not traverse a collimator

At the end of the run a table is printed with the number of events in each of the combinations of the sub-classification types. You may

SPECT histograms: event classification

These histograms are related to the event classification explained above. They are produced if the event classification user action is selected. The name of all these histograms starts with "`SPECTEvtClass:`" and all are written in the file `spect.root/csv`.

The following histograms are written:

- Classification index of event (“SPECT classification”). This index is the one described in the precedent section.
- % For a more compact presentation of the results, those events with an index digit 1,2 are summed up in the bin with index digit 5 (i.e. 0,1 and 2 in bin 5, 10, 11 and 12 in bin 15, 1, 11 and 21 in bin 51, etc.)
- Number of 511-keV reconstructed hits before cleaning if there are more than two and searching for Compton hits, i.e., hits produced merging two crystals (see above) (“N 511 recHits initial”)
- The energy of the 511-keV reconstructed hits (“RecHit energy (keV)”)
- Distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits (=DCA) (“SPECT dist line - vertex (mm)”)
- Distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits in Z axis (=DCA) (“SPECT dist line - vertex Z (mm)”)
- Distance of closest approach between vertex and the line joining the two 511-keV reconstructed hits in R-phi plane (=DCA) (“SPECT dist line - vertex RPHI (mm)”)

There are also a histogram of the DCA and the reconstructed hit energies for each of the sixteen subclassification types (combinations of the 4 subindices) and also for the all the events that are far from vertex, all the events where there were more than two hits, all the events with random coincidences and all the scattered events.

SPECT output for reconstruction

If the event is classified as a SPECT one, it may be dumped in a binary file, if the following parameter is set to 1

```
/gamos/setParam SPECT:DumpEvent MY_FILENAME
```

The file name is given by the parameter

```
/gamos/setParam SPECT:FileName MY_FILENAME
```

that takes the name *spect.out* by default. The variables written are given by the structure

```
struct SPECTOutput
{
char name[8];
float xVtx, yVtx, zVtx, x1, y1, z1, x2, y2, z2, ener, class;
}
```

where *name* is *SPECT*, *xVtx*, *yVtx*, *zVtx* are the coordinates of the event vertex, *x1*, *y1*, *z1* are the coordinates of the reconstructed hit, *x2*, *y2*, *z2* are the coordinates of the collimator centre, *ener* is the hit energy and *class* is the SPECT classification.

The same data that is written to the file can be written in the standard output if the parameter

```
/gamos/setParam SPECT:DumpToCout TRUE
```

is set to true. The positions in the standard output (not in the file) will be written in cylindrical coordinates by default. If you want them in cartesian coordinates you should set to true the parameter

```
/gamos/setParam SPECT:DumpCartesian TRUE
```


COMPTON CAMERA APPLICATION

Before reading this chapter we recommend you to read the chapter on Detector, that contains the utilities that are common to all detector applications.

The Compton camera application is a new addition to the GAMOS framework. The application has been developed for researchers investigating the potential of Compton cameras in nuclear medicine but can be used for a wide variety of fields outside of medicine.

The Compton camera example has a structure very similar to the SPECT application. The example contains a utility to build a Compton camera which is composed of either rings of detectors, or a stack of parallel detectors, through the user defined geometrical parameters. Information relevant to Compton camera applications is accessed using the Compton camera event classifier.

Many of the utilities for Compton camera detectors are related to the sensitive detectors that they contain, so please read the Sensitive Detectors chapter if you have not done so yet.

Compton camera geometry

No standard geometry for Compton cameras exists, and as such, it is impossible to provide the user with a single standard geometry file to model all systems. A typical Compton camera is defined using scatterer detectors (where Compton scattering is the ideal interaction of incident gamma rays) and absorber detectors (where Photoelectric absorption is the ideal gamma ray interaction). It is necessary to define the sensitive detectors as either a scatterer or absorber using the parameter:

```
/gamos/SD/assocSD2LogVol GmSDSimple Scatterer DETECTOR_LOGICAL_NAME
```

```
/gamos/SD/assocSD2LogVol GmSDSimple Absorber DETECTOR_LOGICAL_NAME
```

The Compton camera directory contains a utility to build a Compton camera system in either (i) a dual ring configuration or as (ii) a stack of detectors. Although it is expected that these two examples will cover most users needs, it is still possible for the user to define their system via the GAMOS geometry text file input.

1. The ring system is composed of an inner ring of scatterer detectors and outer ring of absorber detectors. The rings can be replicated along the axial direction, so that a full body system can be simulated. The following parameters are used to define the RING system: - Scatterer Ring:
 - *NSCATRINGS*: Number of rings of blocks
 - *NSCATBLOCKS*: Number of blocks of crystals per ring
 - *NSCATCRYSTALS_transaxial*: Number of crystals in each block, transaxial
 - *NSCATCRYSTALS_axial*: Number of crystals in each block, axial
 - *SCATCRYST_transaxial*: Crystal size, transaxial
 - *SCATCRYST_axial*: Crystal size, axial

- *SCATDIAMETER*: Detector ring diameter
 - *SCATCRYS_MATE*: Scatterer detector material
 - Absorber Ring: - *NABSRINGS*: Number of rings of blocks - *NABSBLOCKS*: Number of blocks of crystals per ring - *NABSCRYSTALS_transaxial*: Number of crystals in each block, transaxial - *NABSCRYSTALS_axial*: Number of crystals in each block, axial - *ABSCRYNS_transaxial*: Crystal size, transaxial - *ABSCRYNS_axial*: Crystal size, axial - *ABSDIAMETER*: Detector ring diameter - *ABSCRYNS_MATE*: Absorber detector material
2. The stack system is composed of a number of scatterer and absorber detectors parallel to each other. The stack is produced through the axial (z) direction and is defined using the following parameters: - Scatterer layers:
- *NSCATLAYERS*: Number of scatterer detector layers
 - *NSCATCRYSTALS_X*: Number of crystals in each layer, X direction
 - *NSCATCRYSTALS_Y*: Number of crystals in each layer, Y direction
 - *NSCATCRYSTALS_Z*: Number of crystals in each layer, Z direction
 - *SCATCRYS_X*: Crystal size, X direction
 - *SCATCRYS_Y*: Crystal size, Y direction
 - *SCATCRYS_Z*: Crystal size, Z direction
 - *SCATDIST*: Distance in Z from centre of first scatterer detector to world origin
 - *SCATSEP*: Separation distance in Z between the centres of neighbouring scatterer detectors
 - *SCATCRYS_MATE*: Scatterer detector material
 - Absorber layers: - *NABSLAYERS*: Number of absorber detector layers - *NABSCRYSTALS_X*: Number of crystals in each layer, X direction - *NABSCRYSTALS_Y*: Number of crystals in each layer, Y direction - *NABSCRYSTALS_Z*: Number of crystals in each layer, Z direction - *ABSCRYNS_X*: Crystal size, X direction - *ABSCRYNS_Y*: Crystal size, Y direction - *ABSCRYNS_Z*: Crystal size, Z direction - *ABSDIST*: Distance in Z from centre of first absorber detector to world origin - *ABSSEP*: Separation distance in Z between the centres of neighbouring absorber detectors - *ABSCRYNS_MATE*: Absorber detector material

To use this utility the user must choose to read the geometry from a text file:

```
/gamos/geometry GmGeometryFromText
```

The user must also provide the *FILE_NAME* which contains the parameters used to define Compton camera geometry

```
/gamos/setParam GmGeometryFromText:FileName FILE_NAME
```

A few example files are included of simple Compton camera systems, with suffix *.geom* in the Compton camera tutorial, for example *CCGeometryRingEx1.geom*. Within this text file, the user must then choose whether to read the Ring or Stack geometry text file by typing

```
#include CCGeometryRing.geom
```

for a ring geometry or

```
#include CCGeometryStack.geom
```

or a stack geometry. Note that different user parameters are required to define a stack and ring geometry.

Compton camera Event Classification

The class *CCEventClassifierUA* classifies the events as useful events for Compton imaging by looking at the reconstructed hits. It is a GAMOS user action and can be activated with the command

/gamos/userAction CCEventClassifierUA

It is currently possible to output data for single/single type Compton imaging events, where there is one reconstructed scatterer detector hit and one reconstructed absorber detector hit.

The event classifier counts how many of these single/single reconstructed hits have deposited the total incident gamma-ray energy within a precision given by the parameter:

/gamos/setParam CC:EvtClass:PhotopeakEPrec GATE

The total gamma-ray energy is set with the parameter

/gamos/setParam CC:EvtClass:GammaEnergy ENERGY

which by default takes a value of 141 keV

where the minimum energy will be $(1-GATE)*\text{photopeak energy}$, that by default takes a value of $0.9*\text{photopeak energy}$ and the maximum energy will be $(1+*GATE)*\text{photopeak energy}$, that by default takes a value of $1.1*\text{photopeak energy}$. You can set independently the minimum and maximum interval values with

/gamos/setParam CC:EvtClass:PhotopeakEPrecMin VALUE

/gamos/setParam CC:EvtClass:PhotopeakEPrecMax VALUE

Only hits whose relative time difference is less than the value given by the parameter

/gamos/setParam CC:EvtClass:CoincidenceTime COINCIDENCE_TIME

will be taken into account to make a Compton imaging event, (it is assumed that one of the two started the trigger and the other must be in the coincidence time open at that moment).

To utilise events in which multiple interactions occur within a detector, it is necessary to identify the appropriate interactions to reconstruct. The most simple example is if a gamma ray Compton scatters in the scatterer detector and then in the absorber detector it Compton scatters and undergoes Photoelectric absorption. This event can be reconstructed if the 1st hit (ie the Compton scattering interaction) is correctly identified in the absorber detector. It is possible to recover these hits when one of several Compton interactions have occurred by examining those that are close. You may set the distance to examine multiple hits within the scatterer and absorber detectors, respectively with the parameters

/gamos/setParam CC:EvtClass:ComptonRecHitDistScat DIST

/gamos/setParam CC:EvtClass:ComptonRecHitDistAbs DIST

DIST takes by default a value of 0, what means that no identification of multiple Compton hits will be done. If *DIST* is > 0 you may select the position and energy of the 1st hits identified through the algorithms described in the Identifying Compton Interactions section of the PET chapter. By default, the algorithm used is the one which identifies the 1st hit as the one with biggest energy, however other algorithms may be selected with the parameter

gamos/setParam CC:EvtClass:1stHitAlgorithm ALGORITHM

You may select a different algorithm for the first and second hit (classified by time) with the parameters

gamos/setParam CC:EvtClass:1stHitAlgorithmFirst ALGORITHM

gamos/setParam CC:EvtClass:1stHitAlgorithmSecond ALGORITHM

If two reconstructed hits have a summed energy which is calculated to lie within the photopeak gates, the event is classified as a good Compton imaging event. Then sub classification is carried out:

- *More than two reconstructed hits:* These events are maximum when no identification of 1st hit multiple interactions is carried out.
- **Random coincidence:** It is checked that each of the two reconstructed hits is built only from tracks from the same 'original' gamma, and that the hits come from the same event.

- *Scattered*: The event is classified as scattered (outside the detector volumes) if the gamma has suffered an interaction in the list of volumes defined by the parameter `/gamos/setParam DetCountScatteringUA:VolumeNames VOLUME_1 VOLUME_2 ...` and this interaction is of one of the process types defined by the parameter `/gamos/setParam DetCountScatteringUA:ProcessNames PROCESS_1 PROCESS_2 ...` (by default Compton are Rayleigh interactions are counted); and it has lost in the volumes more energy than the parameter `/gamos/setParam DetCountScatteringUA:EnergyMin ENER_MIN` (by default the minimal energy is 0.)

The `ClassifyCC` method returns an integer with several digits containing the event classification:

- 0 if it is not a useful Compton imaging event, 1 if it is a coincident event but not fully absorbed, 2 if it is a fully absorbed coincident event and 3 if it is a fully absorbed Compton imaging event with a single interaction in the scatter detector and in the absorber detector. The imaging events are equal to the absorbed events, ie if no multiples examination is carried out, the absorbed events are the singles. If multiples are carried out then the number of absorbed events increases by the number of useable multiple events which can now be used for imaging.
- 10*1 if the search for reconstructed hits found more than 2
- 100*1 if the event is a random coincidence event
- 1000*1 if the event is a scattered event

At the end of the run a table is printed with the number of events in each of the combinations of the sub-classification types.

Compton camera histograms: event classification

These histograms are related to the event classification explained above. They are produced if the event classification user action is selected. The name of all these histograms starts with `"CCEvtClass: "` and all are written in the file `comptoncamera.root`.

The following histograms are written:

- Classification index of event ("Compton camera classification"). This index is the one described in the precedent section.
- The scatterer addback energy for the classified events
- The absorber addback energy for the classified events
- Number of reconstructed hits in the scatterer detector
- Number of reconstructed hits in absorber detector
- Sum energy for the classified events

Compton camera output for reconstruction

To write data to a file, the following parameter must be set to 1

```
/gamos/setParam CC:EvtClass:DumpEvent 1
```

If the event is classified as a Compton imaging one, it is dumped in a file, given by the name

```
/gamos/setParam CompCam:FileName MY_FILENAME
```

You can select the file format to be a binary file or a text file by setting to 1 or 0 the parameter

```
/gamos/setParam CompCam:BinFile 1/0
```


that takes the name *compcam.out* by default. It is necessary to define whether the output file should contain only data for single/single Compton imaging events (2 reconstructed hits) or those events which contain the multiple reconstructed hits, through the parameter

```
/gamos/setParam CC:EvtClass:DumpSingles TRUE/FALSE
```

```
/gamos/setParam CC:EvtClass:DumpMultiples TRUE/FALSE
```

where TRUE writes the data to the file *compcam.out*. The variables written are given by the structure

```
struct COutput
{
char name[9];
float x1, y1, z1, e1, x2, y2, z2, e2
}
```

where *name* is *COMPCAM*, *x1*, *y1*, *z1*, *e1* are the coordinates and energy of the reconstructed hit in the scatterer detector and *x2*, *y2*, *z2*, *e2* are the coordinates and energy of the reconstructed hit in the absorber detector. The same data that is written to the file can be written in the standard output if the parameter

```
/gamos/setParam CompCam:DumpToCout TRUE
```

is set to true. The positions in the standard output (not in the file) will be written in cylindrical coordinates by default. If you want them in cartesian coordinates you should set to true the parameter

```
/gamos/setParam CompCam:DumpCartesian TRUE
```


IMAGE RECONSTRUCTION UTILITIES

Several utilities are provided for helping the user to process projection data and reconstruct PET images. These utilities are installed along with the rest of GAMOS code and their source code included in the $\$/GAMOSINSTALL/analysis/NuclMed/PET/$ directory. They are available as executables as mentioned in the following subsections.

List-mode to projection data: *lm2pd*

This program is devised to convert previously stored PET list-mode output files (.out) in projection data (.hs/s or .hv/v). Its functionality is similar to the *ProjDataMgr* class (see “Projection data file” section), using the following arguments:

- *-a*: axial field of view in millimeters (by default 100.0).
- *-d*: diameter of the transaxial field of view, in millimeters (by default 300.0).
- *-m*: maximum ring difference (by default -1 = number of planes).
- *-n*: name of the output file (without extension).
- *-o*: output format, 0 for GAMOS ad-hoc format (by default, extension .hv/v compatible with GAMOS analytic image reconstructor) or 1 for STIR projection data (extension .hs/s).
- *-p*: number of axial planes (by default 40).
- *-r*: number of tangential bins (by default 81).
- *-t*: theta, number of angular views (by default 80).
- *-v*: verbosity (by default 0=silent, 3=debug).
- *-x*: maximum number of coincidences to process (by default -1 = no limit).

For example, the following line describes how to convert a list-mode file “pet.out” in projection data (file “MY_PROJDATA”) covering a 100-mm axial and 250-mm transaxial field of view; using 48 axial planes (rings), 160 angular views and 151 tangential bins. Output projection data is written in STIR format, with a maximum ring difference of 12.

```
lm2pd pet.out -a 100.0 -d 250.0 -p 48 -t 160 -r 151 -m 12  
-n MY_PROJDATA -o 1
```

Summing projection data files: *sumProjdata*

Merging output files is necessary after simulating different jobs under the same setup. This is straightforward when dealing with list-mode PET files, that can be concatenated with a command like *cat* in Linux; but projection data files

must be summed position-by-position (every position of each file contains the coincidences stored by the same PET line of response), this can be done thanks to the utility *sumProjdata*.

To use it you have to write a file containing the list of projection data file (without extension, it searches for files *.hs/*.s or *.hv/*.v), one per line, for example:

```
projdata_000
projdata_001
projdata_002
```

Then you just have to run the executable

```
sumProjdata INPUT_FILE_LIST OUTPUT_FILE
```

where *INPUT_FILE_LIST* is the name of the file containing the list of files to add and *OUTPUT_FILE* is the name of the output file that will contain the sum of all the files. Two files will be generated: the header, *OUTPUT_FILE.hs* (or .hv) and the binary data *OUTPUT_FILE.s* (or .v).

Analytic image reconstruction: *ssrb_fbp*

This program applies a SSRB-FBP2D[biblio.SSRB]_ method to reconstruct PET images. First, a Single Slice re-binning algorithm (SSRB converts 3-dimensional projection data in a set of 2-dimensional sinograms, one per each axial image slice; next, a Filtered Back Projection (image reconstruction algorithm that makes use of the Radon transform[biblio.imageRecoAlgo]_) is performed in each slice.

As input data, the program requires the name of the GAMOS projection data file without extension (.hv/.v). These files can be generated by GAMOS (see “PET output for reconstruction” section) or converted from list-mode files using the *lm2pd* utility. The command-line options are the following:

- *-m*: SSRB Maximum ring difference (by default: n_planes-1). The user can discard too axially tilted lines of response by selecting a lower value .
- *-r*: SSRB Normalization (by default: 1=yes, 0=no).
- *-i*: Image size, number of X and Y pixels (by default: number of sinogram bins).
- *-x*: Transaxial pixel size (by default: sinogram bin size).
- *-f*: Type of apodization window for filtering [biblio.toft] (low frequency pass): - 1, Generalized Hamming (value by default, it includes Hann window and ramplak). - 2, Butterworth. - 3, Shepp-Logan.
- *-a*: Alpha parameter for generalized Hamming window (by default: 1, ramplak filter).
- *-b*: Order for Butterworth window (by default 4).
- *-c*: Cutoff frequency, relative to Nyquist freq: bin_size/2 (by default: 0.75).
- *-n*: Name of the output image file (“MY_NAME”, without extension). The program will write a “MY_NAME.img” data file and two associated header text files: “MY_NAME.hv” and “MY_NAME.hdr”. The .hv file has a STIR-like Interfile format [biblio.Interfile], whereas the .hdr file uses Interfile 3.3 conventions; it is done to make possible opening the image by different programs (see “Visualization tools” section).
- *-h*: Help, printing arguments list.
- *-v*: Verbosity (by default:0 for silent, 3 for debugging).

The following example will apply a SSRB-FBP2D reconstruction to a projection data file named “MY_PROJDATA_ALL.hv” using a Butterworth filter of order=6 and cutoff frequency=0.5 Nyquist units, image and pixel sizes take values from the projection data (tangential bins):

```
ssrb_fbp MY_PROJDATA_ALL -f 2 -b 6 -c 0.5 -n MY_IMAGE
```

To use this utility, you have to install the FFTW library at the path where the variable `FFTW_BASE_DIR` points (type `echo $FFTW_BASE_DIR`). You can download this library at <http://www.fftw.org>

Visualization tools

Currently GAMOS does not include any utility to visualize the reconstructed images (or projection data), instead of it, we recommend the installation of one of the following free programs to visualize or analyze images: AMIDE and ImageJ.

AMIDE[*biblio.AMIDE*]₁ is a free tool for viewing, analyzing, and registering volumetric medical imaging data sets. It has been written on top of GTK+, and runs on any system that supports this toolkit, including Linux. Both *.hv and *.hdr image header files can be opened by AMIDE, using the option: File -> Import File (guess). AMIDE also is also recommended to visualize STIR images.

ImageJ [*biblio.ImageJ*] is an image processing tool written in Java, which allows it to run on Linux, Mac OS X and Windows. ImageJ and its Java source code are freely available and in the public domain. To open *.hv images the program needs a plug-in to read Interfile formats; next it is described how to install its version 1.44 (bundled with 32-bit Java), with the Interfile decoder plug-in, in Linux:

```
# Installing ImageJ
cd ${HOME}
wget http://rsb.info.nih.gov/ij/download/linux/ij144-x86.tar.gz
gunzip ij144-x86.tar.gz
tar xvf ij144-x86.tar
# Installing plug-in (.jar file) to read Interfile images
cd ImageJ/plugins
wget http://www.med.harvard.edu/JPNM/ij/plugins/download/Interfile_TP.jar
cd ..
```

To run ImageJ execute the './run' script. Images with extension .hv can be opened by the "NucMed Open" command (in menu Plugins->NucMed), alternatively the user can open it ("MY_IMAGE.hv") from the command line using the following expression:

```
${HOME}/ImageJ/jre/bin/java -Xmx512m -jar ${HOME}/ImageJ/ij.jar
-ijpath ${HOME}/ImageJ -eval "run('NucMed Open', 'open=MY_IMAGE.hv');
```

Stochastic Image Ensemble method

Introduction

About this Document

This manual refers to the *SOE_ImageRecon* program, version 0.0.0 of February 2012.

Complaints or questions about installation, compilation and use of this program can be sent to mkolstein@ifae.es.

Introduction to SOE

The SOE (Stochastic Origin Ensemble) method, is based on the article: *Stochastic Image Reconstruction Method for Compton Camera*, by *Andriy Andreyev, Arkadiusz Sitek, and Anna Celler* published in *IEEE Nuclear Science Symposium Conference Record, 2009*. [*biblio.SOEPaper*]

The SOE algorithm works on data files that contain coincidences of two hits. The exact format of these files is described in the Getting Started section. The data can be from a Compton Camera detector or from a PET detector.

In the case of Compton Camera data, one hit should be from the “Scatter detector” and one hit from the “Absorber” detector. Combining the information of these hits (both the positions and the deposited energies), Compton cones can be constructed. The original gamma source for an event lies somewhere on the corresponding Compton cone.

In the case of PET data, the two hits should be on opposing sides of the PET detector, corresponding to the two back-to-back gamma particles caused by an electron positron annihilation. Combining the position information of these hits, a “Line of Response” (LOR) can be reconstructed. The original gamma source for an event lies somewhere on the LOR.

Both for Compton camera data as for PET data, the SOE algorithm starts with a random ensemble of states. This means that at initialization, for each event a random position on the corresponding Compton cone or LOR is assigned. A 3D density matrix is defined, where the solution space is divided in voxels and the density per voxel depends on the number of initial event positions in that voxel.

Subsequently, the SOE algorithm performs a number of iterations. At each iteration, for each event a new position on the corresponding Compton cone or LOR is calculated. The new position is accepted according to the density value of the voxel corresponding to the new position, compared to the density at the old position. The density matrix is updated accordingly.

The iterations are repeated until the algorithm has converged.

By switching on particular parameters as explained later, the *Resolution Recovery* method can be used. This method tries to recover for possible deviations in the hit-positions, the hit-energies, and the Compton cone angle (due to the spatial resolution, the energy resolution and the Doppler broadening effect). By randomly smearing these variables, for each event, the method tries to find their optimum values.

Getting Started

Running SOE

SOE is installed in GAMOS at the directory *analysis/Detector/SOE*.

The SOE algorithm works on data files that contain coincidences of two hits. The exact format of these files is described below in the section called “Input files”. The data can be either from a Compton camera detector or from a PET detector.

To run the code for Compton Camera data, type:

```
SOE_ImageRecon -Compton
```

or, for PET data:

```
SOE_ImageRecon -PET
```

The complete list of possible flags is:

```
SOE_ImageRecon
SOE_ImageRecon
SOE_ImageRecon -PET (for PET data)
SOE_ImageRecon -Compton (for Compton data)
SOE_ImageRecon -PETGauss (for PET data, using a Gauss distribution for guessing points on the LORs)
SOE_ImageRecon -seed < seedNumber > (selecting specific seed)
SOE_ImageRecon -extraIterInfo (additional output files for intermediate iterations)
SOE_ImageRecon -averageStatesOut (additional output each 100 iterations before final iterations)
SOE_ImageRecon -readDensityMatrix (read density matrix 'density_matrix.img' at initialisation of pro
```

```
SOE_ImageRecon -readCurrentState (read current state file 'currentstate.dat' at initialisation of pro
SOE_ImageRecon -help (for this help output)
```

Input files

In the directory in which you run the code, you need the following input files:

```
ir_soe_userparameters.conf (parameters file)
CC_img.out (data file with any kind of name, as given in the parameters file).
```

In the data file, every line should contain the following:

```
Hit1.Z (mm) Hit1.Y (mm) Hit1.X (mm) Hit1.Energy (keV) Hit2.Z (mm) Hit2.Y (mm) Hit2.X (mm) Hit2.Ene
```

In the case of Compton camera data, “Hit1” corresponds to the hit in the Scatter detector and “Hit2” corresponds to the hit in the Absorber. In the case of PET data, the hits correspond to the two back-to-back hits in opposite regions of the detector.

Note 1: Note that the order of the coordinates in the files is: Z, Y, X, Energy!

Note 2: The geometry used in the SOE algorithm is as follows. The z-axis is the axis that separates the Scatter detector from the Absorber detector. The front of the Scatter detector is located at $z = 0$ mm. The gamma source must be at a location with a negative value for the z coordinate. The Absorber detector is at positive z with respect to the Scatter detector.

The parameter file *ir_soe_userparameters.conf* has the following contents:

```
m_iterations      100000          //
m_bins_z          100             //
zPosition1        -95.0           //
zPosition2        -105.0          //
m_bins_x          100             //
m_xmin            -5.0            //
m_xmax            5.0             //
m_bins_y          100             //
m_ymin            -5.0            //
m_ymax            5.0             //
m_sourceEnergy    511             //
DataFileName      0               ../../../../CCs_Machiel/CC_img.out_Gamma511_AllOFF
EventSetSize      0               //
m_maxNumberOfEvents 1000          //
UseEnergyResolutionFactorFWHM 0.0 //percentage
UseSpatialDeltaX  0.0             //mm
UseSpatialDeltaY  0.0             //mm
UseSpatialDeltaZ  0.0             //mm
UseDopplerEffectSigmaTheta 0.0    //sigma
```

Each line should contain three fields:

- the first field is the name of the parameter
- the second field is the value (except in the case of DataFileName, where the value is a dummy)
- the third field is a dummy string (except in the case of DataFileName, where it gives the name of the input file). It should NOT contain any blanks (empty spaces).

Output Files

The output files are:

- *data_beginstate_xy.dat* This file contains a list (for all events) with the initial positions assigned to each cone. (Each position represents a guess of the real location of the gamma source). It has the following format:

```
xposition yposition zposition
```

e.g.:

```
260.788 142.073 -97.8255
-0.722199 -18.968 -96.3066
2.74312 -43.1098 -95.7843
...
```

- *data_endstate_xy.dat* This file contains a list (for all events) with the final positions associated with each cone, after $< N >$ iterations. (Each position represents a guess of the real location of the gamma source). In the perfect case, all positions in the list would be identical to the real position of the gamma source. It has the following format:

```
xposition yposition zposition
```

e.g.:

```
0.44799 0.412472 -100.032
0.491477 0.46389 -100.056
0.459591 0.449933 -99.9168
...
```

- *density_matrix.hdr* This is a header file for the AMIDE program.
- *density_matrix.img* This is the binary image file for the AMIDE program.

Program Parameters and Flags

PET or Compton camera

In order to run on PET data, run the program with either one of the following flags:

```
SOE_ImageRecon -PET
```

or:

```
SOE_ImageRecon -PETGauss
```

With the *-PET* flag the possible locations are taking uniformly from the entire LOR.

With the *-PETGauss* flag the possible locations are taking from a Gauss distribution on the entire LOR, where the sigma corresponds to 16% of the LOR length and the mean is in the middle of the LOR. (Note: this actually only makes sense in the case of a single source located at the centre of the PET detector and is mainly meant for testing purposes.)

Input Data

The input parameters are set in the parameter file *ir_soe_userparameters.conf*.

DataFileName	0	../../../../../../../../CCs_Machiel/CC_img.out_Gamma511_AllOFF
EventSetSize	0	//

With *DataFileName* only the third field is important, the second field should be an integer that is ignored by the code.

The *EventSetSize* parameters sets the size of the number of events that are maintained in memory at the same time. The remaining number of events are stored on disk. What is lost in time because of the I/O is won in time (and memory capacity) because of smaller data set handling. For data sets smaller than a few million events, it should be set to 0 (i.e., all events are kept in memory).

Iterations

The number of iterations is set in the parameter file *ir_soe_userparameters.conf*.

m_iterations	100000	//
--------------	--------	----

Geometry

The geometry parameters are set in the parameter file *ir_soe_userparameters.conf*.

m_bins_z	100	//
zPosition1	-95.0	//
zPosition2	-105.0	//
m_bins_x	100	//
m_xmin	-5.0	//
m_xmax	5.0	//
m_bins_y	100	//
m_ymin	-5.0	//
m_ymax	5.0	//
m_sourceEnergy	511	//

These parameters set the number of bins and the Field Of View (FOV) of the image solution space.

Resolution Recovery

The resolution recovery parameters are set in the parameter file *ir_soe_userparameters.conf*.

UseEnergyResolutionFactorFWHM	0.0	//percentage
UseSpatialDeltaX	0.0	//mm
UseSpatialDeltaY	0.0	//mm
UseSpatialDeltaZ	0.0	//mm
UseDopplerEffectSigmaTheta	0.0	//sigma

The *UseEnergyResolutionFactorFWHM* parameter sets the Gaussian variation in Energy used to recover for resolution loss due to energy resolution.

The *UseSpatialDeltaX*, *SpatialDeltaY*, and *SpatialDeltaZ* parameters set the Uniformly distributed variation in x,y and z coordinates of the detector hits, used to recover for resolution loss due to spatial resolution.

The *UseDopplerEffectSigmaTheta* parameter sets the Gaussian distributed variation in the Compton angle, used to recover for resolution loss due to Doppler broadening.

Special running flags

Flags to run with the code:

SOE_ImageRecon -seed <seedNumber>	(selecting specific seed)
SOE_ImageRecon -extraIterInfo	(additional output files for intermediate iterations)
SOE_ImageRecon -averageStatesOut	(additional output each 100 iterations before final iteration)
SOE_ImageRecon -readDensityMatrix	(read density matrix 'in_density_matrix.img' at initialisation)
SOE_ImageRecon -readCurrentState	(read current state file 'currentstate.dat' at initialisation)
SOE_ImageRecon -help	(for this help output)

The *seed* flag sets the seed for the SOE algorithm. This way, a SOE run can be repeated with exactly the same random numbers.

With the *extraIterInfo* and the *averageStatesOut* flags there will be additional output (for debugging purposes).

With the *readDensityMatrix* and *readCurrentState* flags the “densitymatrix.img” and “currentstate.dat” (of the format as the standard output file “data_endstate.dat”), are read in at initialisation (for debugging purposes).

With the *help* flag, the list of all flags is printed to screen.

Analysis

In the root/ directory, there are various ROOT programs that can be used for the analysis. One of the most important ones is *SOE_img3D.C*. This ROOT program visualises the output of a SOE 3D image reconstruction.

At start-up will ask the user whether the real location of the gamma source is known. If it is not, just type 0. If it is, type 1, and, subsequently give the x, y, and z coordinates of the source location.

Each plot is drawn in its own separate canvas:

- *m_c1: 3dcontour plot.* Shows a 3D contour plot.
- *m_c2: X profile plot.* Shows the density along the X axis, with $Y = y_{\max}$, and $Z = z_{\max}$, the y and z coordinates of the voxel with maximum density (unless the real source location is given, in which case this is used).
- *m_c3: Y profile plot* Shows the density along the Y axis, with $X = x_{\max}$, and $Z = z_{\max}$, the x and z coordinates of the voxel with maximum density (unless the real source location is given, in which case this is used).
- *m_c4 Z profile plot* Shows the density along the Z axis, with $X = x_{\max}$, and $Y = y_{\max}$, the x and y coordinates of the voxel with maximum density (unless the real source location is given, in which case this is used).
- *m_c5 R profile plot.* The density as a function of the radius is calculated. The radius is taken from the voxel with the maximum density (unless the real source location is given, in which case this is used). The bin-contents in this histogram are normalised with the 3D volume corresponding to each bin. (I.e, for $R = [0, \text{bin1}]$, the volume is $4/3 * \pi * R_{\text{bin1}}^3$, for $R = [\text{bin1}, \text{bin2}]$, the volume is $4/3 * \pi * (R_{\text{bin2}}^3 - R_{\text{bin1}}^3)$).
- *m_c6 Accumulative R plot.* The same as *m_c5*, but without normalising with the corresponding volume of the solution space. This way, it can be clearly seen how many events reach the center bin, and how many events are outside it.

RADIOTHERAPY APPLICATION

The Radiotherapy application contains some utilities for the simulation of a teletherapy linear accelerator and dose calculations in the patient. Many of the utilities can also be useful in the simulation of brachytherapy treatments.

Geometrical modules

Two geometrical modules are defined related to Radiotherapy: JAWS and MLC (multileaf collimators). They can be defined in the usual geometry text file (see chapter on *Geometry*) with the format described below.

JAWS module

This module serves to describe the jaws of a radiotherapy accelerator. The jaws are described by their dimensions, and their rotations are calculated automatically using the parameters of the projection into a plane. It is important to remember that the beam is assumed to run along the negative Z axis, following the IEC 61217 convention (if you prefer to define the beam along the positive Z axis, you have to define the environmental variable *BEAMZPOS* and recompile GAMOS).

The endleaf straight implies a circular leaf shape, and the movement is made along a circle. The endleaf rounded implies a straight leaf shape and the movement is made along a straight line. The rounded type has a subtype, when the center of the circle that shapes the rounded leaf tip is not at the centre of the leaf, so that the tip is not symmetrical with respect to the tip half plane. And the displaced rounded tip has in turn a subtype, which allows to define the tip positions as a function of the field aperture reading the positions from a file, instead of calculating them automatically (see below).

The endleaf cross positions are described by their projection at the isocenter plane, as it is common in the DICOM RTPlan files. For the straight leaves, this position is calculated so that the line between the focus and the projection at the isocenter plane is parallel to the leaf end. For the rounded leaves, this position is calculated so that this line is tangent to the leaf end. If a half-value layer is defined this line traverses the leaf end by a distance equals to this value. Alternative, *XXX* as a function of the field aperture reading the positions from a file, instead of calculating them automatically.

The following parameters have to be provided (between parenthesis the name of each parameters as they appear in fig. RT1):

:MODULE JAWS

Module name

Module orientation: direction of the jaws. It has to be X or Y .

End leaf type It must be *STRAIGHT*, *ROUND*, *ROUND_DISP*. *ROUND_DISP* means that the centre of the circle that defines the profile is not at the line in the middle of the leaf. It must be *STRAIGHT*, *ROUND*, *ROUND_DISP*, *ROUND_DISP_POSFILE*. *STRAIGHT* means that the leaf profile is a rectangle and it moves along a circle, while

ROUND means that the leaf profile tip closest to the beam has circular shape and it moves along a straight line. *ROUND_DISP* means that the centre of the circle that defines the profile is not at the line (perpendicular to the beam axis) in the middle of the leaf. And *ROUND_DISP_POSFILE* means that the endleaf positions are a function of the field aperture and are read from a file, instead of calculating them automatically.

- *X half dimension*
- *Y half dimension* (LONGDIM)
- *Z half dimension* (ZDIM)
- *If the end leaf type is ROUND or ROUND_DISP, end leaf radius* (TIP_RADIUS)
- *If the end leaf type is ROUND_DISP, distance between leaf top and radius centre* (DISP).

If the end leaf type is *ROUND*, the radius center is at the middle of the leaf, therefore this parameter is not needed.

- *If the end leaf type is ROUND or ROUND_DISP, half value layer* (HVL).

This half value layer should correspond to the distance at which only half of the gammas from the accelerator traverse the material of the leaf. It is used to displace the leaves so that the line from the focus to the aperture position in the isocenter given below traverses a distance of the rounded leaf tip equal to this value.

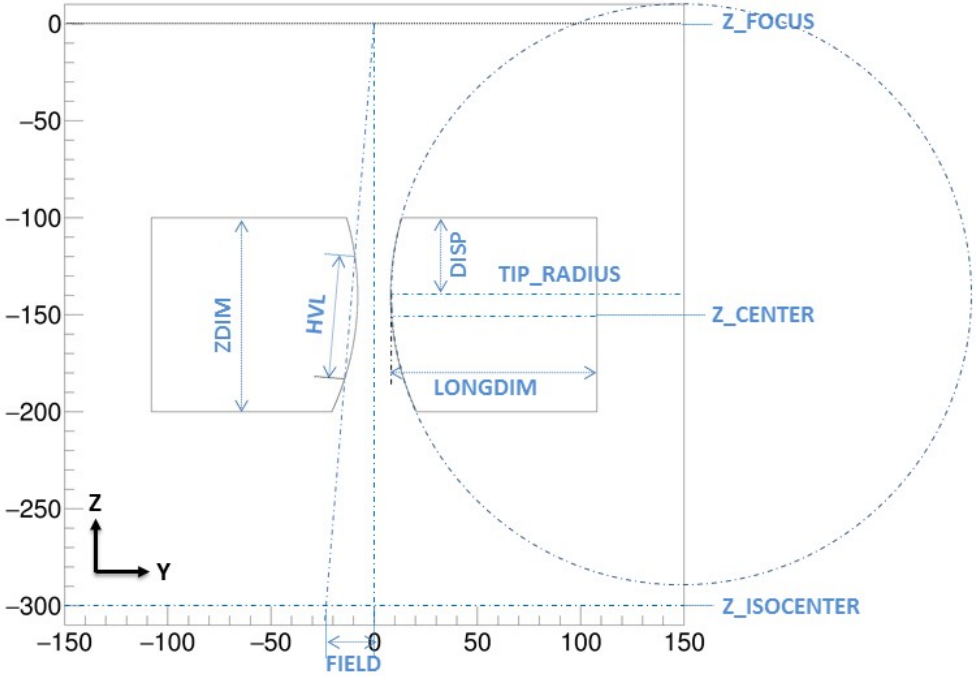
- *Z position of the focus* (Z_FOCUS)
- *If the end leaf type is STRAIGHT, radius of rotation of the aperture movement* (usually the same as the distance between the focus and the volume centre)
- *Z of the position where the center of the volume would be if the field were 0 (no rotation)* (Z_CENTER)
- *Z position of the isocenter, where the projections are calculated* (Z_ISOCENTER)
- *Field negative projection* (usually a negative value) (FIELD)
- *Field positive projection* (usually a positive value)
- *Material*
- *Parent volume name*

Fig. RT1. XZ plane view of a cut along Y=40 mm of the jaws created from the file below

Module Jaws example

On the next lines we include a simplified geometrical description, which can be used as example. On figure RT1 it can be seen the corresponding representation.

```
JAWS_Y Module_name
Y ROUND_DISP Orientation, Tip type
:P JAWS_ZPOS -150.
:P JAWS_Z 100./2.
:P JAWS_TIP_CIRCLE_Z 40.
:P JAWS_TIP_CIRCLE_R 150.
:P JAWS_HVL 70.
:P JAWS_FIELD 5.*cm/2.
:MODULE JAWS
10.*cm/2 10.*cm/2 $JAWS_Z // Half-dimensions
```



```
$JAWS_TIP_CIRCLE_R $JAWS_TIP_CIRCLE_Z // Tip_circle_radius Tip_circle_centre_Z
$JAWS_HVL // Half value layer
0. $JAWS_ZPOS-$JAWS_Z -30.*cm // Z focus Z_centre Z_isocentre(field)
-$JAWS_FIELD $JAWS_FIELD // Field apertures: RIGHT & LEFT
RTUW ACCEL // Material ATERIAL
```

Projection in the isocenter plane of the light field, XL(i), and leaf tip position, W(i)

MLC module

Introduction

A MLC (Multi Leave Collimator) is a collimator system composed of a set of leaf pairs usually a few millimeters thick which can be open or close to form a hole in the field of view of an irradiation treatment beam. By opening the different leaves a different distance an irregular treatment field can be shaped.

This module can describe a high variety of MLC models, whether the leaf profiles are focused to a point in the Z axis or to an off-axis point.

Two endleaf types are supported: straight or rounded. See Module Jaws for details.

Any leaf cross profile (i.e. the 2D profile in the direction orthogonal to the leaf movement) shape can be described by enumerating the list of 2-dimensional points that describe it. These points have to be given in clock-wise order. When the leaf is placed, the profile shape is deformed so the lines that are not perpendicular to the Z axis are focused towards the focus point; this has to be taken into account so that you do not define lines not perpendicular if you do not want that they are modified to point to the focus).

Geometrical module definition and variables list

The MLC module description has to start with the two words (words have to be separated by blank spaces, and follow the order described here, but there is no constraint on the number of lines they may occupy). The following parameters have to be provided (between parenthesis the name of each parameters as they appears in fig. RT2 and RT3):

:MODULE MLC

Then the following parameters have to be filled

- *Module name*

- *Module type.* The type of MLC has to be *FOCUSED*, what means that all leaves profiles in the cross plane are focused to a point. Type *UNFOCUSED* will be supported in future releases.

- *Leaves orientation.* Orientation of the leaves, i.e. the axis of movement, can be *X or Y*

- *Leaf tip type.* It must be *STRAIGHT, ROUND, ROUND_DISP, ROUND_DISP_POSFILE*. As explained above *STRAIGHT* means that the leaf profile is a rectangle and it moves along a circle, while *ROUND* means that the leaf profile tip closest to the beam has circular shape and it moves along a straight line. *ROUND_DISP* means that the centre of the circle that defines the profile is not at the line in the middle of the leaf. And *ROUND_DISP_POSFILE* means that the tip positions are a function of the field aperture and are read from a file, instead of calculating them automatically.

- *Leaf half dimension in the inline plane (LONGDIM).* It is X half dimension if the orientation is X and Y half dimension if the orientation is Y. If the endleaf is curved this length is defined as the maximum extent. If the endleaf is straight, and therefore it has a circular shape, the length is defined as the arc length of the upper part of the leaf (the side closest to the source).

- If the leaf tip type is *ROUND*, *ROUND_DISP* or *ROUND_DISP_POSFILE*, leaf tip radius (*TIP_RADIUS*)
- If the leaf tip type is *ROUND_DISP* or *ROUND_DISP_POSFILE*, distance between leaf top (part closest to the beam origin) and radius centre (*DIST*). If the leaf tip type is *ROUND*, the radius center is at the middle of the leaf, so this parameter is not needed.
- If the leaf tip type is *ROUND* or *ROUND_DISP*, half value layer (*HVL*). This half value layer should correspond to the distance at which only half of the gammas with the spectrum generated by the accelerator traverse the material of the leaf. It is used to move the leaves so that the line from the focus to the aperture position in the isocenter traverses a distance of the rounded leaf tip equal to the half value layer.
- If the leaf tip type is *ROUND_DISP_POSFILE*, name of file with leaf aperture (the number that defines the field) vs. leaf positions (the intersection of the leaf tip at the center of curvature with the isocenter plane). In figure RT1 these two magnitudes correspond to the light field edge, *XL(i)*, and leaf tip position, *W(i)*.
- *Z* position of the focus
- Cross position of the focus. If the leaves orientation is X the cross position is in the Y axis and viceversa.
- *Z* of position of the isocenter. The XY plane at this Z is used to calculate the cross positions that define the leaf profiles as well as the positions that define the leaf apertures. Following the IEC 61217 convention, the beam is assumed to run along the negative Z axis, therefore this position should be a value smaller than the Z position of the focus. F
- *Z* of position of the top of the leaves (the position closer to the beam source) (*Z_TOP*). This position will be used as the 0 reference to place the leaves. Indeed, it is used as the Z=0 position for defining the leaf profiles; therefore the Z positions that define the leaf profile should be negative (further from the beam source).
- Distance in Z between *Z_TOP* and the plane where leaf gap is defined (*Z_GAP*). Remember that if you define it at the bottom of the leaves, it should have a negative value.
- Gap between leaves in the cross direction (*CROSS_GAP*). Separation between leaves defined at *Z_GAP*.
- Cross leaf starting point. Cross position of the leaf situated at the most negative position. It is defined at the same Z plane as the leaf gap. (*CROSS_START*)
- Number of different types of leaf cross profiles.

For each leaf cross profile type the following data has to be defined:

- - Leaf type. It will be a LEAF or BLOCK, but this options is not implemented yet, so it must be LEAF
- - Number of 2-dimensional points that define the cross profile
- For each point the Z and cross coordinates must be defined - *Z coordinate* - *Cross coordinate*

Number of leaf pairs

For each leaf pair the following data has to be defined:

- Leaf type number 1, 2, 3, ...; following the order of leaf types defined
- - Cross projection of the left part (negative coordinate) of the leaf on isocenter plane The projection is taken as that of the leaf profile point that gives an smaller (more negative) coordinate.
- - Opening distance of negative leaf, projected on the isocenter plane In case of rounded leaf tips, the point projected is not the end of the leaf, but a line is traced from the source towards the isocenter plane traversing the leaf, so that the distance traversed is the same as the half value layer given above.
- - Opening distance of positive leaf, projected on the isocenter plane

Finally the material of the leaves have to be defined

material

Mother volume name

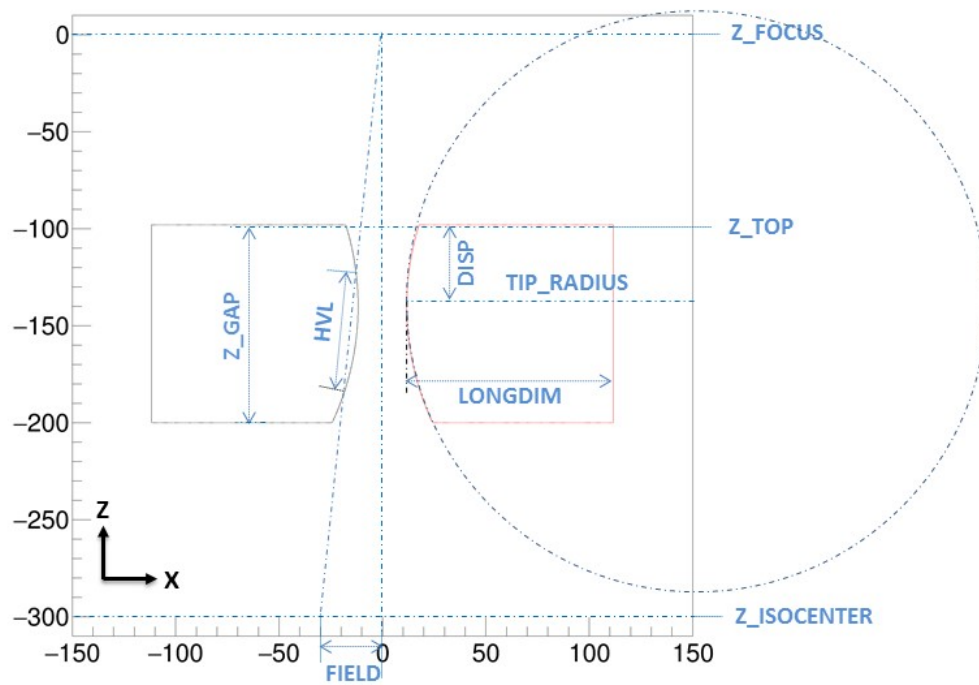


Fig. RT2. YZ plane view of a cut along X=30 mm of the MLC created from the file below

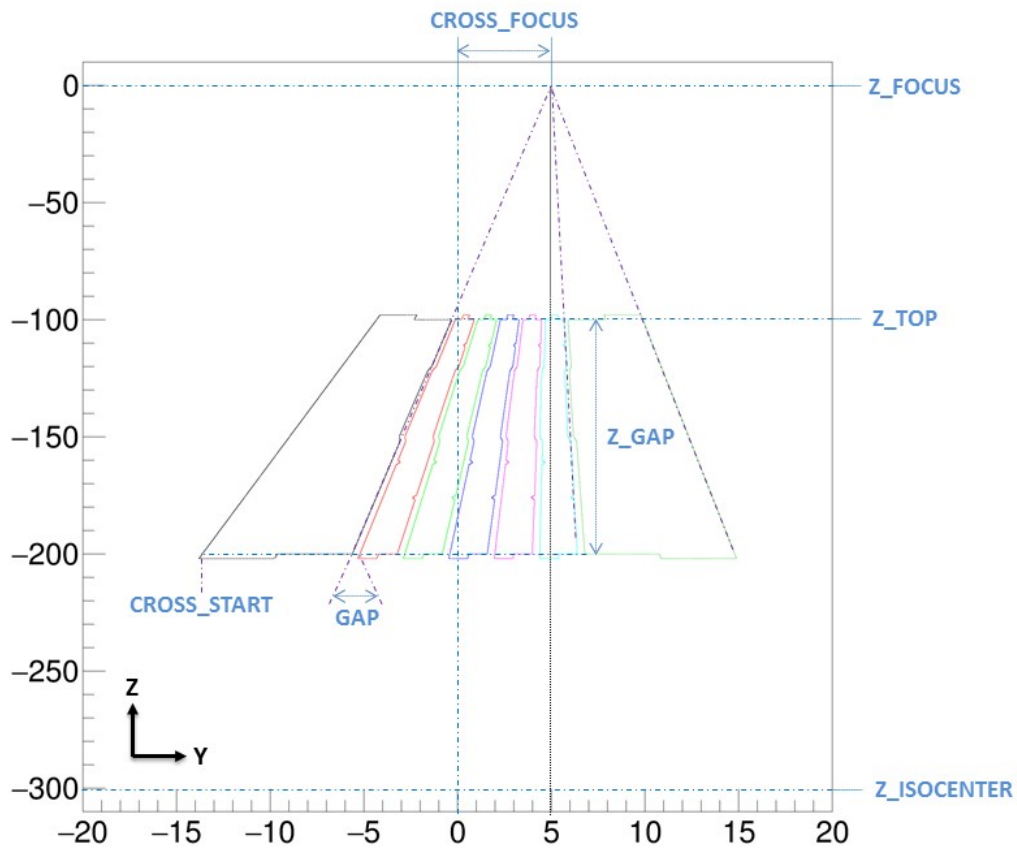


Fig. RT3. XZ plane view of a cut along Y=9 mm of the MLC created from the file below

Fig. RT4. Leafs geometry of the MLC created from the file below

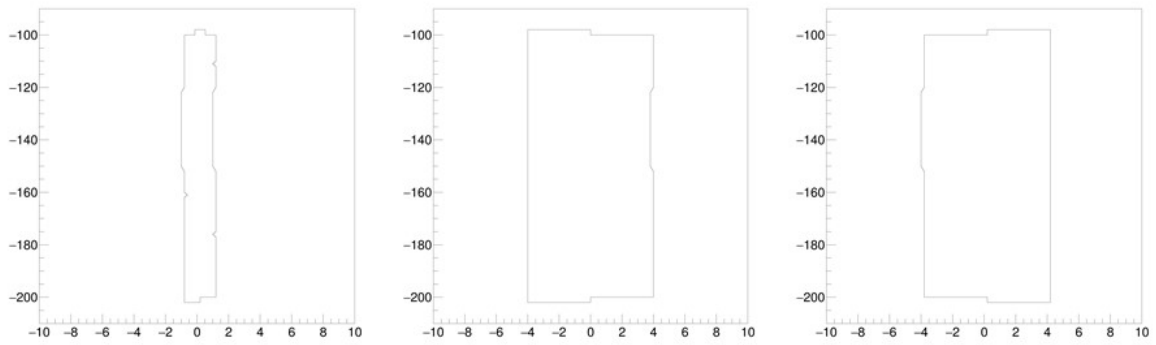
Module MLC example

On the next lines we include a simplified geometrical description, which can be used as example. On figures RT2, RT3 and RT4 it can be seen the corresponding representation.

```

:P MLC_LH 2.
:P MLC_LVD 2.
:P MLC_LHD 0.2
:P MLC_LX 2.
:P MLC_LVN1 20.
:P MLC_LVN2 50.
:P MLC_LVN3 60.
:P MLC_LVN4 100.

```



```

:P MLC_LVP1 10.
:P MLC_LVP2 $MLC_LVN1
:P MLC_LVP3 $MLC_LVN2
:P MLC_LVP4 75.
:P MLC_LVP5 $MLC_LVN4
:P MLC_TIP_CIRCLE_R 150
:P MLC_TIP_CIRCLE_Z 40.
:P MLC_ZPOS_TOP -100
:P MLC_CROSS_FOCUS 5.
:P MLC_IsocentrePositionZ -300.
:P MLC_GAP 0.2
:P MLC_NLEAF_PAIRS 7
:P MLC_HVL 70.
:P MLC_FIELD 30.
:MODULE MLC
MLC_X Module_name
FOCUSED X ROUND_DISP_POSFILE // Type, Orientation, LeafTip type
100./2. // Leaf long half-length
$MLC_TIP_CIRCLE_R $MLC_TIP_CIRCLE_Z // Tip_circle_radius Tip_circle_centre_Z
$MLC_HVL // Half value layer
    0. $MLC_CROSS_FOCUS $MLC_IsocentrePositionZ // Z focus, CrossLeaf focus, Z isocentre
$MLC_ZPOS_TOP // Z top
-$MLC_LVN4 $MLC_GAP -($MLC_LH*($MLC_NLEAF_PAIRS-2)+($MLC_LH*4)*2+$MLC_GAP*($MLC_NLEAF_PAIRS-
1))/2. // Z gap, Cross Leaf Gap, Cross Leaf Start Point
3 // N Types of Leaf Cross Profiles
LEAF 27 // Leaf type 1: Leaf Mode, N Leaves Cross Points
    0. -$MLC_LH/2.
    0. -$MLC_LH/6.
$MLC_LX -$MLC_LH/6.
$MLC_LX $MLC_LH/6.
    0. $MLC_LH/6.
    0. $MLC_LH/2.
-$MLC_LVP1 $MLC_LH/2.
-($MLC_LVP1+$MLC_LVD/2.) $MLC_LH/2.-$MLC_LHD
-($MLC_LVP1+$MLC_LVD) $MLC_LH/2.
-$MLC_LVP2 $MLC_LH/2.

```

$-(\$MLC_LVP2+\$MLC_LVD) \$MLC_LH/2.-\MLC_LHD
 $-\$MLC_LVP3 \$MLC_LH/2.-\$MLC_LHD$
 $-(\$MLC_LVP3+\$MLC_LVD) \$MLC_LH/2.$
 $-\$MLC_LVP4 \$MLC_LH/2.$
 $-(\$MLC_LVP4+\$MLC_LVD/2.) \$MLC_LH/2.-\MLC_LHD
 $-(\$MLC_LVP4+\$MLC_LVD) \$MLC_LH/2.$
 $-\$MLC_LVP5 \$MLC_LH/2.$
 $-\$MLC_LVP5 0.$
 $-(\$MLC_LVP5+\$MLC_LX) 0.$
 $-(\$MLC_LVN4+\$MLC_LX) -\$MLC_LH/2.$
 $-(\$MLC_LVN3+\$MLC_LVD) -\$MLC_LH/2.$
 $-(\$MLC_LVN3+\$MLC_LVD/2.) -\$MLC_LH/2.+\MLC_LHD
 $-\$MLC_LVN3 -\$MLC_LH/2.$
 $-(\$MLC_LVN2+\$MLC_LVD) -\$MLC_LH/2.$
 $-\$MLC_LVN2 -\$MLC_LH/2.-\$MLC_LHD$
 $-(\$MLC_LVN1+\$MLC_LVD) -\$MLC_LH/2.-\MLC_LHD
 $-\$MLC_LVN1 -\$MLC_LH/2.$

LEAF 12 // Leaf type 2: Leaf Mode, N Leaves Cross Points

$\$MLC_LX -\$MLC_LH*2.$

$\$MLC_LX 0.$

0. (a)

0. $\$MLC_LH*2.$

$-\$MLC_LVN1 \$MLC_LH*2.$

$-(\$MLC_LVN1+\$MLC_LVD) \$MLC_LH*2.-\MLC_LHD

$-\$MLC_LVN2 \$MLC_LH*2.-\$MLC_LHD$

$-(\$MLC_LVN2+\$MLC_LVD) \$MLC_LH*2.$

$-\$MLC_LVN4 \$MLC_LH*2.$

$-\$MLC_LVN4 0.$

$-(\$MLC_LVN4+\$MLC_LX) 0.$

$-(\$MLC_LVN4+\$MLC_LX) -\$MLC_LH*2.$

LEAF 12 // Leaf type 3: Leaf Mode, N Leaves Cross Points

0. $-\$MLC_LH*2.$

0. (a)

$\$MLC_LX 0.$

$\$MLC_LX \$MLC_LH*2.$

$-(\$MLC_LVN4+\$MLC_LX) \$MLC_LH*2.$

```

-( $\$MLC\_LVN4+\$MLC\_LX$ ) 0.
- $\$MLC\_LVN4$  0.
- $\$MLC\_LVN4 -\$MLC\_LH*2$ .
-( $\$MLC\_LVN2+\$MLC\_LVD$ ) - $\$MLC\_LH*2$ .
- $\$MLC\_LVN2 -\$MLC\_LH*2-\$MLC\_LHD$ 
-( $\$MLC\_LVN1+\$MLC\_LVD$ ) - $\$MLC\_LH*2-\$MLC\_LHD$ 
- $\$MLC\_LVN1 -\$MLC\_LH*2$ .
 $\$MLC\_NLEAF\_PAIRS$  // Number of leaves
2 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
1 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
1 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
1 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
1 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
1 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
3 - $\$MLC\_FIELD$   $\$MLC\_FIELD$  // Leaf type, negative and positive aperture at isocentre
RTUW ACCEL // Material mother_volume

```

Using phase spaces

A very common utility in teletherapy simulation is the writing of a *phase space*, i.e. the set of particles that reach a certain plane in Z, and the later starting of the next simulation from this set of particles. This technique can save a lot of time in cases several calculations share some accelerator parts and in the case where the accelerator simulation is very slow compared to the dose calculation.

Writing phase spaces

The writing of phase space can be done automatically in GAMOS by selecting the user action

```
/gamos/userAction RTPPhaseSpaceUA
```

When a particle crosses any of the planes defined by the parameter

```
/gamos/setParam RTPPhaseSpaceUA:ZStops Z_1 Z_2 Z_3 ...
```

its information is stored in a file whose named is given by the parameter

```
/gamos/setParam RTPPhaseSpaceUA:FileName MY_FILENAME
```

plus a suffix *.IAEAphsp*. The default value of this parameter is *test*. If there are several Z planes, the Z value of each plane will be added to the name, with a “_” in front, so that each phase space will go to a different file. If there is only one Z defined, the Z value may be written in the file name if the following parameter is set

```
/gamos/setParam RTPPhaseSpaceUA:ZStopInFileName TRUE
```

If the plane crossed is the one with maximum Z, the particle may be stopped, if the following parameter is set

```
/gamos/setParam RTPPhaseSpaceUA:KillAfterLastZStop TRUE
```

The format of the phase space files is the one defined by the IAEA, generated using the official C files from IAEA. First there is a header file, that will have the same name, but with the suffix *.IAEAheader*. It is generated by the IAEA code and you can find the description of it at [\[biblio.iaeaphsp\]](#).

The variables stored for each particle are the following

- X coordinate (cm)
- Y coordinate (cm)
- X direction cosine
- Y direction cosine
- Kinetic energy (MeV)
- Particle statistical weight
- Type of the particle (1 = gamma, 2 = electron, 3 = positron, 4 = neutron, 5 = proton)
- Z direction cosine * particle charge
- Is new history
- Extra integers (0, 1 or 2 values)
- Extra floats (0, 1 or 2 values)

The Z value may optionally be stored if the following parameter is set

```
/gamos/setParam RTPPhaseSpaceUA:StoreZ TRUE
```

The header file may be written each N events, so that if the job ends abnormally the first N events may be recovered in the phase space. The number of events elapsed between header writing is controlled with the parameter

```
/gamos/setParam RTPPhaseSpaceUA:NEventsToSave TRUE/FALSE
```

By default this parameter takes a value of -1 and no header is saved until the end of the run.

It is also possible to limit the number of particles in a file. This can be useful if you want to store a phase space at different Z values and you do not want too many particles at the smallest Z planes. To do this you can set the parameter:

```
/gamos/setParam RTPPhaseSpaceUA:MaxNTracksInFile NTRACKS
```

The number of tracks stored in each file will be stopped at *NTRACKS*, but it may continue in other files.

As the Z plane is probably not a physical plane in the geometry, particle steps will start before the plane and end after the plane. Therefore, the position and energy are rescaled as if the particle would have stopped at the Z plane (by a simple linear interpolation).

By default particles that travel towards the negative Z direction are not stored, as it is assumed that the original direction was towards positive Z and therefore these particles (or their ancestors) should already have been stored. You may nevertheless eliminate this check and store all particles by setting the parameter:

```
/gamos/setParam RTPPhaseSpaceUA:NotStoreBackwards FALSE
```

You can limit the number of particles stored in each phase space (what may be useful when you are filling several phase spaces in a job and do not want that the first phase spaces end occupying too much disk space. To do this you have to set the parameter:

```
/gamos/setParam RTPPhaseSpaceUA:MaxNTracksRecorded NTRACKS
```

Phase space text file

You can also write the phase space in a simple text format. The following variables will be written for each track that reaches the phase space plane: *particle_code kinetic_energy(MeV) x(cm) y(cm) z(cm) direction_x direction_y direction_z weight extra_information_word_1 extra_information_word_2 ... extra_information_word_n*. The name of the file is set with the parameter:

```
/gamos/setParam RTPPhaseSpaceUA:TextFileName FILE_NAME
```

Phase space histograms

When a phase space is generated, a set of histograms are produced to give you some information about the particles in the phase space. The name of all these histograms start with *PhaseSpace:* and they are all dumped into the file *phaseSpace.root/csv*. For each of the following particle types a different set of histograms are produced (containing the particle type in the histogram name:) “*gamma*”, “*e-*”, “*e+*”, plus a set of histograms for all particles, named “*ALL*”. Also histograms for “*neutron*” and “*proton*”, will be produce if the following parameter is set

```
/gamos/setParam RTPPhaseSpaceHistos:Hadrons TRUE
```

A full set of histograms is produced for each Z plane, with the Z value on its name.

The following histograms are produced:

- Position X (cm) (“X at Zstop”)
- Position Y (cm) (“Y at Zstop”)
- Position X and Y (cm) (“XY at Zstop”)
- Radial position in the XY plane (cm) (“R at Zstop”)
- Theta angle of direction (degrees) (“Direction Theta at Zstop”)
- Phi angle of direction (degrees) (“Direction Phi at Zstop”)
- Energy (MeV) (“Energy at Zstop”)
- X direction cosine (“Vx at Zstop”)
- Y direction cosine (“Vy at Zstop”)
- Z direction cosine (“Vz at Zstop”)
- Radial position in the XY plane (cm) vs theta angle of direction (degrees) (“R vs Direction Theta at Zstop”);
- Radial position in the XY plane (cm) vs energy (MeV) (“R vs Energy at Zstop”);
- Theta angle of direction (degrees) vs energy (MeV) (“Direction Theta vs Energy at Zstop”);

The user can choose not to produce any of these histograms by setting the following parameter

```
/gamos/setParam RTPPhaseSpaceUA:Histos FALSE
```

The name of the histogram file can be controlled with the parameter

```
/gamos/setParam RTPPhaseSpaceHistos:FileName MY_FILENAME
```

that by default is “phaseSpace”.

use the commands in */gamos/analysis/* (see section on *Histogramming*) to control the minimum, maximum and number of bins of the histograms

Reading phase spaces

To use the generated phase space you have to define as your primary generator

```
/gamos/generator RTGeneratorPhaseSpace
```

You can change the filename (by default *test*) with the parameter

```
/gamos/setParam RTGeneratorPhaseSpace:FileName MY_FILENAME
```

The particles in the phase space can be translated or rotated by using the parameters

```
/gamos/setParam RTGeneratorPhaseSpace:InitialDisplacement POS_X POS_Y POS_Z
```

```
/gamos/setParam RTGeneratorPhaseSpace:InitialRotAngles ANG_X ANG_Y ANG_Z
```

the position of the particles is first changed by the initial displacement, then the momentum vector is rotated around the X axis by *ANG_X*, around the Y axis by *ANG_Y* and around the Z axis by *ANG_Z*; and finally the position vector is rotated around the X axis by *ANG_X*, around the Y axis by *ANG_Y* and around the Z axis by *ANG_Z*.

An alternative way to provide the initial rotation and displacement is by setting the transformation. The transformations are set with the parameter:

```
/gamos/setParam RTGeneratorPhaseSpace:Transformations TYPE_1 VAL1_1 VAL2_1 VAL3_1 TYPE_2 VAL1_2 VAL2_2 VAL3_2
```

Several transformations can be set at the same type with this parameter, and they will be executed in the order provided. For each transformation four parameters have to be supplied: *transformation type, first value, second value, third value*.

Three types of transformation are supported:

- *Displacement (D)*: the three values are the displacement in X, Y and Z
- *Rotations around XYZ (RXYZ)*: the three values are the angles of rotation around the X, Y and Z axis (always executed in this order)
- *Rotation around accelerator axis (RTPS)*: the three values correspond to the angles of rotation in theta, phi and around the accelerator axis itself (in other words, a rotation around the Z axis is done with angle VAL3, then a rotation around Y with angle VAL1 and finally a rotation around Z with angle VAL2)

If the number of phase space particles is not enough to calculate the dose in the phantom with enough precision you may reuse the particles several times by setting the parameter

```
/gamos/setParam RTGeneratorPhaseSpace:MaxNReuse N
```

to a value bigger than 1. In fact, if this parameter is not explicitly set to 1, GAMOS calculates it automatically by dividing the number of events asked for by the number of events in the input phase space.

Alternatively you may recycle the full phase space several times (i.e. when all particles in the phase space are read, the file is closed and restarted again). The number of times a phase space is recycled is controlled by the parameter

```
/gamos/setParam RTGeneratorPhaseSpace:MaxNRecycle N
```

which must take a value bigger than 1. If no reusing is explicitly set, the number of reuses will be automatically calculated as mentioned above and the number of recyclings will be set to 1, so that no recycling is done.

Caution should be taken when recycling phase spaces, as the error correlations due to the reusing of the same particles is not taken account. Therefore you should first consider reusing instead of recycling.

If you think your phase space has X/Y symmetry you may reduce the correlations due to reusing the same particle several times by mirroring your particles each time they are reused, by setting the parameter

```
/gamos/setParam RTGeneratorPhaseSpace:MirrorWhenReuse OPT
```

where *OPT* can be X or Y or XY. If it is X it means that when a particle is used an even time the X original value of position and momentum direction will be changed sign, while when it is an uneven time it will not be changed. If it is

Y the same but mirroring in Y. And if it XY the second time a particle is used the X values will be changed sign, the third time the Y values will be changed sign, the fourth time both X and Y values will be changed sign, the fifth time it will be no change, and the cycles restarts.

When reading a phase space file you may skip the first N events by using the parameter

```
/gamos/setParam RTGeneratorPhaseSpace:NEventsSkip N
```

You can produce the same histograms that were produced when writing the phase space file by setting the following parameter

```
/gamos/setParam RTGeneratorPhaseSpace:Histos TRUE
```

It is also possible to read a phase space in EGS format, by using as your primary generator

```
/gamos/generator RTGeneratorPhaseSpace_EGS
```

Adding extra information to a phase space

The IAEA format allows to store two long integers and two floats in the phase space file as extra information. In GAMOS we have extended this functionality by putting the 32+32 bits of the two integers in a continuous stream, that the user can divide in groups of bits of the desired length to store several informations. The user can even add more sets of 32 bits by changing the the following line at *source/RadioTherapy/include/iaeaRecord.hh*:

```
#define NUM_EXTRA_LONG 2
```

**

and recompiling (*cd source/RadioTherapy; make*).

To store some information in this format the user has to instantiate one of the following user actions

- *RTEExtraInfoProviderCrossings*: fills a bit if the track has crossed the corresponding region before reaching the phase space
- *RTEExtraInfoProviderInteractions*: fills a bit if the track has interacted in the corresponding region before reaching the phase space
- *RTEExtraInfoProviderOrigin*: fills a bit if the track has been created in the corresponding region before reaching the phase space

The user may select how many bits each information must occupy by using the GAMOS parameter :

```
/gamos/setParam EXTRA_INFO_NAME:NBits NBITS
```

where *EXTRA_INFO_NAME* is the name of the extra information class (see above). GAMOS will check that the index to be filled by a class is not bigger than the number of bits reserved for it. And it will also check that the total number of bits is not bigger than the available quantity ($32 * \text{NUM_EXTRA_LONG}$).

The order of declaration of the extra information user actions sets the order of bits occupancy. At the end of the job each of these extra information user actions fills a file explaining the information contained in each bit. By default this file is called *RTEExtraInfoProvider.summ*, but the user may change the name of it with the parameter

```
/gamos/setParam RTEExtraInfoProviderLong:FileName FILE_NAME
```

An example of a file is the following one:

```
RTEExtraInfoProviderOrigin INDEX/REGION 0 DefaultRegionForTheWorld
RTEExtraInfoProviderOrigin INDEX/REGION 1 targetReg
RTEExtraInfoProviderOrigin INDEX/REGION 2 collimatorReg
RTEExtraInfoProviderOrigin INDEX/REGION 3 filterReg
RTEExtraInfoProviderOrigin INDEX/REGION 4 monitorReg
```

```
RTEExtraInfoProviderOrigin INDEX/REGION 5 shieldingReg
RTEExtraInfoProviderOrigin INDEX/REGION 6 jawsReg
```

The float extra information providers fill one of the two available words in the order they have been defined.

The user can add more float words by changing the the following line at *source/RadioTherapy/include/iaeaRecord.hh*:

```
#define NUM_EXTRA_FLOAT 2
```

and recompile the GAMOS code after that.

To store some information as float the user has to instantiate one of the following user actions

- *RTEExtraInfoProviderZOrigin*: stores the Z position of the origin of the track
- *RTEExtraInfoProviderZLast*: stores the Z position of the last interaction before reaching the phase space
- *RTEExtraInfoProviderZLastNoMaterial*: stores the Z position of the last interaction before reaching the phase space, but you may define a list of materials that will not be taken into account. by default this list is *G4_AIR*, but it can be changed with the parameter */gamos/setParam RTEExtraInfoProviderZLastNoMaterial:Materials MATERIAL_1 MATERIAL_2 ...*

At the end of the job each of these extra information user actions fills a file explaining the information contained in each float. By default this file is called *RTEExtraInfoProvider.summ*, but the user may change the name of it with the parameter

```
/gamos/setParam RTEExtraInfoProviderFloat:FileName FILE_NAME
```

If you have written a phase space file in a 32-bit machine and try to read it in a 64-bit one, you will not be able to recover the numbers properly. You may avoid it with the parameter

```
/gamos/setParam IAEARecord:ExtraLongSize 32
```

Similarly if you wrote the file in a 64-bit machine and try to read it in a 32-bit one.

Filtering and classifying on extra information

It is possible to define a filter and a classifier (see sections on *Filters and Classifiers*) based on the long extra information.

The filter name is *RTEILongFilter*. When it is defined it must receive as extra argument the list of extra info values that are accepted:

```
/gamos/filter MYFILTER RTEILongFilter VALUE_1 VALUE_2 ...
```

The classifier is named *RTClassifierByEILong*. It will read the names of the indices of the extra long information from a file named *RTEExtraInfoProvider.summ*. The name of this file can be changed with the parameter:

```
/gamos/setParam CLASSIFIER_NAME:FileName VALUE
```

Reusing a particle at a phase space without filling the phase space file

It is common in radiotherapy treatment simulations that in a first job a phase space file is created and in a second job the particles therein are read and reused several times to calculate the dose in the patient. GAMOS provides the possibility of reusing particles in a phase space file without having to divide the jobs in two. To use this utility the user has to instantiate the user action

```
/gamos/userAction RTReuseAtZPlaneUA
```

and then use the command

/gamos/RT/ReuseAtZPlane

and before the user has to define the z value of the plane where particles will be replicated, with the parameter

/gamos/setParam RTReuseAtZPlane:ZReusePlane Z_POS

and the number of times particles will be reused (indeed the particles is copied NREUSE-1 times)

/gamos/setParam RTReuseAtZPlane:NReuse NREUSE

The user may independently create the phase space file or not at the same Z position or at others.

An important thing to take into account is that the particle is copied from the position of the PreStepPoint. A possible option would be to extrapolate it to the Z plane and copied it at that position, but if the extrapolation is done with a straight line, it would not take into account the straggling of charged particles.

If you think your phase space has X/Y symmetry you may reduce the correlations due to reusing the same particle several times by mirroring your particles each time they are reused, by setting the parameter

/gamos/setParam RTReuseAtZPlane:MirrorWhenReuse OPT

where *OPT* can be *X* or *Y* or *XY*. If it is *X* it means that when a particle is used an even time the *X* original value of position and momentum direction will be changed sign, while when it is an uneven time it will not be changed. If it is *Y* the same but mirroring in *Y*. And if it *XY* the second time a particle is used the *X* values will be changed sign, the third time the *Y* values will be changed sign, the fourth time both *X* and *Y* values will be changed sign, the fifth time it will be no change, and the cycles restarts.

Optimisation of a linac simulation

To optimise your simulation in GEANT4 you may tune the physics parameters (production cuts, special cuts, multiple scattering options, ...) as well as try some reduction variance techniques.

Cuts optimisation

Please read the sections on automatic optimisation of production cuts and user limits for accelerator and dose calculation simulations.

Electromagnetic parameters optimisation

There are many parameters that control the speed of the electromagnetic physics vs its precision. Please see the web page <http://fismed.ciemat.es/GAMOS/RToptim> to get a list of the best electromagnetic physics parameters that can optimise your simulation.

Particle splitting

Particle splitting is a non-biased variance reduction technique that may reduce the CPU time of your accelerator simulation by a big factor. It basically consists on splitting the secondary particles (in radiotherapy mainly gammas generated by bremsstrahlung) *N* times, so that each time a bremsstrahlung gamma is created, another *N-1* gammas are created at the same position, with weight $1/N$, re-sampling the energy and/or angle distribution. As most of the particles that reach a patient in a radiotherapy accelerator are bremsstrahlung gammas, by using this technique you can spare the time spent simulating the original electron and also the time spent tracking gammas that have small possibility of reaching the patient.

There are three splitting techniques implemented in GAMOS:

- Uniform bremsstrahlung splitting
- Directional bremsstrahlung splitting
- Equal weight particle splitting

The three techniques are explained in the sections on *Biasing operations* in the chapter on *Variance reduction techniques*.

Killing particles at big X/Y

This utility serves to kill the particles that would probably not reach your detector because they have too big X and Y positions (it is assumed that that your detector is described along Z and that your initial particles move in this direction in the positive sense).

It makes a list of the volumes that are placed directly on the world volume and computes the minimum and maximum extension in X and Y (using the method *G4VSolid::GetExtent()*). This rectangular area extends from the minimum Z value of this volume until the minimum Z value of the next volume, so that all tracks that are outside it will be killed. This utility is activated by selecting the user action

```
/gamos/userAction RTZRLimitsAutoUA
```

You may argue that too many particles (or too few) are killed with this automatic definition of limits. You may simply changed the dimensions of the volumes placed at world (adding container volumes made of air will not change your physics). or you can define the limits your self by selecting the user action

```
/gamos/userAction RTZRLimitsUA
```

Then you have to write a file named *xylimits.dat* with the list of values you want to use. The format of that file should be the following: a set of lines with three numbers representing the Z value of the plane and then the X and Y limits. The planes will be extended in Z until the previously defined Z (for the minimum Z defined the world negative Z limit will be used).

If you want to change the name of the limits file, you can do it with the command (remember to define a parameter always before selecting the user action)

```
/gamos/setParam RTZRLimitsUA:FileName MY_FILENAME
```

The user has also the option that particles rejected are not killed, but Russian Roulette is played with them and only if they are rejected they are killed. If they are accepted their weight will be increased by the corresponding value. To use this option the following parameter must be set:

```
/gamos/setParam RTVZRLimitsUA:UseRussianRoulette TRUE
```

The value of the Russian roulette threshold is set with the parameter

```
/gamos/setParam RTVZRLimitsUA:RussianRouletteValue VALUE
```

which by default takes a value of 100.

Scoring dose in phantom

To use a phantom geometry you can use the GAMOS utilities to read phantom geometries in EGS or GEANT4 formats or build simple phantoms with a few user commands. The scoring of the dose in the phantom volumes can be done using the scorer *GmG4PSDoseDeposit* and selecting as detector the voxels, that are named *patient*. For example you can use the following commands:

```
/gamos/scoring/createMFDetector DoseDet patient
```

```
/gamos/scoring/addScorer2MFD DoseScorer GmG4PSDoseDeposit DoseDet
```

This is all you need to get on the standard output the dose by event deposited in each voxel. When you use a phase space as input the number of events is not the number of events run in your job (what would equal be the number of particles in the phase space multiplied by the number of times each particles is reused), but GAMOS uses as number of events the original number of events that were used to generate the phase space. In fact, GAMOS gets the ratio of particles written in the phase space per original event transported through the accelerator and multiplies this ratio by the number of phase space particles used. This allows to get the best approximation to the correct number of events when the phase space is not used fully, or when particles are reused (it is indeed not the exact number if for example you use only the 10 first particles in the phase space: the number of events that generated those 10 particles may be bigger or smaller than the number of events that generated the following 10 particles, due to statistical fluctuations).

If you are using the option of skipping voxel frontiers when the material does not change (see section on *Reading DICOM files*) you may take into account the possibility of tuning the distributions of dose deposited in the voxels traversed by one step (see section on *Scoring in voxelised phantoms*).

Saving scores and scores squared in binary file

The second available file is a binary file where it is written the dose and dose squared in each voxel. The binary format allows for a faster writing and reading and the fact of writing the dose squared instead of the error serves to calculate in a proper way the error correlations when the doses from several jobs are added. To obtain it you just have to add the scorer printer *GmPSPrinterSqdose* to your scorer, for example

```
/gamos/scoring/addPrinter2Scorer GmPSPrinterSqdose DoseScorer
```

The output file is named by default *sqdose.out*. You may change it with the parameter

```
/gamos/setParam PRINTERNAME_SCORERNAME:FileName MY_FILENAME
```

where **PRINTERNAME** is the name of the printer, by default *GmPSPrinterSqdose*, and *SCORERNAME* is the name you have given to the scorer.

All variables are of type *float* and the format is the following:

- Number of events (original number of events used to generate a phase space file is phase space file is used as primary generator)
- Number of voxels in the X, Y and Z directions (e.g. *n_x*, *n_y*, *n_z*)
- Array of voxel boundaries (cm) in the X direction (*n_x*+1 values)
- Array of voxel boundaries (cm) in the Y direction (*n_y*+1 values)
- Array of voxel boundaries (cm) in the Z direction (*n_z*+1 values)
- Array of dose values (Gy/cm³) (*n_x n_y n_z* values)
- Array of dose squared values ((Gy/cm³)²) (*n_x n_y n_z* values)

To optimise the disk space and the memory needed to read the file back, it is possible to only store the dose and dose squares of those voxels that are filled. This can be done by setting the parameter:

```
/gamos/setParam GmSqdose:FileType FILLED
```

The default value of this parameter is *ALL*.

Saving scores and score errors in text file

You can also store the dose in each voxel in a file, what allows to calculate the average dose calculated with several jobs (see dose analysis section). Nevertheless we recommend you to use the previous format instead of this one, as it is faster for writing and reading.

The first file is a text file where it is written the dose and dose error in each voxel. To obtain it you just have to add the scorer printer *GmPSPrinter3ddose* to your scorer, for example

```
/gamos/scoring/addPrinter2Scorer GmPSPrinter3ddose DoseScorer
```

The output file is named by default *3ddose.out*. You may change it with the parameter

```
/gamos/setParam PRINTERNAME_SCORERNAME:FileName MY_FILENAME
```

where **PRINTERNAME** is the name of the printer, by default *GmPSPrinter3ddose*, and *SCORERNAME* is the name you have given to the scorer.

The format is the same as the one 3ddose format used in DOSXYZnrc, except that the first line contains the number of events:

- Number of voxels in the X, Y and Z directions (e.g. *n_x, n_y, n_z*)
- Array of voxel boundaries (cm) in the X direction (*n_x+1* values)
- Array of voxel boundaries (cm) in the Y direction (*n_y+1* values)
- Array of voxel boundaries (cm) in the Z direction (*n_z+1* values)
- Array of dose values (Gy/cm³) (*n_x n_y n_z* values)
- Array of dose relative error values (*n_x n_y n_z* values)

Saving scores in histograms

If you want to store the scores in the phantom in an histogram file, you can use the scorer printer *RTPSPDoseHistos*

```
/gamos/scoring/addPrinter2Scorer RTPSPDoseHistos DoseScorer
```

The name of the histogram file is by default *dose_PRINTER_NAME*, but it may be changed with the parameter

```
/gamos/setParam PRINTER_NAME:FileName VALUE
```

The number of bins in the histograms will be equal to the number of voxels in the phantom (*nVoxel*) and the histogram limits are the $-nVoxel/2$ to $nVoxel/2$.

The name of all these histograms start with *RTPSPDoseHistos*: and they are all dumped into the file *dose.root/csv*.

The following histograms are produced by default:

- Dose in X direction (“Dose Profile X_merged”)
- Dose in Y direction (“Dose Profile Y_merged”)
- Dose in Z direction (“Dose Profile Z_merged”)
- Dose in XY direction (“Dose XY_merged”)
- Dose in XZ direction (“Dose XZ_merged”)
- Dose in YZ direction (“Dose YZ_merged”)
- Dose of each voxel (“Dose”)
- Integrated dose of each voxel, i.e. all the voxels that have dose equal or greater than a given bin fill that bin (“Dose-volume”)

The word *merged* refers to the fact that this histograms count the dose in one direction (or two) integrating the dose in all the voxels in the other two (or one) direction.

If you want to obtain other histograms out of the dose in the voxels, you may list what you want in a file in the following way: first give the name of your file with the parameter:

`/gamos/setParam RTPSPDoseHistos:HistosFileName FILE_NAME`

In this file you have to fill a line for each histogram you want with the following information:

- Histogram type: it must be 1X, 1Y, 1Z, 2XY, 2XZ or 2YZ
- Histogram name
- Minimum voxel in X
- Maximum voxel in X
- Minimum voxel in Y
- Maximum voxel in Y
- Minimum voxel in Z
- Maximum voxel in Z

For the minimum and maximum voxel value you may use the total number of voxels by writing one of the words *NVoxelX*, *NVoxelY* or *NVoxelZ*. Also arithmetic expressions are comment (starting the line with // are allowed. One example file can be the following (for a phantom with number of voxels 101 101 100):

```
// recovering the 1D merged histograms
1X "RTPSPDoseHistos:Dose Profile X_merged" 0 NVoxelX-1 0 NVoxelY-1 0 NVoxelZ-1
1Y "RTPSPDoseHistos:Dose Profile Y_merged" 0 NVoxelX-1 0 NVoxelY-1 0 NVoxelZ-1
1Z "RTPSPDoseHistos:Dose Profile Z_merged" 0 NVoxelX-1 0 NVoxelY-1 0 NVoxelZ-1
// Y profiles at several depths
1Y "RTPSPDoseHistos:Dose Profile Y 1.400 cm" 50 50 0 NVoxelY-1 6 6
1Y "RTPSPDoseHistos:Dose Profile Y 10.000 cm" 50 50 0 NVoxelY-1 49 49
// PDDs merging voxels
1Z "RTPSPDoseHistos:Dose Profile Z 1x1" 50 50 50 50 0 NVoxelZ-1
1Z "RTPSPDoseHistos:Dose Profile Z 3x3" 49 51 49 51 0 NVoxelZ-1
```

It is also possible to produce in a simple way a full set of 2-dimensional plots covering a whole phantom in any of the three dimensions. To do one must use the following parameter:

`/gamos/setParam RTPSPDoseHistos:AllHistos2D 2DIM_2 2DIM_2 ..`

The 2DIM_i values can be *XY*, *XZ* or *YZ* (you can use one of several of these three in your command). To visualise them in a simple way you may use the *printAll.C* (see Appendix) ROOT utility to convert all the histograms in graphics files.

Analysis utilities

This is a set of utilities that may serve in the analysis of phase space and dose files, for example to sum phase space or dose files from different jobs, to get basic information of the file contents, to fill histograms out of the files or to compare files from two jobs.

All these utilities are under the directory *analysis*. They are compiled by default with the rest of GAMOS code and after that they are available as executables as mentioned in the following subsections.

Summing phase space files

This utility serves to sum phase space files corresponding to different jobs with the same setup. To use it you have to write a file containing the list of header phase space files, one file per line, for example


```
ps.20000.IAEAheader
ps.20001.IAEAheader
ps.20002.IAEAheader
```

Then you just have to run the executable

```
sumPS INPUT_FILE_LIST_NAME OUTPUT_FILE_NAME
```

where *INPUT_FILE_LIST_NAME* is the name of the file containing the list of files to add and *OUTPUT_FILE_NAME* is the name of the output file that will contain the sum of all the files (two files indeed as usual for IAEA phase space files: *OUTPUT_FILE_NAME.IAEAheader* and *OUTPUT_FILE_NAME.IEAephsp*).

When running you will see on the screen something similar to this:

```
.. COMMENT: \\tiny
Opening phase space contained in      ps.20000.IAEAheader of type IAEA
PARTICLES 225437 NPART_TOT 225437 NPARTORIG_TOT 5000000
RATIO 0.0450874 +- 0.000101661 RATIO_TOT 0.0450874
Opening phase space contained in      ps.20001.IAEAheader of type IAEA
PARTICLES 224635 NPART_TOT 450072 NPARTORIG_TOT 10000000
RATIO 0.044927 +- 0.000101455 RATIO_TOT 0.0450072
Opening phase space contained in      ps.20002.IAEAheader of type IAEA
PARTICLES 224813 NPART_TOT 674885 NPARTORIG_TOT 15000000
RATIO 0.0449626 +- 0.000101501 RATIO_TOT 0.0449923
* * * * N Particles      = 674885
* * * * N Photons       = 673804
* * * * N Electrons     = 1057
* * * * N Positrons     = 24
* * * * N Original Histories = 1.5e+07
```

For each phase space files after the name of the file comes a line with the file statistics: number of particles, accumulated number of particles of all files, accumulated number of original histories, ratio particles/original histories +- ratio error, accumulated ratio particles/original histories. And at the end the statistics on the total number of particles in total, photons, electron and positrons, and the total number of original histories in all summed files.

Making histograms out of a phase space file

You can make histograms out of the particles contained in a phase space file by running gamos with the input script that you can find in *RadioTherapy/analysis/phaseSpace/analysePS/rt.analysePS.in*. If you edit it and change the name of the input phase space file, at */gamos/setParam RTGeneratorPhaseSpace:FileName*, and run

```
gamos rt.analysePS.in
```

You will get a file named *phaseSpace.root*, that contains the same histograms that you get when you run the job to write the phase space file (see section on *Phase space histograms*).

Alternatively you can directly analyse a phase space file with the executable *analysePS*. You just have to type in your shell:

```
analysePS FILE_NAME
```

where *FILE_NAME* is the name of the phase space header file (the one that ends with *.IAEAheader*; you may indeed omit the suffix *.IAEAheader* if you want).

When running you will see on the screen something similar to this:

```
.. COMMENT: \\tiny
READING FILE ps.ubs.10x10.6
ORIGINAL HISTORIES= 2000000
N PARTICLES= 5.13303e+07 PER ORIG_HIST= 25.6651 +- 0.0184982
```



```
N GAMMAS= 5.11882e+07 PER ORIG_HIST= 25.5941 +- 0.0184479
N ELECTRONS= 137257 PER ORIG_HIST 0.0686285 +- 0.000191492
N POSITRONS= 4839 PER ORIG_HIST= 0.0024195 +- 3.48235e-05
Reading PHSP track 0
...
Reading PHSP track 51000000
=== saving histograms in file === phaseSpace.root
```

The output contains the total number of original histories and after the number of particles, gammas, electrons and positrons in the phase space file, and these numbers divided by the number of original histories, with the error. Also an histogram file equal to the file that was created when producing the phase space file will be created.

Several arguments can be supplied to the executable in the standard Unix format:

- **-f** phase space file name (in this case do not use the file name alone as first argument as before)
- **-NRead** number of particles to be read from the phase space file
- **-fOut** output file name
- **-EMax** maximum limit of the energy histograms
- **-RMax** maximum limit of the position histograms
- **-NBins** number of bins of histograms
- **-verb** verbosity: it sets the RTVerbosity. Default is *warning*, that will print the above lines; *debug*, that will print each particle read from the phase space file
- **-help** prints the set of arguments

Merging 'sqdose' files

This utility serves to merge dose files in the format *sqdose* corresponding to different jobs with the same setup, and getting the average dose of them. To use it you have to write a file containing the list of header phase space files, one file per line, for example

```
sqddose.water.20000.out
sqdose.water.20001.out
sqdose.water.20002.out
```

Then you just have to run the executable

```
mergeSqdose INPUT_FILE_LIST_NAME OUTPUT_FILE_NAME
```

where *INPUT_FILE_LIST_NAME* is the name of the file containing the list of files to merge and *OUTPUT_FILE_NAME* is the name of the output file that will contain the merging of all the files.

When merging files it will be checked that they correspond to the same phantom by checking the number of voxels and voxel limits.

Merging '3ddose' files

This utility serves to merge dose files in the format *3ddose* corresponding to different jobs with the same setup, and getting the average dose of them. To use it you have to write a file containing the list of header phase space files, one file per line, for example

```
3ddose.water.20000.out
3ddose.water.20001.out
3ddose.water.20002.out
```

Then you just have to run the executable

```
merge3ddose INPUT_FILE_LIST_NAME OUTPUT_FILE_NAME
```

where *INPUT_FILE_LIST_NAME* is the name of the file containing the list of files to merge and *OUTPUT_FILE_NAME* is the name of the output file that will contain the merging of all the files.

When merging files it will be checked that they correspond to the same phantom by checking the number of voxels and voxel limits.

Making histograms out of a 'sqdose' file

You can make histograms out of dose information contained in a *sqdose* file by running

```
analyseSqdose SQDOSE_FILE_NAME
```

When running you will see on the screen something similar to this:

```
READING FILE sqdose.ubs.10x10.6.out
GmSqdoseHeader::Read NEvent 2.5e+08
GmSqdoseHeader::Read NVoxels 21 112 150
GmSqdose::Read type 1
USING std::map to store doses
Number of voxels= 352800
RTPSPDoseHistos nvoxel 21 112 150
RTPSPDoseHistos dim 1 1 1
RTPSPDoseHistos translation (0,0,-4)
RTPSPDoseHistos rotation
[ (      1      0      0)
  (      0      1      0)
  (      0      0      1) ]
MINIMUM DOSE 2.352e-15
MAXIMUM DOSE 4.76001e-13
RTPSPDoseHistos AVERAGE ERROR 20% = 0.0139839
RTPSPDoseHistos AVERAGE ERROR 50% = 0.0111726
RTPSPDoseHistos AVERAGE ERROR 90% = 0.00996564
=== saving histograms in file === dose_analyseSqdose.root
N EVENTS IN SOURCE 2.5e+08
```

First it is printed the *sqdose* file header information: number of events (original events run) and number of voxels in X, Y and Z. Then the type of the *sqdose* (see above). Then the STL container that will be used to store the doses and dose errors (see argument descriptions below). Then the total number of voxels to be read and the information from the class *RTPSPDoseHistos*: number of voxels in X, Y and Z, voxel dimensions in X, Y and Z, translation and rotation applied to phantom when dose file was written and the statistics, i.e. minimum and maximum voxel dose, and average dose error in 20%, 50% and 90% highest dose voxels.

You will also get a file named *dose_analyseSqdose.root*, that contains the same histograms that you get when you run the job to write the dose file using the scorer printer *RTPSPDoseHistos*.

Several arguments can be supplied to the executable in the standard Unix format:

- **-f** phase space file name (in this case do not use the file name alone as first argument as before)
- **-NRead** number of particles to be read from the phase space file

- **-fOut** output file name
- **-fHistos** name of file with list of histograms
- **-DoseMin** minimum limit of dose histograms
- **-DoseMax** maximum limit of the dose histograms
- **-cont** type of the STL container that will be used to store the doses and dose errors. It can be *MAP* or *map*, that uses a `std::map`; it is the default one, the one used by the Geant4 scorers.

It can also be *VECTOR* or *vector*, that uses a `std::vector`; it occupies about ten times less than the `std::map`. The `std::map` container occupies a big amount of space, about 500 Mb for 10 million voxels, so we recommend you that you use `std::vector` if your phantom is big

- **-NVoxels** total number of voxels in phantom (argument needed if sqdose file is of type *FILLED*)
- **-verb** verbosity: it sets the RTVerbosity. Default is *warning*, that will print the above lines; *debug*, that will print each particle read from the phase space file
- **-help** prints the set of arguments

Automatic determination of production cuts for an accelerator simulation

The method used in GAMOS to determine the best production cuts is based on what we can call an ‘inverse reasoning’. We count each particle that reaches a given Z plane (corresponding to the phantom surface) and we calculate first the range of the particle in the region where it is created. Then we can know that if we put a range cut in that region smaller than the calculated range, that particle would not reach our target plane. We also compute the range of the mother particle in the region where it was created and the same consecutively for all the ancestors. We know then that if we set in any of the regions where each of the ancestor particles are created a cut smaller than the corresponding range, we would stop the chain of particles and therefore we would have no particle in the target plane. After running a good number of tracks we can know for each particle type and for each region which is the biggest range we can put if we do not want to lose any particle. Indeed we may allow to lose a small amount of particles if this speeds up our simulation. To know easily which is the biggest cut you can use to lose less than a given percentage of particles, GAMOS provides a set of plots (one per each particle type and per each region) and a simple script to get automatically the cut values.

One warning is due here: as mentioned above when a track reaches the target, its range fills a histogram, but also the range of all the ancestors of this track. It may happen then that when you set a certain cut and the abovementioned script gives you how many tracks would be killed, more than one killed track correspond to the same track reaching the target (i.e., with a cut you kill the track that reaches the target and the parent track). Therefore you might have an overcounting of the number of tracks killed by a cut. To avoid this the total number of tracks (the last lines of output) is not computed as the sum of tracks in the region. This number uses a histogram that contains only one entry per track reaching the target, the one corresponding to the track with the smallest range. If you want to set a different cut for each region and are worried for this double counting, you may have a look at the histogram named “trackInfos per Track in target”, that plots per each track reaching the target how many track informations are kept in the histograms. Another useful histogram for this case may be the 2D histogram “trackInfo Region vs trackInfo Region”, that plots all the region number of all the pairs of track informations that correspond to the same track reaching the target (you can get a list of which region number corresponds to which region at the end of the standard output file).

To use this utility in GAMOS it is only needed to add this command in your script:

```
/gamos/userAction GmProdCutsStudyUA RTCutsStudyFilter
```

or that will use as target condition that a track reaches a plane perpendicular to the Z axis defined with the parameters

```
/gamos/setParam RTCutsStudyFilter:PlaneZ ZPOS
```

```
/gamos/setParam RTCutsStudyFilter:PlaneXDim XDIM
```

```
/gamos/setParam RTCutsStudyFilter:PlaneYDim YDIM
```

This command will produce at the end of run a table and a histogram file with the needed information. The table will contain the minimum range that can be applied for each region/particle/process not to lose any track reaching the target, and it will look like this

```
##### PRODUCTION CUTS STUDY RESULTS
GmProdCutsStudyUA: REGION= DefaultRegionForTheWorld PARTICLE=
gamma PROCESS= ALL MIN RANGE= 353161.38
GmProdCutsStudyUA: REGION= DefaultRegionForTheWorld PARTICLE=
gamma PROCESS= eBrem MIN RANGE= 353161.38
```

To get the cuts values for not losing a given percentage of particles in the target plane you can execute the ROOT script that can be found at `GamosCore/GamosPhysics/GamosCuts/getProdCutsEffect.C` :

```
*root -b -p -q .x getProdCutsEffect.C++\\(\\ "prodcuts.root\\",percentage\\) *
```

and look at the last lines of output, those that contain the word 'FINAL', like the following ones

```
PARTICLE= e+ FINAL= 17 / 19
PARTICLE= e- FINAL= 72 / 34185
PARTICLE= gamma FINAL= 72 / 34184
```

Automatic determination of production cuts for a dose in a phantom simulation

The use of production cuts in a dose computation may introduce a bias when a particle is killed (and then its energy is deposited locally) and it has enough energy to reach the next phantom voxel, or enough energy to create a particle that reaches the next phantom voxel (this happens mainly for electrons creating gammas, which have a much higher range).

To calculate automatically the best production cut, that is the one that gives the smallest CPU while biasing the dose computation a minimal amount, GAMOS uses an inverse reasoning. For a given set of cuts for electron and gamma it does not apply them but tags the particles that would have been killed by them. It also tags the voxel in which the particle is produced and then it computes all the dose deposited by the tagged particle or any of its children in a voxel that is not the same as the tagged voxel.

To use this utility in GAMOS you just have to associate to your dose scorer a filter of type `GmProdCutOutsideVoxelFilter`, passing to it as arguments the gamma cut and the electron cut, like in the following example

```
/gamos/scoring/addFilter2Scorer ProdCutFilter GmProdCutOutsideVoxelFilter PDDscorerPC10.1. 10.*mm 1.*mm
```

You should add another scorer without filter to get the total dose. After running your job with as many scorer-filter combinations as you like, you can look at the total dose deposited by each scorer and compare it with the total dose. It may happen that the dose lost with certain cuts, despite being a small proportion of the total dose, is distributed in a different manner than the total dose, introducing some bias in some region that you consider not acceptable. To check in detail the dose produced with a certain filter you can add a scorer printer of type `RTPSPDoseHistos`, that will produce several histograms of the dose (PDD, X & Y profiles, dose, dose-volume):

```
/gamos/scoring/addPrinter2Scorer RTPSPDoseHistos DoseScorerPC10.1.
```

The name of the printer will be passed to the name of the file containing the histograms.

Automatic determination of user limits for an accelerator simulation

The method used in GAMOS to determine the minimum range user limits is similar to the one used to determine the best production cuts. The main difference is that when a track reaches the target we do not have to look at the range it

had when created, but at the range it had in every step. This is because even if we want the minimum step, the track may have crossed several regions and the smallest range may not correspond to the last step. What we do nevertheless is only consider the last step when there are a set of contiguous steps in the same region. Also for the ancestor tracks we have to store the information of each step, starting of course with the one when the track that reached the target (or its n-th ancestor if we are looking at the (n+1)-th ancestor) was created.

The same warning as for the production cuts should be mentioned here, but in this case it is more than a mere warning: when a track reaches the target, we accumulate one-track information of the last step in each region, for each of the ancestor tracks. Therefore it is very likely that there are more than one track information per track reaching the target, and therefore there will be overcounting of the number of tracks killed by a cut. As for the production cuts you should keep an eye on this.

To use this utility in GAMOS it is only needed to add the command in your script:

```
/gamos/userAction GmMinRangeLimitsStudyUA RTCutsStudyFilter
```

what will produce at the end of run a table and a histogram file with the needed information.

```
root -b -p -q .x getMinRangeCutsEffect.C++\\(\\\"prodcuts.root\\\",percentage\\) \\& tee out
```

and look at the last lines of output, those that contain the word 'FINAL', like the following ones

```
PARTICLE= e+ FINAL= 17 / 19
PARTICLE= e- FINAL= 72 / 34185
PARTICLE= gamma FINAL= 72 / 34184
```

Automatic determination of user limits for a dose in phantom simulation

The method used in GAMOS to determine the minimum range user limits is similar to the one used to determine the best production cut.

To use this utility in GAMOS you just have to associate to your dose scorer a filter of type `GmProdCutOutsideVoxelFilter`, passing to it as arguments the gamma cut and the electron cut, like in the following example

```
/gamos/filter ProdCutFilter GmMinRangeCutOutsideVoxelFilter 10.*mm 1.*mm
```

```
/gamos/scoring/addFilter2Scorer ProdCutFilter DoseScorerPC10.1.
```

After running your job with as many scorer-filter combinations as desired, you can look at the total dose deposited and compare it with the dose with very small cuts. It may happen that the dose with certain cuts, despite being a small number, is distributed in a different manner than with very small cuts, introducing some bias that you consider not acceptable. To check in detail the dose produced with a certain filter you can add a scorer printer of type `RTPSPDoseHistos`, that will produce several histograms of the dose (PDD, X & Y profiles, dose, dose-volume):

```
/gamos/scoring/addPrinter2Scorer RTPSPDoseHistos DoseScorerPC10.1.
```

The name of the printer will be passed to the name of the file containing the histograms.

SHIELDING APPLICATION

The shielding application contains a set of utilities that may be useful for shielding calculations.

Studying penetration

You may get the Nth value layer, i.e. the layer width to get reduction of 1/N in the number of original particles, of any particle in any material with the user action:

/gamos/userAction SHNthValueLayerUA

To use it you may follow the example in *MY_GAMOS_DIR/tutorials/ShieldingTutorial/exercise1/exercise1a*. As you can see there, a beam of particles should be set along the Z direction into a geometry made of a slabs stacked along Z which are very wide in X and Y dimensions. *SHNthValueLayerUA* counts the number of particles that exit each slab and, at the end of run, prints these numbers divided by the total number of events, in a format similar to the following one:

```
%%% SHNthValueLayerUA_NSTEPS: shield:0 = 0.96763
%%% SHNthValueLayerUA_NSTEPS: shield:1 = 0.92561
%%% SHNthValueLayerUA_NSTEPS: shield:2 = 0.88194
%%% SHNthValueLayerUA_NSTEPS: shield:3 = 0.83882
%%% SHNthValueLayerUA_NSTEPS: shield:4 = 0.79542
...
```

At the end it makes an exponential fit to the histogram of the number of particles exiting the slabs as a function of the penetration

The fit is done using ROOT utilities, so you must install GAMOS with ROOT to get this result, and the result is printed in a format similar to the following:

```
FCN=6713.22 FROM MIGRAD      STATUS=CONVERGED      51 CALLS
52 TOTAL
EDM=6.48994e-08      STRATEGY= 1      ERROR MATRIX ACCURATE
EXT PARAMETER                                STEP      FIRST
NO.   NAME      VALUE      ERROR      SIZE      DERIVATIVE
1   Constant    1.16833e+01  1.02026e-03  2.99800e-05  4.71215e-01
2   Slope      -6.40772e-02  4.29841e-05  1.26302e-06  5.92494e+00
%%% SHNthValueLayerUA_exitShieldLayerF_copyNoC: gamma G4_Pb 1 MeV :
FIT chi2= 6713.22 nDoF 162
```

From the fitted function, it calculates the width corresponding to each of the reduction factors in the list given by the parameter:

/gamos/setParam USER_ACTION_NAME:Reductions VALUE_1 VALUE_2 ...

This width is also calculated interpolating, with a lin-log interpolation, in the list of values of number of particles exiting each slab: The two widths are printed in a format similar to the following:

```
%%% SHNthValueLayerUA_exitShieldLayerF_copyNoC: gamma G4_Pb 1 MeV :  
REDUCTION FROM FIT= 2 reached at depth= 10.8174 +- 0.00725647 mm  
%%% SHNthValueLayerUA_exitShieldLayerF_copyNoC: gamma G4_Pb 1 MeV :  
REDUCTION FROM SCORES= 2 reached at depth= 12.0863 mm  
%%% SHNthValueLayerUA_exitShieldLayerF_copyNoC: gamma G4_Pb 1 MeV :  
REDUCTION FROM FIT= 10 reached at depth= 35.9345 +- 0.0241055 mm  
%%% SHNthValueLayerUA_exitShieldLayerF_copyNoC: gamma G4_Pb 1 MeV :  
REDUCTION FROM SCORES= 10 reached at depth= 36.864 mm
```

From looking in detail to the example in you can find some important features that you must follow:

- The stacked slab solid must be a box and must be placed with a parameterisation along the Z axis. *SHNthValueLayerUA* will look for a parameterised volume and from it it will extract the layer width as the Z box length. If not found, you have to define this layer width with the parameter: */gamos/setParam USER_ACTION_NAME:LayerWidth VALUE* Also the material name to be printed in the final report will be extracted from the parameterised volume. If not found, you have to define this material name the parameter: */gamos/setParam USER_ACTION_NAME:MaterialName NAME*
- The generator direction distribution should be along the Z axis, unless you want to study some special behaviour.
- The secondary particles should be stopped before being tracked, because if not they will be counted as particle exiting the shield layers. You can do it with the command: */gamos/userAction GmKillAtStackingActionUA GmSecondaryFilter*
- *SHNthValueLayerUA* should be accompanied by a filter and a classifier. The filter should be on the particles exiting the slab volume, for example: */gamos/filter exitShieldLayerF GmExitLogicalVolumeFilter shield* The classifier should classify the particles by the copy number of the slab, therefore it should be of the type *GmClassifierBy1Ancestor: /gamos/classifier copyNoC GmClassifierBy1Ancestor 0*

Activation studies

The user action *SHActivationUA* serves to make activation studies. It counts the number of all radioactive isotopes produced in a job and calculates their activity at any given time as given by the user. You can find an example of its use in *MY_GAMOS_DIR/tutorials/SHieldingTutorial/exercise4/exercise4a*.

The first thing you should do is to define the rate of particles generated. You may use for example the time generator distribution

```
/gamos/generator/timeDist GmGenerDistTimeConstantChange TIME_INTERVAL
```

To define when you want the activity to be calculated, you can use the parameter:

```
/gamos/setParam USER_ACTION_NAME:Times TIME_1 TIME_2 TIME_3
```

You may also define the periods when the source of particles is on or off. You can this by listing the period durations, starting at time 0 and for each one set if the source is on (1) or off (0):

```
/gamos/setParam USER_ACTION_NAME:Periods PERIOD_1 1 PERIOD_2 0 PERIOD_3 1 PERIOD_4 0 ...
```

If the period pattern is complicated you may define them in a file name containing a line per period with the duration followed by 1 or 0. The periods file name is set with the parameters:

```
/gamos/setParam USER_ACTION_NAME:PeriodsFileName FILE_NAME
```

If no period is defined, it will be assumed that the source is on all time. Similarly, if the sum of all the period durations is smaller than the time when you want the activity to be calculated, it will be assumed that the during that time difference the source is on.

The output will be similar to the following one:

```

%%%%%%%% SHActivationUA_inWindowF  NUMBER_OF_EVENTS_PER_SECOND= 6.25001e+12 = 1e+06 events in 160 nanos
%%%%%%%% SHActivationUA_inWindowF  PERIODS: 300 1 600 0
%%% SHActivationUA_inWindowF  TIME= 300 sec Sc43[0.0]  ACTIVITY= 920809.7 decays/sec  LIFETIME= 2021
%%% SHActivationUA_inWindowF  TIME= 300 sec Sc44[0.0]  ACTIVITY= 2189774.6 decays/sec  LIFETIME= 203
%%% SHActivationUA_inWindowF  TIME= 300 sec V46[0.0]  ACTIVITY= 2.2500023e+08 decays/sec  LIFETIME= 0
%%%%%%%% SHActivationUA_inWindowF  TIME= 300 sec  TOTAL ACTIVITY= 0
%%% SHActivationUA_inWindowF  TIME= 900 sec Sc43[0.0]  ACTIVITY= 907243.47 decays/sec  LIFETIME= 202
%%% SHActivationUA_inWindowF  TIME= 900 sec Sc44[0.0]  ACTIVITY= 2157807.2 decays/sec  LIFETIME= 203
%%% SHActivationUA_inWindowF  TIME= 900 sec Sc46[0.0]  ACTIVITY= 1795.27 decays/sec  LIFETIME= 10443
%%% SHActivationUA_inWindowF  TIME= 900 sec V46[0.0]  ACTIVITY= 3.5688296e-206 decays/sec  LIFETIME=

```

It contains first a line with the number of events per second, as calculated from the two following numbers: the number of events simulated and the time that correspond to these events. The following line print the periods when the beam is on or off. Then come a set of lines for each time when you requested the activity to be calculated. Each of these lines gives the time value, the isotope chain name, the activity, the lifetime of the last isotope in the chain, the number of atoms of the same type than the last isotope in the chain created in the job and the number of atoms that would be present at the time requested.

Two warnings are due here: the first is that only chains with two decays are considered currently; the second is that GEANT4 does not create metastable isotopes in an interaction, so that the activity of these isotopes will not be taken into account.

Sum activation results and plot them

The command *sumActivation* serves two join results from different runs and to make histograms out of them. The result of this command will be a series of graphics files in GIF format with the curves of isotope activities vs. time, and a file where the activity results from the files to be summed will be written.

Several arguments control its behaviour:

- *-f FILE_NAME*: name of the file with the output of the activation job
- *-fl FILE_LIST_NAME*: name of the file that contains the list of files with output of activation jobs. The files will be merged, recalculating the activity of each isotope as the average of the activities in the outputs. This argument cannot be used with the *-f* argument. The list file has to contain a list of file names, one name per line.
- *-fOut FILE_NAME*: name of the file where the combined results are written. If this argument does not appear, the name *activation.out* is used.
- *-fPeriods PERIODS_FILE_NAME*: name of periods file (when isotope generation is on or off). If used these periods are used instead of those in the output files
- *-fTimes TIMES_FILE_NAME*: name of times file (times when activity will be printed/plotted). If used these periods are used instead of those in the output files
- *-nIsotopesPerPage N*: number of isotopes that will be printed in each plot (each plot becomes a gif file). Isotopes are ordered looking at the highest activity reached at any time and plotted in descending order
- *-timeUnit UNIT*: unit in which times are plotted. The default unit is *second*
- *-hYMin YMIN*: minimum Y value of the histograms
- *-hYMax YMAX*: maximum Y value of the histograms
- *-exemption FILE_NAME*: Draw isotope exemption activity values as a line. The exemption values are read from a file; the file *exemption.EU.lis* contains the exemption file as provided by the European Union
- *-verb VERB_LEVEL*: verbosity: sets the ShieldingVerbosity

- *-h*: print this list of arguments

It is also possible to run *plotActivation* with only one argument: the file name; in this case the default arguments will be used. If it is desired to change an argument, then the file name has to be given preceded by *-f*.

Correcting decay time to take into account isotope activity

By default when a radioactive decay happens Geant4 changes the final step time and the time of the secondary particle sampling the decay time of the isotope that decayed. This behaviour does not take into account the isotope activity, that is what you probably expect. In other words: if you have a sample with an activity for example 1000, you expect that the radioactive decays happen with a separation close to $1./1000$, so that after one second you have 1000 decays. To take the isotope activity into account you have to use:

```
GmNoUseG4RadDecayTimeUA
```

You may control the minimum activity to apply this correction (for example because you want to spare some time for isotopes with very small activity) using the parameter:

```
/gamos/setParam USER_ACTION_NAME:MinimumActivity VALUE, which by default takes a value of 1.E-30 becquerel
```

If you want to postpone the tracking of particles with a big assigned time (because your number of events do not encompass this time), you may use the parameter:

```
/gamos/setParam USER_ACTION_NAME:MaxTracksTime VALUE, which by default takes a value of 1E-8 second
```

Print channel by channel cross sections

You may print the hadronic cross sections with the user action

```
/gamos/userAction SHPrintPHPChannelXSUA
```

If the physics model selected is *neutron_hp* or *particle_hp*, it will also print the channel by channel cross sections, and, if required the particle yields as read from the database.

This user action invokes the GEANT4 internal methods to get the cross sections, so there is no need to simulate an event. Nevertheless, to profit from the GAMOS utilities, you should set up an input file similar to other GAMOS input files. You can see an example on the use of this utility at *MY_GAMOS_DIR/tutorials/SHieldingTutorial/exercise5/exercise5a.in*.

This example prints the cross section of neutrons, as given by the *neutron_hp* or *particle_hp* GEANT4 package, for several energies in several materials. The first thing it does is to replace the geometry *GmGeometryFromText* by *GmGeometryUseMaterials*. This geometry class will generate a world for each of the materials given by the parameter:

```
/gamos/setParam GmGeometryUseMaterials:Materials MATE_1 MATE_2
```

If any of the materials is not a GEANT4 one, you will need a geometry file where the material is defined. The name of this file is given with the parameter:

```
/gamos/setParam GmGeometryUseMaterials:FileName FILE_NAME
```

The *SHPrintPHPChannelXSUA* user action can print the cross sections for several energies. The energies to be printed are controlled by defining a minimum and a maximum energy and the number of energy steps:

```
/gamos/setParam USER_ACTION_NAME:minE VALUE
```

```
/gamos/setParam USER_ACTION_NAME:maxE VALUE
```

```
/gamos/setParam USER_ACTION_NAME:nStepsE VALUE
```

If you want that the steps are taken logarithmically, you can set the parameter:

```
/gamos/setParam USER_ACTION_NAME:logE 1
```

If the model defined for the inelastic or neutron capture processes belong to the *neutron_hp* or *particle_hp* GEANT4 packages, you will also get the cross sections channel by channel, unless you set the parameter:

```
/gamos/setParam USER_ACTION_NAME:OnlyTotalXS 1
```

The channel cross sections may be printed as a fraction of the total cross section if you set the parameter:

```
/gamos/setParam USER_ACTION_NAME:NormaliseToTotalXS 1
```

It is possible to discard products with high atomic number and mass, but setting the maximum isotope number: atomic number + 1000 * atomic mass (for example discard all isotopes higher than alphas by setting this number to 2004). This number is set with the parameter:

```
/gamos/setParam USER_ACTION_NAME:ProductMaxAZ VALUE
```

Also the particle yields can be printed (the mean number of particles of a given type produced in a reaction channel) and the particle yield multiplied by the channel cross sections (i.e. the particle cross section of production of the given particle), if you select the parameter:

```
*/gamos/setParam USER_ACTION_NAME:ParticleYields 1*
```

The *SHPrintPHPChannelXSUA* user action writes also a set of histograms with the cross sections as a function of energy channel by channel for each isotope in each material. The set of energies is the one selected with the *minE*, *maxE* and *nStepsE* parameters. If it is requested to print the particle yields, it will also

Each of the histogram needs to have a well identified internal number. This puts a limit to the number of materials (10), isotopes (50) and products (200). They may be changed by the parameters:

```
/gamos/setParam USER_ACTION_NAME:MaterialIndexMax VALUE
```

```
/gamos/setParam USER_ACTION_NAME:IsotopeIndexMax VALUE
```

```
/gamos/setParam USER_ACTION_NAME:ProductIndexMax VALUE
```

Counting hadronic cross sections

Although the previous user action may be used to get the inelastic cross sections channel by channel, it only serves for the *neutron_hp* and *particle_hp* physics; and only for those cases where the database contains channel by channel cross sections, i.e. not for those where only total cross sections and particle yields are given (only data in the *F02* directory).

The user action *SHGetHadronicChannelXSUA* serves to get the inelastic or neutron capture cross sections per reaction in any case, as it simulates a number of events and counts how often each reaction happens. It also serves to print the average particle yields of the inelastic collisions.

You can see an example on the use of this utility at *MY_GAMOS_DIR/tutorials/SHieldingTutorial/exercise5/exercise5b.in*. The user action first prints the total cross section of the hadronic processed defined, in a line showing the material, the primary particle energy, the process name, the total and the microscopic cross sections:

```
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic TOTAL_XS(macro)=
0.00119941 cm-1 TOTAL_XS(micro)= 1.41256 barn
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV nCapture TOTAL_XS(macro)=
2.91782e-07 cm-1 TOTAL_XS(micro)= 0.000343635 barn
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV nFission TOTAL_XS(macro)=
0 cm-1 TOTAL_XS(micro)= 0 barn
```

Then it prints the number of times each reaction channel happens in the job. For a reaction channel it is understood each combination of target nucleus plus secondaries. The description of the channel is printed by enumerating the list of particles together with the number of times they appear; at the end it is printed the count number of the channel. The target nucleus is easily identified, as it starts with a “[” character:

```
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL
gamma 2 alpha 1 \[Fe54 1 Cr51[0.0] 1 = 3
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL
gamma 4 alpha 1 \[Fe54 1 Cr51[0.0] 1 = 1
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL
gamma 1 alpha 1 \[Fe56 1 Cr51[0.0] 1 = 3
```

After all the channels are printed, they are merged, putting together those that have the same particles without taking into account the gammas. The channel name is printed in a compact form, with the target nucleus first, the projectile particle, the list of non-ion particles and the list of ions (usually only one). For each channel it is printed the number of times it happens divided by the number of times a collision with the target nucleus happens and the total process microscopic cross section multiplied by this ratio:

```
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL_ALL=
Fe54 (n,2n)Fe56 = 2/2974*XS (micro)= 0.00094994
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL_ALL=
Fe54 (n,a)Cr51 = 4/2974*XS (micro)= 0.00189988
@@SHGetHadronicChannelXSUA G4_Fe 10 MeV NeutronInelastic CHANNEL_ALL=
Fe54 (n,a)Cr53 = 12/2974*XS (micro)= 0.00569964
```

Finally, if you set the parameter

```
/gamos/setParam USER_ACTION_NAME:ParticleYield 1
```

it will print the particle yield, i.e. the number of times a particle is produced for each collision with a target nucleus:

As you can see in this example, the data may be printed for several materials and several energies in a single job. But to do this you have to follow some rules:

- The geometry should be *GmGeometryUseMaterials* (see *Geometry* chapter).
- The generator should be *GmGeneratorChangeEnergyAndMaterial* (see *Generator* chapter). And you have to define a primary particle source with a particle of the type you want to study.
- You should delete the hadronic elastic process, unless you are interested in studying it. And, if you are studying charged particle collisions, also the electromagnetic processes: */gamos/physics/removeProcessesByType Electromagnetic /gamos/physics/removeProcessesByName hadElastic*

In case you do not want to take into account the gammas, you can set the parameter:

```
/gamos/setParam USER_ACTION_NAME:NotGammas 1
```

In case you want to take into account only ions, you can set the parameter:

```
/gamos/setParam USER_ACTION_NAME:OnlyIons 1
```

Make histograms of secondary particles from the neutron_hp or particle_hp databases

The user action:

```
/gamos/userAction SHPlotPHPDBSpectraUA
```

serves to print the energy and theta angle spectra of secondary particles as read from the nuclear databases. This means that the energy and theta angle are taken directly from the database without applying thermal or Lorentz boost corrections.

As this user action invokes the GEANT4 internal methods there is no need to simulate an event. Nevertheless, to profit from the GAMOS utilities, you should set up an input file similar to other GAMOS input files. You can see an example on the use of this utility at *MY_GAMOS_DIR/tutorials/SHieldingTutorial/exercise5/exercise5c.in*.

This example plots the energy and angle spectra from all the secondary particles that would be created at the collision of neutrons, as given by the *neutron_hp* or *particle_hp* GEANT4 package, for several energies in several materials. The first thing it does is to replace the geometry *GmGeometryFromText* by

GmGeometryUseMaterials. This geometry class will generate a world for each of the materials given by the parameter:

```
/gamos/setParam GmGeometryUseMaterials:Materials MATE_1 MATE_2
```

If any of the materials is not a GEANT4 one, you will need a geometry file where the material is defined. The name of this file is given with the parameter:

```
/gamos/setParam GmGeometryUseMaterials:FileName FILE_NAME
```

The *SHPlotPHPDBSpectraUA*> user action can make histograms with several energies at the same job. The energies used in the histograms are defined by giving a minimum and a maximum energy and the number of energy steps:

```
/gamos/setParam USER_ACTION_NAME:minE VALUE
```

```
/gamos/setParam USER_ACTION_NAME:maxE VALUE
```

```
/gamos/setParam USER_ACTION_NAME:nStepsE VALUE
```

If you want that the steps are taken logarithmically, you can set the parameter:

```
/gamos/setParam USER_ACTION_NAME:logE 1
```

To make the histograms the energy and theta variables are sampled as GEANT4 would do it after reading the database. The number of times the distributions are sampled is set with the parameter:

```
/gamos/setParam USER_ACTION_NAME:NEvents VALUE
```

Each of the histogram needs to have a well identified internal number. This puts a limit to the number of materials (10), isotopes (50) and products (200). They may be changed by the parameters:

```
/gamos/setParam USER_ACTION_NAME:MaterialIndexMax VALUE
```

```
/gamos/setParam USER_ACTION_NAME:IsotopeIndexMax VALUE
```

```
/gamos/setParam USER_ACTION_NAME:ProductIndexMax VALUE
```

It is possible to discard products with high atomic number and mass, but setting the maximum isotope number: atomic number + 1000 * atomic mass (for example discard all isotopes higher than alphas by setting this number to 2004). This number is set with the parameter:

```
/gamos/setParam USER_ACTION_NAME:ProductMaxAZ VALUE
```

Print yields of production of secondary particles from charged particles traversing a thich material

The user action:

```
/gamos/userAction SHPrintPHPChannelThickYieldUA
```

prints the average yield of secondary particles produced when a charged particle traverses a thick yield of a material as it loses energy from a maximum until a minimum energy value. It will show you the yield for each hadronic channel independently.

As this user action invokes the GEANT4 internal methods there is no need to simulate an event. Nevertheless, to profit from the GAMOS utilities, you should set up an input file similar to other GAMOS input files. You can see an example on the use of this utility at *MY_GAMOS_DIR/tutorials/SHieldingTutorial/exercise5/exercise5f.in*.

This example print the thick yield of a proton in several materials, using the particle_hp GEANT4 package, for several energies in several materials. The first thing it does is to replace the geometry *GmGeometryFromText* by

GmGeometryUseMaterials. This geometry class will generate a world for each of the materials given by the parameter:

```
/gamos/setParam GmGeometryUseMaterials:Materials MATE_1 MATE_2
```

If any of the materials is not a GEANT4 one, you will need a geometry file where the material is defined. The name of this file is given with the parameter:

```
/gamos/setParam GmGeometryUseMaterials:FileName FILE_NAME
```

The *SHPrintPHPChannelThickYieldUA* user action accumulates the yield of secondary particle from an initial energy down to a minimum energy. These energies are set with the parametes:

```
/gamos/setParam USER_ACTION_NAME:minE VALUE
```

```
/gamos/setParam USER_ACTION_NAME:maxE VALUE
```

To calculate the average yields in a fast way, it does not simulate N particles, but just computes them from the channel cross sections. The number of intermediate energy that it will use for these calculations is given by the parameter (take into account that more steps means bigger precision but longer running time):

```
/gamos/setParam USER_ACTION_NAME:nStepsE VALUE
```

If you want that the steps are taken logarithmically, you can set the parameter:

```
/gamos/setParam USER_ACTION_NAME:logE 1
```

Analysis utilities

Summing scoring results

Summing activation results

MANAGING DICOM FILES

GAMOS is able to read the patient data resulting from a CT scan in DICOM format. It can also read RT structures in DICOM format as well as RT plans, RTIon plans and RT doses.

Managing DICOM CT geometry files

Converting DICOM CT geometry files to GAMOS simulation files

The pixel values of a DICOM image represent CT Hounsfield numbers (related to the electronic density) and to be used in a Monte Carlo simulation they should be converted, first, to a material density and then to a material type. Each file with DICOM Modality *CT* will be treated as a CT file corresponding to an image Z slice. All the CT files processed will be merged to form a unique patient file in Geant4 format. To do this conversion the slices should be contiguous in Z, else a warning message will be thrown. The format of this file is described below. To do the conversion it is needed a metadata file, which controls how the conversion is done and whose format is described below.

To do this conversion you can use the utility at *analysis/DICOM/DICOM2G4* (available from any directory, as it is in the GAMOS path), which is based on the Geant4 example *examples/extended/medical/DICOM*.

This utility uses the *DCMTK* (<http://dicom.offis.de/dcmTk.php.en>) package to treat DICOM files. The DCMTK tool is installed automatically with GAMOS, under the directory *MY_GAMOS/external/dcmTk* (where *MY_GAMOS* is the directory where you installed GAMOS).

Writing the metadata file

The first step to convert a DICOM file to GAMOS is to write a metadata file, that you may call *Data.dat*.

This metadata file should be passed as first argument of the *DICOM2G4* executable or as a parameter:

```
DICOM2G4 Data.dat
```

or

```
DICOM2G4 -f Data.dat
```

The information in this file will drive the conversion from DICOM to GAMOS format. The format of this file is based on tags (similary to the geometry files). The following tags should be used:

Output file name

The name of the file that contains the DICOM images in GAMOS format. If this tag does not exists, the file will be named *testCT.g4dcm*.

Example:

```
:FILE_OUT mytest.g4dcm
```

Compression level

:COMPRESSION

- compression level

This is the number of voxels that will be merged into one in the X and Y dimensions. The Hounsfield numbers of the voxels merged are averaged to give the resulting value for the new voxel.

Example:

```
:COMPRESSION 4 // 4 X 4 voxels will be merged, so that the number of voxels in X and Y dimensions will be reduced by a factor 4X4
```

DICOM files

:FILE

- file name

These are the list of files (one line per file) in DICOM format that will be treated. They can be of modality *CT*, *RTSTRUCT*, *RTPLAN*, *RTIonPLAN*, *PET* or *RTDOSE* (the code will automatically detect its modality and treat it correspondingly).

Example:

```
:FILE Image/Patient_CT_slice_1.dcm
:FILE Image/Patient_CT_slice_2.dcm
:FILE Image/Patient_CT_slice_3.dcm
```

Coefficients for Hounsfield number to material density conversion

:CT2D

- Hounsfield number
- density

These sets of value pairs build the calibration curve (linearly interpolating between them). In other words, each Hounsfield number is given a material density using a function that is built interpolating between this list of value pairs.

Example:

```
:CT2D -5000 0.
:CT2D -1000 0.01
:CT2D -400 0.602
:CT2D 300 1.145
:CT2D 2000 1.856
```


The values here are just a simple example. You may find more realistic values at [*Hounsfield_Schneider*] and [*Hounsfield_duPlessis*]. In any case, the values found in the image file must lie between the minimum and maximum values that you define.

Material identification through Hounsfield numbers

:MATE

- material name
- upper bound of material density interval

This serves for the assignment of materials to voxels following the voxel material density. The voxels with a material density between 0 and the first upper bound will be assigned to the first material, those with a material density between the first upper bound and the second upper bound will be assigned to the second material, etc.

Example:

```
:MATE G4_AIR 0.207
:MATE G4_LUNG_ICRP 0.919
:MATE G4_WATER 1.
:MATE G4_MUSCLE_WITH_SUCROSE 1.05
:MATE G4_B-100_BONE 1654
```

CT geometry file format

The DICOM CT geometry files are converted to a simple ASCII format. You may also create your own file by hand with the following format (see e.g. testCT.g4dcm):

- A line with the number of materials
- A line for each material with its index and name (the same name of materials that you construct as G4Material's)
- A line with the patient positioning (not mandatory):
 - HFP = Head First-Prone
 - HFS = Head First-Supine
 - HFDR = Head First-Decubitus Right
 - HFDL = Head First-Decubitus Left
 - FFDR = Feet First-Decubitus Right
 - FFDL = Feet First-Decubitus Left
 - FFP = Feet First-Prone
 - FFS = Feet First-Supine
- A line with the number of voxels in X, Y and Z
- A line with the minimum and maximum extension in X (mm)
- A line with the minimum and maximum extension in Y (mm)
- A line with the minimum and maximum extension in Z (mm)
- A number of lines containing the nVoxelX*nVoxelY*nVoxelZ material indices (one per voxel). Each line contains a number of words equal to nVoxelZ

- A number of lines containing the $nVoxelX * nVoxelY * nVoxelZ$ material densities (one per voxel). Each line contains a number of words equal to $nVoxelZ$

Optionally, you may add information on DICOM structures, see section below.

Reading a DICOM CT geometry file in a GAMOS job

To read the CT geometry file produced by the DICOM2G4 utility you should define as your geometry the class

```
/gamos/geometry GmReadPhantomG4Geometry
```

You should have a file with the format described in the previous section containing your phantom description. The name of this file is set with the parameter

```
/gamos/setParam GmReadPhantomGeometry:Phantom:FileName MY_FILENAME
```

and another file where you describe the rest of your geometry (at least the world volume where the phantom is placed). The name of this file is set with the parameter

```
/gamos/setParam GmReadPhantomGeometry:FileName MY_FILENAME
```

In a phantom file, the voxels of the same material may have a different density. In Geant4 this means that it is necessary to create a different material for each different value of the density, and then a set of physics tables for each material, what occupies a lot of memory and has a penalty in the CPU time. By default GAMOS uses the density for each voxel from the DICOM file, but to avoid creating too many materials (what will increase the memory), GAMOS groups densities in intervals, creating a new material for each combination material name - density interval. The width of the interval can be selected with the parameter

```
/gamos/setParam GmReadPhantomGeometry:Phantom:DensityStep DENSITY_INTERVAL
```

which by default takes a value of 0.1 g/cm³.

If you want to use the original material density creating a different material for each density value in the DICOM file, you have to set to false the parameter

```
/gamos/setParam GmReadPhantomGeometry:RecalculateMaterialDensities 0
```

The navigation in the voxels is done using the Geant4 algorithm, *G4RegularNavigation*, that is the optimal one for regular geometries (see [\[biblio.g4doc-regularNav\]](#)). The user may select whether when a track navigates through contiguous voxels with the same material the frontier between them will be skipped, or not, by using the parameter

```
/gamos/setParam GmReadPhantomGeometry:Phantom:SkipEqualMaterials 1/0
```

which by default takes a value of 1.

For testing purposes, other navigation algorithms may be selected, namely voxel navigation with 1-dimensional optimization (that occupies similar memory as *RegularNavigation* but is very slow)

```
/gamos/setParam GmReadPhantomGeometry:Phantom:RegularStructureID 0
```

or 3-dimensional optimization (that occupies a lot of memory but it is almost as fast as *G4RegularNavigation* when no skipping of frontiers of voxels with equal materials is used)

```
/gamos/setParam GmReadPhantomGeometry:Phantom:OptimAxis kUndefined
```

GAMOS is also able to read the EGSnrc/DOSXYZnrc format for DICOM files. Simply use

```
/gamos/geometry GmReadPhantomEGSGeometry
```

and the rest of parameters are the same as for the Geant4 DICOM files.

By default every phantom is placed in the world volume. If you want to place it into another physical volume, you can set the parameter

```
/gamos/setParam GmReadPhantomGeometry:MotherName PHYSVOL_NAME
```

By default the name of the voxels is *phantom*, and this is the name that you should use in you commands. You may change it with the parameter

```
/gamos/setParam GmReadPhantomGeometry:VoxelName VOXEL_NAME
```

It is also possible to displace the phantom from its (0,0,0) position in the mother volume. To do it you have to set the parameter:

```
/gamos/setParam GmReadPhantomGeometry:InitialDisplacement DISP_X DISP_Y DISP_Z
```

And to rotate it around the X axis, then Y axis and finally Z axis around the phantom centre you can use the parameter:

```
/gamos/setParam GmReadPhantomGeometry:InitialRotAngles ANGLE_X ANGLE_Y ANGLE_Z
```

As commented above there are eight possible patient rotations in the DICOM format (what is called *Patient Position*, see <https://www.dabsoft.ch/dicom/3/C.7.3.1.1.2>). By default the beam travels along the negative Z direction, what means, that if the phantom file is not rotated it will be simulated so that the beam enters first the patient head. If you want to conform to the orientation as given by the DICOM file you should them add the following *InitialRotationAngles* in your script:

- HFS : 90 0 0
- HFDL : 90 0 90
- HFDR : 90 0 -90
- HFP : 90 0 180
- FFS : -90 0 180
- FFDL : -90 0 -90
- FFDR : -90 0 90
- FFP : -90 0 0

Managing DICOM CT structure files

A DICOM CT structure is used to define the voxels that belongs to a given volume, for example an organ, like brain, lungs, thyroid, etc., or also a treatment volume. In each Z slice a contour is drawn to define the area that belongs to the structure, which includes the voxels that are totally or partially inside the structure (indeed a voxel is considered to be inside a structure if the center of the voxel is); this operation is repeated in a set of contiguous Z slices to define the full set of voxels belonging to the structure.

DICOM CT structures can be managed in GAMOS, but before they have to be converted into GAMOS format. Sections below explain how do this, how to use these files in GAMOS and also several utilities to check and draw DICOM structures.

Converting CT structure files

The DICOM CT structures, also known as Regions Of Interest (ROI), can be converted to a simple ASCII format by adding a new line in the metadata file, *Data.dat*, used with *DICOM2G4*. As for the CT files, this line has to start with the tag *:FILE*.

Example:

```
:FILE Image/Patient_CT_slice_1.dcm
:FILE Image/Patient_CT_slice_2.dcm
:FILE Image/Patient_CT_slice_3.dcm
```

```
:FILE RT.structures.dcm
```

The code will loop through all the voxels in the CT files and check in which CT structures they are included. In the same file as the CT information (*testCT.g4dcm* by default), after the list of the material indices and the material densities, it will be written an index for each of the voxels setting the structure it belongs to. If the voxel does not belong to any structure a "0" will be written.

If a voxel belongs to several ROI's, it will be assigned an index which contains the list of ROI's indices in the form $INDEX_1+INDEX_2*100+INDEX_3*100*100+...$. As this index is stored as an unsigned integer, the maximum value it can take is 2^{64} , this means that no more than 9 ROI's indices can be stored for a voxel. If you need more you have to set the *-structureIDShift* argument to a lower value.

```
DICOM2G4 -f Data.dat -structureIDShift 50
```

After all the voxel indices, there is a list of two-column lines to assign a number to each structure name found in the DICOM file, for example:

```
1 "Head tumor"  
2 "Heart"  
3 "lungs"
```

The algorithm used to determine if a voxel is inside a structure is a complex one. If you find that there are some precision problems we recommend you to set the following *DICOM2G4* parameter to a number bigger than the default value of 2:

```
-structureNCheck NEW_VALUE
```

Using files with CT structures in your GAMOS job

To use the phantom structure it is necessary to select a different geometry, depending on the format:

```
/gamos/geometry GmReadPhantomG4withPSGeometry
```

or

```
/gamos/geometry GmReadPhantomEGSwithPSGeometry
```

It is important to remember that, if you have changed the maximum number of structures when building the Geant4 ASCII file, you have to put the same number when reading it, by using the parameter:

```
/gamos/setParam GmPhantomStructure:IDShift VALUE
```

In the case of the EGS format, by default the number of digits used to store the index is two, so if more than one structure is assigned to a voxel, it is needed to increase this number, by changing the parameter:

```
GmReadPhantomEGSwithPSGeometry:PVNumberOfDigits VALUE
```

Filtering on DICOM structures

It is possible to filter when a track or a track step Enter/Exit/Start/Stop/In/Traverse a DICOM structure by using the corresponding filter: *GmEnterPhantomStructureFilter*, *GmExitPhantomStructureFilter*, *GmStartPhantomStructureFilter*, *GmStopPhantomStructureFilter*, *GmInPhantomStructureFilter*, *GmTraversePhantomStructureFilter*. These filters must receive an argument, the list of phantom structures names, for example:

```
/gamos/filter InBrainF GmInPhantomStructureFilter brain "optical nerve"
```

Managing DICOM PET image files

Also PET image files can be treated with GAMOS. Similarly to other files, all that is necessary is to add the file name in the metadata file, *Data.dat*, with the tag *:FILE*. The intensities in the DICOM file will be written as a file similar format to the CT file:

After transforming them to GAMOS ASCII format, they can be used to define the position of a source of particles proportionally to the intensity of the image pixels.

Converting DICOM PET files to GAMOS simulation files

PET images in DICOM format can be transformed to GAMOS ASCII format by simply adding them in the metadata file of the *DICOM2G4* utility (see section on *Converting DICOM CT geometry files to GAMOS simulation files*). The PET image files will be transformed into a single ASCII file, with default name *testPET.g4dcm*, containing with the following data:

- A line with the number of voxels in X, Y and Z
- A line with the minimum and maximum extension in X (mm)
- A line with the minimum and maximum extension in Y (mm)
- A line with the minimum and maximum extension in Z (mm)
- A number of lines containing the $nVoxelX * nVoxelY * nVoxelZ$ intensity indices (one per voxel). Each line contains a number of words equal to $nVoxelZ$.

The default output name can be changed in the metadata file with the tag:

:FILE_OUT fileName

Reading a DICOM PET image file in a GAMOS job

The information contained in the ASCII file described above can be used as the distribution of the position of the primary particles in a GAMOS job. The positions will be distributed in the voxels defined in the file, proportionally to the intensity assigned to each voxel. Inside a voxel the position will be sampled uniformly in each of the three dimensions. To create this distribution, to the primary generator of type *GmGenerator* it has to be assigned the position distribution *GmGenerDistPositionPETImage*, adding the PET file name with the parameter:

```
/gamos/generator/positionDist MY_SOURCE GmGenerDistPositionPETImage FILE_NAME
```

Managing RT plan data

Transforming DICOM RT plan data to GAMOS ASCII format

You can extract the data from a DICOM RT plan file, by adding the file name in the control metadata file, *Data.dat*, when using *DICOM2G4*. This command will write the interesting data into a set of files with the format of the text geometry, so that they can be used in your job. If you use the file at *analysis/DICOM/DICOMExample/Plan/Patient_plan.dcm* you will get a file *RTPlan* with the following lines

```
:P FractionGroup_1_Number 1
```

```
:P FractionGroup_1_NumberOfBeams 2
```

```
:P FractionGroup_1_NumberOfBrachyApplicationSetups 0
```

:P FractionGroup_1_NumberOfFractionsPlanned 25

For each beam a file *RTPlanBeam_ii* will be written with the information on the beam sequence:

```
:P BeamDose 1.3895
:P BeamDoseSpecificationPoint_X -18.067
:P BeamDoseSpecificationPoint_Y -155.136
:P BeamDoseSpecificationPoint_Z -1175.69
:P BeamMeterset 234
:P BeamNumber 1
:P FinalCumulativeMetersetWeight 1
:P HighDoseTechniqueType 1
:P NumberOfBlocks 0
:P NumberOfBoli 0
:P NumberOfCompensators 0
:P NumberOfControlPoints 28
:P ReferencedPatientSetupNumber 1
:P SourceAxisDistance 1000
:PS BeamDescription "ARC 1"
:PS BeamName "1"
:PS BeamType "DYNAMIC"
:PS InstitutionName "Clinica Universitaria de Navarra"
:PS Manufacturer "Elekta"
:PS PrimaryDosimeterUnit "MU"
:PS RadiationType "PHOTON"
:PS TreatmentDeliveryType "TREATMENT"
:PS TreatmentMachineName "VERSA"
:P BLD_MLCX_Z 401.77
:P BLD_MLCX_NumberOfLeafJawPairs 1
:P BLD_ASYMY_Z 509
:P BLD_ASYMY_NumberOfLeafJawPairs 1
```

And for each control point of each beam you will get a file named *RTPlanControlPoint_ii_jj* with the information on the control point sequence:

```
:P ControlPointIndex 1
:P NominalBeamEnergy 10
:P IsocenterPositionX 0
:P IsocenterPositionY -155.136
:P IsocenterPositionZ -1175.69
:P SourceToSurfaceDistance 908.127
:P GantryAngle 189
:P BeamLimitingDeviceAngle 20
:P PatientSupportAngle 0
:P TableTopEccentricAngle 0
:P SourceToSurfaceDistance 908.127
:P MetersetWeight 1.57651e-52
:P GantryPitchAngle 0
:P SurfaceEntryPoint 0
:P TableTopEccentricAxisDistance 0
:P TableTopLateralPosition 0
:P TableTopLongitudinalPosition 0
:P TableTopPitchAngle 0
:P TableTopRollAngle 0
:P TableTopVerticalPosition 0
:P ASYMY_1_POS -45
:P ASYMY_2_POS 40
:P MLCX_1_POS -2.5
:P MLCX_2_POS -2.5
```

```
...
:P MLCX_160_POS 2.5
```

Using DICOM RT plan data in a GAMOS job

To use the above files into a GAMOS job, i.e. to simulate an RTPlan, you have to use as primary generator the GAMOS generator, *GmGenerator*, and as particle source *RTPlanSource*, with initial particle an electron

```
/gamos/generator/addRTPlanSource mySource e-
```

```
/gamos/setParam          source:ControlPointFileNamePrefix      RTPlanControlPoint      /gamos/setParam
source:PlanFileNamePrefix RTPlan /gamos/setParam source:BeamFileNamePrefix RTPlanBeam /gamos/setParam
source:NEventsPerMSUnit   1000000 #/gamos/setParam source:MSUnitsToSkip 1 /gamos/setParam
source:OpeningAngleX     0.*deg /gamos/setParam source:OpeningAngleY 0.*deg /gamos/setParam
source:DiscSigma 0. /gamos/setParam source:EnergySigma 0. /gamos/generator/addRTPlanSource source e-
#/gamos/generator/addSingleParticleSource source geantino 1 /gamos/generator/RTPlan/associatePlanVolumes
source MLCX MLC_X /gamos/generator/RTPlan/associatePlanVolumes source ASYMY JAWS_Y
/gamos/generator/RTPlan/defineAcceleratorVolume source ACCEL
```

Managing RTIon plan data

Transforming DICOM RTIon plan data to GAMOS ASCII format

Similar to the CT plan data (see section on *Managing RT plan data*), a plan of an active scanning of a proton or ion beam can be transformed to a format usable by GAMOS, and later it be used to define the energy, position and direction of the beam.

The DICOM file containing the plan is automatically transformed to GAMOS ASCII format by simply adding it in the metadata file, *Data.dat* of the *DICOM2G4* utility. This will produce four sets of files, which correspond to the RTIon DICOM data structure:

- *RTIonPlan*: contains the basic informations about the fractions groups. The only relevant information for GAMOS is:

```
:P NumberOfBeams 1
```

- *RTIonPlanBeam_Nb*, where Nb is the beam number: contains the information about beam Nb. The only relevant information for GAMOS is:

```
:P NumberOfControlPoints 42
:P FinalCumulativeMetersetWeight 2446.76
```

(the meterset is the “intensity” of the beam)

- *RTIonPlanControlPoint_Nb_Ncp* where Ncp is the control point number: contains the information about control point Ncp of beam Nb. The relevant information for GAMOS is:

```
:P GantryAngle 270
:P GantryPitchAngle 0
:P LimitingDeviceAngle 0
:P MetersetWeight 30.7095
:P NominalBeamEnergy 198.9
:P NumberOfPaintings 1
:P NumberOfScanSpotPositions 321
:P PatientSupportAngle 0
:P ScanningSpotSize 9.29121
```

```
:P SnoutPosition 650 ??
:P TableTopLateralPosition 0 ??
:P TableTopLongitudinalPosition 0 ??
:P TableTopPitchAngle 0 ??
:P TableTopRollAngle 0 ??
:P TableTopVerticalPosition 0 ??
ScanSpotPositions
-47.9622 60.3654 0
... (NumberOfScanSpotPositions times)
-41.2028 60.3654 0
```

Each line of the ScanSpotPositions contains X_position, Y_position and meterset (the meterset is the “intensity” of the beam)

Using DICOM RTIOn plan data in a GAMOS job

To use the RTIOn plan data in a GAMOS job you have to add a source in the primary generator of type *GmGenerator* of type:

```
/gamos/generator/addRTIOnPlanScanSpotSource MY_SOURCE PARTICLE
```

This source will distribute the Geant4 primary generator particles in each scan spot of each control point of each beam of each fraction group, so that the number of events is proportional to the meterset of the scan spot. Indeed this will only happen with enough precision if the number of events requested is very big. By default, the events will be distributed in the scan spots rounding to an integer and if the number of events is small it may even happen that some scan spots are not assigned any event. To force a different behaviour you can use the following parameter:

```
/gamos/setParam SOURCE_NAME:NEventsPerMSUnit NEV
```

With this parameter the number of events assigned to each scan spot will be $NEV * \text{meterset}$, so that when the number of events requested is exhausted the run will be stopped, leaving the last scan spots with no event. If you want to continue the simulation of the scan spots, you may use the parameter:

```
/gamos/setParam SOURCE_NAME:NEventsToSkip NEV
```

or alternatively

```
/gamos/setParam SOURCE_NAME:MSUnitsToSkip MSUNITS VALUE
```

By default the beam is simulated as a disk, using the position distribution *GmGenerDistPositionDisc*, with radius equal to half of the value of the parameter *ScanningSpotSize* of each control point. But there is the possibility of simulating the beam as a disk with a 2D Gaussian profile, using the position distribution *GmGenerDistPositionDiscGaussian*, with sigma equal to half of the value of the parameter *ScanningSpotSize* of each control point. To use this option you have to set the parameter:

```
/gamos/setParam SOURCE_NAME:DiscGaussian 1
```

It is also possible to make a figure representing the scan spots as circles with diameter equal to the the parameter *ScanningSpotSize*. For each control point a figure will be made, with name *RTIOnPlanScanSpot_Nb_Ncp.gif*, where Nb is the number of the beam and Ncp is the number of the control point. To use this option you have to set the parameter:

```
/gamos/setParam SOURCE_NAME:PlotScanSpots 1
```

Visualizing geometry superimposed to DICOM images

It is possible to visualize the Geant4 geometry superimposed on the DICOM CT, PET or dose images by using the ScanVis 2D geometry visualization utility. This utility serves to make 2D images in the XY, XZ or YZ planes, which

can be superimposed on the DICOM images by the procedure described below.

The first thing is to do is running a gamos job selecting as primary particle generator the *GmGeneratorScanVis* so that you write a set of images in the XY, XZ or YZ planes. You can see an example at *PATH_TO_MY_GAMOS_DIRECTORY/analysis/DICOM/DicomExample/geometry/scanVis.XY.in* (or *scanVis.XZ.in* * or **scanVis.YZ.in*). See the chapter on *2D Visualization* to understand all the commands in these scripts. On top of creating the 2D figures, running these scripts will produce a file in binary format containing all the points that appear in the figure, with file name *scanVis.XY.out** (or *scanVis.XZ.out* * or **scanVis.YZ.out*). This file can be input to the *drawG4dcm* by using the parameter *-fGeometry FILE_NAME*. Another parameter to be used is the one that selects one of the planes: *-geometryPlane XY*, the default one, or *XZ* or *YZ*. For example, if you are creating XY images, each time a DICOM image is produced, *drawG4dcm* will look from the list of points with Z coordinate closer to the center of the image plane, given that this Z coordinate lies in the range between the minimum and maximum Z of the image. It is important to take into account that only one Z value will be taken from to draw the geometry on the image; this means that you should use an *ScanVis* file that contains planes created in the same orientation defined with the *-geometryPlane* parameter. You can find more details in the *README* file at the **PATH_TO_MY_GAMOS_DIRECTORY/analysis/DICOM/DicomExample/geometry/** directory, which explains the example.

Partial phantom geometries

The DICOM files always describe a parallelepiped geometry, including air around the real patient, animal or phantom. It may be that you have a CT DICOM file that you want to simulate inside a PET scanner or a radiotherapy accelerator and it does not fit because this air overlaps with the machine. In this case you may use a part of your *.g4dcm* file, taking the parallelepipedal DICOM geometry and intersecting it with some volume to create a new file. The volume can be one in the Geant4 volume or a virtual one you create. To do this you can follow the example in *analysis/DICOM/buildPartialDICOM*. You have to run a GAMOS job with an input file similar to the one in that directory *buildPartial_G4Volume.in*. The physics and primary generator can be anyone, but for the geometry you have to use *GmReadPhantomG4Geometry* to read your *.g4dcm* file. Then you have to add a command:

```
/gamos/geometry/DICOM/intersectWithG4Volume VOLUME_NAME
```

where *VOLUME_NAME* is the name of a volume defined in your geometry.

Alternatively you may use a virtual volume you create for the occasion, in a similar way as it is done at *buildPartial_userVolume.in*. In this case you have to add a command:

```
/gamos/geometry/DICOM/intersectWithUserVolume POS_X POS_Y POS_Z ANG_X ANG_Y ANG_Z SOLID_TYPE SOLID_PARAM_1 SOLID_PARAM_2 ...
```

where *POS_X POS_Y POS_Z* is the position of the centre of the volume,

ANG_X ANG_Y ANG_Z are the angles of rotation around X, Y and Z axis, and then it comes the solid type and parameter with the same format as the one used to define a solid in the geometry text file format.

The file format is slightly different than that of the *.g4dcm* files. The start is the same: list of materials, number of voxels and phantom dimensions (as if it were the original parallelepiped). Then it comes the information about which voxels are used and which not: for each Z slice and for each Y slice there is a line indicating the X index of the first voxel used and that of the last voxel used (-1 -1 if no voxel is used in that X row). The rest of the file is also similar: the list of voxel material indices and the list of voxel densities.

To read this file the geometry to be used is:

```
/gamos/geometry GmReadPhantomPartialG4Geometry
```

Visualizing all types of DICOM files

The utility *drawG4dcm* serves to visualize the DICOM files after transforming them into GAMOS format. You can visualize CT images, PET images, structures (and superimpose them to CT, PET or dose images), doses (and superimpose isodose lines to CT, PET or dose images). The figures produced will be 2-D images of all slices in the XY, XZ or YZ plane. We describe in the following the parameters that should be used to visualize each type of file.

Visualizing CT images

To visualize CT images you have to run *drawG4dcm* with the argument *-fG4DCM FILE_NAME*, where *FILE_NAME* is the name of a set of CT images transformed into GAMOS text format (by using the *DICOM2G4* command). This will produce a set of images where the intensity will be proportional to the density. The name of the images will be *hisDrawG4dcm_*. The name of the files will be *hisDrawG4dcm_SliceXY_Z.gif* where *Z* is the *Z* position of the center of the XY slice (similarly for the XZ and YZ slices).

You can choose instead to produce images with a different color for each different material by adding the parameter *-bByMate 1*. The name of the files will be *hisDrawG4dcm_Mate_SliceXY_Z.gif* where *Z* is the *Z* position of the center of the XY slice (similarly for the XZ and YZ slices).

To better visualize the different materials, it is also possible to produce in the same job another set of images, one per each material, by adding the parameter *-bMate1Fig 1*. The name of the files will be *hisDrawG4dcm_Mate_MATE_NAME_SliceXY_Z.gif* where *MATE_NAME* is the material name and *Z* is the *Z* position of the center of the XY slice (similarly for the XZ and YZ slices).

Visualizing PET images

To visualize PET images you have to run *drawG4dcm* with the argument *-fG4DCMPET FILE_NAME*, where *FILE_NAME* is the name of a set of PET images transformed into GAMOS text format (by using the *DICOM2G4* command). This will produce a set of images where the intensity will be proportional to the PET intensity. The name of the files will be *hisDrawG4dcm_SliceXY_Z.gif* where *Z* is the *Z* position of the center of the XY slice (similarly for the XZ and YZ slices).

Visualizing RT structures

To visualize RT structures that are closed polygonal lines you have to run *drawG4dcm* with the argument *-fSt FILE_NAME*, where *FILE_NAME* is the name of an RTSTRUCT file in DICOM format. This will draw the structures superimposed in each of the CT, PET or Dose images in the XY plane. It is therefore mandatory that you define a CT, PET or Dose images together with this argument. By default all structure lines will be drawn, each one with a different color. If you want to select which structures to draw, and the drawing style of each structure, you can use the parameter *-bStLines FILE_NAME*, where *FILE_NAME* is the name of a file where you describe the drawing styles by adding a line for each structure in the following format:

- Structure name
- Drawing option: 0 = no drawing, 1 = drawing, 2 = drawing line and name (name will be written at each 90:sup:o, tangent to the structure)
- Line and name color: 0 = white, 1 = black, 2 = red, 3 = green, 4 = blue, 5 = yellow, 6 = magenta, 7 = cyan, ... Default color is the ROI number (you can find the full set of colors at <https://root.cern/doc/v610/classTColor.html>)
- Line style: 1 = continuous, 2 = points, 3 = continuous grey, 4 = small and big points, ... Default line style is 1 (you can find the full set of line styles at <https://root.cern.ch/doc/master/classTAttLine.html>)

- Line width: Default line width is 1 (you can find the full set of line widths at <https://root.cern.ch/doc/master/classTAttLine.html>)

If you want to check in detail the structures, you may also draw a figure for each XY plane and each structure containing only the structure contour, plus another file with the structure contour plus a point representing the center of the voxels that are included in that contour. To do this you have to use the argument `-fDrawOnlySt 1`. You may choose to draw only these figures, but not the structures superimposed on CT, PET or Dose images, by setting the argument `-fStOnImage 0`. The name of the files will be `hisDrawG4dcm_ROI_NAME*onlySt_Z.gif` and `hisDrawG4dcm_ROI_NAME*St_Z.gif` where `ROI_NAME` is the name of the structure, and `Z` is the Z position of the center of the XY slice.

Visualizing Doses

You can visualize dose files running `drawG4dcm` whether in GAMOS text format, by using the argument `-fG4DCMDose FILE_NAME`, or in GAMOS `sqdose` format, by using the argument `-fSqDose FILE_NAME`. These arguments will automatically dose figures in the XY, XZ and YZ planes, and isodose lines in all CT, PET or Dose images. The name of the files will be `hisDrawDose_XY_Z.gif` where `Z` is the Z position of the center of the XY slice (similarly for the XZ and YZ slices).

The isodose lines are built localising the voxel with maximum dose and then tracing lines from this voxel center in different directions and calculating in each line the point where the dose is equal to $FRACTION * MAXIMUM_DOSE$, where `FRACTION` is by default 0.9, 0.5 and 0.1. These lines by default are traced in directions separated 1 degree, so that each isodose line is drawn joining 360 points. You may change the number of points with the argument `-isodNAngles N_ANGLES`. Similarly to the structures you can write a file which contold the drawing of the isodose lines, by using the parameter `-bIsodLines FILE_NAME`. This file contains lines with the following format:

- Fraction of maximum dose
- Drawing option: 0 = no drawing, 1 = drawing, 2 = drawing line and name (name will be written at each 90:sup:o, tangent to the structure)
- Line and name color: 0 = white, 1 = black, 2 = red, 3 = green, 4 = blue, 5 = yellow, 6 = magenta, 7 = cyan, ... Default color is the ROI number (you can find the full set of colors at <https://root.cern/doc/v610/classTColor.html>)
- Line style: 1 = continuous, 2 = points, 3 = continuous grey, 4 = small and big points, ... Default line style is 1 (you can find the full set of line styles at <https://root.cern.ch/doc/master/classTAttLine.html>)
- Line width: Default line width is 1 (you can find the full set of line widths at <https://root.cern.ch/doc/master/classTAttLine.html>)

You may choose not to draw the figures that contain only the doses, by using the argument `-bDrawOnlyDose 0`. Or you may choose not to draw the isodose lines superimposed on CT, PET or Dose images, by setting the argument `-fIsodOnImage 0`.

Drawing options

There are several arguments that serve for all types of figures:

- `-bDrawXY 0*`: do not write figures in XY plane
- `-bDrawXZ 0*`: do not write figures in XZ plane
- `-bDrawYZ 0*`: do not write figures in YZ plane
- `-nPixels N_PIXELS`: number of pixels used for each figure. Default is 1000.
- `-palette`: palette of colors used for each figure. 62 color palettes are available (see <https://root.cern/doc/v610/classTColor.html>). The default palette in GAMOS is number 52, i.e. black/white. The default ROOT palette is number 0 in GAMOS.

- *-imageFormat FORMAT*: available format are *gif* (default), *png*, *eps*, *ps*, *pdf*, *svg*, *xpm*, *jpg* and *tiff*
- *-fRoot FILE_NAME*: name of the ROOT file containing the histograms (you may later open it and manipulate the figures). The default name is “drawG4dcm.root”. If you do not want to write this file, you have to use the argument *-fRoot “”*
- *-verb VERBOSITY_LEVEL*: this may serve to debug the application in case you find a problem, or a behaviour you want to better understand. Verbosity levels are: *silent*, *error*, *warning*, *info* (default), *debug* and *test*.
- *-h* or *-help*: running *drawG4dcm* with only this argument will print on the screen the set of parameters with a short explanation

Modifying the material indices of voxels inside a CT structure

You can modify the material indices of all the voxels that are inside a DICOM CT structure. You may find this useful if you are simulating for example a boron neutron capture therapy treatment, and you want to change the material composition of some structure to include the presence of boron, or simply because you are not satisfied with the automatic assignment of material from Hounsfield numbers and want to change the material inside a selected structure. To run this utility you have to run the command:

```
changeMaterialSt PHANTOM_FILE_NAME STRUCTURE_NAME NEW_MATERIAL_NAME
```

This utility can be found in the directory *analysis/DICOM/changeMaterialSt*. If you run it with the arguments :

```
changeMaterialSt ../DICOM2G4/test.c8.g4dcm CTV myNewMaterial
```

you will get the new GAMOS DICOM file *test.g4dcm.new*. You may change this file name with the argument *-fOut NEW_DICOM_FILE_NAME*. As usual if you define an extra argument, you cannot set the other arguments without explicitly defining what they are, i.e. you have to use the command

```
changeMaterialSt -DCM PHANTOM_FILE_NAME -st STRUCTURE_NAME -mate NEW_MATERIAL_NAME -fOut NEW_DICOM_FILE_NAME
```

Other arguments that you can use are:

- *-verb* : verbosity: sets the verbosity
- *-help* : prints the set of arguments

Use simple phantom geometries

The user may build simple regular phantom geometries without the need of writing a DICOM file by using

```
/gamos/geometry GmSimplePhantomGeometry
```

The number of voxels is defined with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:N Voxels NVOXEL_X NVOXEL_Y NVOXEL_Z
```

The minimum and maximum extensions in the three axes are defined with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:PhantomDims MIN_X MAX_X MIN_Y MAX_Y MIN_Z MAX_Z
```

Then you can divide the phantom in different regions along the Z axis with the parameter

```
/gamos/setParam GmSimplePhantomGeometry:MaterialZVoxels NZ_1 NZ_2 ...
```

where *NZ_i* is the number of voxels along Z of the *i* region.

Then you can assign the material and material densities of each Z region with the parameters

/gamos/setParam GmSimplePhantomGeometry:MaterialNames MATERIAL_1 MATERIAL_2 ...

/gamos/setParam GmSimplePhantomGeometry:MaterialDensities DENSITY_1 DENSITY_2 ...

OPTIMISING THE CPU TIME OF YOUR APPLICATION

We recommend you to spend some time in optimising the CPU time spent by your application, as you may save a big amount of CPU time with not too much effort.

Knowing where the time is spent

Before starting your optimisation it may be very useful to understand where the time is spent. For example if most of the time is spent tracking some type of particle, some volumes, or particles created by some process. To get a report of where the time is spent GAMOS provides the user action *GmTimeStudyUA*, that you can combine with one or more classifiers to get a detailed report.

Production cuts

The production cuts are the minimum energy of the electrons produced by ionisation and of gammas produced by bremsstrahlung (and also of the positrons produced by pair production process of muons if you are using these particles). If the cut values are very big, it means that you are not creating electrons or gammas of high energy but instead you count all their energy as energy deposited at the last step position.. so you should check that your value is not too high. On the other side, the number of these particles produced grows exponentially as you diminish these cuts, so you should check that your value is not too low.

The cut is given in Geant4 as a distance value, meaning that you are not creating an electron or gamma if (in average) it is not going to travel a distance longer than the cut value. A rule of thumb may be that you set a cut value of the order of 1/10 of the resolution you have, so that your results are not affected much (of course if you want to have a very high precision the effect of the cut should be reduced even further). The default cut value in GAMOS is 0.1 mm. You may change it with the Geant4 command:

```
/run/setCut VALUE mm
```

You should also take into account the possibility of using different cut for different parts of your geometry (regions). For example a bigger cut in regions where you do not measure anything and a lower cut in your sensitive regions; or a big cut if the particles created in a region are far from your detectors so that they have a very small probability of reaching them. To set cuts per region see section *production cuts by region*.

To understand if the changing of cuts may have an important effect in your CPU time, you may do first a time study checking if the particles created by ionisation or bremsstrahlung are taking a big proportion of the time, by adding the command:

```
/gamos/userAction GmTimeStudyUA GmClassifierByCreatorProcess
```

If your application is a nuclear medicine detector or a radiotherapy treatment, we recommend you to read the sections of automatic optimisation of production cuts, which may help you in getting the best cut values.

By default the production cuts only affect ionisation and bremsstrahlung, but you may force that they are also used in all processes by using the command:

```
/gamos/physics/applyCutsForAllProcesses 1
```

Killing particles of small energy

An alternative or complementary approach of the optimisation of production cuts is the optimisation of the user limits that kill the particles when their energy becomes small (see section on *User limits*). It is likely logic that if you put a production cut to kill particles of small range, you kill the particles when their energy becomes small so that they will not travel more than that range value. For efficiency reasons we recommend that you use the user limit:

```
/gamos/physics/userLimits/setMinEKinByRange USER_LIMITS_NAME LOGICAL_VOLUME_NAME PARTICLE_NAME MIN_RANGE
```

If your application is a nuclear medicine detector or a radiotherapy treatment, we recommend you to read the sections of automatic optimisation of user limits, which may help you in getting the best values.

Killing particles

The two user actions *GmKillAtStackingUA* and *GmKillAtSteppingUA* combined with the filtering mechanism can help you to kill those particles that you know are not affecting your result by a sensible amount. For example you may kill the neutrinos produced in a radioactive decay, or the electrons in an application where you are scoring neutrons.

Optimising the particle generation

GAMOS provides a wide variety of primary generator distributions with which to simulate in detail the primary particles that correspond to your application. But you may consider that some particles will not reach your detector (or will reach it with a very small probability) and modify your distributions so that those particles are never produced. Another alternative is to use a biasing distribution (see section on *Biasing generator distributions*).

Limiting the number of user actions

Some of the user actions (for example those that fill lots of histograms) may consume a non negligible amount of CPU time. Therefore a simple recommendation is to check that you deactivate all the user actions you do not need when you are making a big production of events.

Using variance reduction techniques

Several variance reduction techniques are provided by GAMOS to optimise the CPU time for specific applications. See the section on *Variance reduction techniques* or the corresponding application.

APPENDIX A: MISCELLANEOUS

Using parameters

As you can see through this guide, many algorithms in GAMOS use parameters to let the user change their behaviour. All the parameters in GAMOS have a default value and the user may change it in the user command file with the command:

Checking the usage of parameters

Many of the GAMOS classes or your own classes have a different behaviour depending on the value of some parameters. You can see many example of this throughout this guide.

You have to remember always to set a parameter before you invoke any code that may use it (we recommend you to set all the parameters at the beginning of your command file). The parameters are read usually in the class constructors; therefore, if you set a parameter after the class has been constructed, it will take no effect and the default value will be used.

To guarantee that you have done it this way, you will get at the end of a GAMOS job the information on the usage of parameters. You will always get a list of the parameters that have not been used in the code. This is probably an indication that you have mistyped a parameter name. This list appears at the end of your output and looks similar to this one:

```
%%%%%%%% PARAMETERS NOT USED (DEFINED IN SCRIPT BUT
NOT USED BY C++ CODE)
%%%      MAYBE YOU HAVE MISPELLED THEM?
PARAMETER: GmGeometryText:FileNam
```

Other lists are available at user request to get a more detailed information. You may get them anywhere in your simulation by using the command

/gamos/base/printParametersUsage LEVEL

If *LEVEL* takes a value ≥ 0 you will get the same list as above. If *LEVEL* takes a value ≥ 1 you will get a list of parameters that are using the default value (you may then check if this list contains one of the parameters whose value you thought you have changed). This list looks similar to this one:

```
%%%%%%%% PARAMETERS USING DEFAULT VALUE (DEFINED IN C++
CODE BUT VALUE NOT DEFINED IN SCRIPT)
PARAMETER: Generator:Isotope:FileName
PARAMETER: GmCountTracksUA:FirstEvent
```

If *LEVEL* takes a value ≥ 2 you will get a list of how many times each parameter have been used. This list looks similar to this one:

```
##### NUMBER OF TIMES EACH PARAMETER IS USED IN C++ CODE
PARAMETER GmCountTracksUA:EachNEvent TIMES USED= 1
PARAMETER GmGeometryFromText:FileName TIMES USED= 1
```

Managing the input data files

To run an example you will probably need some input data, like for example the file describing the geometry, the list of isotopes, etc.

You can set the name of your file in your script, but you do not need to tell the path where to look for it. An environmental variable, called *GAMOS_SEARCH_PATH* contains the list of directories where GAMOS will look for your file. The directories are separated by a colon, and their order in this variable reflects the order in which they will be searched. This variable is set up when you configure GAMOS, and takes a default value of

```
.:MY_GAMOS_DIR/data
```

You may change this value at your will, but remember not to reset the GAMOS configuration after that, because it will reset it to its original value.

```
/gamos/setParam PARAMETER_NAME PARAMETER_VALUE(S)
```

There are four types of parameters in GAMOS:

- One number
- A list of numbers
- One string
- A list of strings

The above command can be used for any kind of parameters, and GAMOS will detect which of the four types it is by analysing the parameter values. There may be though some peculiar cases where this automatic identification may fail. For example if you want to set a parameter for a list of volume names like *VOL_A1*, *VOL_B1*, *VOL_C1* and you use a parameter to set its value equal to the three like:

```
/gamos/setParam PARAMETER_NAME *1
```

GAMOS will mistakenly think that is a parameter of type number number. For this cases it is possible to use a command specific for each of the four types:

```
/gamos/setParamN PARAMETER_NAME NUMBER
```

```
/gamos/setParamLN PARAMETER_NAME NUMBER_1 NUMBER_2 ...
```

```
/gamos/setParamS PARAMETER_NAME STRING
```

```
/gamos/setParamLS PARAMETER_NAME STRING_1 STRING_2 ...
```

Random number seeds

If you want to run several jobs with the same configuration but different random seeds each, you can use the GEANT4 random number management, or do it the GAMOS way.

GAMOS offers two ways, to set the initial random seed or to restore it from a file. Both options are explained below.

Setting the initial random number seeds

GAMOS offers a single command with which you can give a different initial random seed to your job, so that the results are statistically independent:

```
/gamos/random/setSeeds INITIAL_SEED N_RANDOM_NUMBERS
```

where two numbers are given to better guarantee the independence of the results. *INITIAL_SEED* is the initial random seed and *N_RANDOM_NUMBERS* is the number of times a random seed is sampled before starting the simulation. You may use different or equal numbers for these two, and use continuous numbers (e.g. 1001, 1002, 1003, ...) if you want.

Restoring the initial random number seeds

Although the previous method should guarantee that two jobs have independent random numbers, it may happen that the numbers chosen happen to give some correlation. To avoid this GAMOS provides a set of initial random number seeds that are independent. They have been produced by running contiguous sets of 1E10 random numbers and storing the final random numbers (or, in other words, running random numbers and storing them each 1E10). The command to use this utility is :

```
/gamos/random/restoreSeeds SEED_SET_NUMBER N_RANDOM_NUMBERS
```

where *SEED_SET_NUMBER* the first number correspond to the file name where the seed is stored, by default: *data/initialSeeds/initialSeeds.1e+10.SEED_SET_NUMBER*; it can be from 0 to 999. *N_RANDOM_NUMBERS* is the number of times a random seed is sampled before starting the simulation.

GAMOS provides 1000 random seed files, separated by 1e+10 random numbers using the JamesRandom engine. You may build your own set, but you have to follow some rules:

- All files should be put in the same directory, whose name is given by the parameter : */gamos/setParam GmRunManager:SeedFileDir DIR_NAME* The directory will be looked for in the current directory or in the *data* directory, or you may use an absolute path (starting by ******).
- The file names should be *initialSeeds.NNUMBERS.SEED_SET_NUMBER*, where *NNUMBERS* is the number of random numbers thrown for each file. You have to set this number with the parameter (by default it is *1.e+10*): */gamos/setParam GmRunManager:RandomSeedsInFile NUMBER*

Changing the random engine

The random engine (the algorithm that gets the random numbers) may be changed with the command:

```
/gamos/random/setEngine ENGINE_NAME
```

Any of the engines available in CLHEP can be chosen, i.e. *DRand48Engine DualRand Hurd160Engine Hurd288Engine HepJamesRandom MTwistEngine NonRandomEngine RandEngine RanecuEngine Ranlux64Engine RanluxEngine RanshiEngine TripleRand*.

Sending several jobs in the same machine

It may often happen that you have a multi-core machine and you want to run several jobs at the same time on it to accumulate statistics. The approach that is explained here is to send the same job several times with different random seeds. Another possible approach, which will be available in future GAMOS releases, is to use multi-threading, what will spare the initialisation time and reduce the memory by sharing it among the different jobs.

There are many ways of sending many jobs together in a machine. Here we propose a simple but flexible script that may facilitate this task. This script is written in the Unix command language *bash*, and it just needs the utility *awk*, a data extraction and reporting tool which is available on any Unix or MacOS operative system.

The example has four input parameters that the user has to give: energy, random seed, number of events, number of jobs. The four parameters will be converted to internal variables:

```
### set the variables read from the command line
ENERGY=$1
SEED=$2
NEV=$3
NJOBS=$4
```

Then the loop to the number of jobs is started:

```
### start the loop of jobs
nj=0
while
test $nj -lt $NJOBS
do
```

A different suffix for each job is created, which will be added to the new name of the input file, as well as to the output file name defined inside the input file:

```
### set the suffix of the output files
SUFFIX=$1"."$2"."$3"."$nj
echo " SUFFIX = $SUFFIX "
```

The input file is copied into a new one, so that you can keep a track of the different files that are run:

```
### copy the input file into a new one (so that you can keep a track of the different files that are
new_inputfile="exercise2."$SUFFIX
log_inputfile=zz_"$new_inputfile
echo " The new input file = " $new_inputfile
```

The *awk* tool is used to substitute the scrip input variables in the GAMOS input file:

```
### substitute in the input file the variables from the command line
awk -v ENERGY=$ENERGY -v SEED=$SEED -v NEV=$NEV -v nj=$nj -v SUFFIX=$SUFFIX '{
if($1=="/run/beamOn") {printf("%s %s \n",$1, NEV) }
else if ($1=="/gamos/random/setSeeds") {printf("%s %s %s\n",$1, SEED, SEED+nj) }
else if ($2=="RTPhaseSpaceUA:FileName") {printf("%s %s %s\n",$1, $2, "test."SUFFIX) }
else if ($1=="/gamos/generator/addSingleParticleSource") {printf("%s %s %s %s\n",$1,$2,$3,ENERGY"\*M
else { print $0 }
}' "exercise2b.in" > $new_inputfile".inn"
```

You can observe that each of the variables that are going to be used by *awk* have to be passed with the *-v* option. The *awk* will loop through the lines in the input file and will make the substitutions. For example the line

```
else if ($2=="RTPhaseSpaceUA:FileName") {printf("%s %s %s\n",$1, $2, "test."SUFFIX) }
```

means that it looks for a line whose second word is *RTPhaseSpaceUA:FileName* and then substitutes this line by three words (three *%s*), the first two are left intact, and the third one is substituted by the value of the *SUFFIX*, preceded by *test.*.

Finally the job is sent in background. You may run it in foreground, what means that you have to wait until a job finish to start the next one:

```
### run job in background
gamos $new_inputfile".inn" 2>&1 | tee $log_inputfile &
```

```
### run job in foreground
# gamos $new_inputfile".inn" 2>&1 | tee $log_inputfile
```

If, for example, you want to run 40 jobs in your 4-core machine, you should not run them all in background at the same time, as they will have to share the CPU and memory, wasting your computer resources, or even saturating your memory. A smarter approach would be to type four times a *sendjobs* command running jobs in foreground:

```
sh sendjobs 6. 1111 10000 10 &
sh sendjobs 6. 2111 10000 10 &
sh sendjobs 6. 3111 10000 10 &
sh sendjobs 6. 4111 10000 10 &
```

The full *sendjobs* file can be found in your GAMOS distribution, under the directory *tutorials/RTTytutorial/exercise2*.

Identifying touchables

As explained in several points through this guide, you can use the concept of touchable available in GAMOS. We will explain first in a few lines the concept of touchable in Geant4 and GAMOS:

In Geant4 there are several geometrical objects [*biblio.g4doc-geometry*] :

- *G4VSolid*: A solid is a geometrical object that has a shape and specific values for each of the shape dimensions
- *G4LogicalVolume*: A logical volume contains the volume's full properties. It includes the geometrical properties of the solid, and adds physical characteristics: the material of the volume, whether it contains any sensitive detector elements, the magnetic field, etc.
- *G4VPhysicalVolume*: A physical volume is a volume placed already in another volume, that is a volume with position and rotation matrix
- *G4VTouchable*: A touchable is each copy of a volume. To understand the difference with a physical volume, we put an example: If you place a volume A in 5 places and a volume B in 12 places, you will have 5 + 12 physical volumes, each one with a distinct position and rotation matrix. But you will have 5 X 12 = 60 individual copies, that is 60 touchables

To save memory the *G4VTouchable* are instantiated in Geant4 only when a track traverses the corresponding volume in space, and they are immediately deleted when the track leaves. In GAMOS you have the possibility of accessing any individual touchable whenever you like. When you need it you can ask GAMOS to create a *GmTouchable*, which will have the same characteristics as the corresponding *G4VTouchable* that would be created when a track reaches it.

To identify the touchable you want to use, you have to use the following notation:

For example the name *CRYSTAL* identifies all individual crystals of your detector that have name ‘‘*CRYSTAL*’’, while *BLOCK:2/CRYSTAL:1* refers only to the crystal(s) with copy number 1 in block(s) with copy number 2. *RING:3/BLOCK/CRYSTAL:1* refers to all the crystal(s) with copy number 1 in all the block(s) whatever copy number they have in the ring(s) that have copy number 3. Don't forget that the full touchable name will start with a slash followed by the name of the world volume, e.g. */world:0/RING:3/BLOCK/CRYSTAL:1*. You may get the full list of touchables in your geometry with the command:

```
/gamos/geometry/printTouchables *
```

If you are writing some new C++ code for GAMOS you can have easy access to the list of touchables with a given name with the line

```
GmGeometryUtils::GetInstance()->GetTouchables( touch_name, itExists )
```

If the touchables do not exist in your geometry you will get a warning in case *itExists* is false and an exception if *itExists* is true.

There is a limitation on the use of touchables: they cannot be used for assembly volumes, as Geant4 creates internally the physical volumes.

Using asterisks to get volume, particle and material names

In many commands described in this guide, you give the name of a logical volume, physical volume, touchable, particle or material, so that GAMOS finds the corresponding Geant4 object. In case you want to apply your command to several volumes with similar names, you can use an asterisk that would mean 'any character'. For example if you have the volumes named CRYSTAL_BGO_1, CRYSTAL_BGO_2, CRYSTAL_LUYAP_1 and CRYSTAL_LUYAP_2

```
/gamos/SD/assocSD2LogVol GmSDSimple Calor CRYSTAL*
```

will associate a sensitive detector to the four volumes, while

```
/gamos/SD/assocSD2LogVol GmSDSimple Calor CRYSTAL_BGO_*
```

will associate a sensitive detector to the two volumes CRYSTAL_BGO_1 and CRYSTAL_BGO_2, and

```
/gamos/SD/assocSD2LogVol GmSDSimple Calor CRYSTAL_*1
```

will associate a sensitive detector to the two volumes CRYSTAL_BGO_1 and CRYSTAL_LUYAP_1.

Using particle names

Each particle type in Geant4 is identified by a unique name. No particle is created by Geant4 unless the user code does it explicitly. At any time in your command file you can ask for a list of created particles with the command */run/particle/dumpList*. If you do this after instantiating the GAMOS electromagnetic physics list, you will get the following list:

```
anti_nu_e, chargedgeantino, e+, e-,
gamma, geantino, nu_e, opticalphoton
```

If you use the GAMOS hadronic physics list, you will get the following list:

```
GenericIon, He3, alpha, anti_neutron
anti_nu_e, anti_nu_mu, anti_nu_tau, anti_proton
chargedgeantino, deuteron, e+, e-
eta, eta_prime, gamma, geantino
mu+, mu-, neutron, nu_e
nu_mu, nu_tau, opticalphoton, pi+
pi-, pi0, proton, tau+
tau-, triton,
```

These are the names that should be used in the commands that need a particle name. If you want to use hadrons, GAMOS also provides the possibility of grouping them, so that with a single name you can identify the whole group. The groups defined are the following:

- *lightMeson*: Mesons that only contain up and down quarks

```
pi+, pi-, pi0, eta, eta_prime, kaon+, kaon-, kaon0, kaon0L,
kaon0S, a0(\*), a1(\*), a2(\*), k(\*), k1(\*), k2(\*), k_star(\*),
k0_star(\*), k2_star(\*), k3_star(\*), anti_k(\*), anti_k0(\*),
anti_k1(\*), anti_k2(\*), anti_k_star(\*), anti_k2_star(\*),
anti_k3_star(\*), b1(\*), f0(\*), f1(\*), f2(\*), f2_prime(\*),
```

```
h1(\*), eta(\*), eta2(\*), phi(\*), phi3(\*), pi(\*), pi2(\*),
rho(\*), rho3(\*)
```

- *charmMeson*: Mesons that contain a charm quark

```
D+, D-, D0, anti_D0, Ds+, Ds-, J/psi
```

- *bottomMeson*: Mesons that contain a bottom quark

```
B+, B-, B0, anti_B0, Bs0, anti_Bs0
```

- *meson*: All mesons
- *lightBaryon*: Baryons that only contain up and down quarks

```
proton, anti_proton, neutron, anti_neutron, N(\*), anti_N(\*),
delta(\*), anti_delta(\*)
```

- *strangeBaryon*: Baryons that contain a strange quark

```
lambda, anti_lambda, sigma0, anti_sigma0, sigma+,
anti_sigma+, sigma-, anti_sigma-, xi0, anti_xi0, xi-,
anti_xi-, omega-, anti_omega-, lambda(\*), anti_lambda(\*),
sigma(\*), anti_sigma(\*), xi(\*), anti_xi(\*), omega(\*),
omega3(\*)
```

- *charmBaryon*: Baryons that contain a charm quark

```
lambda_c+, anti_lambda_c+, sigma_c0, anti_sigma_c0,
sigma_c+, anti_sigma_c+, sigma_c++, anti_sigma_c++, xi_c+,
anti_xi_c+, xi_c0, anti_xi_c0, omega_c0, anti_omega_c0
```

- *baryon*: All baryons
- *ion*: ions

```
GenericIon, alpha, He3, deuteron, triton
```

- *ALL*: All particles

APPENDIX B: EXTENDING GAMOS FUNCTIONALITY WITH C++ UTILITIES

Converting a Geant4 example into a GAMOS example

Converting a Geant4 example into a GAMOS examples usually requires only adding a few lines to transform the different simulation components into plug-in's. In *examples/N02* you can see the official Geant4 example *novice/N02* transformed into a GAMOS example. We will use it to illustrate the procedure to follow.

The first thing to do is to substitute the Geant4 *GNUMakefile* by a GAMOS *GNUMakefile*. You can use the file in this example for any other example you want to transform, just substitute the line

```
name := exampleN02
```

by the name of your example (indeed you can use any name you want, it will be automatically detected by GAMOS).

Then you have to add a file, that we called *src/plugin.cc* where all the plug-in's are created. The first line of this file (after the corresponding 'includes'), must be

```
#include "Reflex/PluginService.h"
```

For the detector construction it is only needed to add a line (plus the corresponding include files)

```
PLUGINSVC_FACTORY(ExN02DetectorConstruction,G4VUserDetectorConstruction*());
```

For the physics, in a similar way, we add

```
PLUGINSVC_FACTORY(ExN02PhysicsList,G4VUserPhysicsList*());
```

The primary generator requires some more changes. As you can see *ExN02PrimaryGeneratorAction* constructor receives as argument the detector construction class, so that it can then ask it for the dimensions of the world. This is not needed in GAMOS because all volumes are available in any class through the singleton class *GmGeometryUtils*. Therefore we have deleted the detector construction argument in the constructor, and then we have substituted the line

```
G4double position = -0.5*(myDetector->GetWorldFullLength());
```

by this one

```
G4Box* worldBox = (G4Box*) (GmGeometryUtils::GetInstance()->
GetLogicalVolumes("World")[0]->GetSolid());
G4double position = -0.5*worldBox->GetXHalfLength();
```

After this in the *src/module.cc* a line has to be included

```
PLUGINSVC_FACTORY(ExN02PrimaryGeneratorAction,G4VUserPrimaryGeneratorAction*());
```

For the user actions, we have first to transform then into GAMOS user actions, what requires simply to edit the *.hh* classes and make them inherit from *GmUserXXXAction*, instead of *G4UserXXXAction*. Then we have to add the following lines in *module.cc*

```
PLUGINSVC_FACTORY(ExN02RunAction,GmUserAction*());
PLUGINSVC_FACTORY(ExN02EventAction,GmUserAction*());
PLUGINSVC_FACTORY(ExN02SteppingAction,GmUserAction*());
```

It is not needed to convert the sensitive detector into a plug-in, that could be called in the user command file, because it is explicitly called in the detector construction class. Indeed, if you want to do it, you should delete the lines that instantiate it there and then you can write

```
PLUGINSVC_FACTORY(ExN02TrackerSD,G4VSensitiveDetector*(G4String));
```

The main class, *exampleN02.cc* is not needed any more. We use the GAMOS main and a user command file, that we may *exampleN02_GAMOS.in*, with the user commands that select the example components, like the following one:

```
/gamos/geometry ExN02DetectorConstruction
/gamos/physicsList ExN02PhysicsList
/gamos/generator ExN02PrimaryGeneratorAction
/gamos/userAction ExN02RunAction
/gamos/userAction ExN02EventAction
/gamos/userAction ExN02SteppingAction
/run/initialize
/run/beamOn 10
```

You can then run *gamos exampleN02_GAMOS.in* and you will get the same results as if you run the original Geant4 example.

Creating your plug-in

There are several “factories” in GAMOS that take care of the different plug-in types. To transform your class into a plug-in you have to follow the instructions in this section, using the relevant “factory” and class as indicated in the relevant section of this guide (e.g. *GmPhysicsFactory* and *G4VUserPhysicsList* for a geometry plug-in, *GmVerbosityFactory* and *GmVerbosityMgr* for a verbosity plug-in, etc).

To write a new plug-in of any type follow these steps

1. Create your class or use one of the existing Geant4 classes. This class should inherit from the corresponding Geant4 or GAMOS class: - G4VUserDetectorConstruction: geometry - G4VUserPhysicsList: physics list - G4VUserPrimaryGeneratorAction: primary generator - GmVGenerDistPosition: primary particles position distribution - GmVGenerDistDirection: primary particles direction distribution - GmVGenerDistEnergy: primary particles energy distribution - GmVGenerDistTime: primary particles time distribution - GmVUserXXXAction: user action (XXX can be Run, Event, Tracking, Stepping or Stacking) - G4VSensitiveDetector: sensitive detector - GmVDigitizer: hits digitizer - GmVRecHitBuilder: reconstructed hits builder - GmVPrimitiveScorer: scorer - GmVPSPrinter: scorer printer - GmVFilter: filter - GmVClassifier: classifier - GmVVerbosityMgr: verbosity If you are creating a new class you can use as example one of the classes at directory *examples/PlugInTemplates*, that are nearly-empty classes that contain the necessary methods that you have to implement for each plug-in type. .. COMMENT: %\item {\bf GmGeometryFactory, GmPhysicsFactory, GmGeneratorFactory, GmGenerDistEnergyFactory, GmGenerDistTimeFactory, GmGenerDistPositionFactory, GmGenerDistDirectionFactory, GmUserActionFactory, GmSensDetFactory, GmDigitizerFactory, GmRecHitBuilderFactory, GmPrimitiveScorerFactory, GmFilterFactory, GmClassifierFactory, GmPSPrinterFactory and GmVerbosityFactory}
2. Include the ROOT plug-in module definition:

```
#include "Reflex/PluginService.h"
```

Then you simply you have to create a plugin writing which is the base class, for example, if it is a geometry (detector construction in Geant4's terminology)

```
PLUGINSVC_FACTORY (MY_GEOMETRY, G4VUserDetectorConstruction* ( ))
```

And the plug-in name will be the same as your class name

3. Once this is done, you can select your geometry with the command: `/gamos/xxx MY_GEOMETRY`

NOTE: If you are creating a plug-in in a new directory you have created, you have to be sure to have the “plugin” option in the *GNUmakefile*, as explained in the section *Compiling your new code*.

Using a parameter in your C++ code

We describe in this section how to create and use a new parameter if you are creating a new C++ class.

GAMOS provides an utility that allows you to change the value of a parameter in the input file, together with the line commands, and use it in any of your classes (even in several of them). There are four classes of parameters: numbers, string, list of numbers, list of strings.

To change the value of a parameter (of any of the four types) in your input file, you have to use the command

```
/gamos/setParam MY_PARAM_NAME MY_PARAM_VALUE(S)
```

Then you can use this parameter in your class with a line like this:

```
G4double value = GmParameterMgr::GetInstance()->
GetNumericValue("MY_PARAM_NAME", DEFAULT_VALUE);
```

if it is a number, or

```
G4String value = GmParameterMgr::GetInstance()->
GetStringValue("MY_PARAM_NAME", "DEFAULT_VALUE");
```

if it is a string, or

```
std::vector < G4double > vdefault;
std::vector < G4double > values = GmParameterMgr::GetInstance()->
GetVNumericValue("MY_PARAM_NAME", vdefault);
```

if it is a list of numbers, or

```
std::vector < G4String > vdefault;
std::vector < G4String > values = GmParameterMgr::GetInstance()->
GetVStringValue("MY_PARAM_NAME", vdefault);
```

if it is a list of strings.

Event classification by interaction types

There is a utility in GAMOS that helps you in counting and classifying the tracks by the type of interactions they have suffered. You just have to create at each step a new *GmTrajPoint* and at the end of track pass this list to a *GmVSimuEventClassifier* that will return the classification.

You can see an example at *GamosCore/GamosAnalysis/src/GmHistosGammaAtSD.cc*, that we explain here in detail:

This class counts the type of interaction of the photons in the sensitive detectors of your geometry.

The first thing it does, at the *PreUserTrackingAction* is checking if the current track is an 'original' gamma. To do this it gets the help of the *GmCheckOriginalGamma* class, that classifies the gammas as

- 0: not an 'original' gamma
- 1: it is a primary particle, created at the beginning of the event
- 2: is is created at the annihilation of the positron (it is assumed that the positron is a primary particle)

At each step, the *UserSteppingAction* method checks that it is inside a volume declared as sensitive detector. In this case, it adds a new *GmTrajPoint* for this track, with all the information of the track at this moment (plus it adds at the beginning another point with the vertex information).

At the end of track, if it is an original gamma, it asks the class *GmClassifierByInteraction* to classify it based on the type and number of interactions. The method *Classify()* of this class returns an integer with the meaning: $100 * 100$ Number of LowEnPhotoElec interactions + $100 * \text{Number of LowEnCompton interactions}$ + Number of LowEnRayleigh interactions*.

See PET section for more details on the concrete class *GmHistosGammaAtSD*.

Structure of GAMOS

Although the origin of GAMOS is the medical physics field, the latest additions, specially the many utilities to extract detail information of many types, make it likely that you can profit from GAMOS to do your simulation in other field of physics, as already several people are doing.

Although you will not need to know anything about the C++ code, unless you want to develop new code for GAMOS, we explain here how the GAMOS source code is organized in the subsystems, each one subdivided in the packages:

- *GamosCore*: core software covering main Geant4 functionality - *GamosBase*
 - *Base*: parameter manager, analysis manager, CSV histograms and other basic functionalities
 - *Filters*: filters
 - *Classifiers*: classifiers
 - *GamosGeometry*: geometry-related utilities and support for text detector description
 - *GamosMovement*: support for displacements, rotations or general movements of volumes during a job
 - *GamosSD*: classes for sensitive detectors, hits and digitization
 - *GamosGenerator*: utilities to support single particle and isotope source generators as well as different initial particle distributions
 - *GamosPhysics* - *PhysicsList*: example of
 - electromagnetic physics list, supporting standard, low-energy and Penelope classes, and example of hadronic physics list (meant for hadrontherapy)
 - * *OtherPhysicsLists*: other examples of electromagnetic and hadronic physics lists
 - * *Cuts*: management of production cuts and user limits, including tools to automatically optimise them
 - * *VarianceReduction*: implementation of several variance reduction techniques
 - *GamosUserActionMgr*: user action management, to allow several user actions of the same type, selectable by user commands

- *GamosScoring*: scoring manager and messenger and examples of scoring plug-in classes (scorers, filters and printers) - *Management*: scoring manager and messenger and base class for scorer and scorer printers - *Scorers*: scorers - *Printers*: scorer printers - *PointDector*: point detector scoring
- *GamosData - Management*: data management and base data classes - *Data*: data - *Users*: data users
- *GamosAnalysis*: utilities that can help the advanced user to analyse results
- *GamosReadDICOM*: code to read in DICOM files containing patient data
- *GamosUtilsUA*: user action utilities (tracking verbosity control, track counting, process counting, time study, ...)
- *GamosUtils*: general C++ utilities
- *GamosVisIguana*: support of IGUANA visualisation
- *GamosApplication*: GAMOS run manager and the ‘‘main’’ program
- *ITACA*: example of ITACA simulation
 - *ITACA geometry*: ITACA geometry
 - *ITACA histograms and ITACA event classifier*: ITACA histograms and ITACA event classifier
- *PET*: example of PET simulation - *PETGeometry*: example to simulate the most common PET devices by defining its properties in a few-lines text file
 - *PETAnalysis*: PET event classifier and PET histograms
- *RadioTherapy*: example of radiotherapy simulation, including writing and reading phase space files

Each package has the following subdirectories:

- *src*: the source code
- *include*: the header files

You do not need to follow this file distribution if you want to create a new package, but we recommend you to do so.

Phys. Med. Biol 41: 111-124 (1996)

INDICES AND TABLES

- [genindex](#)
- [modindex](#)
- [search](#)

BIBLIOGRAPHY

- [biblio.geant4] <http://www.cern.ch/geant4>
- [biblio.g4doc] <http://www.geant4.org/geant4/support/userdocuments.shtml>
- [biblio.clhep] <http://proj-clhep.web.cern.ch/proj-clhep>
- [biblio.root] <http://root.cern.ch>
- [biblio.penelope] Penelope - A Code System for Monte Carlo Simulation of Electron and Photon Transport Workshop Proceedings Issy-les-Moulineaux, France 5-7 November 2001, AEN-NEA
- [biblio.g4doc-geometry] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/Detector/detector.html>
- [biblio.g4doc-regularNav] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/Detector/Geometry/geomNav.html#fast-navigation-in-regular-patterned-geometries>
- [biblio.g4doc-generator] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/GettingStarted/eventDef.html#g4vprimarygenerator>
- [biblio.g4doc-physics] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/PhysicsReferenceManual/html/index.html>
- [biblio.g4doc-physics-ref] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/PhysicsReferenceManual/html/PhysicsReferenceManual.html>
- [biblio.Colombino] Colombino et al., Nuovo Cimento, vol.38 num.2, 16 (1965)
- [biblio.g4phys-lists] http://www.slac.stanford.edu/comp/physics/geant4/slac_physics_lists/G4_Physics_Lists.html
http://geant4.web.cern.ch/geant4/physics_lists
- [biblio.g4doc-nist-materials] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/Detector/detector.html>
- [biblio.g4doc-solids] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/Detector/geomSolids.html>
- [Hounsfield_Schneider] Schneider U., Pedroni E., Lomax A., *The calibration of CT Hounsfield units for radiotherapy treatment planning*,
- [Hounsfield_duPlessis] du Plessis F. C. P., Willemse C. A., Loetter M.G., *The indirect use of CT numbers to establish material properties needed for Monte Carlo calculation of dose distributions in patients*, Med. Phys. 25:1195-201 (1998)
- [biblio.g4doc-scoring] <http://geant4.web.cern.ch/geant4/UserDocumentation/UsersGuides/ForApplicationDeveloper/html/Fundamentals/biasing.html#scoring-geometrical-importance-sampling-and-weight-roulette>

- [biblio.beamnrc] <http://www.irs.inms.nrc.ca/BEAM/beamhome.html>
- [biblio.iaephsp] <http://www-nds.iaea.org/phsp/phsp.htmlx>
- [biblio.dbs] Kawrakow I., Rogers D. W. O., Walters B. R. B., *Large efficiency improvements in BEAMnrc using directional bremsstrahlung splitting*, Med. Phys. 31(10):2883-98 (2004)
- [biblio.STIR] <http://stir.sourceforge.net/main.htm>
- [biblio.SSRB] Daube-Witherspoon M. E., Muehllehner G., *Treatment of axial data in three-dimensional PET*, J. Nucl. Med., vol. 28, pp. 1717-24 (1987)
- [biblio.imageRecoAlgo] Herman G. T., *Fundamentals of Computerized Tomography: Image Reconstruction from Projections (2nd ed.)*, Springer, ISBN 978-1-85233-617-2 (2009)
- [biblio.toft] Toft P., *The Radon Transform - Theory and Implementation*, Ph.D. thesis, Department of Mathematical Modelling, Technical University of Denmark, 326 pages (1996). <http://petertoft.dk/PhD>
- [biblio.AMIDE] *AMIDE: Amide's a Medical Imaging Data Examiner* <http://amide.sourceforge.net/>
- [biblio.ImageJ] *ImageJ: Processing and Analysis in Java* <http://rsbweb.nih.gov/ij>
- [biblio.Interfile] <http://www.med.harvard.edu/JPNM/ij/plugins/Interfile.html>
- [biblio.SOEpaper] Andreyev A., Arkadiusz S., Celler A., *Stochastic Image Reconstruction Method for Compton Camera*, IEEE Nuclear Science Symposium Conference Record (2009)