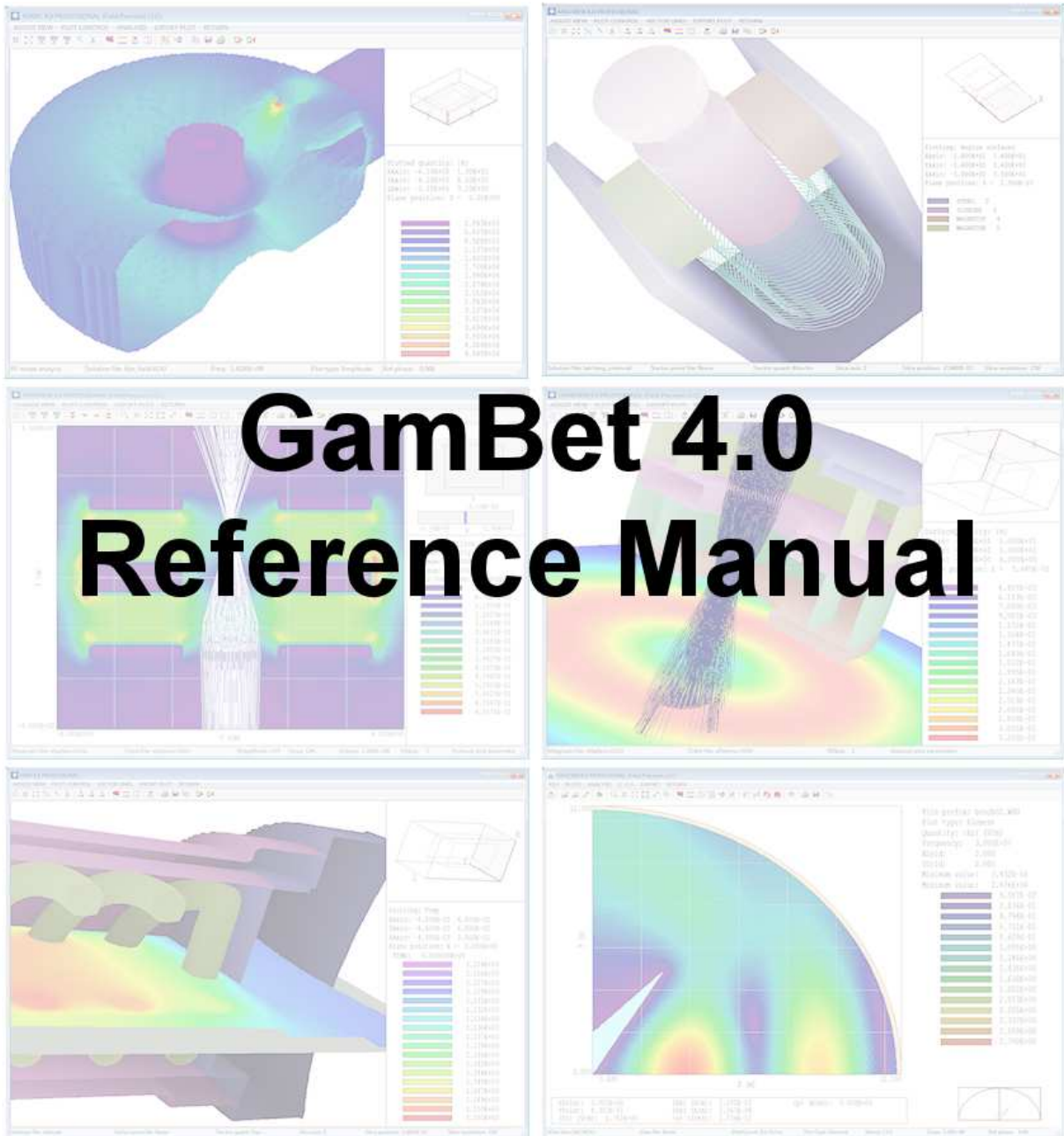




Field Precision LLC



PO Box 13595, Albuquerque, NM 87192 U.S.A.
Telephone: +1-505-220-3875
Email: techinfo@fieldp.com URL: <http://www.fieldp.com>

Read this first

With the impressive graphics and computational power of modern computer codes, we tend to forget that they don't always give the right answer. Sources of error may include invalid input parameters, misinterpretation of results and program bugs. Computer "simulations" should not be considered the absolute arbiter in technical applications. Cross checks and careful analysis are essential. This is particularly important in radiation transport calculations that could affect personnel safety.

- **GamBet** can be a useful aid in the design of radiation shielding, but the simulations alone are insufficient. The results must be combined with analytic estimates, practical guidelines and careful monitoring to ensure safety.
- Although **GamBet** is well suited to research investigations of new approaches to radiation therapy, the code was not designed for planning radiation treatments in a clinical setting. **GamBet is not certified or recommended for use in clinical radiation therapy.**

Although we make all possible efforts to improve and to correct **GamBet**, it is impossible to guarantee that a code of its complexity is absolutely free of error. The user is responsible for checking the validity and inputs and results. Field Precision assumes no liability for applications of **GamBet** beyond a refund of the purchase price.

Contents

1	Introduction	5
2	GBLaunch	8
2.1	Setting the program and data directories	8
2.2	Creating and running tasks	8
2.3	Tools	10
3	Simulation modes	12
4	Structure of the GamBet control script	14
5	Using the setup dialog to create a script	16
6	Geometry commands	22
6.1	Loading the geometry file	23
6.2	Loading and modifying electric field information	23
6.3	Loading and modifying magnetic field information	24
7	Composition commands	26
8	Source commands	29
8.1	Particle input from source files	30
8.2	Particle input from Trak and OmniTrak	31
8.3	Number of showers and source properties	32
9	Process commands	34
9.1	Penelope control	34
9.2	Variance reduction control	36
9.3	Simulation control	37
10	Running GamBet: interactive and background modes	41
11	GamBet: parallel operation	44
12	Input/output files	48
13	Contents of the listing file	51
13.1	Source information	51
13.2	Penelope setup information	51
13.3	Primary particle statistics	52
13.4	Energy conservation	53
13.5	Dose assignment statistics	53
13.6	Particle flux statistics	54

14 GBView2	55
14.1 File menu	55
14.2 Spatial plots menu	56
14.3 Dose analysis menu	59
14.4 Scan plot menu	61
15 GBView3	63
15.1 File menu	63
15.2 Dose analysis menu	64
15.3 Plane plots	66
15.4 Slice plots	68
15.5 Analysis in a slice	70
15.6 Surface plots	71
15.7 Saving and loading views	72
16 GBView2 and GBView3 script operation	74
17 Penelope predefined materials	76
17.1 Elements	76
17.2 Compounds and mixtures	77
18 Penelope data files	82

1 Introduction

GamBet (gamma and beta particles) is a comprehensive software suite to simulate the transport of energetic electrons, photons and positrons through matter over the energy range from 50 eV to 1 GeV. The program tracks the history of primary and secondary particles produced by material interactions by the Monte Carlo method. **GamBet** is an essential tool for applications such as X-ray imaging, radiation detectors, electron-beam generation of bremsstrahlung radiation and research on radiation medical treatments. **GamBet** may be employed as an extension to the **Trak** and **OmniTrak** charged-particle codes or as a stand-alone Monte Carlo simulator. The program produces files of power deposition that may be ported to **TDiff** or **HeatWave** for thermal analyses of radiation effects.

The program `gambet.exe` combines Field Precision technology developed for finite-element codes with the **Penelope** radiation-physics package¹. The **Penelope** routines perform the Monte Carlo tasks in **GamBet**: generation of atomic cross sections, prediction of single-particle interactions with matter and the creation of secondary particles. The Field Precision routines handle run control and all matters related to the division of space. Tasks include the organization of input/output data for large distributions of particles, calculation of statistics, identification of material boundaries, generation of escape-particle records, variance reduction techniques, creation of records of spatial variations of dose and control of parallel operation. Other programs of the **GamBet** package address mesh generation and interactive analysis of results. The basic **GamBet** suite includes the components listed in Table 1. Table 2 lists the Field Precision programs that produce compatible input for **GamBet**.

GamBet employs a unique approach to Monte Carlo calculations with several advantages over other programs:

1. Conformal meshes are used for the division of the material volume, giving high-accuracy representations of slanted or curved boundaries.
2. The effects of arbitrary 2D and/or 3D electric and magnetic fields may be included.
3. Sophisticated graphical pre- and post-processors are available for mesh generation and the analysis of dose distributions and particle orbits.
4. The program accepts direct input of electron or positron distributions from the **Trak** and **OmniTrak** codes.
5. Dose quantities are recorded on the conformal mesh; therefore, each element acts as a subzone. The user can make analysis decisions in the post-processing stage, reducing the need to re-run simulations.
6. Dose distribution files may be ported directly to **TDiff** and **HeatWave** for static or dynamic thermal calculations.

¹ Francesc Salvat, Jos M. Fernández-Varea, and Josep Sempau, **PENELOPE-2006: A Code System for Monte Carlo Simulation of Electron and Photon Transport**, (Proceedings of a Workshop/Training Course, July 4-7, 2006), NEA Number 6222.

Table 1: Components of the **GamBet** code suite

Program	Function
<code>mesh.exe</code>	Definition of 2D geometries, preparation of conformal triangular meshes.
<code>geometer.exe</code>	Interactive definition of 3D geometries
<code>metamesh.exe</code>	Creation of 3D conformal hexahedron meshes from geometric information
<code>gambet.exe</code>	Main technical program for Monte Carlo simulations
<code>gambetmp.exe</code>	A background version of the technical program called for runs with parallel processing.
<code>gbview2.exe</code>	Interactive post-processor for plots and analysis of 2D dose files and trajectory records
<code>gbview3.exe</code>	Interactive post-processor for plots and analysis of 3D dose files and trajectory records
<code>gendist.exe</code>	Utility to create or to analyze large input particle distributions

Table 2: Field Precision programs that produce input files compatible with **GamBet**

Program	Function
<code>estat.exe</code>	2D electric field solutions
<code>permag.exe</code>	2D magnetic field solutions
<code>trak.exe</code>	Simulation of 2D electron guns and transport devices, creation of input particle distributions
<code>hiphi.exe</code>	3D electric field solutions
<code>magnum.exe</code>	3D magnetic field solutions
<code>omnitrak.exe</code>	Simulation of 3D electron guns and transport devices, creation of input particle distributions

7. Particle output files may be used as input for subsequent runs or ported to **Trak** and **OmniTrak**.
8. Advanced variance-reduction techniques are employed, including interaction forcing and splitting/reduction of particles that enter specified regions of the solution volume.
9. The code supports efficient parallel processing on multi-core Windows computers.

To install the **GamBet** package, download and run `gambetsetup.exe`. The installation utility creates a new directory and then adds programs, instruction manuals, material data files and examples. The executables and the instruction manuals are located in the directory `C:\GAMBET`. The Penelope data files are in the directory `C:\GAMBET\REFERENCE`. The installer also adds a shortcut to `gblaunch.exe` on the desktop.

The programs must be activated before they can be used. Instructions are supplied with the package. Note that activation is independent of installation. You can update the programs without affecting activation of the computer. To avoid update problems, we recommended that you do not store data and other files to the program directory `C:\GAMBET`.

2 GBLaunch

The first step to run any of the programs in the **GamBet** package is to set up `gblaunch.exe`, the program controller. The utility helps you to organize your work in two ways:

- Only one shortcut is required on your desktop, minimizing clutter.
- All programs open in a specified data directory, eliminating redundant trips through the directory tree.

2.1 Setting the program and data directories

Figure 1 shows the screen display of the **GBLaunch** controller. To begin, click the *Program folder* button in the *Control* group. In the dialog move to the directory that contains the executables and click *OK*. The default location is `\fieldp\gambet`. The buttons in the technical group should all become active. An inactive button indicates a missing executable. Setting the program directory also sets a registry entry so that **GamBet** can locate the folder that contains Penelope data files.

It is a good practice is to collect all input and output files for related **GamBet** solutions in a specific data directory. Use the *Data Folder* button to set the current location. Programs that are opened after the change will read and write to the directory. (Note that the setting does not affect previously-opened programs.) The *Information* area at the bottom of the control box shows: 1) the current data directory, 2) the last operation performed and 3) the program directory.

2.2 Creating and running tasks

GamBet programs are optimized for the latest generation of multi-core or multi-processor PCs. The Professional solution programs (**MetaMesh**, **GamBet**,) feature true parallel operation. A second feature in both the Basic and Professional programs is the capability to run multiple independent calculations simultaneously. All solution programs can run in the background if launched from a Windows batch file. Background operation is automatic and faster than running in a window. The *Create task* and *Run task* commands in **GamBet** make it easy to use batch files. With the commands you can 1) quickly define multi-step calculations (*tasks*) in an interactive dialog, 2) launch simultaneous tasks in the background and 3) find out which tasks are running.

The *Create task* button calls up the dialog of Figure 2. Supply a file prefix `FPREFIX` that indicates the function of the task. The task information will be stored in a DOS batch file `FPREFIX.BAT` created in the current **GamBet** data directory. Commands in the file are compatible with all recent Windows versions including Windows 7. Each row represents an operation (batch file command). The first column defines the action. Clicking on a cell brings up a menu that includes all **GamBet** programs capable of background operation. In addition, several relatively safe DOS commands are included (`ERASE`, `COPY`, `MOVE`, `RENAME` and `REM`). All commands operate on a file (*FileIn* column). The DOS commands `COPY`, `MOVE` and `RENAME`

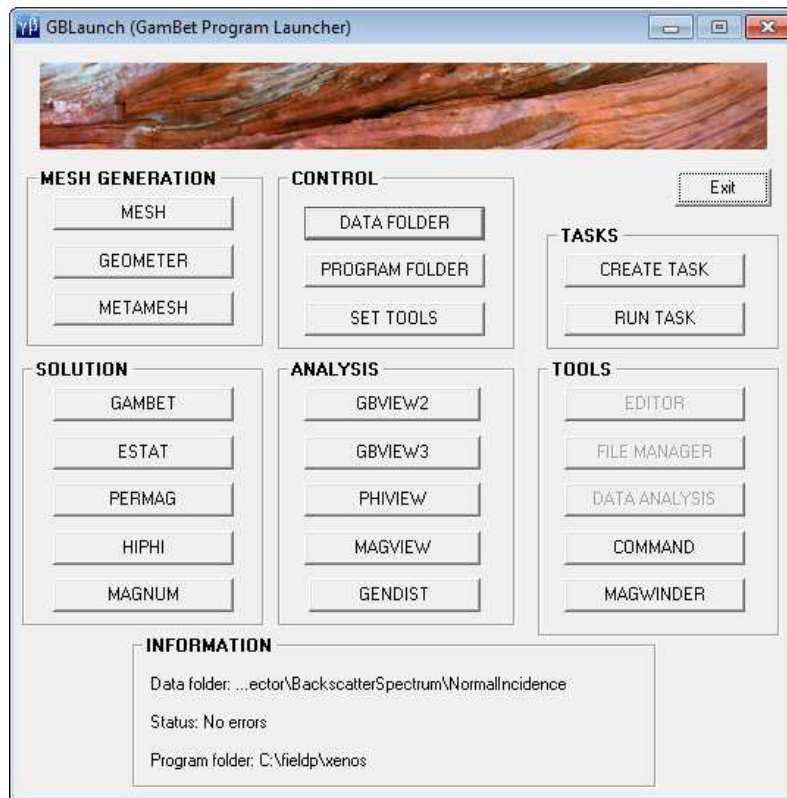


Figure 1: GamBet program launcher

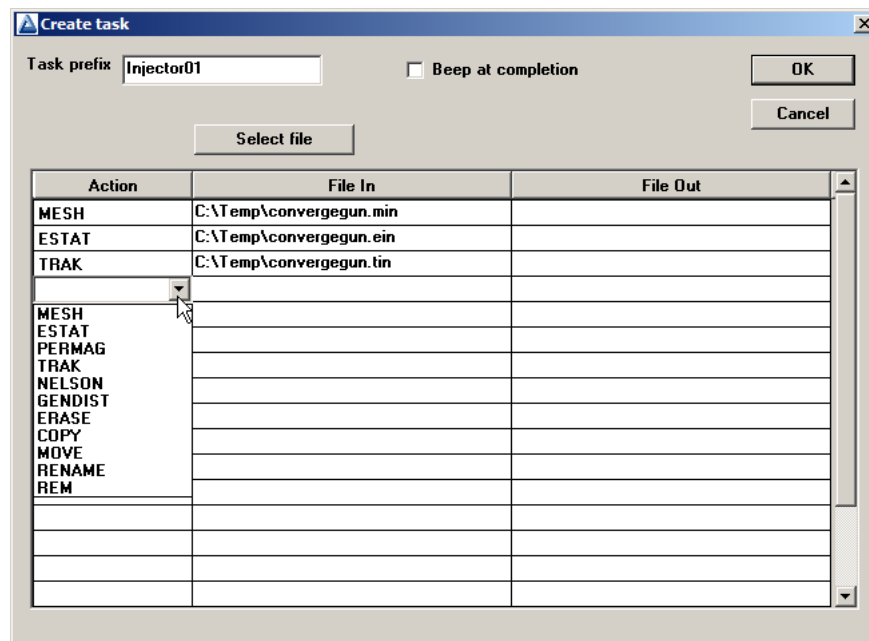


Figure 2: Create task dialog

require a second file name (*FileOut* column). You can type file names in the cells. By default, the files are in the **GamBet** working directory, but you can include path information if the files are in other directories. Alternatively, you can click in a cell and then pick the *Select file* command to use the standard Windows dialog for choosing files anywhere on the computer. For the **GamBet** programs, the dialog displays only files with appropriate suffixes (*e.g.*, *.GIN for **E**).

Click the *OK* button when the sequence is complete to create the batch file. Here is an example:

```
REM GamBet batch file, Field Precision
START /B /WAIT C:\fieldp\gambet\mesh64.exe C:\Temp\convergegun
START /B /WAIT C:\fieldp\gambet\gambet64.exe C:\Temp\convergegun
START /B /WAIT C:\fieldp\gambet\gbview264.exe C:\Temp\convergegun
ERASE *.?ls
START /B /WAIT C:\fieldp\gambet\notify.exe
IF EXIST Electrode01.ACTIVE ERASE Electrode01.ACTIVE
```

The operations listed perform a complete **GamBet** calculation in the background and then erase all listing files. The example has some notable features:

- The operations are performed sequentially because data from one operation may be used in the next. To run calculations in parallel, define and run multiple tasks.
- You can modify the file with an editor if you are familiar with DOS commands.
- The DOS commands recognize the standard wildcard conventions (* for any character grouping, ? for any character).
- The programs adds the command `notify.exe` to the task sequence if *Audio alarm* was checked. In this case, the computer beeps when a task is completed.
- The final command to erase a file `FPREFIX.ACTIVE` is added to all batch files. The presence of the file indicates that the task is running.

Click the *Run task* button when you have created tasks or moved predefined task files to the data directory. The dialog (Figure 3) organizes tasks into two groups: ones that are available to run and ones that are currently running (*i.e.*, `FPREFIX.ACTIVE` has been detected). To launch a task, choose one from the top list and click *OK*. The program creates a file `FPREFIX.ACTIVE` and runs the batch file. The program sequence runs silently in the background. In the meantime, you can prepare other inputs or run other tasks.

2.3 Tools

You will probably use additional software tools working with **GamBet** and other technical programs. You can set **GamBet** to launch your favorite utility programs. To define a text editor, press the *Setup tools* button in the *Control* section. In the dialog, choose the *Editor* option. Then, move to the appropriate directory and select the program. When you exit the procedure, the *Editor* button in the *Tools* section becomes active.



Figure 3: Run task dialog

You can use the *Set tools* command to define two other utilities. For the *File manager* option, select a program like **Windows Explorer** or **FreeCommander**. Use the *Data analysis* button for a spreadsheet, plotting program or mathematical analysis software. The *textitCommand* button opens a DOS window if you want to run programs from the command prompt or under the control of your own batch files.

3 Simulation modes

There are three available modes for a **GamBet** simulation:

- SingleParticle
- ContinuousBeam
- PulsedBeam

The mode determines units of data quantities recorded in the output files. **GamBet** sets the mode depending on entries in the source or particle input lists and the presence of the TPULSE command (see Table 3). The single-particle mode corresponds to a standard Monte Carlo run where quantities are weighted according to the number of input particles. In the continuous-beam mode, a current or flux is associated with each model particle. In this case, **GamBet** can calculate dose rates and particle flux between regions. In the pulsed-beam mode, the model particle current is active for a specified time. In this case, the code calculates dose and integrated flux (total number).

Table 3: **GamBet** simulation modes

Mode	Current/flux entry in PLIST or SLIST data	TPULSE specified
SingleParticle	No	(No effect)
ContinuousBeam	Yes	No
PulsedBeam	Yes	Yes

The units of deposited dose (Table 4) depend on the calculation mode. The standard SI dose unit is the gray ($1 \text{ Gy} = 1 \text{ J/kg} = 100 \text{ rads}$). The following considerations apply to the calculation of deposited dose in planar 2D calculations.

- In the single-particle mode, each incident primary particle is treated as one particle/m uniformly distributed in z .
- In the continuous-beam and pulsed-beam modes, the flux value for a primary particle is in A/m for electrons and positrons and particles/s/m for photons. The flux is assumed to be uniformly distributed in z .

The unit of particle flux between regions is particles/primary for all calculation modes and symmetries. Table 5 shows units of energy flux between regions.

Table 4: Recorded dose quantities

Mode	Recorded quantity
SingleParticle	eV/kg/primary
ContinuousBeam	J/kg/s (Gy/s)
PulsedBeam	J/kg (Gy)

Table 5: Recorded energy-flux quantities

Mode	Symmetry	Recorded quantity
SingleParticle	Planar2D,Cylindrical2D,3D	eV/primary
ContinuousBeam	Planar2D	J/s/m
	Cylindrical2D,3D	J/s
PulsedBeam	Planar2D	J/m
	Cylindrical2D,3D	J

4 Structure of the GamBet control script

The **GamBet** script is a text file with data lines containing commands and parameters. The program makes no distinction between upper and lower case. Entries on a line may be separated by the following delimiters:

- Space
- Comma [,]
- Tab character
- Colon [:]
- Equal sign [=]
- Left parenthesis [(
- Right parenthesis [)]

You may use any number of delimiters in a line. This means that you can add indentations and customize the appearance of the script. **GamBet** ignores blank lines and comment lines. Comment lines begin with the symbol '*' (asterisk).

Most parameters are real numbers. The following formats are valid.

- 1.000
- 5.67E6
- 6.8845E+09
- 5

The final number is interpreted as 5.0.

The **GamBet** script has the following organization:

```
GEOMETRY
  (Geometry commands)
END

COMPOSITION
  (Composition commands)
END

SOURCE
  (Source commands)
END

PROCESS
  (Process commands)
END
```

The four sections must appear in the order shown. Allowed commands may be placed in any order within a section. **GamBet** reads and analyzes all commands of a section before proceeding with calculations. The exception is the **DUnit** command for coordinate conversion. The command affects only subsequent commands.

The script sections have the following functions:

GEOMETRY Load a 2D or 3D mesh that defines the material geometry. Optionally, you can also load **TriComp** or **AMaze** solution files to define applied electric and magnetic fields.

COMPOSITION Define material properties for Penelope calculations and associate materials with the regions of the geometric mesh.

SOURCE Specify the properties of incident particles (electrons, photons and positrons).

PROCESS Control the Monte Carlo calculation.

The following example illustrates a simple **GamBet** script:

```
* File: COPPERSHEET.GIN
GEOMETRY
  DUnit = 1.0E6
  GFile2D = CopperSheet.MOU (Rect)
END
COMPOSITION
  Material = 29
  Region(1) = 1
END
SOURCE
  SList
    E  20.0E6  0.00 0.00 0.00  1.00  0.00  0.00
  End
  NPMult = 100000
END
PROCESS
  EAbs(Electron) = 1.0E5
  EAbs(Photon) = 1.0E5
  EAbs(Positron) = 1.0E5
  DsMax(1) = 0.20
  PlotOn(1)
END
ENDFILE
```

5 Using the setup dialog to create a script

The setup dialog is a quick and easy way to construct basic **GamBet** control scripts. The interactive environment helps you remember commands and syntax rules. You can use the dialog to create a basic structure and then add details or modify parameters with a text editor. This chapter briefly summarizes the function of fields in the dialog. Chapters 6 through 9 give detailed information on standard and advanced commands.

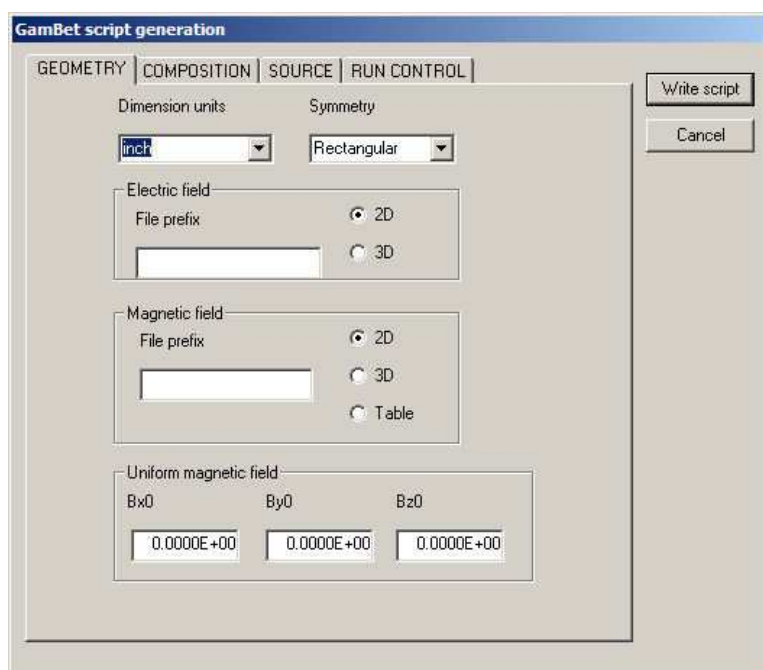


Figure 4: Setup dialog, Geometry tab

GamBet calculations require a geometry mesh. A two-dimensional mesh file (`FPREFIX.MOU`) or a three-dimensional file (`FPREFIX.MDF`) must be available to use the setup dialog. In the main menu, click on the **Setup** command or tool. In the file selection dialog, choose the input mesh file. **GamBet** checks the number of regions in the mesh and reads the region names if they are available. If the file is a valid mesh, the dialog of Fig. 4 appears. Note the tabs at the top of window: there is one tab for each of the command groups in the script (Chap. 4).

The *Geometry* tab is active in Figure 4. The following fields apply to all scripts.

Dimension units

This entry informs **GamBet** of the length units used in mesh file. Click the down-arrow to display the full list of options. Choose *Custom* if the desired unit is not included in the list. In this case, the program makes a dummy entry in the script that you can complete with a text editor.

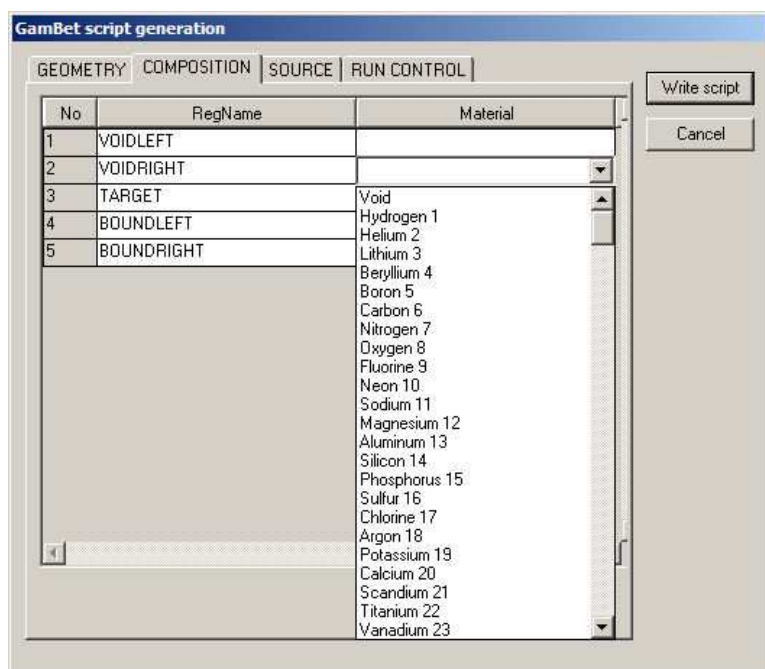


Figure 5: Setup dialog, *Composition* tab

Symmetry

This field is active only for two-dimensional meshes. The *Rectangular* option implies that there are variations in x and y with infinite length along z . For the *Cylindrical* option, the system has azimuthal symmetry with variations along r and z .

Enter values in fields of the electric and magnetic field groups if you want to load information from output files generated by **EStat**, **HiPhi**, **PerMag** or **Magnum**. Electrostatic and magnetostatic fields act on electrons and positrons in the calculation.

File prefix

Enter the prefix of the field data file. The string must be from 1 to 46 characters in length. Depending on the setting of 2D/3D radio buttons, the program adds the appropriate suffix. For electric fields, the resulting file names are **FPREFIX.EOU** (**EStat**, 2D) and **FPREFIX.HOU** (**HiPhi**, 3D). The file names for magnetic fields are **FPREFIX.POU** (**PerMag**, 2D) and **FPREFIX.MOU** (**Magnum**, 3D). For the *Table* option in the magnetic field group, the resulting file name is **FPREFIX.TAB**.

Bx0, By0, Bz0

Define uniform values of magnetic field along the x , y and z direction. Enter component values in tesla.

Figure 5 shows the display for the second tab. Values set the the material composition of regions in the simulation volume. The grid has one row for each region of the mesh. The columns of the grid show the region number, the region name (if available) and the material composition. To define a material, move the mouse into a *Material* box and click the left button.

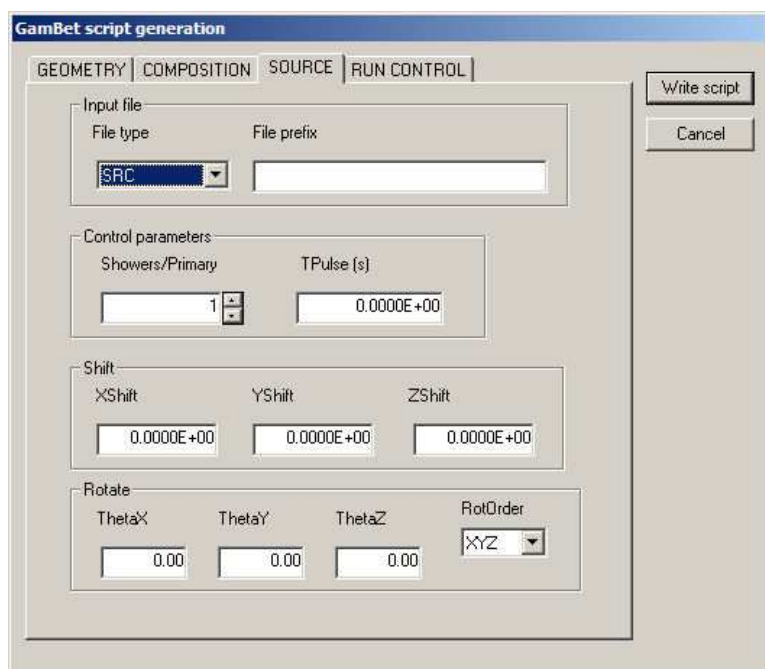


Figure 6: Setup dialog, *Source* tab

The drop-down list shows the full set of Penelope materials (Chap. 17) and the additional choice of *Void* (vacuum). Scroll through the list to find an appropriate material and then click the left mouse button to set the composition for the box. Note that the Penelope numbers initially follow the atomic number of elements. After that, compounds are listed roughly alphabetically. Chapter 17 summarizes the choices. Each box in the column must be specified. The program issues an error message if there is a missing definition for a region.

Fields in the *Source* tab define the properties of injected particles. The following quantities must be defined for each primary particle: type (electron, photon or positron), kinetic energy, position and direction. The quantities are contained in a list with one data line for each primary particle. The list may be either incorporated in the script or imported from a separate file. There are two list formats:

- The native **GamBet** format, where the list file has a name of the form `FPREFIX.SRC` (Sect. 8.1).
- The input/output format of **Trak** and **OmniTrak**, where the list file has a name of the form `FPREFIX.PRT` (Sect. 8.2).

The following two fields appear in the *Input file* group.

File type

Specifies whether input will be in the *SRC* or *PRT* format.

File prefix

The prefix of a file (FPREFIX.SRC or FPREFIX.PRT) of particle data lines available in the working directory. Sections 8.1 and 8.2 describe the file formats. If there is no entry in this file, **GamBet** creates a dummy entry in the script of the form:

```
SList
*   Type   Kinetic   x   y   z   ux   uy   uz   [Current/flux]
*           energy
*   =====
*   (Add particle parameters)
End
```

In this case, you must add particle data lines with a text editor.

The *Control parameters* group has the following entries:

Showers/primary

Set the parameter $NPMult$, the number of showers to initiate per primary particle. Larger values give improved statistics. Regarding the assignment of deposited dose, if a primary particle carries current I , then the current per shower particle is I/N_{pmult} .

TPulse

Set the pulselength Δt_p for the input beam of primary particles. Enter the value in seconds. This parameter is active only if current or flux is associated with the primary particles. If a value is supplied, **GamBet** records accumulated dose in the output file. If Δt_p is not defined, the program records the dose rate.

The entries in the final two groups (*Shift* and *Rotation*) may be used to apply global translations or rotations to the primary particles.

XShift, YShift, ZShift

Shift the initial positions of particles relative to the geometry mesh. Values are modified according to $x' = x + X_{shift}, \dots$. Enter values in units set by the specified dimension units. Note that rotations are applied before shifts.

XRot, YRot, ZRot, RotOrder

Rotate the particle direction vectors relative to the geometry mesh. Enter the values of angles in degrees. The optional string parameter *RotOrder* specifies the order of rotations. Rotations are applied before shifts.

Values in fields of the *Run* control tab of Fig. 7 control the Monte Carlo calculation.

Maximum energy

The maximum energy E_{max} for the calculation of cross sections by Penelope. Enter the value

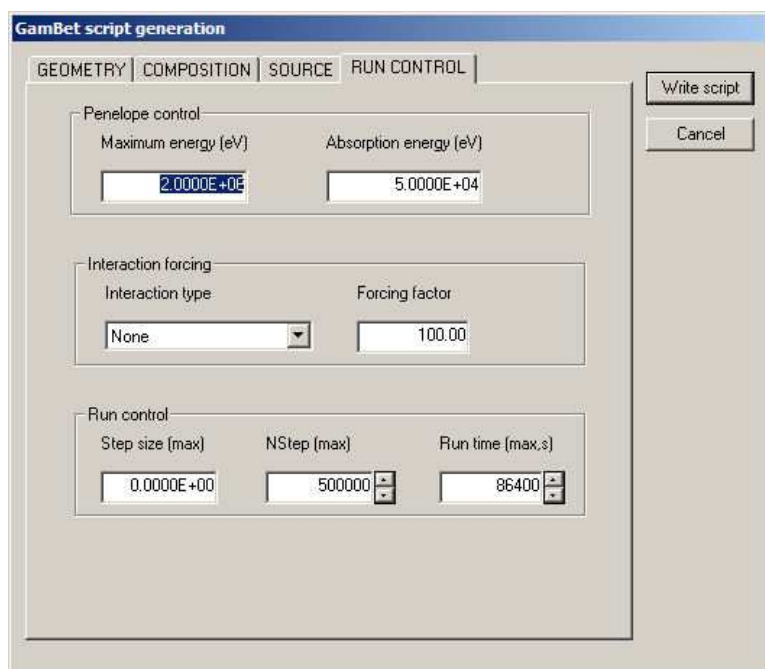


Figure 7: Setup dialog, *Process* tab

in eV (electron volts). Usually, E_{max} is slightly higher than the maximum energy of primary particles. In a solution with electric fields be sure to include the possibility of electron or positron acceleration.

Absorption energy

A particle is absorbed in a material region when its energy drops below a value E_{abs} . In this case, the remaining particle kinetic energy is deposited in the currently-occupied element. The value of E_{abs} must be greater than or equal to 50.0 eV, the lower limit in the Penelope tables. Higher values substantially reduce the run time.

Interaction forcing improves statistics for low-probability events. The program adjusts particle weights and interaction probabilities to maintain unbiased statistics.

Interaction type

The two choices are bremsstrahlung and pair production. The first option is useful for modeling X-ray targets while the second option may be applied to positron sources.

Forcing factor

Enter a real number greater than 1.0, typically 10.0 to 100.0.

The fields in the last group control aspects of the Monte Carlo simulation.

Step size

This field is the maximum allowed transit distance for particles per event step. Enter a value in the specified dimension units. The maximum step size should be about 1/10 of the dimension of the smallest material region.

NStep

The maximum number of event steps in a shower. The purpose of the limit is to prevent infinite calculations (*i.e.*, electron in a void trapped by a magnetic field). The default value is 500,000. Increase the number if particles stop early. You can check for the condition by inspecting the energy conservation statistics in the listing file (GLS).

Run time

The maximum time for the calculation. Enter the integer value in seconds. **GamBet** stops at the maximum run time and records data for the completed showers. This value may be useful for long runs under batch file control.

6 Geometry commands

COMMAND SUMMARY

```
DUNIT DUnit
GFILE2D FileName [Rect, Cylin]
GFILE3D FileName
EFILE2D FileName [EMult]
EFILE2D FileName BTHETA [EMult] [BBMult]
EFILE3D FileName [EMult]
BFILE2D FileName [BMult]
BFILE3D FileName [BMult]
BTABLE: FileName [ZOffset,Zult, BOffset, BMult]
BUNI: Bx0 By0 Bz0
SHIFT [E,B] XShift YShift ZShift
ROTATE [E,B] RotX, Roty, Rotz
```

GamBet can employ geometric and field information from three independent conformal meshes.

GFILE [Required] Geometry of materials used for Penelope operations.

EFILE [Optional] Applied electric field.

BFILE [Optional] Applied magnetic field.

You can combine 2D and 3D meshes in a single simulation. You can also combine 2D meshes having different symmetries (planar or cylindrical). The output dose file has the same dimensionality and symmetry as the **GFILE**.

The following considerations apply when you combine meshes.

- By default the origins of the meshes are coincident. Here the origin is the point where $(x, y, z) = (0, 0, 0)$. The **GFILE** defines the reference coordinate system. Use the **SHIFT** command to translate the **EFILE** or **BFILE** meshes.
- By default, the coordinate axes are colinear. A cylindrical mesh has z along the z direction in 3D space and r lies in the x - y plane. A planar mesh has variations in x - y and infinite length along z . Use the **ROTATE** command to rotate the **EFILE** and **BFILE** meshes relative to the **GFILE** mesh.
- If the **GFILE** is two-dimensional, the output dose distributions have two-dimensional symmetry, even if the particle distribution in the simulation is not symmetric. Non-symmetric particle distributions may be specified in the **SOURCE** section or could result from motion in a field with different symmetry. The dose in a file with planar symmetry is integrated along z ; therefore, only the x - y variations are displayed. In a cylindrical file, energy is assigned to r - z elements independent of the model particle azimuth.

DUNIT Unit

Enter a conversion factor for coordinates that appear in subsequent commands. The conversion also applies to coordinates in the **GFILE** for input from **Mesh** and **MetaMesh**. In this case, ensure that the **DUNIT** command appears before the **GFILE2D** or **GFILE3D** commands. The **Unit** argument may be one of the following strings for common units: **angstrom**, **nanometer**, **micrometer**, **mil**, **mm**, **cm**, **inch**, **foot**, **yard**, **meter**, **kilometer** or **mile**. For custom units, enter a real number equal to the number of **Mesh** or **MetaMesh** units per meter. The quantity is the number of length units per meter. For example, if coordinates in the **GFILE** were specified in microns, set $DUnit = 1.0 \times 10^6$ and enter coordinates in subsequent commands in microns.

6.1 Loading the geometry file

GFILE2D FileName [Rect, Cylin]

Define the material geometry for the Monte Carlo calculation. The command requires an output file from **Mesh** (**FName.MOU**) or the **TriComp** solution programs **EStat** (**FName.EOU**), **PerMag** (**FName.POU**) or **Pulse** (**FName.POU**). The file must be available in the working directory. The keywords **RECT** or **CYLIN** must be included when loading a file from the 2D Mesh program. In the **RECT** option, the program interprets the geometry as variable in x - y and infinite in z . In the **CYLIN** option, the program interprets the geometry as varying in r - z with symmetry along θ . When loading information from a field solution file, **Gambet** extracts the mesh information and reads a file parameter to determine the symmetry.

GFILE3D FileName

Define the material geometry used for the Monte Carlo calculation. The command requires an output file from **MetaMesh** (**FName.MDF**). The program interprets the geometry as varying in x - y - z .

6.2 Loading and modifying electric field information

EFILE2D FileName [EMult]

Load an electric field solution from **EStat**. Note that the mesh need not have the same symmetry or cover the same volume as the **GFile**. The electric field equals zero at points outside the **EFile** boundaries. The optional parameter **EMult** is a field scaling factor. Values of electrostatic potential are multiplied by **EMult** as they are loaded. For example, a value **EMult** = 0.5 reduces electric field components by a factor of 2.0.

EFILE2D FileName BTHETA [EMult] [BBMult]

The special form of the command loads an electric field solution created with the **EDUMP** command by the **Trak** code running in the **RELBEAM** mode. **Trak** has been modified to record values of the beam-generated toroidal field (B_θ) at nodes in addition to the electrostatic potential. The extra information will be loaded into **Gambet** in response to the keyword **BTHETA**. In this

case, electron orbits will include the effect of the toroidal magnetic field. Values of ϕ and B_θ may be scaled with the optional parameters **EMult** and **BBMult**.

EFILE3D FileName [EMult]

Load an electric field solution from **HiPhi**. Note that the mesh need not have the same symmetry or cover the same volume as the **GFile**. The electric field equals zero at points outside the **EFile** boundaries. The optional parameter **EMult** is a field scaling factor.

ROTATE E RotEx, RotEy, RotEz [RotOrder]

Rotate the electric field solution relative to the geometry mesh. Enter the values of angles in degrees. The optional string parameter **RotOrder** specifies the order of rotations (*i.e.*, 'ZX', 'ZYX'). Only the operations specified in **RotOrder** are performed. The default is **RotOrder** = 'XYZ'. Note that rotations are applied before translations.

SHIFT E XEShift YEShift ZEShift

Shift coordinates of the electric field solution relative to the geometry mesh. Enter values in units set by **DUnit**. Note that rotations are applied before translations.

6.3 Loading and modifying magnetic field information

BFILE2D FileName [BMult]

Load a magnetic field solution from **PerMag** or **Pulse**. Note that the mesh need not have the same symmetry or cover the same volume as the **GFile**. The magnetic field equals zero at points outside the **BFile** boundaries. The optional parameter **BMult** is a field scaling factor.

BFILE3D FileName [BMult]

Load a magnetic field solution from **Magnum**. Note that the mesh need not have the same symmetry or cover the same volume as the **GFile**. The magnetic field equals zero at points outside the **BFile** boundaries. The optional parameter **BMult** is a field scaling factor.

BTABLE: FileName [ZOffset,ZMult, BOffSet, BMult]

Determine an applied solenoid-type magnetic field from a table of values along the z axis at $r = 0.0$. The field must have cylindrical symmetry. The parameter **FileName** is the full name of a tabular file. The file contains up to 256 data lines of the form:

```
z    Bz(0,z)
```

The file terminates with an **EndFile** command. The table may also contain comment and blank lines. The positions z should be specified in units set by the current setting of **DUnit**. Supply values of B_z in tesla, or you can use the **BMult** parameter to convert from other units. For example, if the data are given in gauss, use **BMult**= 1.0×10^{-4} . The following illustrates a table


```

* Table from SOL_LENS.BOU
*      z      Bz
* =====
-9.000E+00  4.421E-06
-8.852E+00  5.926E-05
-8.703E+00  1.173E-04
-8.555E+00  1.780E-04
...
 9.258E+00  1.014E-04
 9.406E+00  7.838E-05
 9.555E+00  5.670E-05
 9.703E+00  3.685E-05
 9.852E+00  1.767E-05
1.000E+01  7.111E-11
ENDFILE

```

Optionally, you can supply four real numbers as parameters in the command: **ZOffSet**, **ZMult**, **BOffSet** and **BMult**. The parameters modify table values entered into the program according to

```

zprog = ZMult*ztab + ZOffSet,
Bzprog = BMult*Bztab + BOffSet

```

A cubic spline routine is used to interpolate values of B_z along the axis and numerical derivatives are used to calculate off-axis values $B_r(r, z)$ and $B_z(r, z)$. As a result, estimates of B_z are limited to terms of order r^2 . This means that variations of B_r are limited to terms of order r (linear variation).

BUNI: Bx0 By0 Bz0

Set uniform components of the magnetic field. Enter the values in tesla. This command may appear in conjunction with **BFile** or **BTable** commands. In this case, the magnetic field is the sum of all sources.

ROTATE B RotBX, RotBy, RotBz

Rotate the magnetic field solution relative to the geometry mesh. Enter the values of angles in degrees. The optional string parameter **RotOrder** specifies the order of rotations (*i.e.*, 'ZX', 'ZYX'). Only the operations specified in **RotOrder** are performed. The default is **RotOrder** = 'XYZ'. Note that rotations are applied before translations.

SHIFT B XBSHift YBSHift ZBSHift

Shift coordinates of the magnetic field solution relative to the geometry mesh. Enter values in units set by **DUnit**. Note that rotations are applied before translations.

7 Composition commands

COMMAND SUMMARY

```
MATERIAL MatNo [CONDUCTOR,INSULATOR]
MATERIAL
    NAME
    COMPONENT [Symbol or atomic number] [Fraction]
    DENSITY
    CONDUCTOR
END
REGION RegNo MatNo
REGION RegNo VOID
REGION RegNo EXCLUDE
ECALC RegNo [ON, OFF]
BCALC RegNo [ON, OFF]
FULLINFO
```

Commands of the *Composition* section create the temporary file **GAMBET.MAT** (used for the Monte Carlo calculations) and associate regions of the geometric mesh with materials. Each *Material* statement defines a different material. Materials are numbered in the order they appear in script. The maximum number of materials is 25. You can associated multiple regions with a single material. There are two forms of the command:

MATERIAL MatNo [Conductor, Insulator]

The integer parameter *MatNo* corresponds to the number of a predefined Penelope material (1-279). For a list of available materials, see Sect. 17 or the **View material file** command in the **GamBet File** menu. The optional string parameter [Conductor or Insulator] overrides the default material state for the generation of Penelope cross sections (metallic elements in the range 1-99 are set as conductors, compound materials in the range 100-280 are set as insulators). If an **EFile** has been loaded, the **Conductor/Insulator** setting affects the default status for electric and magnetic field calculations inside the material. By default, the electric field is taken as zero inside the volume of conducting regions of the geometric mesh, independent of region definitions in the electric field file. The electric field has non-zero default values inside insulators and voids. Static magnetic fields are unaffected by the conductivity status. The material status for field calculation may be changed with the **ECalc** and **BCalc** commands.

MATERIAL

NAME

COMPONENT [Chemical symbol or atomic number] [Fraction]

DENSITY

INSULATOR

END

A *Material* command with no parameters signals the beginning of a material section. You can define new materials with this construct. The section must terminate with the **End** command. The following commands may appear in any order within the material section:

NAME MatName

Enter a name for the material (1-80 characters). The name should not contain any of the standard delimiters. Examples of valid names: *Carbon_Dioxide*, *High_Density_Lucite*, ...

COMPONENT [Chemical symbol or atomic number] [Fraction]

A material section may contain up to 20 **Component** commands. Each command defines an elemental component of the material. The second parameter (string or integer) is the chemical symbol or the atomic number (*i.e.* Ga or 31, C or 12). The third parameter (real) is the number of atoms per molecule or stoichiometric fraction. Fractional values are permitted for alloys and mixtures. For example, the following lines describe acetic acid ($\text{CH}_3\text{CO}_2\text{H}$):

Component C 2.0

Component H 4.0

Component O 2.0

DENSITY

Material density in gm/cm^3 .

INSULATOR, CONDUCTOR

Conductivity state of the material

REGION RegNo MatNo

This command associates a region of the geometric mesh with a defined material. The integer parameter **RegNo** is the region number and **MatNo** is the material number. For example, to specify that region 6 consists of the third material to appear in the **Composition** section, use the command:

```
REGION(6) = 3
```

Gambet issues an error message if any regions remain unassigned or if *MatNo* exceeds the number of defined materials.

REGION RegNo VOID

Assign the *void* property to region *RegNo*. Particles move through voids without interaction. Electric and magnetic fields may act on electrons or positrons in voids. You can speed calculations by representing a region with infrequent interactions (*i.e.*, a low-density gas) as a void.

REGION RegNo EXCLUDE

Assign the *exclusion* property to region *RegNo*. Particles that enter an exclusion region are immediately stopped and their properties at the region boundary are recorded in the escape file. In other words, they are treated in the same way as particles that leave the solution volume. This command is useful for defining a non-planar exit boundary in a three-dimensional solution. It may also be applied to find the distribution of particles entering a region without tracking interactions inside the region.

ECALC RegNo [ON, OFF]

Over-ride default settings to specify whether electric fields are zero [*Off*] or non-zero [*On*] inside a region. This command has an effect only when an **EFile** has been loaded.

BCALC RegNo [ON, OFF]

Over-ride default settings to specify whether magnetic fields are zero [*Off*] or non-zero [*On*] inside a region. This command affects all sources of magnetic field (**BFILE**, **BTABLE**, **BUNI**).

FULLINFO

If this command is issued, **GamBet** records full information reported by the Penelope sub-routines during the creation of the file **GAMBET.MAT**. This information is useful only to users familiar with the intricacies of Penelope.

8 Source commands

COMMAND SUMMARY

```
SLIST
  (Data lines)
END
SFILE SourceFilePrefix
PLIST
  (Data lines)
END
PFILE PartFilePrefix
NPMULT NPerPrimary
TPULSE TPulse
SHIFT XPShift YPShift ZPShift
ROTATE RotPX RotPY RotPZ [RotOrderP]
```

Source commands define the set of primary particles that initiate showers of electrons, photons and positrons. Here, we use the term *shower* to refer to the set of interactions and secondary particles that occur for each instance of an injected primary particle. For example, if there is a single primary particle and the parameter $N_{pmult} = 500$, then **GamBet** initiates 500 showers by injecting the primary particle 500 times with a statistical weighting factor of $1/500$.

There are four ways you can specify primary-particle information:

1. Lists of particles that you prepare with an editor, spreadsheet or your own software.
2. Source files with large numbers of particles generated by the **GenDist** utility from script commands.
3. The particle escape file from a previous **GamBet** run.
4. Output files of electron or positron parameters from the programs **Trak** or **OmniTrak**.

The maximum number of unique primary particles is 1,000,000. You can initiate multiple showers for each primary particle with the *NPMult* command.

It is important to clarify the definition of a primary particle:

- Each data line in the **SList**, **SFile**, **PList** or **PFile** commands defines a primary particle.
- The total particle flux into the solution volume is the sum of the fluxes of primary particles. For example, if the list contains 10 electrons that each carry 1 mA of current, the injected current is 10 mA.

- Primary particles may be split into multiple showers using the **NPMult** command for improved statistics. If $N_{pmult} > 1$, then each primary particle is injected $NPMult$ times with a weight of $1.0/N_{pmult}$. For example, if the list contains one electron that carries current $2.0 \mu\text{A}$ and $N_{pmult} = 20$, then 20 shower calculations are initiated by identical electrons that each carry $0.1 \mu\text{A}$.

8.1 Particle input from source files

SLIST

(Data lines)

END

The purpose of the *SList* structure is to read input particle data from source data lines in the **Gambet** script. Each primary particle in the *SList* structure is defined by a data line. A line may contain 8 or 9 quantities:

```
Type  Kinetic  x    y    z    ux    uy    uz  [Current/flux]
      energy
```

Line entries may be separated by any valid delimiters. The quantities in the line have the following meanings:

Type

A string quantity that gives the type of particle. The following entries are recognized:

Electron: "1", "E" or "E-"

Photon: "2" or "P"

Positron: "3" or "E+"

Kinetic energy

The particle initial kinetic energy in electron volts ($1.0\text{eV} = 1.6021 \times 10^{-19}$ joules).

x, y, z

The initial particle position in meters or units set by **DUnit**. The initial position must be inside the boundaries of the geometric mesh. If you want to start particle at a boundary, pick a position slightly inside the solution volume.

ux, uy, uz

A vector parallel to the initial particle direction. **Gambet** converts the quantities to a unit vector.

Current/flux

An optional parameter to represent the current carried by primary electrons and positrons or the flux of photons. For cylindrical or 3D geometric meshes, enter the current in A or flux in photons/s. For a planar geometric mesh, enter the line current density J in A/m or the line flux in photons/s/m. Note: if a primary particle carries current I and **Gambet** creates $NPMult$ showers per primary, the input current assigned to each shower is I/N_{pmult} .

As an example, the following line designates a 1.7 MeV electron beam moving in the z direction at position (1.0, 2.0, 0.0) carrying a current of 1.0 μA :

```
E- 1.7E6 1.00 2.00 0.00 0.00 0.00 1.00 1.0E-6
```

SFILE SPrefix

This command directs **GamBet** to read particle data lines from an external file with a name of the form **SPrefix.SRC**. The file prefix contains from 1 to 32 characters. The file must be available in the working directory. A particle file may consist of blank lines, comments and from 1 to 20000 data lines. Data lines have the same format as those of the **SList** command. The file must terminate with the **End** or **EndFile** command.

8.2 Particle input from Trak and OmniTrak

Standard particle list files (**PName.PRT**) created by **Trak** and **OmniTrak** contain a data line for each particle. The list may contain blank and comment lines and must terminate with the **End** command. Comment lines are marked by the asterisk (*) symbol. Particle information lines may be inserted directly into the **GamBet** script with the **PList** construct or they can be read from an independent file with the **PFile** command. For both the **PList** and **PFile** commands, **GamBet** expects a series of text lines consisting of nine or ten real numbers.

PLIST

(Data lines)

END

The **PList** construct reads particle parameters from data lines in **Trak** or **OmniTrak** format in the **GamBet** script. Data lines have the following components:

```
Rest  Charge  Kinetic  x   y   z   ux   uy   uz   [Current]
mass                energy
```

Line entries may be separated by any valid delimiter. The quantities in the line have the following meanings:

Rest mass

The particle rest mass in AMU (atomic mass units), where $1.0 \text{ AMU} = 1.65979 \times 10^{27} \text{ kg}$. The rest mass of electron and positrons is $5.488 \times 10^{-4} \text{ AMU}$. If 0.00 appears in this position, **GamBet** will insert the rest mass of the electron/positron. The program issues an error message if the list contains particles other than electrons and positrons.

Charge

The particle charge in units of elementary charge, $1.60210 \times 10^{-19} \text{ coulombs}$. **GamBet** issues an error message if this entry does not equal -1.0 (electrons) or +1.0 (positrons).

Kinetic energy

The initial particle kinetic energy in electron volts ($1.0 \text{ eV} = 1.6021 \times 10^{-19} \text{ joules}$).

x, y, z

The initial particle position in meters or units set by **DUnit**. The initial position must be inside the boundaries of the geometric mesh.

ux, uy, uz

A vector parallel to the initial particle direction. **GamBet** normalizes the values to create a unit vector.

Current

An optional parameter to represent the current associated with the primary particle. Specify the current in amperes for cylindrical or 3D geometric meshes. For a planar geometric mesh, the current represents the line current density J of a sheet particle in A/m.

PFILE FPrefix

This command directs **GamBet** to read particle parameter lines from an external file with a name of the form **FPrefix.PRT**. The file prefix contains from 1 to 46 characters. The file must be available in the working directory. A particle file may consist of blank lines, comments and from 1 to 1,000,000 data lines. Data lines have the same format as those of the **PList** command. The file must terminate with the **EndFile** command.

8.3 Number of showers and source properties

NPMULT NPMult

The integer parameter $NPMult$ is the number of showers to initiate per specified primary particle. For example you could represent a uniform current-density electron beam with 200 primary particles distributed in radius and choose $N_{pmult} = 500$ to achieve good statistics. In this case, the total number of showers is $200 \times 500 = 100,000$. For a specified primary particle, each particle that initiates one of the 500 showers starts in the same position but follows a statistically-independent history. The initiating particle has a weight of $1.0/500$ for the calculation of dose and flux.

TPULSE TPulse

This command affects how dose is recorded in the output files. Enter the value of the beam pulselength in seconds. For example, suppose the sum of primary particle currents is 2.5 A. In the absence of the **TPulse** command, **GamBet** calculates the dose rate that would result from a continuous 2.5 A beam. If the command $T_{pulse} = 2.0 \times 10^{-3}$ appears, the code calculates the accumulated dose resulting from an injected charge of 5.0×10^{-3} coulomb.

ROTATE PRotX, PRotY, PRotZ [RotOrder]

Rotate the particle direction vectors relative to the geometry mesh. Enter the values of angles in degrees. The optional string parameter *RotOrder* specifies the order of rotations (*i.e.*, 'ZX', 'ZYX'). Only the operations specified in **RotOrder** are performed. The default is *RotOrder* = 'XYZ'. Note that rotations are applied before position shifts.

SHIFT XPSHift YPSHift ZPSHift

Shift the initial positions of particles relative to the geometry mesh. Enter values in units set by `DUnit`. Note that rotations are applied before shifts.

9 Process commands

COMMAND SUMMARY

Penelope control

```
EABS [Electron,Positron,Photon] 10000.00 [MatNo]
C1 0.1 [MatNo]
C2 0.1 [MatNo]
WCC 100.0 [MatNo]
WCR 100.0 [MatNo]
RSEED ISeed1 ISeed2
EMAX EMax
FORCE [BREMS,PAIR,...] FFactor [NReg]
ENHANCE NReg NSplit
REDUCE NReg NKill
```

Run control

```
DSMAX DsMax
DSMAX NReg DsMax
TIME 600
STEPMAX 20000
PLOTON [NPInterval]
      [ElecP,PhotP,PosiP,ElecS,PhotS,PosiS]
NSEARCHWIDTH G [2]
STEPLIMIT
MCA NBins Reg01 Reg02 ... Reg10
NOESCAPE
ESCAPEFILTER Condition01 Condition02
```

9.1 Penelope control

The first set of commands discussed in this section control operation of the Penelope package. To obtain the detailed Penelope manual, click on the **Penelope manual** command in the **Help** menu of **GamBet**. Although a detailed knowledge of the underlying physics of Penelope is not required to run **GamBet**, you should be aware that the parameters may affect run accuracy and speed. The following material is adapted from the Penelope manual.

EABS [Electron,Positron,Photon] 10000.00 [MatNo]

A particle in material *MatNo* is assumed to be absorbed when its energy drops below a value *EAbs* (in eV). With this command, you can set independent absorption energies for electrons, photons and positrons. Because the interaction database is limited to energies above 50 eV, absorption energies *EAbs* must be larger than this value. If a value for *MatNo* is not given, the absorption energy is applied in all materials. Note that positrons undergo annihilation and emit two photons when they are absorbed. (Default value: 10 keV for electrons, photons and positrons).

The absorption energy values, particularly those of electrons and photons, can significantly affect the solution run time. Generally, there is no reason to track low-energy electrons with range small compared to the system size if you are not interested in details of the low-energy particle spectra. Furthermore, small values of *EAbs* for electrons and positrons can result in poor solution accuracy in the presence of magnetic fields (see the discussion under the *DsMax* command).

C1 0.1 [MatNo]

The parameter *C1* marks the division between discrete and continuous calculations for electron or positron scattering. The quantity equals the average angular deflection ($1 - \cos(\theta)$) produced by multiple elastic scattering along a path length equal to the mean free path between hard elastic events. The default value is $C_1 = 0.05$, and the maximum allowed value is 0.2. If *MatNo* is not specified, the value is applied in all materials of the solution volume.

C2 0.1 [MatNo]

The parameter *C2* marks the division between discrete and continuous calculations of electron or positron energy loss. The quantity equals the maximum average fractional energy loss between consecutive hard elastic events. The default value is $C_2 = 0.05$, and the maximum allowed value is 0.2. If *MatNo* is not specified, the value is applied in all materials of the solution volume.

WCC 100.0 [MatNo]

The parameter *Wcc* is the cutoff energy loss (in eV) for hard inelastic electron collisions. If *MatNo* is not specified, the value is applied in all materials of the solution volume. The default is $W_{cc} = 1000$ eV.

WCR 100.0 [MatNo]

The parameter *Wcr* is the cutoff energy loss (in eV) for hard bremsstrahlung emission. If *MatNo* is not specified, the value is applied in all materials of the solution volume. The default is $W_{cr} = 1000$ eV.

The control parameters determine the accuracy and speed of the simulation. For highest accuracy at low energy the quantities *C1* and *C2* should have small values (0.01). With larger values the simulation gets faster at the expense of some loss in accuracy. The cutoff

energies W_{cc} and W_{cr} mainly influence the simulated energy distributions. The simulation speeds up by using larger cutoff energies, but if these are too large the simulated low-energy distributions may be somewhat distorted. In practice, simulated energy distributions are found to be insensitive to the values of W_{cc} and W_{cr} when they are less than the bin width used to tally the energy histograms. Thus, the desired energy resolution determines the maximum allowed cutoff energies. The reliability of the whole simulation rests on a single condition: the number of steps per primary track must be 'statistically sufficient' (*i.e.* larger than ~ 10).

RSEED ISeed1 ISeed2

Set a random seed for the simulation by supplying two integer values. You can use this command to ensure that a simulation generates identical results each time it is run. The default is to choose seed values set by the current clock time to generate statistically independent results in each run.

EMAX EMax

Sets a maximum energy for the cross-sections generated in **GAMBET.MAT**. In a solution with electric fields, be sure to allow some slack to account for acceleration and/or numerical inaccuracies in the orbit simulation. If the **EMax** command does not appear, **Gambet** estimates a value from the energies of particles defined in the *Source* section. For an incident positron with kinetic energy T_e , the maximum energy is taken as $E_{max} = 1.21 \times (T_e + 5.11 \times 10^5)$ eV to account for the kinetic energy of annihilation photons.

9.2 Variance reduction control

The next set of commands addresses processes often gathered under the term *variance reduction*. Here the intent is to improve statistics for a given run time by devoting resources to critical interactions of interest while de-emphasizing reactions that have little effect on the results. You must perform sufficient checks to ensure that results derived under the following commands are physically correct for your application (*i.e.*, making test runs with and without the application of variance reduction).

FORCE [BREMS,PAIR] Factor [NReg]

Define a forcing factor to increase the number of particles generated as the result of low-probability events. **Gambet** automatically adjusts particle weights and interaction probabilities to maintain unbiased statistics. The second keyword gives the interaction type. Two options are available: *Brems* (generation of bremsstrahlung radiation by energetic electrons and photons) and *Pair* (creation of electron/positron pairs by energetic photons). The third parameter is the forcing factor for the interaction (a real number larger than 1.0). The final integer parameter *NReg* is the number of the region in the geometric mesh where forcing should be applied. If *NReg* does not appear, forcing applies in all regions. Note that large forcing factors (> 100) may skew the statistics and also exhaust the available storage in the secondary-particle stack. **Gambet** includes a warning in the listing file if stack overflow errors occurred during a run.

ENHANCE NReg NSplit [ElecP, PhotP, PosiP, ElecS, PhotS, PosiS]

You can use this command to improve statistics in particular regions of the solution volume. Any particle that enters region *NReg* is split into *NSplit* particles with weights adjusted by a factor $1.0/N_{split}$ to preserve unbiased statistics. The integer quantity *NSplit* (the enhancement factor) must be greater than 1. This command is most effective if it is applied to a remote region that receives a small fraction of the particle flux created in the solution. The script may contain multiple **Enhance** commands. If you include from 1-6 optional string parameters, **GamBet** applies enhancement only to the specified classes of particles : *ElecP* (primary electrons), *PhotP* (primary photons), *PosiP* (primary positrons), *ElecS* (secondary electrons), *PhotS* (secondary photons), *PosiS* (secondary positrons).

REDUCE NReg NKill [ElecP, PhotP, PosiP, ElecS, PhotS, PosiS]

This command reduces the code resources devoted to particle tracking in region *NReg* by applying a Russian-roulette technique. A particle that enters the region is killed with probability $P_{kill} = 1.0 - 1.0/N_{kill}$. If it survives, its weight is increased by a factor $1.0/(1.0 - P_{kill})$. The integer quantity *NKill* (the reduction factor) must be greater than 1. This command is useful if the solution volume contains a large, remote region that makes only a small contribution to interactions in the solution volume. A script may contain multiple **Reduce** commands. If you include from 1-6 optional string parameters, **GamBet** applies reduction only to the specified classes of particles : *ElecP* (primary electrons), *PhotP* (primary photons), *PosiP* (primary positrons), *ElecS* (secondary electrons), *PhotS* (secondary photons), *PosiS* (secondary positrons).

9.3 Simulation control

The remaining commands control general features of the simulation.

DSMAX [NReg] DsMax

GamBet tracks electrons and positrons that interact with materials and move in electric and magnetic fields by dividing the trajectory into a number of small steps of length *Ds*. This command may be used to set a global value or individual region values for the maximum step size. The default value for all regions is the diagonal length of the solution volume divided by 100.0. Enter the value in units set by *DUnit*. In the default program mode, this command has no effect on photon orbits which generally move in large steps between discrete interactions.

Use the following guidelines to pick values of *DsMax*.

1. The Penelope routines usually assign small values of *Ds* for electrons moving in materials. In this case, the value of *DsMax* has little effect on calculations with no applied fields. For high-energy electrons incident on thin foils, set *DsMax* less than or equal to 1/10 of the foil thickness.
2. The value of *Ds* is critical in simulations with applied magnetic fields. It must be small compared to the radius of gyration of the minimum-energy electron or positron (usually determined by *EAb*s). A larger value of *Ds* gives inaccurate or unstable orbits with poor energy conservation. Because small values of *Ds* may increase the run time, pick *EAb*s as large as possible in calculations with strong magnetic fields.

3. Penelope does not determine values of Ds in VOID regions. You must set a value for $DsMax$ that ensures accuracy of the numerical orbit integrations if there are applied fields.
4. Check the energy conservation statistics computed by **GamBet** to determine whether Ds is small enough for accurate orbit calculations in applied magnetic fields. With inaccurate orbit integrations, the output energy may be higher than the input energy (global energy conservation). Section 10 discusses entries for energy conservation in the listing files.
5. By default, the maximum step size also applies to photons except when the **NoStepLimit** command appears in the script. Eliminating the step limit for photons saves time, but could cause invalid results when the solution volume contains a spatial region that 1) is surrounded by another region, and 2) has width comparable to or less than the photon mean-free path. Without a limit on step-size, photons may jump over the small region with sensing a boundary crossing, giving errors in the assigned dose.

STEPMAX

Depending on the settings of $EAbs$, $C1$, $C2$, Wcc and Wcr , the number of steps in an electron or positron history may be quite large. The parameter $StepMax$ sets a maximum value to prevent infinite run times. The default is $StepMax = 500,000$. If a particle stops early, either increase $StepMax$ or change the values of the Penelope control parameters. The listing file contains a warning message if any particle trajectory exceed $StepMax$.

TIME 600

Use this command to set a maximum run time for the calculation. Enter the value in seconds. You can use this command to set a maximum time instead of a maximum number of showers. **GamBet** records complete information for showers calculated within the allowed time and completes the present shower before terminating. In a parallel run, the primary and all child processes will stop at the time limit.

NSEARCHWIDTH [G, E, B] NSearchWidth

GamBet uses high-speed, high-reliability methods to search meshes for particle positions. The default search width of 3 elements should be sufficient for all circumstances. Increase $NSearchWidth$ if the program reports a mesh search error. For simple meshes, the program may run faster with a lower value. You can set individual values for searches on the geometric, electrical and magnetic meshes.

PLOTON [NPShower] [ElecP, PhotP, PosiP, ElecS, PhotS, PosiS]

In response to this command, **GamBet** records selected trajectories of primary and secondary particles. The trajectories may be displayed in **GBView2** and **GBView3**. The integer quantity $NPShower$ is the number of showers to record for each primary. The default is $N_{pshower} = 1$. If you include from 1-6 optional string parameters, **GamBet** records only trajectories for certain types of particles: *ElecP* (primary electrons), *PhotP* (primary photons), *PosiP* (primary positrons), *ElecS* (secondary electrons), *PhotS* (secondary photons), *PosiS* (secondary

positrons). Use **this command with caution**: a run with a large number of primaries or with $N_{pmult} \gg 1$ can generate a huge plot file. Plotting is deactivated in parallel runs.

PARALLEL NProc [TStart TEnd]

Run multiple instances of **GamBet** on a multi-processor computer. The integer parameter *NProc* equals the total of the primary and child processes, usually equal to the maximum number allowed on the computer (*e.g.*, for a Core i7 machine, the maximum value is 8). The integer *TStart* is the maximum time (in seconds) to wait for the start of each child process, and *TEnd* is the maximum time to wait for the conclusion. Section 11 gives a detailed description of the parallel procedure and factors that influence the choice of parameters.

NOSTEPLIMIT

The maximum step size applies to photons except when the **NoStepLimit** command appears. Eliminating the step limit for photons may save time, but large photon steps can lead to invalid results when the solution volume contains small regions that are immersed in another region. In this case, **GamBet** may fail to sense a boundary transition for steps that cross the small region but have starting and ending points in the surrounding region.

ESCAPEOFF

Suppress creation of the file of escaping particle parameters.

ESCAPEFILTER Condition01 Condition02 ...

By default, the escape file contains all particles that leave the solution volume. Filters may be applied that limit particles recorded in the escape file by species, exit position and kinetic energy. This command is useful to reduce the size of escape files by limiting data to particles of interest, thereby reducing the load time in **GenDist**. The nature of the filter is determined by specified conditions that follow the command. A condition is short string that does not contain any of the standard delimiters. Conditions are separated by spaces. Any number of conditions may be included in the line. The following conditions determine the included particle types (primary or secondary): **ElecP**, **PhotP**, **PosiP**, **ElecS**, **PhotS** and **PosiS**. If none of the conditions appear, all particle types are included. If one or more conditions appears, only the specified types are included. For example, this command includes only secondary electrons and photons:

```
ESCAPEFILTER ElecS PhotS
```

The following expressions define pass conditions for particles according to their exit positions: $X > 10.56$, $Y < 1.0E-8$, $Z > 15$, $R > 30.5$, The expression may not include spaces or other delimiters. The condition must begin with the character *X*, *Y*, *Z* or *R*. The quantity *R* is given by

$$R = \sqrt{X^2 + Y^2}. \quad (1)$$

The next character must be either $>$ (greater than) or $<$ (less than). The remainder of the condition is a real number in any valid format. Specify the coordinate in units given by the *DUnit* parameter. As an example, the command

```
ESCAPEFILTER X>0.0 Y>0.0
```

passes only particles that exit in the first quadrant of the x - y plane. In this case, no filtering is applied in z . Finally, conditions of the form $T<5.0E6$, $T<1500$,.... pass particles according to their kinetic energy. Specify energy values in eV. As an example, consider a 500 keV photon beam passing through a slab of material with $z_{max} = 60.0$ mm. To calculate the attenuation coefficient, we want to find the number of full-energy, primary photons that emerge from the exit face. The appropriate filter command is:

```
ESCAPEFILTER PhotP T>4.999E5 Z>59.999
```

MCA NBins Reg01 Reg02 ... Reg10

The main use of the MCA (Multi-channel Analyzer) command is to characterize detectors. When the command is issued, **GamBet** opens a file `RunName.PLS` and records total deposited energy in selected regions for each shower. At the end of run, the program uses values in the file to create a histogram for each region showing the number of showers per range of deposited energy. The histograms are recorded in the listing file `RunName.GLS`. The integer parameter $NBins$ is the number of energy intervals. The allowed range is $N_{bins} = 2$ to 250. **GamBet** automatically sets minimum and maximum values for the energy range. The integer parameters $Reg01$, $Reg02$,... are the numbers of any regions in the geometry mesh with non-zero volume. You can enter a maximum of 10 regions. The MCA command functions only in the `SingleParticle` mode. Also, the command may not be used in a run with the `Parallel` command.

A histogram listing is shown below. A plot of the number of showers versus energy looks like the display of a multichannel analyzer monitoring a scintillation or solid state detector.

Deposited energy distribution for Region 1

Energy(min):	0.0000E+00	
Energy(max):	2.1670E+06	
dEnergy:	2.1670E+04	
EDn	EUp	NEvent
(ev)	(eV)	
=====		
0.0000E+00	2.1670E+04	10996
2.1670E+04	4.3340E+04	59
4.3340E+04	6.5010E+04	62
6.5010E+04	8.6680E+04	54
...		
2.0803E+06	2.1020E+06	2
2.1020E+06	2.1237E+06	0
2.1237E+06	2.1453E+06	4
2.1453E+06	2.1670E+06	2287

10 Running GamBet: interactive and background modes

A basic **GamBet** run involves the following activities:

1. Create an input script (**RunName.MIN**) describing the system geometry. Run the **Mesh** (2D) or **MetaMesh** (3D) program to convert the geometric specifications to a conformal mesh file.
2. Create an input script (**RunName.GIN**) to control the program and to define the properties of material and sources. Run **GamBet** to generate output files that describe spatial variations of dose, particle orbits within the solution volume and parameters of particle that escape.
3. Analyze the output files using **GBView2**, **GBView3** or **GenDist**.

The section describes the second step, operation of **GamBet** in either the interactive or background modes.

The program **gambet.exe** can run interactively in a Window. In this mode you can carry out several solutions in a session or leave the program while you work in other Windows applications. The most convenient way to run **GamBet**, **GBView2** or **GBView3** is from the **GBLaunch** program launcher. You can also run the executable programs individually from Windows Explorer or add shortcuts to the desktop or the Windows Start Menu.

GamBet has three popup menus: **File**, **Run** and **Help**. The following commands appear in the **File** menu.

CREATE INPUT FILE

This command calls up a full-featured internal Windows editor to create an input script. Supply a run prefix (1-46 characters). The allowed commands appear as a set of comment lines. Save or abandon the file to return to the solution program.

EDIT INPUT FILE

EDIT LISTING FILE

EDIT FILE

These commands call the editor to inspect or to modify **GamBet** input and output files. Choosing a file from an alternate directory does not change the working directory. The **Edit input file** command shows a list of all files with names of the form **FPREFIX.GIN**. The **Edit listing file** command displays files with names **FPREFIX.GLS**.

The **Run** menu contains four commands:

START RUN

Pick an input file (`FPREFIX.GIN`) to start a solution. The working directory is changed if you pick a file from an alternate directory. The run begins if the geometric mesh file and all required data files are present. The small text window displays information on the run status.

PAUSE RUN

The intensive calculations of the Monte Carlo simulation demand the full resources of your computer, causing other tasks to run slowly. If you need to perform critical work, you can pause the solution program and reactivate it later without loss of data.

STOP RUN

This command terminates the program and saves information after completing the current shower.

The *Help* menu has the following entries:

GAMBET REFERENCE MANUAL

Open this manual in the default PDF viewer.

PENELOPE MANUAL

Open the users' manual for Penelope 2006 in the default PDF viewer.

PHYSICAL CONSTANTS

Display a useful list of physical constants in the default PDF viewer.

NIST ELECTRON DATA

Go to the NIST EStar site in the default browser to find electron cross sections and ranges.

NIST PHOTON DATA

Go to the NIST XCom site in the default browser to find photon cross sections.

FIELD PRECISION WEBSITE

Go to the Field Precision home page in the default browser to check on other products or to contact us.

To run **GamBet** in the background, go to the command prompt from Windows (*i.e.*, run `CMD`) and log to the data directory that contains the control script (`FPREFIX.GIN`) and data input files. For example, suppose `COPPER_FOIL.MOU` and `COPPER_FOIL.GIN` are stored in `SIMULATIONS\WORKING` and that the program `gambet.exe` is in the directory `\GAMBET`. From `\SIMULATIONS\WORKING` type

```
C:\GAMBET\GAMBET COPPER_FOIL <Enter>
```

The program runs silently, writing detailed information in the listing file `COPPER_FOIL.GLS`. If the solution is successful, the program may write `COPPER_FOIL.G2D` or other files to the data directory. During lengthy runs you can perform other tasks in Windows.

The main function of the command mode is autonomous operation under batch file control. This feature is useful if you want to make an extended series of solutions overnight or in the background. As an example, suppose you have prepared the input files `SWT01.MIN`, ..., `SWT08.MIN` and `SWT01.GIN`, ..., `SWT08.GIN` in the directory `\SIMULATIONS\WORKING`. Furthermore, you have written the following batch file `SWRUN.BAT` in the working directory using a text editor.

```
@ECHO OFF
ECHO Main switch data run
START /WAIT C:\GAMBET\MESH.EXE SWT01
START /WAIT C:\GAMBET\GAMBET.EXE SWT01
START /WAIT C:\GAMBET\MESH.EXE SWT02
START /WAIT C:\GAMBET\GAMBET.EXE SWT02
...
START /WAIT C:\GAMBET\MESH.EXE SWT08
START /WAIT C:\GAMBET\GAMBET.EXE SWT08
```

Note the use of `/WAIT` option. The mesh data must be available before the **GamBet** calculations. Type

```
SWRUN<Enter>
```

to generate all solutions without the need for further user input. Sometimes you can make efficient use of a multicore computer by eliminating the wait `/WAIT` directive and launching multiple instances of the program. In this case, you must ensure that individual processes do not try to access the same input/output files.

11 GamBet: parallel operation

GamBet run times may be significantly reduced on modern personal computers with parallel techniques. Field Precision programs are designed for Windows-based machines with multiple processors or cores – a *shared-memory architecture*. Accordingly, we have integrated **OpenMP** in our field solution programs such as **HiPhi** and **Magnum**. For these codes, the matrix-inversion process that occupies the bulk of the run time is relative easy to parallelize. Threads work on separated parts of the matrix so there is no possibility of memory conflicts.

Although at first glance a Monte Carlo code would seem to be a good candidate for parallel operation, there are potential issues in **GamBet**:

- Primary and secondary particles update deposited dose on a single conformal mesh. With complex trajectories, it is not possible to ensure that particles from parallel showers do not occupy the same cell.
- The **Penelope** library was not written for parallel operation and there are potential conflicts in shared resources like the secondary stack.

For these reasons, we decided to take the *distributed memory* approach with **GamBet**. In this case, multiple instances of the program with independent storage run in individual threads controlled by the operating system. The resulting data files are combined at the end to a single representation with improved statistics. This approach is practical for current computers with large memories. Furthermore, it is easily extended to cluster computers.

The **GamBet** procedure is as follows:

1. In a parallel run, the executable `gambet.exe` (the *primary* process) may run either interactively or in the background. In turn, it launches multiple instances of the background program `gambetmp.exe` (*child* processes). All processes utilize the same input files (control script, geometry and field meshes,...).
2. A new command in the **Process** section of the input script signals that the primary process should start a parallel run:

```
PARALLEL NProc [TStart TEnd]
```

The integer parameter *NProc* specifies the number of threads. For a Core i7 processor, the maximum value is $NProc = 8$. Guidelines for picking *NProc* are discussed below. The parameter *TStart* is the number of seconds to wait for a signal from each child process before abandoning the launch. The parameter *TEnd* is the maximum number of seconds to wait for each child process to finish before combining the output data files.

3. If the primary process detects the **PARALLEL** command, it creates a common **Penelope** material input file `GAMBET.MAT` and then sequentially initiates ($NProc-1$) instances of the program `gambetmp.exe`. After launch, a child process reads the shared input files. The

process sends a start signal to confirm that the input files have been closed, a precaution against access conflict. The signal is the creation of a blank file. If the control script has the name `FPREFIX.GIN`, the start signal files have names of the form `FPREFIX01.START`, `FPREFIX02.START`,.... When the primary process detects a start signal, it notes a successful launch and proceeds to the next child process. The primary process seeks the start signal for each child process for a maximum of $TStart$ seconds.

4. When all child processes are running, the primary process proceeds with its own calculation, issuing standard screen messages for interactive runs.
5. When the calculation is complete, the primary process checks that all valid child processes have sent an end signal (*i.e.*, created a file with a name of the form `FPREFIX01.END`). The program waits a maximum of $TEnd$ seconds for each process to finish.
6. The data of the primary process and successful child processes are combined. Start and end signals and unnecessary data files are deleted.

Although the sequence of a parallel run is involved, all operations are organized by the primary process. The user need only add the `PARALLEL` command. The primary process creates a text report, `MPSTATUS.DAT`. Check this file if there are any problems.

Data files are combined according to the following rules:

- Each process creates an escape file listing parameters of particles that escape from the solution volume (`FPREFIXESC.SRC`). Data lines from the child files are appended to the primary escape file. If particles have an associated current or flux, the values are divided by $NProc$. When complete, the child escape files are deleted. To interpret the number of entries in the combined escape file, note that the total number of showers per primary particle is $NPMult \times NProc$.
- Dose files describe the spatial distribution of deposited energy in the solution volume. They have names of the form `FPREFIX.G2D` (two-dimensional) and `FPREFIX.G3D` (three-dimensional). Information from dose files created by the primary and child processes is accumulated, averaged and then written to the primary dose file. The child dose files are deleted.
- The listing files from the primary and child processes contain run statistics, information on deposited energy and values for particle and energy flux between regions. These quantities are accumulated, averaged and then written to the primary listing file (`FPREFIX.GLS`). Child listing files are deleted.
- **GamBet** can record sample electron, photon and positron trajectories in a file with a name of the form `FPREFIX.GPL`. The purposes of trajectory files are to check run physics and to create presentations. They are generally not generated in production Monte Carlo runs. A run with millions of trajectories, each with thousands of orbit vectors, could result in unmanageably large files. Therefore, trajectory plots are automatically deactivated in parallel runs. To create plots, use a serial run and limit the number of trajectories per primary particle.

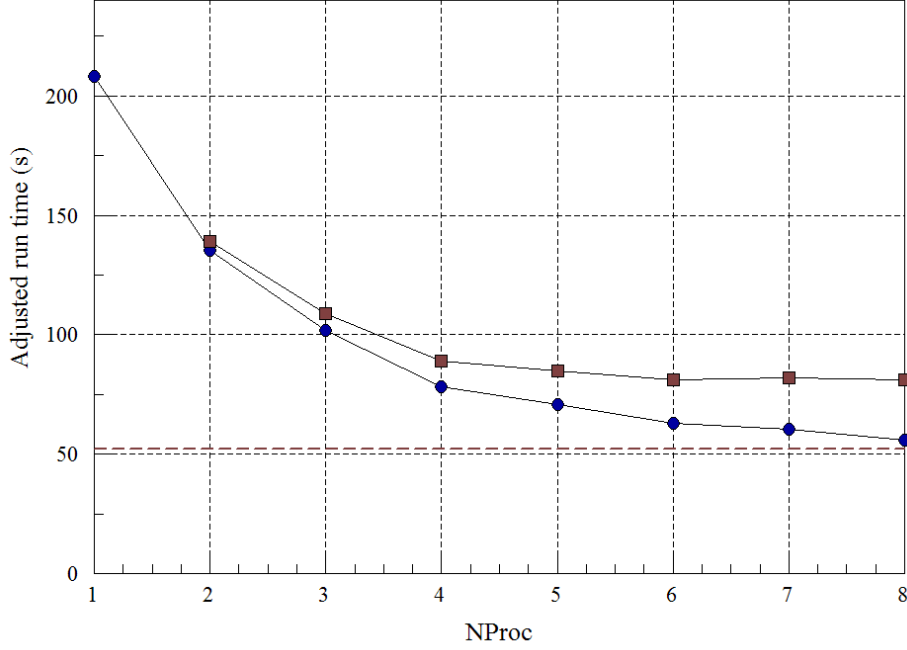


Figure 8: Benchmark test: run time as function of the number of processes. Red: Total run time. Blue: Time with input/output operations subtracted.

Next, we shall consider how to pick parameters in the `PARALLEL` command and how much performance improvement to expect. The maximum value of `NProc` is 99. The parameter should not be set higher than the maximum number of independent threads available on the computer. First consider the value of `TStart`. Each child process must read input files, and two processes cannot access a file at the same time. Therefore, the primary process must wait for a signal that the child process has read and closed all files. This normally requires only a few seconds, so the default `TStart` = 10 seconds is sufficient. If the simulation involves a complex three-dimensional mesh or a large input `SRC` file, you may need to increase `TStart`. With regard to run completion, the child processes should be finished before the primary process if all threads have the same run time. The parameter `TEnd` is included for any operating-system vagaries. If the primary process does not detect the end signal from a child process, it will wait `TEnd` seconds before abandoning the attempt to combine files. The default is `TEnd` = 10 seconds. Again, increase `TEnd` if there are problems reported in `MPSTATUS.DAT`.

To illustrate the effect of the `NProc` parameter, consider a benchmark example of an electron beam incident on an aluminum target. The total number of showers from all processes was fixed at 2,000,000. The run was performed on a Core i7 machine running at 2.93 GHz. The total run time as a function of `NProc` is shown as the red squares in Fig. 8. The serial run time was 208 seconds. Note that the total run time did not decrease above `NProc` = 6, saturating at a reduction factor of 1.72. The time required to read input files and to combine output files negated the speed increase from additional threads. A measurement showed that input/output operations occupied about 3.6 seconds/process. Subtracting this time gives the actual computation time, shown as the blue circles in Fig. 8. At `NProc` = 8, the time reduction approached the theoretical value of 4.0. The implication is that the full benefits of parallel operation can be achieved when the serial run time exceeds about 5 minutes.

Some notes on parallel operation:

- The programs `gambet.exe` and `gambetmp.exe` have been modified from Version 2.0 to ensure that there is a unique random number seed for the primary process and each child process. Otherwise, each process would generate identical information that would be summed and then divided by *NProc*. Without a unique seed, there would be no point to a parallel run.
- To minimize problems, always make sure that a run operates correctly in the serial mode before attempting a large parallel production run. For testing, remove the `PARALLEL` command and use a reduced value of *NpMult*.
- After launch, child processes run independently of the primary process. Therefore, the `Stop run` command in an interactive run will only stop the primary process. You must use the Windows task manager to terminate child processes.

12 Input/output files

During a run **GamBet** may generate four types of output data file

LISTING FILE

In the default mode, simulations generate a listing file in text format with a name of the form `RunName.GLS`. The listing file contains extensive information about the run setup and also calculated statistics. The following section discusses information contained in the listing file.

DOSE FILE

All simulations generate a dose file. Depending on the symmetry of the input *GFile*, the dose file has a name of the form `RunName.G2D` or `RunName.G3D`. Files of type `RunName.G2D` are analyzed with **GBView2** and files of type `RunName.G3D` with **GBView3**. The dose file contains the following information: parameters of the geometric mesh, region identities of nodes and elements, node coordinates, total dose in elements and dose contributions from primary and secondary electrons, photons and positrons and region properties. The units of recorded dose depend on the simulation mode. The **TriComp** program **TDiff** can import files of the type `RunName.G2D` to perform thermal analyses. In this case, the recorded element quantities must be dose rates; therefore, the calculation must be performed in the continuous beam mode (Chap. 3). Here, the current or flux information should be specified for the primary particles and *TPulse* must be undefined. A temporal function may be specified in **TDiff** to modulate the energy deposition rate. Similarly, files of the type `RunName.G3D` may be exported to the **AMaze** program **HeatWave**.

ESCAPE FILE

In the absence of the **ESCAPEOFF** command, a simulation generates an escape file. This file lists properties of particles that terminate by leaving the material boundaries of the solution region. The file has a name of the form `RunNameECS.SRC` and the same format as the source file described in Sect. 8.1. Therefore, an escape file can be used as the input file for a subsequent **GamBet** run or analyzed with the **GenDist** program. An escape event occurs when either 1) the particle moves outside the bounds of the geometric mesh, 2) enters a element with *RegNo* = 0 or 3) enters a region with the *Exclude* property (Sect. 7). The following quantities are included in comment lines at the beginning of the escape file:

DUNIT

Conversion factor to convert particle coordinates to meters.

NPRIMARY

Number of primary particles in the input list of the **GamBet** run.

NShower

Number of showers created in the **GamBet** run, where $N_{shower} = N_{pmult} \times N_{primary}$.

Table 6: Interpretation of flux in the **GamBet** escape file

Particle type	Mesh symmetry	Current/flux quantity
Electron/positron	3D, 2D cylindrical	A
	2D planar	A/m
Photon	3D, 2D cylindrical	photon/s
	2D planar	photon/m/s

Each data line in the file represents a simulation particle and contains the following entries:

Particle type (E,P or E+)
Kinetic energy (eV)
Coordinates x, y and z (units defined by $DUnit$)
Direction unit vector components u_x , u_y and u_z
Current/flux

The interpretation of the current/flux quantity depends on the geometric mesh symmetry and particle type (see Table 6).

TRAJECTORY PLOT FILE

A shower trajectory file is created when the `PlotOn` command appears in the script. The file has a name of the form `RunName.GPL`. The header contains the quantities $DUnit$ and $NShower$. The file contains trajectories of all primary and secondary particles in one or more showers. Each particle orbit starts with the heading:

```
=====
ShowerNo:  667
PrimaryNo:  4
PartType: [Electron, Photon, Positron]
      x      y      z      NReg
-----
```

and contains a number of data lines of the form

```
x,y,z,NReg
```

where the coordinates are given in units specified by $DUnit$. The line

```
-1.0000E+00 -1.0000E+00 -1.0000E+00      0
```

signifies the end of an orbit. Trajectories may be superimposed on dose plots in **GBView2** and **GBView3**. The file is in text format. The content may be inspected with an editor or ported to your own analysis programs.

DEPOSITED ENERGY DISTRIBUTION FILE

The format of the file `RunName.PLS` is shown below. Values for the first region are listed in column 1, the second region in column2 and so forth. As an alternative to the histogram in the listing file, you can use the data directly in your own statistical analysis routines. There are six lines in the heading. The quantity *NLines* is the number of data lines, equal to the number of showers (*NPMult*). The unit (eV/p) denotes electron-volts per primary particle. Each data line contains from 1 to 10 entries, one for each region specified in the `MCA` command. The values are the deposited energy in each region for each shower.

GamBet Pulse-height Distribution File (Field Precision)

RunName: NaIDetect2

NLines: 20000

Regions: 3 6

Units: eV/p

0.0000E+00	0.0000E+00
8.2165E+05	1.3454E+06
0.0000E+00	0.0000E+00
0.0000E+00	7.7453E+05
0.0000E+00	0.0000E+00
...	

13 Contents of the listing file

All Field Precision programs generate listing files that report on the progress of the run. The **GamBet** file `RunName.GLS` records run parameters and contains important results of the calculations.

13.1 Source information

After analyzing the commands of the *Source* section, **GamBet** lists the properties of the primary particle distribution. If the particle lists contain current or flux information, the reported averages are weighted by these quantities.

```
Input distribution properties
  Number of primary particles:      1
  Shower calculations per primary: 50000
  Total number of shower calculations: 50000
  Input energy: 1.0000E+06 (eV/m)
```

```
Electron properties
  Total number: 1.00
  Average energy: 1.0000E+06 (eV)
  Average position
    X: 1.0000E-04 (cm)
    Y: 0.0000E+00 (cm)
    Z: 0.0000E+00 (cm)
  Average direction unit vector
    U: 1.0000
    V: 0.0000
    W: 0.0000
```

13.2 Penelope setup information

After reading commands of the *Process* section, **GamBet** lists quantities generated by Penelope while building material tables and region properties.

Penelope setup

Specified control parameters

```
(Random number seeds: 12345 54321)
MatNo  EAbs(Elec)  EAbs(Phot)  EAbs(Posi)  C1
      (eV)      (eV)      (eV)
-----
1  1.0000E+05  1.0000E+05  1.0000E+05  1.0000E-01

      C2      WCC      WCR
      (eV)      (eV)
-----
1.0000E-01  1.0000E+05  1.0000E+05
```

Set up material tables

```
Energy range, EMin: 1.0000E+05 (eV)  EMax: 1.0000E+06 (eV)
```

Processing Material No.: 1

```
Electron absorption energy: 1.0000E+05 (eV)
Photon absorption energy: 1.0000E+05 (eV)
Positron absorption energy: 1.0000E+05 (eV)
C1: 1.0000E-01, C2: 1.0000E-01  WCC: 1.0000E+05 (eV)  WCR: 1.0000E+05 (eV)
Mass density: 1.9320E+01 (g/cm3)
Number of elements in the molecule: 1
Element: Au (Z=79), atoms/molecule = 1.00000000E+00
Mean excitation energy = 7.9000E+02 eV
```

Maximum step sizes for electrons and positrons

```
NReg      DsMax
-----
1  5.0000E-02 (cm)
```

Integrated region masses

```
NReg      Mass (kg/m)
-----
1  5.7959E-01
```

13.3 Primary particle statistics

When the Monte Carlo calculation is complete, **GamBet** lists several categories of statistic information. The primary particle list has the format shown below. Note that escape particles are those primaries that leave the solution volume while absorbed particles stop inside the solution volume. Units depend on the simulation mode.

Primary electrons

```
Escape fraction: 0.4417
Average energy of escape electrons: 7.2260E+05 (eV)
Average pathlength of escape electrons: 1.3666E-02 (cm)
```

```

Average pathlength of absorbed electrons:  3.8120E-02 (cm)
Number of secondary electrons per primary:  0.290
Average secondary electron energy:  1.8428E+05 (eV)
Average energy lost to secondary electrons per primary:  5.3519E+04 (eV)
Number of secondary photons per primary:  0.141
Average secondary photon energy:  2.5054E+05 (eV)
Average energy lost to photons per primary:  3.5326E+04 (eV)
Number of secondary positrons per primary:  0.000
Average energy lost to positrons per primary:  0.0000E+00 (eV)

```

13.4 Energy conservation

Energy conservation statistics compare the input energy to total deposited and escape energies. In the absence of an applied electric field, the quantity *Energy difference* should be small ($\ll 1\%$). Anomalous energy differences may appear under two circumstances:

- With chosen Penelope parameters, the value of *StepMax* may be too small. In this case, particle histories are not followed to completion.
- The step size *Ds* is too large for accurate numerical integrations in the presence of electric or magnetic fields.

To compute energy conservation, **GamBet** compares the total input particle kinetic energy to the sum of the escape energy and the energy deposited in the medium. **NOTE:** The energy difference may not be a small number if particles can gain significant energy within the simulation volume from applied electric fields.

```

Energy conservation
Input energy:  1.0000E+06 (eV/m)
Energy deposited by primaries:  6.0399E+05 (eV/m)
Energy deposited by secondaries:  6.0372E+04 (eV/m)
Primary escape energy:  3.1917E+05 (eV/m)
Secondary escape energy:  1.6206E+04 (eV/m)
Energy difference:  2.6031E-02%
Energy deposited by electrons:  6.5512E+05 (eV/m)
Electron escape energy:  3.2095E+05 (eV/m)
Energy deposited by photons:  9.1827E+03 (eV/m)
Photon escape energy:  1.4430E+04 (eV/m)
Energy deposited by positrons:  0.0000E+00 (eV/m)
Positron escape energy:  0.0000E+00 (eV/m)

```

13.5 Dose assignment statistics

As a check **GamBet** prints a sum of energy deposited on the finite-element mesh. The quantities should equal the sum of primary and secondary deposited energy.

```

Energy/Power deposited on the mesh
Global energy
Primary electrons:  3.3241E+04 (eV)
Primary photons:  0.0000E+00 (eV)
Primary positrons:  0.0000E+00 (eV)

```

Secondary electrons: 2.6175E+03 (eV)
 Secondary photons: 8.8804E-02 (eV)
 Secondary positrons: 0.0000E+00 (eV)
 All particles: 3.5858E+04 (eV)

Region energy (eV)

NReg	Prim elect	Prim phot	Prim posit	...
1	3.3241E+04	0.0000E+00	0.0000E+00	

	Sec elect	Sec phot	Sec posit	Total
	2.6175E+03	8.8804E-02	0.0000E+00	3.5858E+04

13.6 Particle flux statistics

Finally, **GamBet** makes a list of particle flux between regions. In the example listed below the solution volume contains a single region (numbered 1). Region number 0 represents any exterior volume. Therefore, the quantities shown below represent electrons and photons escaping from the solution volume. **NOTE:** In the example the electron flux value of 0.4998 is higher than the primary escape fraction of 0.4417 because of the expulsion of secondary electrons. In a solution with multiple regions, **GamBet** lists results for all boundaries that have non-zero flux.

--- Region fluxes ---

NReg	NReg	Electron	Electron	Photon	Photon
From	To	flux	eng. flux	flux	eng. flux
1	0	4.499800E-01	3.209489E+05	4.750000E-02	1.443030E+04

14 GBView2

GBView2 is an interactive graphical post-processor for analysis of output files generated by two-dimensional **GamBet** runs. The run has two-dimensional symmetry if the command **GFile2D** was used to load a conformal triangular mesh with planar or cylindrical symmetry. In this case, **GamBet** creates an output file with a name of the form `RunName.G2D` with dose deposited in the triangular elements.

GBView2 automatically sets appropriate units for planar or cylindrical symmetry with the following conventions:

- In a planar simulations there is no variation in the z direction. Primary particles represent a planar beam with input current in A/m and flux in photons/s/m. Dose is deposited uniformly along z (*i.e.*, the z coordinates of individual particle orbits are ignored).
- In a cylindrical simulation there is no variation in the θ direction. Primary particles represent an annular distribution of radius $r = \sqrt{x^2 + y^2}$ with input current in A and flux in photons/s. Dose is deposited uniformly in θ at the r and z coordinates of particles, independent of the relative values of x and y .

14.1 File menu

LOAD DOSE FILE

Load a file with a name of the form `RunName.G2D` containing information on deposited dose in the elements of the geometric mesh.

LOAD TRAJECTORY FILE

This command is active if a dose file has been loaded. When a trajectory file is loaded, you can superimpose plots of primary and secondary particle orbits. Trajectory files have names of the form `RunName.GPL` and are created if the *PlotOn* command appears in the script.

CLOSE TRAJECTORY FILE

Close the file and remove particle orbits from plot.

OPEN DATA LISTING FILE

You can save the results of dose calculations performed in **GBView2** in a text file. Supply a file prefix *FPrefix* in the dialog. The output file has a name of the form `FPrefix.DAT`.

CLOSE DATA LISTING FILE

Terminate input to the data file. The file should be closed before opening another file or editing the present file with another Windows program.

RUN SCRIPT

With scripts you can automatically perform complex or repetitive calculations on a set of similar solutions. This command displays a dialog with a list of script files that you have created with the suffix **SCR**. Pick a file and click OK. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions. Note that the script file should be in the same directory as the data files. Section 14 describes script commands.

CREATE SCRIPT

This command allows you to create script files using the internal editor of the post-processor. A dialog box requests a file prefix. The resulting script file will be saved as **FPREFIX.SCR**. Next, the program opens the file in the editor and writes a list of allowed commands for reference. This list follows the *EndFile* command and will be ignored by **GBView2** or **GBView3**. Enter analysis commands above the **EndFile** command. After saving the file, you can run it using the **Run script** command.

EDIT SCRIPT

Use this command to change an existing script file. The dialog lists files in the current directory with the subscript **SCR**. Changing directories does not change the working directory of the program.

EDIT DATA FILE

Use this command to view or to modify files with names of the form **FPREFIX.DAT**.

EDIT FILE

Use this command to view or to modify any text file.

14.2 Spatial plots menu

PLOT TYPE

The postprocessors can create a variety of colored screen and hardcopy plots.

Mesh: outline of the elements in the computational mesh.

Region: computational mesh with elements color-coded by region number.

Contour: lines that follow constant values of dose.

Element: elements of the solution space color-coded according to dose. You can activate or deactivate element boundaries with the *Toggle element outline* command.

Surface: a three-dimensional plot where height is proportional to dose over a region in the x - y or z - r plane. The spatial limits of the plot correspond to the current view window for *Mesh*, *Region*, *Contour*, or *Element* plots.

PLOTTED QUANTITY

There are eight plot quantities: the total dose, the total energy and dose contributions from primary and secondary electrons, photons and positrons.

PLOT LIMITS

In the default autoscale mode **GBView2** picks limits in *Contour*, *Element* and *Surface* plots that span the full range of dose. With this command you can set manual limits. In the dialog uncheck the *Autoscale* box and fill in minimum and maximum values.

TOGGLE GRID

A set of dashed grid lines in x - y or z - r may be superimposed on *Mesh*, *Region*, *Contour* and *Element* plots. **GBView2** automatically chooses intervals and positions so that lines occur at convenient values of x or y (for example, 0.01 rather than 0.01153). Grids corresponding to the axes ($x = 0.0$ or $y = 0.0$) are plotted as solid lines.

SET CONTOUR PLOT STYLE

This command is active only when the current plot type is *Contour*. There are four choices: monochrome, monochrome with labels, colored and colored with labels. In the colored mode, the lines are color-coded according to the value of the plotted quantity. A legend is included in the information window to the right of the plot. In the labeled modes, contour lines are numbered according to their values. Overlapping labels on closely-spaced lines may look better in a zoomed view.

TOGGLE ELEMENT OUTLINE

This command determines whether the element boundaries are included in *Region* and *Element* plots. It may be necessary to deactivate outlines to view large meshes.

TOGGLE MOUSE/KEYBOARD

By default the program uses interactive mouse entry of coordinates for commands like *Line scan* and *Zoom*. This command switches the program between mouse and keyboard input. Enter keyboard coordinates in the distance units defined by *DUnit*. In other words, if the solution program has $DUnit = 1.0 \times 10^6$, enter dimensions in microns.

TOGGLE SNAP MODE

The mouse snap mode is a convenient feature of **GBView2**. When snap mode is active, the mouse returns the coordinate values closest to an integer multiple of the quantity *DSnap*. In other words, if $DSnap = 0.5$ and the mouse position is [5.4331,-2.6253], the returned coordinates are [5.5,-2.5]. By default, snap mode is *ON*. Snap mode is automatically turned off for coordinate input to the commands **Point calculation** and **Element properties**. Otherwise, the program could pick a location closest to the snap point rather than the point at the tip of the mouse arrow, giving misleading information.

SET DSNAP

Set the distance scale for the mouse snap mode.

The following commands change the view limits in *Mesh*, *Region*, *Contour* and *Element* plots. The current view limits of the two-dimensional plots are used when creating three-dimensional surface plots.

ZOOM WINDOW

You can zoom in on any area of the plot by specifying the two corners of a view box with the mouse. The coordinates are displayed in the window below the plot. The returned coordinate values depend on whether snap mode is active. You can also enter view coordinates from the keyboard by issuing the `Toggle mouse/keyboard` command.

ZOOM IN

Magnifies the view about the center of the current plot.

EXPAND VIEW

Expands the view area about the center of the current plot.

GLOBAL VIEW

Returns the plot boundaries to the full solution area.

PAN

Moves the center of the plot. Enter two points to define a displacement vector.

The following commands control the appearance of surface plots. The commands are active only when a surface plot is displayed. You may notice that there is a delay if you choose the *Surface* option in the plot type command or if you change the plotted quantity when a *Surface* plot is active. To create the plot, **GBView2** must map the current quantity to a rectangular grid, performing a large number of interpolations.

ROTATE 3D IMAGE

Rotate the *Surface* plot 90° in the spatial plane.

VIEW ANGLE 3D

Change the angle of view for the *Surface* plot.

SET GRID 3D

To create the plot, the dose is mapped to a rectangular grid with dimensions N_x, N_y . These numbers also determine the total number of grid lines in the *Surface* plot. The default values are $N_x = N_y = 75$. You can change values with this command. Although higher numbers give

plots with more detail, the regeneration time is longer and the screen display may be unattractive.

The next set of commands is used to send the current plot to a printer or file.

DEFAULT PRINTER

Spatial and scan plots may be ported to any installed Windows printer (including network printers, postscript drivers,...). In response to the command, the current screen plot is sent to the default Windows printer. If necessary, change the default printer using the **Settings** command of Windows before running **GBView2**.

SAVE PLOT FILE

Use this command to create a graphics file of the current plot in either Windows Bitmap (BMP) or Portable Network Graphics (PNG) formats. In the dialog, specify the format, the size in pixels and the file prefix. The graphics file is created in the current directory.

COPY TO CLIPBOARD

Copy the current plot to the clipboard in Windows MetaFile format. The plot can then be pasted into a compatible graphics program.

14.3 Dose analysis menu

The commands in the *Dose Analysis* menu generate numerical data. Most of the functions require coordinate input from the user, usually through the mouse. Therefore, the analysis menu is active only when a *Mesh*, *Region*, *Contour* or *Element* plot is displayed.

POINT CALCULATION

To find the dose in an element, click on the command and then point to any position. Note that snap mode is turned OFF for coordinate input. **GBView2** displays dose values for the element that contains the point and also records information if a data file is open. To enter point coordinates by keyboard, use the **Toggle mouse/keyboard** command.

LINE SCAN

The line scan is one of the most useful functions of **GBView2**. After clicking on the command, supply two points by left-clicking the mouse in a view of a *Mesh*, *Region*, *Contour* or *Element* plot to define a scan line. The snap mode is useful in this application (for example, you may want a scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program determines a series of dose values in elements at equal intervals along the line. The information is recorded if a history file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the commands in the *Scan plot* menu. **GBView2** adds fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (like 0.05 or 2.00).

SCAN PLOT QUANTITY

There are seven plot quantities: the total dose and dose contributions from primary and secondary electrons, photons and positrons.

SET NUMBER OF SCAN POINTS

This command sets the number of line scan points used for the screen plot and recorded in the history file. The default value is 50 and the maximum number is 500.

ELEMENT PROPERTIES

Pick a triangular element with the mouse (or keyboard) and the post-processor writes material and dose properties of the element to the screen.

REGION PROPERTIES

To see physical properties associated with a region of the solution space, click the mouse close to any arc or line vector of the region. Partial results are shown on the screen and a complete analysis is included in the data file.

VOLUME INTEGRALS

No input is needed for this command. **GBView2** automatically computes integrals of quantities over the full solution volume and over individual regions. Information is recorded on the screen or in a history file.

MATRIX FILE

This command is useful if you want to write your own analysis routines or port results to mathematical software. In response to the command, **GBView2** performs interpolations over a specified box region on a regular grid of values. It is much easier to use results in this form than to deal with the conformal triangular mesh directly. Clicking on *Matrix file* calls up a dialog. Specify the dimensions of the box along the axes (in units set by *DUnit*) and the number of calculation intervals. To illustrate, calculations are performed at positions X_n with coordinates given by

$$X_n = X_{min} + n(X_{max} - X_{min})/N_x,$$

where $n = 0, 1, 2, \dots, N_x$. You can also specify an output file prefix. The text file is created in the format shown below with suffix **MTX**. Units of dose depend on the mode of the **GamBet** simulation.

Matrix of values from data file Ion_Chamber.G2D

```

ZMin:  0.000E+00    RMin:  0.000E+00
ZMax:  1.000E+01    RMax:  5.000E+00
NZ:  10    NR:  10
      Z          R      NReg    Dose (Gy)
0.000000E+00  0.000000E+00    1  4.409377E+06
1.000000E+00  0.000000E+00    1  4.143115E+06
2.000000E+00  0.000000E+00    1  3.733472E+06
3.000000E+00  0.000000E+00    1  1.859380E+06
4.000000E+00  0.000000E+00    1  1.215571E+06
5.000000E+00  0.000000E+00    1  6.421903E+05
...

```

SMOOTH DOSE

Statistical noise is invariably associated with Monte Carlo calculations. The interactive plotting and calculation functions of **GBView2** make it easy to visualize and to quantify statistical variations of dose. These variations may be significant in solutions where the mesh has a large number of small elements. With this command you can apply averaging to reduce variations between elements at the expense of some loss of spatial resolution. The following procedure is used. The program cycles through *NSmoothParam* cycles. In each cycle, the element dose is set equal to 0.75 times the previous dose plus 0.25 times a volume-weighted average of the doses in the three adjacent elements. You can apply this command several times to achieve the desired level of smoothing. If smoothing becomes excessive, reload the original file.

SMOOTH PARAMETER

Set the value of *NSmoothParam*. Larger values give stronger smoothing. The default value is 10.

SAVE DOSE FILE

You can save a smoothed distribution in a standard two dimensional dose file. Supply a file prefix *FPrefix*. Be sure to use a different prefix if you want to preserve the original data. The file is saved in the current directory under the name **FPrefix.G2D**.

14.4 Scan plot menu

The commands of the **Scan plot** menu become active when a plot is created following the line scan command.

OSCILLOSCOPE MODE

In the oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope. The program superimposes a cross-hair pattern on the graph. Plot values at the intersection are displayed in the information window. Move the marker along the plot by moving the mouse. If you click the left mouse button at a point, the program displays the plot values along with the

numerical derivative and integral of the curve. The definite integral is taken from the left-hand side of the plot to the current point. Values are displayed on the screen and written to the data file if open. Press the right mouse button to exit the oscilloscope mode.

TOGGLE SCAN SYMBOLS

The setting determines whether symbols are added to the scan plot to show calculated points.

TOGGLE GRID

The setting determines whether grid lines are added to the screen and hardcopy scan plots.

CLOSE SCAN PLOT

The scan plot must be closed before you can continue to use the *File* and *Analysis* functions of **GBView2**. This command closes the scan plot and returns the program to the previous spatial plot.

15 GBView3

GBView3 is an interactive graphical post-processor for analysis of output files generated by three-dimensional **GamBet** runs. The run has three-dimensional symmetry if the command **GFile3D** was used to load a conformal hexahedron mesh. In this case, **GamBet** creates an output file with a name of the form **RunName.G3D** with dose deposited in the hexahedron elements.

15.1 File menu

GBView3 has the following popup menus: **File operations**, **Analysis**, **Plane plots**, **Slice plots**, **Surface plots** and **Help**. When the program starts, only the **File** and **Help** menus are active. You must load a data file in order to create plots or to perform analyses. We shall first review options in the **File** menu.

LOAD DOSE FILE

In response to the command, the program displays a dialog with a list of files with names of the form **FPREFIX.G3D**. Changing the directory in the dialog changes the program working directory. Pick an available file and click **OK**. The program loads the solution and updates the status bar. If data retrieval is successful, the analysis and plot menus become active.

DOSE FILE INFORMATION

Display a dialog with information on the currently-loaded dose file.

LOAD TRAJECTORY FILE

This command is active when a dose file is active. When a trajectory file is loaded, you can superimpose plots of primary and secondary particle orbits. Trajectory files have names of the form **RunName.GPL** and are created if the **PLOTON** command appears in the script.

CLOSE TRAJECTORY FILE

Close the file and remove particle orbits from plot.

OPEN DATA LISTING FILE

You can save the results of dose calculations performed in **GBView3** in a text file. Supply a file prefix **FPrefix** in the dialog. The output file has a name of the form **FPrefix.DAT**.

CLOSE DATA LISTING FILE

Terminate input to the data file. The file should be closed before opening another file or editing the present file with another Windows program.

RUN SCRIPT

With scripts you can automatically perform complex or repetitive calculations on a set of similar solutions. This command displays a dialog with a list of script files that you have created with the suffix **SCR**. Pick a file and click *OK*. The script file can load data files, open and close history files, and perform any of the quantitative analysis functions described in this manual. Note that the script file should be in the same directory as the data files. Section 14 describes script commands.

CREATE SCRIPT

This command allows you to create scripts using the internal editor of the post-processor. A dialog box requests a file prefix. The resulting script will be saved as **FPREFIX.SCR**. Next, the program opens the file in the editor and writes a list of allowed commands for reference. This list follows the **EndFile** command and will be ignored by **GBView2** or **GBView3**. Enter analysis commands above the **EndFile** command. After saving the file, you can run it using the **Run script** command.

EDIT SCRIPT

Use this command to change an existing script. The dialog lists files in the current directory with the subscript **SCR**. Changing directories does not change the working directory of the program.

EDIT DATA FILE

Use this command to view or to modify files with names of the form **FPREFIX.DAT**.

EDIT FILE

Use this command to view or to modify any text file.

15.2 Dose analysis menu

The commands in the **Dose analysis** menu generate numerical data. Some functions operate only if a data listing file has been opened.

POINT CALCULATION

To find the dose at a position, supply (x, y, z) coordinates in the dialog in units set by **DUnit** in the **GamBet** run. **GBView3** displays writes dose values for the element that contains the point and also records information if a data file is open.

LINE SCAN

In response to this command, **GBView3** calculates a series of dose values along a specified line. Supply (x, y, z) coordinates of the start and end points. The information is recorded if a history file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the commands in the **Scan plot** menu. **GBView3** adds

fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (like 0.05 or 2.00).

SCAN PLOT QUANTITY

There are seven plot quantities: the total dose, the total energy and dose contributions from primary and secondary electrons, photons and positrons.

SET NUMBER OF SCAN POINTS

This command sets the number of line scan points used for the screen plot and recorded in the history file. The default value is 50 and the maximum number is 500.

REGION PROPERTIES

To see physical properties associated with a region of the solution space, supply the region number in the dialog. Partial results are shown on the screen and a complete analysis is included in the data file.

VOLUME INTEGRALS

No input is needed for this command. **GBView3** automatically computes integrals of dose over the full solution volume and over individual regions. Information is recorded on the screen or in a history file.

CREATE MATRIX FILE

This command is useful if you want to write your own analysis routines or port results to mathematical software. In response to the command, **GBView3** performs interpolations over a specified box region on a regular grid of values. It is much easier to use results in this form than to deal with the conformal hexahedron mesh directly. Clicking on **Create matrix file** calls up a dialog. Specify the dimensions of the box along each axis (in units set by **DUnit**) and the number of calculation intervals. To illustrate, calculations are performed at positions X_n with coordinates given by

$$X_n = X_{min} + n(X_{max} - X_{min})/N_x,$$

where $n = 0, 1, 2, \dots, N_x$. You can also specify an output file prefix. The text file is created in the format shown below. Units of dose depend on the mode of the **GamBet** simulation.

GBView3 matrix file: OutMatrix.MTX

```

XMin:  -7.50000E+00   XMax:   7.50000E+00   NX:      10
YMin:  -7.50000E+00   YMax:   7.50000E+00   NY:      10
ZMin:   0.00000E+00   ZMax:  1.50000E+01   NZ:      10

```

X	Y	Z	Energy	Density	NReg
=====	=====	=====	=====	=====	=====
-7.50000E+00	-7.50000E+00	0.00000E+00	5.93616E+04	1.00000E+00	1
-6.00000E+00	-7.50000E+00	0.00000E+00	5.93616E+04	1.00000E+00	1
-4.50000E+00	-7.50000E+00	0.00000E+00	5.93616E+04	1.00000E+00	1
-3.00000E+00	-7.50000E+00	0.00000E+00	5.93616E+04	1.00000E+00	1
-1.50000E+00	-7.50000E+00	0.00000E+00	5.93616E+04	1.00000E+00	1
...					

SMOOTH DOSE

This command has the same function of the `Smooth dose` command in **GBView2**. You can apply the command several times to achieve the desired level of smoothing. If smoothing becomes excessive, reload the original file.

SMOOTH PARAMETER

Set the value of `NSmoothParam`. Larger values give stronger smoothing. The default value is 5.

SAVE DOSE FILE

You can save a smoothed distribution in a standard three-dimensional dose file. Supply a file prefix `FPrefix`. Be sure to use a different prefix if you want to preserve the original data. The file is saved in the current directory under the name `FPrefix.G3D`.

15.3 Plane plots

Plane plots are two-dimensional plots that show the variation of dose over a plane normal to one of the Cartesian axes. Plane plots provide simple and quick views of the solution space. The technique is to generate a set of values on a rectangular mesh over a specified planar region and then to create plots in a variety of styles. No attempt is made to connect the plot mesh with the conformal mesh of the simulation. Slice plots are also two-dimensional in a plane normal to a Cartesian axis. The difference is that slice plots are built using the computational mesh. They show precise region boundaries and contour lines in true scale. Plot options are more limited because slice plots are more difficult to construct than plane plots.

SET PLANE

This command brings up a dialog to set the plane for the plot. Set the normal axis with the radio buttons at the top. For example, for a choice of z the plot will be created in the x - y plane. You can use the slider bar to set the position along the normal axis or type a value in the box. The range of the slider bar is automatically set to the limits of the solution volume along

the normal axis. The boxes at the bottom determine the plot range in the normal plane. The default settings are the limits of the solution volume. Note that plane plots are constructed to fill the maximum screen area. They do not preserve scaling in the normal plane.

The following commands are in the PLOT CONTROL popup menu:

PLOT STYLE

The command brings up a dialog to set the plot style. The numbers at the bottom give the resolution of the mesh used to create the plot. Higher values give more detail but require longer regenerate times. The default is a 75×75 mesh.

PLOT QUANTITY

The single allowed quantity is the total dose.

PLOT LIMITS

Set limits for the plotted quantity. When Autoscale is active, **GBView3** automatically sets limits based on the range of values in the solution file.

ROTATE PLOT

This command is active only for the **Filled contours 3D** and **Gradient plot 3D** plot styles. You can rotate the plot in 90 degree increments for the best view.

The commands of the **EXPORT PLOT** menu are used to generate hardcopy or to create plot files.

DEFAULT PRINTER

Use this command to send the current plot to any installed Windows printer (including network printers, postscript drivers, PDF drivers...). Note that the destination is the default Windows printer. If necessary, change the default using the **Settings** command of Windows before issuing the command.

SAVE PLOT FILE

Use this command to create a graphics file of the current plot in either Windows Bitmap (BMP) or Portable Network Graphics (PNG) formats. In the dialog, specify the format, the size in pixels and the file prefix. The graphics file is created in the current directory.

COPY TO CLIPBOARD

The current plot is copied to the clipboard in Windows Metafile format. You can then paste the image into graphics software.

15.4 Slice plots

Slice plots are two-dimensional plots that show the variation of quantities over a plane normal to one of the Cartesian axes. In contrast to plane plots, slice plots are based on the structure of the mesh projected to a slice plane. This structure may be quite complex for a conformal mesh; therefore, slice plots require more computational effort. To facilitate the process, slices are at discrete locations along the normal axis corresponding roughly to the planes of the **MetaMesh** foundation mesh. The precise rendering of spatial information allow point-and-click analysis operations (point calculation, line scan, ...) in the slice.

The **Change view** popup menu contains commands to set the slice plane and to adjust the dimensions of the plot.

SET SLICE PLANE PROPERTIES

This command calls up the same dialog as the **Set plane** command in the **Plane plot** menu. You can change the normal axis, change the position along the normal axis, and set plot limits in the normal plane.

SLICE NORMAL TO X

SLICE NORMAL TO Y

SLICE NORMAL TO Z

Quick commands to change the normal axis.

JUMP FORWARD

STEP FORWARD

STEP BACKWARD

JUMP BACKWARD

Move along the slice axis by small or large steps. The small step is approximately one layer of the foundation mesh and the large step is 5 layers. The term forward implies motion toward higher indices of the normal axis. The slider bar in the orientation area to the right of the plot shows the current location.

ZOOM WINDOW

As an alternative to the entries in the **Set slice plane** dialog, you can interactively change plot limits in the normal plane using the mouse. Choose the command and move the mouse pointer into the plot area. The status bar enters coordinate mode. It shows the current mouse position in the plot and the snap mode. Use the left button to pick one corner and then move the mouse to create a view box. Click the left button again, and the plot regenerates. On any coordinate operation, press the **F1** key if want to enter values from the keyboard. Note that the normal plane box in the orientation area to the right of the plot shows the dimensions of the slice plane and the outline of the current zoomed view.

ZOOM IN

Enlarge the plot about the current view center.

EXPAND VIEW

Expand the plot about the current view center.

GLOBAL VIEW

Enlarge the plot boundaries to show the entire normal plane.

PAN

When the plot is zoomed, you can use this operation to shift the current view center. Use the mouse to define a relative starting and ending point for the shift.

The commands in the `Plot control` popup menu are used to set the plot style and mouse options.

SET SLICE PLOT STYLE

This command brings up the dialog to choose the plot style. The *Region plot* style is a cross-section view of the mesh element divisions color-coded by region. In contrast to the logical plane plot of **MetaMesh**, **GBView3** attempts to resolve the exact mesh structure in the plane. The *Contour plot* style shows lines of constant dose, and *Element plot* shows element cross-sections color-coded by dose. The plot option *Region shading* applies to contour plots. When active, **GBView3** adds light background colors to show the boundaries between different material regions. The *Element outline* option applies to element plots. When active, **GBView3** adds the boundaries of elements. The check boxes on the right-hand side control the data included in plots. The *Electron*, *Photon* and *Positron* options are active only if a trajectory plot has been loaded.

PLOT LIMITS

In the autoscale mode, **GBView3** chooses defaults for the minimum and maximum potential values of dose. Deactivate autoscale to set the values manually.

NUMBER OF CONTOUR LINES

Change the number of lines for contour plots.

TOGGLE SNAP MODE

Mouse coordinates for commands such as `Zoom window`, `Pan`, and `Scan in slice` can be entered in two modes. In the normal mode, the returned position corresponds to the mouse position on the screen. In the snap mode, the program picks a point at an even interval close to the mouse position. The returned point depends on the value of the parameter *DSnap*. For example if $DSnap = 0.1$ and the mouse is at position (6.2345,-5.6113), the returned position is (6.2000,-5.6000). The status bar displays the actual or snapped position of the mouse.

SET SNAP DISTANCE

Change the value of *DSnap* from the default value determined by the program.

TOGGLE GRID

A set of dashed grid lines can be superimposed on slice plots. **GBView3** automatically chooses intervals and positions so that the lines occur at convenient values along the horizontal and vertical directions (for example, 0.01 rather than 0.01153). Listings of the grid intervals are included in the axis labels. Grids corresponding to the normal plane axes are plotted as solid lines.

15.5 Analysis in a slice

You can determine dose values at points and along scan lines within the slice using the commands of the **Analysis** popup menu. Results are recorded if a data file is open.

POINT CALCULATION

This command is useful to make quick checks of dose values in the solution volume. After you click the **Point calculation** command, move the mouse into the plot area. The mouse pointer changes to a cross-hair pattern and the status bar enters coordinate mode. Click the left button to specify a point or press the **F1** key to enter the coordinates from the keyboard. Note that mouse coordinates will shift between discrete values if snap mode is active. **GBView3** calculates dose values at the point in the normal plane given by the coordinates.

LINE SCAN

The line scan is one of the most useful **GBView3** capabilities. After clicking on the command, supply two points with the mouse to define a scan line (or press the **F1** key to enter coordinates manually). The snap mode is useful in this application (for example, you may want the scan to extend from 0.000 to 5.000 rather than 0.067 to 4.985.) The program computes a series of values of field quantities in the normal plane at equal intervals along the line. Complete information is recorded if a data file is open. The program also makes a screen plot of the currently-selected quantity versus distance along the scan and activates the **Scan plot** menu. **GBView3** adds fiducial lines to the plot using intelligent grid selection. This means that the plot is adjusted to fill the screen and grids are drawn at useful intervals (like 0.05 or 2.00).

SET SCAN QUANTITY

There are seven plot quantities: the total dose and dose contributions from primary and secondary electrons, photons and positrons.

SET NUMBER OF SCAN POINTS

This command sets the number of line scan points in the screen plot and data file listing. The default value is 50 and the maximum number is 250.

In addition to the standard **Export plot** options, the **Scan plot** menu contains the following command:

OSCILLOSCOPE MODE

In oscilloscope mode, a scan plot assumes characteristics of a digital oscilloscope. **GBView3**

superimposes a cross-hair pattern on the graph. Plot values at the intersection are displayed in the information window. Move the marker along the plot by moving the mouse. If you click the left mouse button at a point, the program records information when a data file is open. Press the right mouse button to exit the oscilloscope mode.

Click *Return* to exit the scan plot and to return to the slice plot.

15.6 Surface plots

Surface plots are three-dimensional views of the solution space. Three types of information may be superimposed: 1) boundaries of regions with foreground lighting, 2) dose values in a slice plane normal to one of the Cartesian axes and 3) dose values of region surface facets. Surface plots are created from the conformal mesh and preserve true spatial scaling.

View controls are the same as those in **Geometer** and **Metamesh**. With the mouse cursor in the center, the left button moves the viewpoint inward (zoom) and the right outward (expand view). Changes are reflected in the orientation box in the upper-right portion of the screen. The main plot is updated when you release the mouse button. With the cursor on the right-hand side of the plot, press the left button to walk around the object and the right button to move in the direction of the cursor arrow.

The commands of the *Plot control* popup menu control the appearance of the plot.

SURFACE PLOT STYLE

This command brings up a dialog with the following options: *Hidden surface/Wireframe* (plot style for region boundary facets), *Include facets* (include or omit the borders of region boundary facets), *Include reference grid* (plot lines with tick marks along the edges of the solution volume) and *Include surface plot* (include a plot of dose in a plane normal to one of the Cartesian axes). If a normal plane dose plot is active, you can use the controls on the right-hand side to set the normal direction of the plane and its position along the normal axis.

DISPLAYED REGIONS

The command brings up a dialog where you can pick region boundaries to superimpose on a plot. The default is that no regions are displayed. You can also determine whether the region boundary is plotted as a perspective view with foreground lighting or with color-coding according to the average local dose. Note that the program issues an error message in the normal plane plot is deactivated and no regions are chosen.

SET CUT PLANES

In a hidden surface plot, internal details may be obscured by surrounding parts or regions. This command brings up a dialog that allows you to adjust the display areas along the x , y and z axes. **GBView3** does not display facets that lie outside the limits. With this feature you can create cutaway views. The default is that cut limits are set equal to the dimensions of the solution volume so that all facets are included.

SET SURFACE PLOT QUANTITY

There are seven available surface quantities: the total dose and dose contributions from primary and secondary electrons, photons and positrons.

15.7 Saving and loading views

The creation of plots for presentations may involve some effort. With the following two commands, you can save all the current view parameters in **GBView3** and immediately restore the plot.

SAVE NAMED VIEW

Save the view parameters for the current plot. Quantities such as the slice axis, slice position and zoom limits are saved for two dimensional plots. Parameters for three-dimensional plots include the viewpoint position, displayed regions and cut planes. The information is stored in a text file in the current directory with a name of the form **FPREFIX.FPV**.

LOAD NAMED VIEW

Load a view file and refresh the plot

The file contains the complete set of plot parameters. This excerpt illustrates the format:

```
Program: GBView3
2D/3D: 2D
DisplayBy: Regions
Outline: On
NSlice:      40
SliceAxis: YAxis
PlotType: LogElemUp
XPMIn: -1.500000E+00
XPMaX:  4.250000E+00
...
```

If a specific solution file is loaded, the plot will be restored exactly. The saved view feature in **GBView3** has two useful features if a different meshes are loaded:

- Dynamic adaptation to different solutions.
- Option for user control of the view parameter set.

Regarding the first feature, there are situations where you want to create consistent views of a set of solutions with different geometries, maintaining a similar appearance. Some plot properties (like the viewpoint rotation matrix) are applicable to any solution, but others (like region cut planes or slice plot limits) depend on the geometry. **GBView3** checks each plot parameter for validity. If a parameter is outside the allowed range for the currently-loaded solution, the program computes an alternative. The goal is to preserve as many features of the view as possible.

You can modify view files with an editor. The order of entries is not rigid. On input, **GBView3** uses a free-form parser. If a parameter is missing, the program simply makes no change from the value current in the program. The implication is that you can modify a saved view to include only elements essential to your application. For example, you could compare a series of assemblies with different sizes, maintaining an orthographic 3D view from the same point in Cartesian space. In this case, the view file would contain only the entries:

```
DView:  1.000000E+37
R11:   8.660253E-01
R12:  -5.000002E-01
R13:   0.000000E+00
R21:   1.669031E-01
R22:   2.890846E-01
R23:   9.426408E-01
R31:  -4.713208E-01
R32:  -8.163510E-01
R33:   3.338061E-01
```

16 GBView2 and GBView3 script operation

Scripts to control **GBView2** or **GBView3** have names of the form **FPREFIX.SCR**. They should be in the same directory as the data files. Scripts are text files that follow the **Gambet** syntax conventions. The program ignores blank lines and indentations. Data lines use the standard delimiters and comment lines begin with '*' (asterisk). Processing stops when the *ENDFILE* command is encountered.

To execute a script, click on the *Run script* command in the *File* menu. The program shows a list of available script files. Pick a file and click OK. The script operates on the presently-loaded data file or you can load other files from within the script. You can also sequentially open one or more data files.

GBView2 and **GBView3** can run autonomously under script file control from the Windows command prompt. Suppose you have a file **GTest.SCR** in the directory **\GAMBET\BUFFER** and that **GBVIEW2.EXE** is in the directory **\GAMBET**. From **\GAMBET\BUFFER**, type

```
../GBVIEW2 GTEST <Enter>
```

The following commands may appear in a script:

INPUT BACKSCAT.G2D

Close the current data file and load a file for analysis. The parameter is the full name of the data file. For the command illustrated, the post-processor would load the file **BackScat.G2D**. You can load several files for sequential analysis.

OUTPUT BACKSCAT

Close the current history file and open an output file **BACKSCAT.DAT**.

POINT 5.65 10.68

POINT 0.00 0.00 10.00

Calculate the dose at the specified point and write the results to the data file. In **GBView2**, the two real number parameters are the coordinates of the point (x - y or z - r) in units set by *DUnit*. **GBView3** requires three real numbers for the x , y , z coordinates.

SCAN 0.00 0.00 10.00 0.00

SCAN (0.00, 0.00, 3.00) (0.00, 0.00, 10.00)

Write calculated dose values on a line between the specified points to the data file. In **GBView2**, the four real-number parameters are the starting and end coordinates in units set by *DUnit*: [$xstart$ $ystart$ $xend$ $yend$] or [$zstart$ $rstart$ $zend$ $rend$]. In **GBView3**, supply six real-number parameters, [$xstart$ $ystart$ $zstart$ $xend$ $yend$ $zend$].

PATH PathName

PATH LargeArc.DAT

Calculate the dose at the coordinates listed in the file *PathName* and write the results to the data file. The path file must be available in the current directory. For **GBView2** each data line contains

two real numbers giving a position (x, y) or (z, r) (depending on the symmetry of the geometry file). For **GBView3** the data lines should contain three real numbers giving (x, y, z) values. Enter the coordinates in units set by *DUnit*. The path file may contain any number of data lines, blank lines and comments (starting with an asterisk). End the list with the *EndFile* command.

ELEMENT 5.65 10.68

ELEMENT 0.00, 1.00, 5.67

Write the properties of the element at the specified point to the data file. In **GBView2**, the two real number parameters are the coordinates of the point $(x-y$ or $z-r)$ in units set by *DUnit*. Enter three real numbers (x, y, z) in **GBView3**.

NSCAN 150

Set the number of points in a line scan. The default is 50 and the maximum number is 500.

REGION 5

Write volume integrals for a region to the data file. The integer parameter is the region number.

VOLUMEINT

Write volume integrals for the full solution and regions to the data file.

MATRIX GOLDFOIL 10 20 0.00 0.00 5.00 10.00

MATRIX GOLDFOIL 0.00 1.00 10 0.00 1.00 10 0.00 5.00 25

Create a file and record dose values on a regular rectangular matrix. In **GBView2**, the command must contain the following seven parameters.

Param 1: The prefix of the matrix file **FPREFIX.MTX** (string).

Param 2: Number of intervals along the x (or z) direction (integer).

Param 3: Number of intervals along the y (or r) direction (integer).

Param 4-7: The coordinates of the corners of a box in the solution volume, $[x_1, y_1, x_2, y_2]$ or $[z_1, r_1, z_2, r_2]$ (real).

GBView3 requires the following 10 parameters:

Param 1: The prefix of the matrix file **FPREFIX.MTX** (string).

Param 2-4: The box dimensions x_{min} and x_{max} (real) and the number of intervals N_x along x (integer).

Param 5-7: The box dimensions y_{min} and y_{max} (real) and the number of intervals N_y along y (integer).

Param 8-10: The box dimensions z_{min} and z_{max} (real) and the number of intervals N_z along z (integer).

ENDFILE

Terminate execution of the script file. You can add descriptive text in any format after this command.

17 Penelope predefined materials

This section lists predefined materials included in the Penelope file PDCOMPOS.P06 file along with their identifying numbers. These material numbers may be used in the

MATERIAL MatNo [CONDUCTOR,INSULATOR]

command of the *COMPOSITION* section.

17.1 Elements

For elements, the material number equals the atomic number

1	Hydrogen	50	Tin
2	Helium	51	Antimony
3	Lithium	52	Tellurium
4	Beryllium	53	Iodine
5	Boron	54	Xenon
6	Amorphous carbon	55	Cesium
7	Nitrogen	56	Barium
8	Oxygen	57	Lanthanum
9	Fluorine	58	Cerium
10	Neon	59	Praseodymium
11	Sodium	60	Neodymium
12	Magnesium	61	Promethium
13	Aluminum	62	Samarium
14	Silicon	63	Europium
15	Phosphorus	64	Gadolinium
16	Sulfur	65	Terbium
17	Chlorine	66	Dysprosium
18	Argon	67	Holmium
19	Potassium	68	Erbium
20	Calcium	69	Thulium
21	Scandium	70	Ytterbium
22	Titanium	71	Lutetium
23	Vanadium	72	Hafnium
24	Chromium	73	Tantalum
25	Manganese	74	Tungsten
26	Iron	75	Rhenium
27	Cobalt	76	Osmium
28	Nickel	77	Iridium
29	Copper	78	Platinum
30	Zinc	79	Gold
31	Gallium	80	Mercury
32	Germanium	81	Thallium
33	Arsenic	82	Lead
34	Selenium	83	Bismuth

35	Bromine	84	Polonium
36	Krypton	85	Astatine
37	Rubidium	86	Radon
38	Strontium	87	Francium
39	Yttrium	88	Radium
40	Zirconium	89	Actinium
41	Niobium	90	Thorium
42	Molybdenum	91	Protactinium
43	Technetium	92	Uranium
44	Ruthenium	93	Neptunium
45	Rhodium	94	Plutonium
46	Palladium	95	Americium
47	Silver	96	Curium
48	Cadmium	97	Berkelium
49	Indium	98	Californium
		99	Einsteinium

17.2 Compounds and mixtures

Compounds and mixtures appear in alphabetical order.

100	Acetone
101	Acetylene
102	Adenine
103	Adipose tissue (ICRP)
104	Air, dry (near sea level)
105	Alanine
106	Aluminum oxide
107	Amber
108	Ammonia
109	Aniline
110	Anthracene
111	B-100 bone-equivalent plastic
112	Bakelite
113	Barium fluoride
114	Barium sulfate
115	Benzene
116	Beryllium oxide
117	Bismuth germanium oxide
118	Blood (ICRP)
119	Bone, compact (ICRU)
120	Bone, cortical (ICRP)
121	Boron carbide
122	Boron oxide
123	Brain (ICRP)
124	Butane
125	N-butyl alcohol
126	C-552 air-equivalent plastic
127	Cadmium telluride

128 Cadmium tungstate
129 Calcium carbonate
130 Calcium fluoride
131 Calcium oxide
132 Calcium sulfate
133 Calcium tungstate
134 Carbon dioxide
135 Carbon tetrachloride
136 Cellulose acetate, cellophane
137 Cellulose acetate butyrate
138 Cellulose nitrate
139 Ceric sulfate dosimeter solution
140 Cesium fluoride
141 Cesium iodide
142 Chlorobenzene
143 Chloroform
144 Concrete, portland
145 Cyclohexane
146 1,2-dichlorobenzene
147 Dichlorodiethyl ether
148 1,2-dichloroethane
149 Diethyl ether
150 N,n-dimethyl formamide
151 Dimethyl sulfoxide
152 Ethane
153 Ethyl alcohol
154 Ethyl cellulose
155 Ethylene
156 Eye lens (ICRP)
157 Ferric oxide
158 Ferrobaboride
159 Ferrous oxide
160 Ferrous sulfate dosimeter solution
161 Freon-12
162 Freon-12b2
163 Freon-13
164 Freon-13b1
165 Freon-13i1
166 Gadolinium oxysulfide
167 Gallium arsenide
168 Gel in photographic emulsion
169 Pyrex glass
170 Glass, lead
171 Glass, plate
172 Glucose
173 Glutamine
174 Glycerol
175 Graphite
176 Guanine

177 Gypsum, plaster of Paris
178 N-heptane
179 N-hexane
180 Kapton polyimide film
181 Lanthanum oxybromide
182 Lanthanum oxysulfide
183 Lead oxide
184 Lithium amide
185 Lithium carbonate
186 Lithium fluoride
187 Lithium hydride
188 Lithium iodide
189 Lithium oxide
190 Lithium tetraborate
191 Lung (ICRP)
192 M3 wax
193 Magnesium carbonate
194 Magnesium fluoride
195 Magnesium oxide
196 Magnesium tetraborate
197 Mercuric iodide
198 Methane
199 Methanol
200 Mix d wax
201 Ms20 tissue substitute
202 Muscle, skeletal (ICRP)
203 Muscle, striated (ICRU)
204 Muscle-equivalent liquid, with sucrose
205 Muscle-equivalent liquid, without sucrose
206 Naphthalene
207 Nitrobenzene
208 Nitrous oxide
209 Nylon, du Pont elvamide 8062
210 Nylon, type 6 and type 6/6
211 Nylon, type 6/10
212 Nylon, type 11 (rilsan)
213 Octane, liquid
214 Paraffin wax
215 N-pentane
216 Photographic emulsion
217 Plastic scintillator (vinyltoluene based)
218 Plutonium dioxide
219 Polyacrylonitrile
220 Polycarbonate (makrolon, lexan)
221 Polychlorostyrene
222 Polyethylene
223 Polyethylene terephthalate (mylar)
224 Polymethyl methacrylate (lucite, perspex, plexiglass)
225 Polyoxymethylene

226 Polypropylene
227 Polystyrene
228 Polytetrafluoroethylene (teflon)
229 Polytrifluorochloroethylene
230 Polyvinyl acetate
231 Polyvinyl alcohol
232 Polyvinyl butyral
233 Polyvinyl chloride
234 Polyvinylidene chloride (saran)
235 Polyvinylidene fluoride
236 Polyvinyl pyrrolidone
237 Potassium iodide
238 Potassium oxide
239 Propane
240 Propane, liquid
241 N-propyl alcohol
242 Pyridine
243 Rubber, butyl
244 Rubber, natural
245 Rubber, neoprene
246 Silicon dioxide
247 Silver bromide
248 Silver chloride
249 Silver halides in photographic emulsion
250 Silver iodide
251 Skin (ICRP)
252 Sodium carbonate
253 Sodium iodide
254 Sodium monoxide
255 Sodium nitrate
256 Stilbene
257 Sucrose
258 Terphenyl
259 Testes (ICRP)
260 Tetrachloroethylene
261 Thallium chloride
262 Tissue, soft (ICRP)
263 Tissue, soft (ICRU four-component)
264 Tissue-equivalent gas (methane based)
265 Tissue-equivalent gas (propane based)
266 A-150 tissue-equivalent plastic
267 Titanium dioxide
268 Toluene
269 Trichloroethylene
270 Triethyl phosphate
271 Tungsten hexafluoride
272 Uranium dicarbide
273 Uranium monocarbide
274 Uranium oxide

275 Urea
276 Valine
277 Viton fluoroelastomer
278 Water, liquid
279 Water vapor
280 Xylene

18 Penelope data files

The following files must be located in the directory `/PATH/GAMBET/REFERENCE`.

eeldx001.p06	eeldx046.p06	eeldx091.p06	pdebr34.p06	pdebr79.p06
eeldx002.p06	eeldx047.p06	eeldx092.p06	pdebr35.p06	pdebr80.p06
eeldx003.p06	eeldx048.p06	eeldx093.p06	pdebr36.p06	pdebr81.p06
eeldx004.p06	eeldx049.p06	eeldx094.p06	pdebr37.p06	pdebr82.p06
eeldx005.p06	eeldx050.p06	eeldx095.p06	pdebr38.p06	pdebr83.p06
eeldx006.p06	eeldx051.p06	eeldx096.p06	pdebr39.p06	pdebr84.p06
eeldx007.p06	eeldx052.p06	eeldx097.p06	pdebr40.p06	pdebr85.p06
eeldx008.p06	eeldx053.p06	eeldx098.p06	pdebr41.p06	pdebr86.p06
eeldx009.p06	eeldx054.p06	eeldx099.p06	pdebr42.p06	pdebr87.p06
eeldx010.p06	eeldx055.p06	pdatconf.p06	pdebr43.p06	pdebr88.p06
eeldx011.p06	eeldx056.p06	pdbrang.p06	pdebr44.p06	pdebr89.p06
eeldx012.p06	eeldx057.p06	pdcompos.p06	pdebr45.p06	pdebr90.p06
eeldx013.p06	eeldx058.p06	pdebr01.p06	pdebr46.p06	pdebr91.p06
eeldx014.p06	eeldx059.p06	pdebr02.p06	pdebr47.p06	pdebr92.p06
eeldx015.p06	eeldx060.p06	pdebr03.p06	pdebr48.p06	pdebr93.p06
eeldx016.p06	eeldx061.p06	pdebr04.p06	pdebr49.p06	pdebr94.p06
eeldx017.p06	eeldx062.p06	pdebr05.p06	pdebr50.p06	pdebr95.p06
eeldx018.p06	eeldx063.p06	pdebr06.p06	pdebr51.p06	pdebr96.p06
eeldx019.p06	eeldx064.p06	pdebr07.p06	pdebr52.p06	pdebr97.p06
eeldx020.p06	eeldx065.p06	pdebr08.p06	pdebr53.p06	pdebr98.p06
eeldx021.p06	eeldx066.p06	pdebr09.p06	pdebr54.p06	pdebr99.p06
eeldx022.p06	eeldx067.p06	pdebr10.p06	pdebr55.p06	pdeel101.p06
eeldx023.p06	eeldx068.p06	pdebr11.p06	pdebr56.p06	pdeel102.p06
eeldx024.p06	eeldx069.p06	pdebr12.p06	pdebr57.p06	pdeel103.p06
eeldx025.p06	eeldx070.p06	pdebr13.p06	pdebr58.p06	pdeel104.p06
eeldx026.p06	eeldx071.p06	pdebr14.p06	pdebr59.p06	pdeel105.p06
eeldx027.p06	eeldx072.p06	pdebr15.p06	pdebr60.p06	pdeel106.p06
eeldx028.p06	eeldx073.p06	pdebr16.p06	pdebr61.p06	pdeel107.p06
eeldx029.p06	eeldx074.p06	pdebr17.p06	pdebr62.p06	pdeel108.p06
eeldx030.p06	eeldx075.p06	pdebr18.p06	pdebr63.p06	pdeel109.p06
eeldx031.p06	eeldx076.p06	pdebr19.p06	pdebr64.p06	pdeel110.p06
eeldx032.p06	eeldx077.p06	pdebr20.p06	pdebr65.p06	pdeel111.p06
eeldx033.p06	eeldx078.p06	pdebr21.p06	pdebr66.p06	pdeel112.p06
eeldx034.p06	eeldx079.p06	pdebr22.p06	pdebr67.p06	pdeel113.p06
eeldx035.p06	eeldx080.p06	pdebr23.p06	pdebr68.p06	pdeel114.p06
eeldx036.p06	eeldx081.p06	pdebr24.p06	pdebr69.p06	pdeel115.p06
eeldx037.p06	eeldx082.p06	pdebr25.p06	pdebr70.p06	pdeel116.p06
eeldx038.p06	eeldx083.p06	pdebr26.p06	pdebr71.p06	pdeel117.p06
eeldx039.p06	eeldx084.p06	pdebr27.p06	pdebr72.p06	pdeel118.p06
eeldx040.p06	eeldx085.p06	pdebr28.p06	pdebr73.p06	pdeel119.p06
eeldx041.p06	eeldx086.p06	pdebr29.p06	pdebr74.p06	pdeel120.p06
eeldx042.p06	eeldx087.p06	pdebr30.p06	pdebr75.p06	pdeel121.p06
eeldx043.p06	eeldx088.p06	pdebr31.p06	pdebr76.p06	pdeel122.p06
eeldx044.p06	eeldx089.p06	pdebr32.p06	pdebr77.p06	pdeel123.p06
eeldx045.p06	eeldx090.p06	pdebr33.p06	pdebr78.p06	pdeel124.p06

pdeel25.p06	pdeel70.p06	pdesi15.p06	pdesi60.p06	pdgph06.p06
pdeel26.p06	pdeel71.p06	pdesi16.p06	pdesi61.p06	pdgph07.p06
pdeel27.p06	pdeel72.p06	pdesi17.p06	pdesi62.p06	pdgph08.p06
pdeel28.p06	pdeel73.p06	pdesi18.p06	pdesi63.p06	pdgph09.p06
pdeel29.p06	pdeel74.p06	pdesi19.p06	pdesi64.p06	pdgph10.p06
pdeel30.p06	pdeel75.p06	pdesi20.p06	pdesi65.p06	pdgph11.p06
pdeel31.p06	pdeel76.p06	pdesi21.p06	pdesi66.p06	pdgph12.p06
pdeel32.p06	pdeel77.p06	pdesi22.p06	pdesi67.p06	pdgph13.p06
pdeel33.p06	pdeel78.p06	pdesi23.p06	pdesi68.p06	pdgph14.p06
pdeel34.p06	pdeel79.p06	pdesi24.p06	pdesi69.p06	pdgph15.p06
pdeel35.p06	pdeel80.p06	pdesi25.p06	pdesi70.p06	pdgph16.p06
pdeel36.p06	pdeel81.p06	pdesi26.p06	pdesi71.p06	pdgph17.p06
pdeel37.p06	pdeel82.p06	pdesi27.p06	pdesi72.p06	pdgph18.p06
pdeel38.p06	pdeel83.p06	pdesi28.p06	pdesi73.p06	pdgph19.p06
pdeel39.p06	pdeel84.p06	pdesi29.p06	pdesi74.p06	pdgph20.p06
pdeel40.p06	pdeel85.p06	pdesi30.p06	pdesi75.p06	pdgph21.p06
pdeel41.p06	pdeel86.p06	pdesi31.p06	pdesi76.p06	pdgph22.p06
pdeel42.p06	pdeel87.p06	pdesi32.p06	pdesi77.p06	pdgph23.p06
pdeel43.p06	pdeel88.p06	pdesi33.p06	pdesi78.p06	pdgph24.p06
pdeel44.p06	pdeel89.p06	pdesi34.p06	pdesi79.p06	pdgph25.p06
pdeel45.p06	pdeel90.p06	pdesi35.p06	pdesi80.p06	pdgph26.p06
pdeel46.p06	pdeel91.p06	pdesi36.p06	pdesi81.p06	pdgph27.p06
pdeel47.p06	pdeel92.p06	pdesi37.p06	pdesi82.p06	pdgph28.p06
pdeel48.p06	pdeel93.p06	pdesi38.p06	pdesi83.p06	pdgph29.p06
pdeel49.p06	pdeel94.p06	pdesi39.p06	pdesi84.p06	pdgph30.p06
pdeel50.p06	pdeel95.p06	pdesi40.p06	pdesi85.p06	pdgph31.p06
pdeel51.p06	pdeel96.p06	pdesi41.p06	pdesi86.p06	pdgph32.p06
pdeel52.p06	pdeel97.p06	pdesi42.p06	pdesi87.p06	pdgph33.p06
pdeel53.p06	pdeel98.p06	pdesi43.p06	pdesi88.p06	pdgph34.p06
pdeel54.p06	pdeel99.p06	pdesi44.p06	pdesi89.p06	pdgph35.p06
pdeel55.p06	pdeflist.p06	pdesi45.p06	pdesi90.p06	pdgph36.p06
pdeel56.p06	pdesi01.p06	pdesi46.p06	pdesi91.p06	pdgph37.p06
pdeel57.p06	pdesi02.p06	pdesi47.p06	pdesi92.p06	pdgph38.p06
pdeel58.p06	pdesi03.p06	pdesi48.p06	pdesi93.p06	pdgph39.p06
pdeel59.p06	pdesi04.p06	pdesi49.p06	pdesi94.p06	pdgph40.p06
pdeel60.p06	pdesi05.p06	pdesi50.p06	pdesi95.p06	pdgph41.p06
pdeel61.p06	pdesi06.p06	pdesi51.p06	pdesi96.p06	pdgph42.p06
pdeel62.p06	pdesi07.p06	pdesi52.p06	pdesi97.p06	pdgph43.p06
pdeel63.p06	pdesi08.p06	pdesi53.p06	pdesi98.p06	pdgph44.p06
pdeel64.p06	pdesi09.p06	pdesi54.p06	pdesi99.p06	pdgph45.p06
pdeel65.p06	pdesi10.p06	pdesi55.p06	pdgph01.p06	pdgph46.p06
pdeel66.p06	pdesi11.p06	pdesi56.p06	pdgph02.p06	pdgph47.p06
pdeel67.p06	pdesi12.p06	pdesi57.p06	pdgph03.p06	pdgph48.p06
pdeel68.p06	pdesi13.p06	pdesi58.p06	pdgph04.p06	pdgph49.p06
pdeel69.p06	pdesi14.p06	pdesi59.p06	pdgph05.p06	pdgph50.p06

pdgph51.p06	pdgph96.p06	pdgpp42.p06	pdgpp87.p06	pdpsi33.p06
pdgph52.p06	pdgph97.p06	pdgpp43.p06	pdgpp88.p06	pdpsi34.p06
pdgph53.p06	pdgph98.p06	pdgpp44.p06	pdgpp89.p06	pdpsi35.p06
pdgph54.p06	pdgph99.p06	pdgpp45.p06	pdgpp90.p06	pdpsi36.p06
pdgph55.p06	pdgpp01.p06	pdgpp46.p06	pdgpp91.p06	pdpsi37.p06
pdgph56.p06	pdgpp02.p06	pdgpp47.p06	pdgpp92.p06	pdpsi38.p06
pdgph57.p06	pdgpp03.p06	pdgpp48.p06	pdgpp93.p06	pdpsi39.p06
pdgph58.p06	pdgpp04.p06	pdgpp49.p06	pdgpp94.p06	pdpsi40.p06
pdgph59.p06	pdgpp05.p06	pdgpp50.p06	pdgpp95.p06	pdpsi41.p06
pdgph60.p06	pdgpp06.p06	pdgpp51.p06	pdgpp96.p06	pdpsi42.p06
pdgph61.p06	pdgpp07.p06	pdgpp52.p06	pdgpp97.p06	pdpsi43.p06
pdgph62.p06	pdgpp08.p06	pdgpp53.p06	pdgpp98.p06	pdpsi44.p06
pdgph63.p06	pdgpp09.p06	pdgpp54.p06	pdgpp99.p06	pdpsi45.p06
pdgph64.p06	pdgpp10.p06	pdgpp55.p06	pdpsi01.p06	pdpsi46.p06
pdgph65.p06	pdgpp11.p06	pdgpp56.p06	pdpsi02.p06	pdpsi47.p06
pdgph66.p06	pdgpp12.p06	pdgpp57.p06	pdpsi03.p06	pdpsi48.p06
pdgph67.p06	pdgpp13.p06	pdgpp58.p06	pdpsi04.p06	pdpsi49.p06
pdgph68.p06	pdgpp14.p06	pdgpp59.p06	pdpsi05.p06	pdpsi50.p06
pdgph69.p06	pdgpp15.p06	pdgpp60.p06	pdpsi06.p06	pdpsi51.p06
pdgph70.p06	pdgpp16.p06	pdgpp61.p06	pdpsi07.p06	pdpsi52.p06
pdgph71.p06	pdgpp17.p06	pdgpp62.p06	pdpsi08.p06	pdpsi53.p06
pdgph72.p06	pdgpp18.p06	pdgpp63.p06	pdpsi09.p06	pdpsi54.p06
pdgph73.p06	pdgpp19.p06	pdgpp64.p06	pdpsi10.p06	pdpsi55.p06
pdgph74.p06	pdgpp20.p06	pdgpp65.p06	pdpsi11.p06	pdpsi56.p06
pdgph75.p06	pdgpp21.p06	pdgpp66.p06	pdpsi12.p06	pdpsi57.p06
pdgph76.p06	pdgpp22.p06	pdgpp67.p06	pdpsi13.p06	pdpsi58.p06
pdgph77.p06	pdgpp23.p06	pdgpp68.p06	pdpsi14.p06	pdpsi59.p06
pdgph78.p06	pdgpp24.p06	pdgpp69.p06	pdpsi15.p06	pdpsi60.p06
pdgph79.p06	pdgpp25.p06	pdgpp70.p06	pdpsi16.p06	pdpsi61.p06
pdgph80.p06	pdgpp26.p06	pdgpp71.p06	pdpsi17.p06	pdpsi62.p06
pdgph81.p06	pdgpp27.p06	pdgpp72.p06	pdpsi18.p06	pdpsi63.p06
pdgph82.p06	pdgpp28.p06	pdgpp73.p06	pdpsi19.p06	pdpsi64.p06
pdgph83.p06	pdgpp29.p06	pdgpp74.p06	pdpsi20.p06	pdpsi65.p06
pdgph84.p06	pdgpp30.p06	pdgpp75.p06	pdpsi21.p06	pdpsi66.p06
pdgph85.p06	pdgpp31.p06	pdgpp76.p06	pdpsi22.p06	pdpsi67.p06
pdgph86.p06	pdgpp32.p06	pdgpp77.p06	pdpsi23.p06	pdpsi68.p06
pdgph87.p06	pdgpp33.p06	pdgpp78.p06	pdpsi24.p06	pdpsi69.p06
pdgph88.p06	pdgpp34.p06	pdgpp79.p06	pdpsi25.p06	pdpsi70.p06
pdgph89.p06	pdgpp35.p06	pdgpp80.p06	pdpsi26.p06	pdpsi71.p06
pdgph90.p06	pdgpp36.p06	pdgpp81.p06	pdpsi27.p06	pdpsi72.p06
pdgph91.p06	pdgpp37.p06	pdgpp82.p06	pdpsi28.p06	pdpsi73.p06
pdgph92.p06	pdgpp38.p06	pdgpp83.p06	pdpsi29.p06	pdpsi74.p06
pdgph93.p06	pdgpp39.p06	pdgpp84.p06	pdpsi30.p06	pdpsi75.p06
pdgph94.p06	pdgpp40.p06	pdgpp85.p06	pdpsi31.p06	pdpsi76.p06
pdgph95.p06	pdgpp41.p06	pdgpp86.p06	pdpsi32.p06	pdpsi77.p06

pdpsi78.p06	pelDX023.p06	pelDX068.p06
pdpsi79.p06	pelDX024.p06	pelDX069.p06
pdpsi80.p06	pelDX025.p06	pelDX070.p06
pdpsi81.p06	pelDX026.p06	pelDX071.p06
pdpsi82.p06	pelDX027.p06	pelDX072.p06
pdpsi83.p06	pelDX028.p06	pelDX073.p06
pdpsi84.p06	pelDX029.p06	pelDX074.p06
pdpsi85.p06	pelDX030.p06	pelDX075.p06
pdpsi86.p06	pelDX031.p06	pelDX076.p06
pdpsi87.p06	pelDX032.p06	pelDX077.p06
pdpsi88.p06	pelDX033.p06	pelDX078.p06
pdpsi89.p06	pelDX034.p06	pelDX079.p06
pdpsi90.p06	pelDX035.p06	pelDX080.p06
pdpsi91.p06	pelDX036.p06	pelDX081.p06
pdpsi92.p06	pelDX037.p06	pelDX082.p06
pdpsi93.p06	pelDX038.p06	pelDX083.p06
pdpsi94.p06	pelDX039.p06	pelDX084.p06
pdpsi95.p06	pelDX040.p06	pelDX085.p06
pdpsi96.p06	pelDX041.p06	pelDX086.p06
pdpsi97.p06	pelDX042.p06	pelDX087.p06
pdpsi98.p06	pelDX043.p06	pelDX088.p06
pdpsi99.p06	pelDX044.p06	pelDX089.p06
pdrelax.p06	pelDX045.p06	pelDX090.p06
pelDX001.p06	pelDX046.p06	pelDX091.p06
pelDX002.p06	pelDX047.p06	pelDX092.p06
pelDX003.p06	pelDX048.p06	pelDX093.p06
pelDX004.p06	pelDX049.p06	pelDX094.p06
pelDX005.p06	pelDX050.p06	pelDX095.p06
pelDX006.p06	pelDX051.p06	pelDX096.p06
pelDX007.p06	pelDX052.p06	pelDX097.p06
pelDX008.p06	pelDX053.p06	pelDX098.p06
pelDX009.p06	pelDX054.p06	pelDX099.p06
pelDX010.p06	pelDX055.p06	
pelDX011.p06	pelDX056.p06	
pelDX012.p06	pelDX057.p06	
pelDX013.p06	pelDX058.p06	
pelDX014.p06	pelDX059.p06	
pelDX015.p06	pelDX060.p06	
pelDX016.p06	pelDX061.p06	
pelDX017.p06	pelDX062.p06	
pelDX018.p06	pelDX063.p06	
pelDX019.p06	pelDX064.p06	
pelDX020.p06	pelDX065.p06	
pelDX021.p06	pelDX066.p06	
pelDX022.p06	pelDX067.p06	

Index

absorption energy, choosing, [35](#)

analysis script, [74](#)

Element, [75](#)

Input, [74](#)

Matrix, [75](#)

NScan, [75](#)

Output, [74](#)

Path, [74](#)

Point, [74](#)

Region, [75](#)

Scan, [74](#)

bremsstrahlung, [36](#)

calculation, stages, [41](#)

command

MCA, [50](#)

commands

BCalc, [28](#)

BFile2D, [24](#)

BFile3D, [24](#)

BTable, [24](#)

BUni, [17](#), [25](#)

C1, [35](#)

C2, [35](#)

DsMax, [21](#), [37](#)

DUnit, [16](#), [23](#)

EAbs, [20](#), [35](#)

ECalc, [28](#)

EFile2D, [23](#)

EFile3D, [24](#)

EMax, [20](#), [36](#)

Enhance, [37](#)

EscapeFilter, [40](#)

EscapeOff, [39](#)

Force, [20](#), [36](#)

FullInfo, [28](#)

GFile2D, [17](#), [23](#)

GFile3D, [23](#)

Material, [26](#)

MCA, [40](#)

NPMult, [19](#), [32](#)

NSearchWidth, [38](#)

Parallel, [39](#), [44](#)

PFile, [19](#), [32](#)

PList, [19](#), [31](#)

PlotOn, [39](#)

Reduce, [37](#)

Region, [27](#)

Rotate, [19](#), [24](#), [25](#), [32](#)

RSeed, [36](#)

SFile, [19](#)

Shift, [19](#), [24](#), [25](#), [33](#)

SList, [19](#), [30](#), [31](#)

StepLimit, [39](#)

StepMax, [21](#), [38](#)

Time, [21](#), [38](#)

TPulse, [19](#), [32](#)

WCC, [35](#)

WCR, [35](#)

Composition section, [15](#), [18](#), [26](#)

deposited energy file, [50](#)

dose file, [48](#)

dose units, [12](#), [32](#), [48](#), [53](#)

DsMax, choosing value, [37](#)

electric field, [22](#), [23](#)

escape file, [28](#), [48](#)

exclusion region, [28](#)

GamBet

background operation, [42](#)

batch file control, [43](#)

file types, [48](#)

parallel operation, [44](#)

program menu, [41](#)

GBLaunch, [8](#), [41](#)

Create and run tasks, [8](#)

GBView2 commands

Close data listing, [55](#)

Close trajectory file, [55](#)

Contour plot type, [57](#)

Create script, [56](#)

Edit data file, [56](#)

Edit file, [56](#)

Edit script, [56](#)

Element outline, [57](#)

Element properties, [60](#)

Line scan, [59](#)

Load dose file, [55](#)

Load trajectory file, [55](#)

Matrix file, [60](#)

- Mouse/keyboard, [57](#)
- Open data listing, [55](#)
- Oscilloscope mode, [62](#)
- Plot export, [59](#)
- Plot limits, [57](#)
- Plot type, [56](#)
- Plotted quantity, [57](#)
- Point calculation, [59](#)
- Region properties, [60](#)
- Run script, [56](#)
- Save dose file, [61](#)
- Save plot file, [59](#)
- Scan quantity, [60](#)
- Smooth dose, [61](#)
- Snap mode, [57](#)
- Toggle grid, [57](#)
- Volume integrals, [60](#)
- GBView3 commands
 - Create script, [64](#)
 - Cut planes, [71](#)
 - Displayed regions, [71](#)
 - Dose file information, [63](#)
 - Edit, [64](#)
 - Line scan, [65](#), [70](#)
 - Load dose file, [63](#)
 - Load named view, [72](#)
 - Load trajectory file, [63](#)
 - Matrix file, [65](#)
 - Open data listing, [63](#)
 - Oscilloscope mode, [71](#)
 - Plot export, [67](#)
 - Plot quantity, [67](#)
 - Plot style, [67](#), [69](#), [71](#)
 - Point calculation, [64](#), [70](#)
 - Region properties, [65](#)
 - Run script, [64](#)
 - Save dose file, [66](#)
 - Save named view, [72](#)
 - Save plot file, [67](#)
 - Set plane, [67](#)
 - Slice plane, [68](#)
 - Smooth dose, [66](#)
 - Volume integrals, [65](#)
- GBView3 plot style
 - plane, [66](#)
 - sclie, [68](#)
 - surface, [71](#)
- geometry mesh, [16](#), [22](#)
- Geometry section, [15](#), [16](#), [22](#)
- installation instructions, [7](#)
- interaction forcing, [20](#), [36](#)
- Listing file, [48](#), [51](#)
- magnetic field, [22](#), [24](#)
 - beam-generated, [24](#)
 - table format, [24](#)
- material
 - defining new, [27](#)
- meshes, rules for combining, [22](#)
- multi-channel analyzer, [40](#), [50](#)
- OmniTrak, [5](#)
 - PRT file, [18](#), [29](#), [31](#)
- package components, [5](#)
- pair production, [36](#)
- parallel operation sequence, [44](#)
- parallel processing, [7](#)
 - child process, [44](#)
 - combining data, [45](#)
 - distributed memory architecture, [44](#)
 - file conflict, [45](#)
 - listing file, [45](#)
 - operation sequence, [44](#)
 - primary process, [44](#)
 - run time reduction, [46](#)
 - shared memory architecture, [44](#)
- particle
 - flux units, [12](#), [49](#), [54](#)
 - primary, definition, [29](#)
 - properties, [30](#), [31](#)
- Penelope, [5](#), [7](#), [15](#), [19](#), [28](#), [37](#)
 - data files, [82](#)
 - manual, [34](#)
 - materials, [18](#), [26](#), [76](#)
 - setup information, [51](#)
- Plot views, saving, [72](#)
- Process section, [15](#), [19](#), [34](#)
- program function, [5](#)
- random number seed, [36](#), [47](#)
- run statistics, [48](#)
- run time
 - optimizing, [36](#)
- script
 - example, [15](#)
 - Setup dialog, [16](#)

- structure, [14](#)
- shower, definition, [29](#), [32](#)
- simulation modes, [12](#)
- source information, generating, [29](#)
- Source section, [15](#), [18](#), [29](#)
- standard delimiters, [14](#)
- statistical quantities, [51](#)
 - dose, [53](#)
 - energy conservation, [53](#)
 - primary particles, [52](#)
- tasks
 - in GBLaunch, [8](#)
- trajectory file, [49](#)
- Trak, [5](#), [23](#)
 - PRT file, [18](#), [29](#), [31](#)
- variance reduction, [36](#)
- void region, [28](#)

Penelope is covered by the following copyright:

PENELOPE/PENGEOM (version 2006)

Copyright (c) 2001-2006

Universitat de Barcelona

Permission to use, copy, modify, distribute and sell this software and its documentation for any purpose is hereby granted without fee, provided that the above copyright notice appears in all copies and that both that copyright notice and this permission notice appear in all supporting documentation. The Universitat de Barcelona makes no representations about the suitability of this software for any purpose. It is provided "as is" without express or implied warranty.