GamBet Reference Manual

Field Precision
PO Box 13595, Albuquerque, NM 87192 U.S.A.
Telephone: 505-220-3975 FAX: 505-294-0222
E mail: techinfofieldp.com Internet: www.fieldp.com

December 2004

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, it is essential to realize that 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

GamBet (**gam**ma and **bet**a particles) is a comprehensive software suite to simulate the transport of energetic electrons, photons and positrons through matter. 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** can be employed as an extension to the **Trak** and **OmniTrak** charged-particle codes or as a stand-alone Monte Carlo simulator.

The core program (`gambet.exe`) combines Field Precision technology developed for finite-element codes with the state-of-the-art **Penelope** radiation-physics package[1]. The physical basis of **Penelope** is described in the reference Francesc Salvat, Jos M. Fernndez-Varea, and Josep Sempau, **PENELOPE, A Code System for Monte Carlo Simulation of Electron and Photon Transport**, (Proceedings of a Workshop/Training Course), OECD/NEA 7-10 July 2003, ISBN: 92-64-02145-0. The document can be downloaded from the Internet at:

`http://www.nea.fr/html/dbprog/penelope-2003.pdf`

Penelope performs the Monte Carlo tasks in **GamBet**: generation of atomic cross sections, prediction of single-particle interactions with matter, creation of secondary particles, and determination of the effects of collisions. 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 and creation of records of spatial variations of dose. 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 available 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.

[1]Penelope is covered by the following copyright:
PENELOPE/PENGEOM (version 2001)
Copyright (c) 2001, 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.

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

Program	**Function**
mesh.exe	Definition of 2D geometries, preparation of conformal triangular meshes.
metamesh.exe	Definition of 3D geometries, preparation of conformal hexahedron meshes
gambet.exe	Main technical program for Monte Carlo simulations
gbview2.exe	Interaction post-processor for plots and analysis of particle source files and 2D dose files
gbview3.exe	Interaction post-processor for plots and analysis of particle source files and 3D dose files
gendist.exe	Utility to create large input particle distributions

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

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

3. Sophisticated graphical pre- and post-processors are available for mesh generation and the analysis of dose and particle orbits.

4. 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.

5. The program accepts direct input of electron or positron distributions from the Trak and OmniTrak codes.

6. The post-processors have extensive interactive capabilities for analysis and plotting of input or output particle distributions.

7. Advanced variance-reduction techniques are employed, including interaction forcing and splitting/reduction of particles that enter specified regions of the solution volume.

8. Particle output files can be used as input for subsequent runs or ported to the Trak or OmniTrak programs.

2 Installing the software

To install the **GamBet** package in Windows, simply copy the directory `/GAMBET` from the distribution CD to any directory (denoted as `PATH`) on the hard disk. The **GamBet** programs (`gblaunch.exe, gambet.exe, gbview2.exe, gbview3.exe`) and this help file (`gambet_reference.pdf`) are located in the directory `PATH/GAMBET`. The Penelope data files must be available in the directory `PATH/GAMBET/REFERENCE`. Make sure that directories and files are not marked as "read-only." In Windows Explorer, right-click on the directory `PATH/GAMBET` and then click on "Properties." Under "Attributes", make sure that the box labelled "Read-only" is not checked. Then click OK. Confirm that changes should be applied to all subdirectories. While in Windows Explorer, click on "Tools/Folder options/View" and make sure that the box "Hide file extensions for known file types" is unchecked.

Set up a shortcut to the **GamBet** program launcher. In Windows Explorer, go to the directory `PATH/GAMBET` and right-click on `gblaunch.exe`. Choose "Create shortcut". Drag the resulting shortcut to your desktop and/or start menu. Run and initialize the program. Be sure that the program directory is set to `PATH/GAMBET` and that the data directory is initially set to

PATH/GAMBET/BUFFER. When you exit gblaunch.exe, the program creates the configuration file gb.cfg in the root directory.

The installation process is similar in Linux. Copy the directory /GAMBET from the distribution CD to any directory (denoted as PATH) on the hard disk. The following executables are located in the program directory: gblaunch, gambet, gbview2 and gbview3. Because names are case sensitive in Linux, all input files should have names in upper case (*i.e.*, ALUMBEAM.GIN). You must set up the environmental variable

```
GAMBET = PATH/GAMBET
```

so that **GamBet** can find Penelope files in the directory PATH/GAMBET/REFERENCE. For convenience, you can create an application link to gblaunch. The configuration file gb.cfg is stored in the program directory.

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).

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

Table 3: **GamBet** simulation modes

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

Table 4: Recorded dose quantities

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

- 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.

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 can be separated by the following delimiters:

```
Space
```

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

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

You can 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 the properties of materials 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
  C1 = 0.10
  C2 = 0.10
  WCc = 1.0E4
  WCr = -1.0E4
  DsMax(1) = 0.20
  PlotOn(1)
END
ENDFILE
```

5 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 [Mod]
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-dimensonal, 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 DUnit

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 quantity *DUnit* 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.

5.1 Loading the geometry file

GFILE2D FileName [Rect, Cylin]
Define the material geometry used for the Monte Carlo calculation. The command requires an output file from **Mesh** (`FName.MOU`) or the TriComp solution programs **EStat** (`FName.EOU`), **BStat** (`FName.BOU`), **PerMag** (`FName.POU`) of **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** abstracts 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.

5.2 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 *RELMODE*. **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 does not affect standard TriComp programs such as **VEStat** or **VTrak**, but 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 can 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.

5.3 Loading and modifying magnetic field information

BFILE2D FileName [BMult]
Load a magnetic field solution from **BStat**, **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. (NOTE: This feature has yet been implemented, 11/20/04)

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 has 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)
```

It must terminate 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 $BMult = 1.0 \times 10^{-4}$. 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.

6 Composition commands

COMMAND SUMMARY
MATERIAL MatNo [CONDUCTOR,INSULATOR]
MATERIAL
NAME
COMPONENT [Chemical symbol or atomic number] [Fraction]
DENSITY
CONDUCTOR
END
REGION RegNo MatNo
REGION RegNo VOID
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 20. 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*] over-rides the default material state for the generation of Penelope cross sections (metallic elements in the range 1-92 are set as conductors, compound materials in the range 100-279 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 COMPO*NENT* 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 stochiometric fraction. Fractional values are permitted for alloys and mixtures. For example, the following lines describe acetic acid (CH_3CO_2H):
Component C 2.0
Component H 4.0
Component O 2.0

DENSITY
Material density in gm/cm3.

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.

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 has an effect only when an *BFile* has been loaded.

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

7 Source commands

COMMAND SUMMARY
SLIST
(Data lines)
END
SFILE SourceFileName
(Data lines)
END
PLIST
(Data lines)
END
PFILE PartFileName
(Data lines)
END
NPMULT NPerPrimary
TPULSE TPulse
SHIFT XPShift YPShift ZPShift
ROTATE RotPX RotPY RotPZ [RotOrderP]

SOURCE commands define the set of primary particles that initiate electron/photon/positron showers. 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 $NPMult = 500$, 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 **Trak** or **Omni-Trak** codes.

The maximum number of unique primary particles is 20,000. 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 can be split into multiple showers using the *NPMULT* command for improved statistics. If $NPMult$ 1, then each primary particle is injected *NPMult* times with a weight of $1.0/NPMult$. For example, if the list contains 1 electron that carries current 2.0 μA and $NPMult = 20$, then 20 shower calculations are initiated by identical electrons that each carry 0.1 μA.

7.1 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 (1.0eV = 1.6021×10^{-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 $1.0/NPMult$.

As an example, the following line designates a 1.7 MeV electron at position (1.0,2.0,0.0) moving in the z direction 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 20 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.

7.2 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 can 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 file 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 1.0AMU = 1.65979×10^{27} kg. The rest mass of electron and positrons is 5.488×10^{-4} 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, 1.60210×10^{-19} 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 (1.0eV = 1.6021×10^{-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.

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 20 characters. The file must be available in the working directory. A particle file may consist of blank lines, comments and from 1 to 20,000 data lines. Data lines have the same format as those of the *PLIST* command. The file must terminate with the *ENDFILE* command.

7.3 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 $NPMult = 500$ 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 $TPULSE = 2.0 \times 10^{-3}$ 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.

8 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]
NSEARCHWIDTH G [2]
STEPLIMIT

8.1 Penelope control

The first set of commands discussed in this section control operation of the Penelope package. The document F. Salvat, J. M. Fernndez-Varea and J Sempau, (available at `http://www.nea.fr/html/dbprog/penelope-2003.pdf`) gives a detailed description of theoretical methods and the effects of control parameters. The file `PENELOPE.TXT` (included with the **GamBet** package) gives a summary. 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 `PENELOPE.TXT`.

EABS [Electron,Positron,Photon] 10000.00 [MatNo]
A particle that moves 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 100 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 $(1 - \overline{\cos(\theta)})$ produced by multiple elastic scattering along a path length equal to the mean free path between hard elastic events. The default value is $C1 = 0.1$, 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 $C2 = 0.1$, 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 $Wcc = 1000$ 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 $Wcr = 1000$ 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 *Wcc* and *Wcr* 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 *Wcc* and *Wcr* 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 control whether a simulation generates identical or randomly-different results each time it is run. The default values are $ISeed1 = 12345$ and $ISeed2 = 54321$. If the default values are used, a calculation will generate the same results reach time it runs on the same type of computer.

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 $E_{max} = 1.21 \times (Te + 5.11 \times 10^5)$ eV to account for the kinetic energy of annihilation photons.

8.2 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 in the present version of **GamBet**: *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.

ENHANCE NReg NSplit
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 $1.0/NSplit$ 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.

REDUCE NReg NKill
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 $PKill = 1.0 - 1.0/NKill$. If it survives, its weight is increased by a factor $1.0/(1.0 - PKill)$. 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.

8.3 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 can 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 *EAbs*). 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 *EAbs* 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. You can 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. The maximum step size also applies to photons when the *STEPLIMIT* command appears in the script. Setting a limit may help **GamBet**

to identify photon boundary crossings in complex geometries. The *STEPLIMIT* command is essential 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.

NSEARCHWIDTH [G, E, B] NSearchWidth

GamBet uses new methods to search meshes for the particle position. The procedures have higher speed and reliability than those in the present versions of **Trak** and **OmniTrak**. 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]

In response to this command, **GamBet** records the trajectories of the primary and secondary particles generated by each injected particle. The trajectories can be displayed in **GBView2** and **GBView3**. The integer quantity *NPShower* is the number of showers to record for each primary. The default is *NPShower* = 1. **Use this command with caution**. A run with a large

number of primaries or with $NPMult \gg 1$ can generate an immense plot file.

STEPLIMIT
In the default mode, the Penelope routines do not apply the limit *DsMax* to photons. Large photon steps can result in errors in dose assignment 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. For this situation, include the *STEPLIMIT* command and choose values of *DsMax* smaller that the width of the enclosed region. The extra operations to advance photon orbits may increase the run time.

9 Running GamBet in the interactive and background modes

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

1. Create an input script (`RunName.MIN`) that describes the system geomety. 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** or **GBView3**.

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 (1-20 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 text input and output files for **GamBet**. 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 has 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.

To run **GamBet** in the background, go to the Command Prompt from Windows 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 GAMBET/BUFFER and that the program gambet.exe is in the directory /GAMBET. From /GAMBET/BUFFER type

```
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. Note that the response of the computer may be slowed considerably.

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 /GAMBET/BUFFER. Furthermore, you have written the following batch file SWRUN.BAT in the data directory using an text editor.

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

Type

```
SWRUN<Enter>
```

to generate all solutions without the need for further user input.

Microsoft has released over thirty versions of it's 32-bit operating system since Windows 95. There is considerable inconsistency in DOS emulation between versions. To ensure reliable batch file operation we supply the utility **GCon** with all our software. The program emulates many DOS commands and has advanced features to organize and to analyze large data runs. To avoid programs, we advise running batch scripts from **GCon** rather than from the Command Prompt.

10 Input/output files

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

LISTING FILE
All 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, electron/positron and photon dose quantities in elements, region properties, particle-flux between regions and energy-flux between regions. The units of recorded dose and flux depend on the simulation mode.

ESCAPE FILE
All simulations generate 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. 7. Therefore, an escape file can be used as the input file for a subsequent **GamBet** run. An escape event occurs when either 1) the particle moves outside the bounds of the geometric mesh or 2) enters a element with RegNo = 0. 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 $NShower = NPMult \times NPrimary$.

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

Particle type	**Mesh symmetry**	textbfCurrent/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 ux, uy and uz
Current/flux

The interpretation of the current/flux quantity depends on the geometric mesh symmetry and particle type (see Table 6). The *DISTRIBUTION* menus of **GBView2** and **GBView3** allow interactive analysis of the content of escape files.

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 can 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.

11 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.

11.1 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
```

11.2 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:   1.0
  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
```

11.3 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 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)
```

11.4 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 (less than 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)
```

11.5 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.

```
Accuracy check: energy/power deposited on the mesh
   Electron/positron deposited energy:   6.5544E+05 (eV/m)
   Photon deposited energy:   9.1825E+03 (eV/m)
   NReg  ElecEnergy  PhotEnergy
           (eV/m)      (eV/m)
   ============================
      1  6.5544E+05  9.1825E+03
```

11.6 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
```

12 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 on 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.

12.1 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.

LOAD SOURCE FILE

Load an input source file or output escape file with a name of the form `RunName.SRC`. Source files are analyzed with the commands of the *SOURCE ANALYSIS* menu. It may take some time to read a large file. When loading has been completed, **GBView2** displays statistics of the distribution in a dialog - click *OK* to continue.

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.

12.2 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 electron or photon 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.

You can also choose whether to display the dose and orbits or electrons, photons and positrons. The orbit feature is active only when a orbit file is loaded.

PLOTTED QUANTITY

You can choose to display the total, electron and photon dose. The total dose (the default) represents the sum of deposited energy from all processes. The photon dose results from direct local absorption of photon energy and does not include the contributions of energetic secondary electrons and positrons produced in the interactions. The electron dose is the sum of contributions from electrons 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 can 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 iterpolations.

ROTATE 3D IMAGE
Rotate the *Surface* plot 90^o 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 NX, NY. These numbers also determine the total number of grid lines in the *Surface* plot. The default values are $NX = NY = 51$. 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 command may cause a program delay because values must be recalculated.

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

DEFAULT PRINTER
Spatial and scan plots can be ported to any installed Windows printer (including network printers, postscript drivers,...). You can generate colored plots if you have a color printer. In response to the command, the current screen plot is sent to the default Windows printer. If necessary, change the default using the Settings command of Windows before running **GBView2**.

PLOT FILE (POSTSCRIPT)
Use this command to make a plot file of the current screen plot in Encapsulated PostScript format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.EPS`.

PLOT FILE (BMP)
Use this command to make a plot file of the current screen plot in Windows Bitmap format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.BMP`.

PLOT FILE (PNG)
Use this command to make a plot file of the current screen plot in Portable Network Graphics format (GIF equivalent). Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.PNG`.

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.

12.3 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 amd 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

With this command you can choose the quantity to display in screen and hardcopy plots of line scans. The choices in **GBView2** are total, electron/positron and photon dose. This setting has no effect on the history file listing which includes both quantities.

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 $n = 0, 1, 2, ..., N_x$. 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     E (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
   ...

              P (Gy)        Dose (Gy)
           0.000000E+00   4.409377E+06
           0.000000E+00   4.143115E+06
           0.000000E+00   3.733472E+06
           0.000000E+00   1.859380E+06
           0.000000E+00   1.215571E+06
           0.000000E+00   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`.

12.4 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 use the *File* and *Analysis* func-

tions of **GBView2**. This command closes the scan plot and returns the program to the previous spatial plot.

13 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 on output file with a name of the form `RunName.G3D` with dose deposited in the hexahedron elements.

13.1 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 if 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.

LOAD SOURCE FILE

Load an input source file or output escape file with a name of the form `RunName.SRC`. Source files are analyzed with the commands of the *SOURCE ANALYSIS* menu. It may take some time to read a large file. When loading has been completed, **GBView3** displays statistics of the distribution in a dialog – click OK to continue.

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.

13.2 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 amd also records information if a data file is open.

LINE SCAN
In response to this command, **GBView3** calcutes 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
With this command you can choose the quantity to display in screen and hardcopy plots of line scans. The choices in **GBView3** are total, electron/positron and photon dose. This setting has no effect on the history file listing which includes both quantities.

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 $n = 0, 1, 2, ..., N_x$. 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          ElecEng
==================================================
 -7.50000E+00 -7.50000E+00  0.00000E+00  5.93616E+04
 -6.00000E+00 -7.50000E+00  0.00000E+00  5.93616E+04
 -4.50000E+00 -7.50000E+00  0.00000E+00  5.93616E+04
 -3.00000E+00 -7.50000E+00  0.00000E+00  5.93616E+04
 -1.50000E+00 -7.50000E+00  0.00000E+00  5.93616E+04

                   PhotEng       Density    NReg
```

```
================================
  4.83820E+02  1.00000E+00     1
  4.83820E+02  1.00000E+00     1
  4.83820E+02  1.00000E+00     1
  4.83820E+02  1.00000E+00     1
  4.83820E+02  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`.

13.3 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 51×51 mesh.

PLOT QUANTITY

Set the quantity to be plotted. The choices are the total dose or the dose components resulting from electrons/positron or photon processes.

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 hard-copy or to create plot files.

DEFAULT PRINTER

With this command, any **GBView3** plot can be ported to any installed Windows printer (including network printers, postscript drivers, PDF drivers...). You can generate colored plots if you have a color printer. Note that the current screen plot is sent to the default Windows printer. If necessary, change the default using the Settings command of Windows before issuing the command.

PLOT FILE (EPS)
Use this command to make a plot file of the current screen plot in Encapsulated PostScript format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.EPS`.

PLOT FILE (BMP)
Use this command to make a plot file of the current screen plot in Windows Bitmap format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.BMP`.

PLOT FILE (PNG)
Use this command to make a plot file of the current screen plot in Portable Network Graphics format. Supply a file prefix in the dialog box. The plot file will be created in the current directory with the name `FPREFIX.PNG`.

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

13.4 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 CONTRO*L 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.

13.5 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

Line scans are 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
With this command you can pick the quantity that will be displayed in screen and exported plots of line scans. Pick the quantity from the list box and click OK. This setting has no effect on the data file listing which includes all field quantities.

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.

13.6 Surface plots

Surface plots are three-dimensional views of the solution space. Two types of information can be superimposed: 1) boundaries of regions, 2) dose values in a slice plane normal to one of the Cartesian axes. Surface plots are created from the conformal mesh and preserve true spatial scaling.

There are two ways to control the three-dimensional view: 1) The active command bar (red arrows) on the right-hand side of screen and 2) The *SET SURFACE VIEW* command in the *ADJUST VIEW* menu. The following options are included on the command bar: *ZM* (zoom or expand), *RX, RY* and *RZ* (rotations about the current x, y or z axis), *X, Y,* and *Z* (translations along the x, y or z axis). To activate a function, move the mouse over an arrow and click (or hold down) the left button. Click the right button or click on the *DRAW* box to update the main plot.

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, *DISPLAY FACETS/HIDE FACETS* (include or omit the borders of region boundary facets), include or omit a plot of dose in a plane normal to one of the Cartesian axes. If a 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. 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

Chose between total, electron/positron and photon dose for the normal-plane plot.

14 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 run 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*].

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, [x1,y1,x2,y2] or [z1,r1,z2,r2] (real).

GBView3 requires the following 10 parameters:

Param 1: The prefix of the matrix file FPREFIX.MTX (string). .
Param 2-4: The box dimensions XMin and XMax (real) and the number of intervals Nx along x (integer).
Param 5-7: The box dimensions YMin and YMax (real) and the number of intervals Ny along y (integer).
Param 8-10: The box dimensions ZMin and ZMax (real) and the number of intervals Nz along z (integer)

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

15 Source analyses in GBView2 and GBView3

The *SOURCE ANALYSIS* menus in **GBView2** and **GBView3** become active when you load a source file (`FPrefix.SRC`) using the *LOAD PARTICLE FILE* command in the *FILE* menu. Two popup menus are displayed when you enter the *Source* menu: *Analysis* and *Plots*.

15.1 Analysis commands

Analysis commands control calculation of numerical data from values in the source file. It is important to note that particle positions in `SRC` files are recorded in units set by *DUnit* (*i.e.*, centimeters, inches, microns,...). The values are converted to meters when the file is loaded into **GBView2** or **GBView3** according to the present value of the *DUnit* parameter. In turn, the programs use the same conversion factor to write positions to output `SRC` files. The unit conversion factor must be known for statistical calculations. Therefore, **GamBet** inserts a comment line of the form

```
* DUnit:   1.00000E+02
```

when creating an output `SRC` file. **GBView2** and **GBView3** check for the comment when a file is loaded and use the value of *DUnit* if the information is available. You can add a *DUnit* comment line to `SRC` files that you construct with a spreadsheet. Alternatively, you can set the value using the *SET DUNIT* command in the *ANALYSIS* menu. Note that the current value of *DUnit* is displayed in the status bar.

SOURCE ANALYSIS

In response to this command, **GBView2** and **GBView3** perform a statistical analysis of particle parameters and record the results in the data file. Only particles that pass the current filter are included. The programs use the current values of the reference axis and *DUnit*. You can modify settings with the *SET REFERENCE AXIS* and *SET DUNIT* commands.

SET REFERENCE AXIS

Distribution quantities may be calculated relative to a beam axis. The choices are the X, Y or Z axes of the solution coordinate system. The quantities *HOrig* and *VOrig* give the position of the beam axis in the horizontal and vertical directions of the normal plane. The association of the horizontal

Table 7: Horizontal and vertical planes for a reference axis

Reference plane	**Horizontal plane**	**Vertical plane**
X	Y	Z
Y	Z	X
Z	X	Y

and vertical directions with coordinate directions is listed in Table 7. The convention follows the right-hand and is applied in several of the distribution plots.

SET DUNIT

Use this command to set a value for the conversion factor that will be used in the distribution analysis. The parameter $DUnit$ gives the conversion between length units used in the `SRC` file to meters. For example, if positions in the `SRC` file are given in inches, set $DUnit = 39.37$.

SET PARTICLE FILTER

This command calls up a dialog to set criteria for including particles in analyses and plots. You can filter particles by type (electron, photon, positron), position along the three axes, the direction vector, and/or kinetic energy.

DISTRIBUTION TRANSFORMATIONS

Sometimes it may be useful to apply spatial transformations to the filtered particles of a distribution. For instance, in a simulation of bremsstrahlung radiation you could collect photons on the surface of a sphere and project the orbits back a uniform distance to determine the effective source size. There are two available back-projection types: 1) a uniform distance and 2) to the position closest to a line parallel to the x, y or z axis. In the former case, specify the distance. In the latter case, specify the line direction and position in the normal plane.

WRITE SRC FILE

This command writes a `SRC` file that contains data lines for particles that pass the current filter. As an application example, suppose you want to inject

only electrons into a second simulation. Use the filter to remove photons and create a source file.

OPEN DATA FILE
GBView2 and **GBView3** will record the results of beam calculations in text format if you open a data file. Supply a complete file name in the dialog. The program will write results of multiple analyses to the file until it is closed.

CLOSE DATA FILE
Close the current data file.

15.2 Plot commands

GBView2 and **GBView3** can create presentation-quality plots based on the content of source files. You can use this feature to display characteristics of distributions generated by **GamBet** and to check the validity of input distributions created by **GenDist**. Note that only particles that pass the current filter are included in the plots.

The programs support the following plot types:

Spatial projections: particle positions projected to a normal plane. The options are x-y, y-z and z-x. For example, an x-y plot shows the position of a particle in the plane normal to z regardless of the particle position in z. You could set a filter to restrict plotted positions to a range in z. The program aborts the plot and displays an error message if the range of values along either the horizontal or vertical axis equals zero.

Phase-space plots: phase-space distributions relative to the reference axis. The options are x-x', y-y' and z-z'. To illustrate the meaning of quantities, suppose the reference axis is z and we choose a plot of x-x'. The abscissa of the plot is $(x - xorig)$ expressed in meters. The ordinate is the quantity $x' = tan^{-1}(ux/uz)$ expressed in radians. This plot type is useful only if particles have orbits that are paraxial with respect to the reference axis (*i.e.*, the distribution constitutes a beam moving along the axis). A plot is not generated if you pick the option z-z' when the reference axis is set to z.

1D bins: histograms of the number of particles or current organized by a quantity. The following plot quantities are available for all SRC files: $f(x), f(y), f(z)$ and $f(T)$. For example, the option $f(T)$ shows the distribution of computational particles in terms of kinetic energy. The option $f(x)$ divides particles into bins according to their position in x, regardless of their position in y and z. You can limit the range of particles in either x, y or z using a filter. The resulting histrogram shows the number of model particles in each bin along x. The following options are available only when the `SRC` file contains information on the model particle current/flux: $Curr(x), Curr(y), Curr(z)$ and $Curr(T)$. In this case, the histogram shows the relative distribution of current in position or kinetic energy. The quality of the plot depends on the number of simulation particles and the number of bins. You can change the number of bins using the *SET NUMBER OF BINS* command. You can also manually set limits on the abscissa using the *SET LIMITS* command.

2D bins: histograms of the number of model particles or the current sorted according to position in a normal plane. The following options are available for all SRC files: $f(x, y), f(y, z)$ and $f(z, x)$. For example, in the $f(x, y)$ option particles are assigned to a two-dimensional array of bins according to their positions in x and y, regardless of the position in z. You can use particle filters to restrict plotted particles to a region of space. The following options are available if the `SRC` file contains information on the particle current: $Curr(x, y), Curr(y, z)$ and $Curr(z, x)$. You can change the number of bins using the *SET NUMBER OF BINS* command. You can also manually set limits on the abscissa using the *SET LIMITS* command.

3D scatter: three-dimensional plot of particle positions, (x, y, z). Only particles that pass the present filter are included. When this plot is active, you can shift the viewpoint using the *ROTATION* and *ELEVATION* commands.

In addition to the standard hardcopy options, the following commands appear in the *PLOTS* popup menu of the *DISTRIBUTION* menu.

SET PLOT TYPE

Pick the plot type from the list described above. The program picks a default plot option valid for the current plot type.

PLOT QUANTITY
Pick a plotted quantity from one of the displayed options. The program displays only options that are valid for the current plot type.

PLOT LIMITS
Manually set limits on the plot. The number of quantities to supply depends on the plot type.

SET NUMBER OF BINS
Set the number of bins for 1D and 2D histograms.

ROTATION (UP)
ROTATION (DOWN)
These commands function only when the current plot type is *Scatter 3D*. Rotate the plot in azimuth.

ELEVATION (UP)
ELEVATION (DOWN)
These commands function only when the current plot type is *Scatter 3D*. Shift the view (polar angle)

16 Penelope predefined materials

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

```
MATERIAL MatNo [CONDUCTOR,INSULATOR]\\
```

command of the *COMPOSITION* section.

16.1 Elements

For elements, the material number equals the atomic number. An following asterisk designates that the element is not usable in Penelope.

```
 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  (*)
```

16.2 Compounds and mixtures

Compunds and mixtures appear in alphabetical order.

```
 99  A-150 tissue-equivalent plastic
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 Ferroboride
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 methacrilate (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 Titanium dioxide
267 Toluene
268 Trichloroethylene
269 Triethyl phosphate
270 Tungsten hexafluoride
271 Uranium dicarbide
272 Uranium monocarbide
273 Uranium oxide
274 Urea

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

17 Penelope data files

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

```
pdatconf.tab   pdebr73.tab   pdeel56.tab   pdgph38.tab   pdgpp21.tab
pdbrang.tab    pdebr74.tab   pdeel57.tab   pdgph39.tab   pdgpp22.tab
pdcompos.tab   pdebr75.tab   pdeel58.tab   pdgph40.tab   pdgpp23.tab
pdebr01.tab    pdebr76.tab   pdeel59.tab   pdgph41.tab   pdgpp24.tab
pdebr02.tab    pdebr77.tab   pdeel60.tab   pdgph42.tab   pdgpp25.tab
pdebr03.tab    pdebr78.tab   pdeel61.tab   pdgph43.tab   pdgpp26.tab
pdebr04.tab    pdebr79.tab   pdeel62.tab   pdgph44.tab   pdgpp27.tab
pdebr05.tab    pdebr80.tab   pdeel63.tab   pdgph45.tab   pdgpp28.tab
pdebr06.tab    pdebr81.tab   pdeel64.tab   pdgph46.tab   pdgpp29.tab
pdebr07.tab    pdebr82.tab   pdeel65.tab   pdgph47.tab   pdgpp30.tab
pdebr08.tab    pdebr83.tab   pdeel66.tab   pdgph48.tab   pdgpp31.tab
pdebr09.tab    pdebr84.tab   pdeel67.tab   pdgph49.tab   pdgpp32.tab
pdebr10.tab    pdebr85.tab   pdeel68.tab   pdgph50.tab   pdgpp33.tab
pdebr11.tab    pdebr86.tab   pdeel69.tab   pdgph51.tab   pdgpp34.tab
pdebr12.tab    pdebr87.tab   pdeel70.tab   pdgph52.tab   pdgpp35.tab
pdebr13.tab    pdebr88.tab   pdeel71.tab   pdgph53.tab   pdgpp36.tab
pdebr14.tab    pdebr89.tab   pdeel72.tab   pdgph54.tab   pdgpp37.tab
pdebr15.tab    pdebr90.tab   pdeel73.tab   pdgph55.tab   pdgpp38.tab
pdebr16.tab    pdebr91.tab   pdeel74.tab   pdgph56.tab   pdgpp39.tab
pdebr17.tab    pdebr92.tab   pdeel75.tab   pdgph57.tab   pdgpp40.tab
pdebr18.tab    pdeel01.tab   pdeel76.tab   pdgph58.tab   pdgpp41.tab
pdebr19.tab    pdeel02.tab   pdeel77.tab   pdgph59.tab   pdgpp42.tab
pdebr20.tab    pdeel03.tab   pdeel78.tab   pdgph60.tab   pdgpp43.tab
pdebr21.tab    pdeel04.tab   pdeel79.tab   pdgph61.tab   pdgpp44.tab
pdebr22.tab    pdeel05.tab   pdeel80.tab   pdgph62.tab   pdgpp45.tab
pdebr23.tab    pdeel06.tab   pdeel81.tab   pdgph63.tab   pdgpp46.tab
pdebr24.tab    pdeel07.tab   pdeel82.tab   pdgph64.tab   pdgpp47.tab
pdebr25.tab    pdeel08.tab   pdeel83.tab   pdgph65.tab   pdgpp48.tab
pdebr26.tab    pdeel09.tab   pdeel84.tab   pdgph66.tab   pdgpp49.tab
pdebr27.tab    pdeel10.tab   pdeel85.tab   pdgph67.tab   pdgpp50.tab
pdebr28.tab    pdeel11.tab   pdeel86.tab   pdgph68.tab   pdgpp51.tab
pdebr29.tab    pdeel12.tab   pdeel87.tab   pdgph69.tab   pdgpp52.tab
pdebr30.tab    pdeel13.tab   pdeel88.tab   pdgph70.tab   pdgpp53.tab
```

pdebr31.tab	pdeel14.tab	pdeel89.tab	pdgph71.tab	pdgpp54.tab
pdebr32.tab	pdeel15.tab	pdeel90.tab	pdgph72.tab	pdgpp55.tab
pdebr33.tab	pdeel16.tab	pdeel91.tab	pdgph73.tab	pdgpp56.tab
pdebr34.tab	pdeel17.tab	pdeel92.tab	pdgph74.tab	pdgpp57.tab
pdebr35.tab	pdeel18.tab	pdeflist.ta	pdgph75.tab	pdgpp58.tab
pdebr36.tab	pdeel19.tab	pdgph01.tab	pdgph76.tab	pdgpp59.tab
pdebr37.tab	pdeel20.tab	pdgph02.tab	pdgph77.tab	pdgpp60.tab
pdebr38.tab	pdeel21.tab	pdgph03.tab	pdgph78.tab	pdgpp61.tab
pdebr39.tab	pdeel22.tab	pdgph04.tab	pdgph79.tab	pdgpp62.tab
pdebr40.tab	pdeel23.tab	pdgph05.tab	pdgph80.tab	pdgpp63.tab
pdebr41.tab	pdeel24.tab	pdgph06.tab	pdgph81.tab	pdgpp64.tab
pdebr42.tab	pdeel25.tab	pdgph07.tab	pdgph82.tab	pdgpp65.tab
pdebr43.tab	pdeel26.tab	pdgph08.tab	pdgph83.tab	pdgpp66.tab
pdebr44.tab	pdeel27.tab	pdgph09.tab	pdgph84.tab	pdgpp67.tab
pdebr45.tab	pdeel28.tab	pdgph10.tab	pdgph85.tab	pdgpp68.tab
pdebr46.tab	pdeel29.tab	pdgph11.tab	pdgph86.tab	pdgpp69.tab
pdebr47.tab	pdeel30.tab	pdgph12.tab	pdgph87.tab	pdgpp70.tab
pdebr48.tab	pdeel31.tab	pdgph13.tab	pdgph88.tab	pdgpp71.tab
pdebr49.tab	pdeel32.tab	pdgph14.tab	pdgph89.tab	pdgpp72.tab
pdebr50.tab	pdeel33.tab	pdgph15.tab	pdgph90.tab	pdgpp73.tab
pdebr51.tab	pdeel34.tab	pdgph16.tab	pdgph91.tab	pdgpp74.tab
pdebr52.tab	pdeel35.tab	pdgph17.tab	pdgph92.tab	pdgpp75.tab
pdebr53.tab	pdeel36.tab	pdgph18.tab	pdgpp01.tab	pdgpp76.tab
pdebr54.tab	pdeel37.tab	pdgph19.tab	pdgpp02.tab	pdgpp77.tab
pdebr55.tab	pdeel38.tab	pdgph20.tab	pdgpp03.tab	pdgpp78.tab
pdebr56.tab	pdeel39.tab	pdgph21.tab	pdgpp04.tab	pdgpp79.tab
pdebr57.tab	pdeel40.tab	pdgph22.tab	pdgpp05.tab	pdgpp80.tab
pdebr58.tab	pdeel41.tab	pdgph23.tab	pdgpp06.tab	pdgpp81.tab
pdebr59.tab	pdeel42.tab	pdgph24.tab	pdgpp07.tab	pdgpp82.tab
pdebr60.tab	pdeel43.tab	pdgph25.tab	pdgpp08.tab	pdgpp83.tab
pdebr61.tab	pdeel44.tab	pdgph26.tab	pdgpp09.tab	pdgpp84.tab
pdebr62.tab	pdeel45.tab	pdgph27.tab	pdgpp10.tab	pdgpp85.tab
pdebr63.tab	pdeel46.tab	pdgph28.tab	pdgpp11.tab	pdgpp86.tab
pdebr64.tab	pdeel47.tab	pdgph29.tab	pdgpp12.tab	pdgpp87.tab
pdebr65.tab	pdeel48.tab	pdgph30.tab	pdgpp13.tab	pdgpp88.tab
pdebr66.tab	pdeel49.tab	pdgph31.tab	pdgpp14.tab	pdgpp89.tab
pdebr67.tab	pdeel50.tab	pdgph32.tab	pdgpp15.tab	pdgpp90.tab
pdebr68.tab	pdeel51.tab	pdgph33.tab	pdgpp16.tab	pdgpp91.tab
pdebr69.tab	pdeel52.tab	pdgph34.tab	pdgpp17.tab	pdgpp92.tab
pdebr70.tab	pdeel53.tab	pdgph35.tab	pdgpp18.tab	pdrelax.tab
pdebr71.tab	pdeel54.tab	pdgph36.tab	pdgpp19.tab	
pdebr72.tab	pdeel55.tab	pdgph37.tab	pdgpp20.tab	