

Getting Started with EGSnrc

R. Townson, F. Tessier, E. Mainegra, B. Walters
Ionizing Radiation Standards
National Research Council of Canada, Ottawa

Printed: June 1, 2018

Abstract

This manual is a “quick start” guide to help new users get familiar with EGSnrc. A number of complete tutorials are presented along with general guidance intended both for new users and for those looking to update their EGSnrc skills. It is suggested that the reader follows the document from top to bottom, as concepts are introduced gradually and through applied examples. The expected outcome is to provide the reader with some practical skills, along with a broader understanding of the capabilities of EGSnrc and the included tools. However, the information provided here is not complete: it will be necessary to refer to the relevant manuals for full technical descriptions.

Contents

1	What is EGSnrc?	4
2	Installation	4
3	EGSnrc overview: the applications	5
3.1	BEAMnrc	6
3.2	DOSXYZnrc	6
3.3	The RZ applications (cylindrical symmetry)	7
3.3.1	DOSRZnrc	7
3.3.2	CAVRZnrc	7
3.3.3	FLURZnrc	7
3.3.4	SPRRZnrc	7
3.4	The SPH applications (spherical symmetry)	8
3.4.1	CAVSPHnrc	8
3.4.2	EDKnrc	8
3.5	g	9
3.6	egs_chamber	9
3.7	egs_cbct	9
3.8	egs_fac	10
3.9	cavity	10
3.10	Tutorials (the tutor codes)	10
4	Getting dirty: writing your first EGSnrc application	11
4.1	Create a new egs++ application	11
4.2	Create an egs++ input file	12
4.2.1	Questions	16
4.3	Compile and run using egs_gui	17
4.4	Add a dose scoring object	17
4.4.1	Questions	18
4.5	Change the type of incident particles	19
4.5.1	Questions	19

4.6	Change the energy of incident particles	20
4.6.1	Questions	20
4.7	Change the material of the plate	20
4.7.1	Questions	20
5	Navigating egs++ input files	22
5.1	Write a new input file	22
5.1.1	Define a cylindrical water phantom	23
5.1.2	Define an isotropic point source	25
5.1.3	Define a dose scoring object	26
5.1.4	Define the run control	26
5.2	Run the simulation	26
5.2.1	Questions	27
5.3	Simulate a collimated source	27
5.3.1	Questions	28
5.4	Generate particles tracks	30
6	Modelling an ionization chamber	32
6.1	Visualize your geometry with <code>egs_view</code>	33
6.1.1	Question	33
6.2	Dig in with clipping planes	35
6.2.1	Questions	35
6.3	Build an egs++ model of the 3C chamber	36
6.3.1	Questions	37
6.4	Build an Exradin A12 chamber	37
6.4.1	Model the spherical chamber tip	37
6.4.2	Model the chamber body	39
6.4.3	Join the chamber body to the chamber tip	40
6.4.4	Questions	42
6.4.5	Add a region label for the cavity	42
6.4.6	Inscribe the chamber in air	43
6.4.7	Add media definitions	44

6.4.8	Add a particle source	45
6.4.9	Add transport parameters and run control	46
6.4.10	Add egs_chamber scoring options	46
6.4.11	Add variance reduction	47
6.4.12	Add ausgab (output) options	48
6.4.13	Run the simulation	49
6.4.14	Questions	49
7	Linear accelerator simulations	50
7.1	Build a new BEAMnrc accelerator	50
7.2	Run a BEAMnrc simulation	55
7.3	Analyze the phase-space with beamdp	56
7.4	Convert CT DICOM data to egspant	58
7.5	Calculate dose with DOSXYZnrc	59
7.6	Extract dose profiles from 3ddose	63
8	Online resources	65
8.1	EGSnrc on github	65
8.2	How to get support	65
8.3	Bug reports	65
8.4	Submitting your code	66
8.5	Documentation	66
8.5.1	Compiling documentation locally	66
9	EGSnrc file structure	67
9.1	The \$HEN_HOUSE	67
9.1.1	Configuring for different machines	69
9.2	The \$EGS_HOME	69

1 What is EGSnrc?

EGSnrc is a software toolkit used to perform Monte Carlo simulation of ionizing radiation transport through matter. It models the propagation of photons, electrons and positrons with kinetic energies between 1 keV and 10 GeV, in homogeneous materials. EGSnrc is an extended and improved version of the Electron Gamma Shower (EGS) software package originally developed at the Stanford Linear Accelerator Center (SLAC) in the 1970s. Most notably, it incorporates significant refinements in charged particle transport, better low energy cross sections, and the egs++ class library to model elaborate geometries and particle sources.

EGSnrc is distributed with a wide range of applications (previously referred to as “user codes”) that utilize the radiation transport physics to calculate specific quantities. These codes have been developed by numerous authors over the lifetime of EGSnrc to support the large user community. For a brief description of each application, see section 3.

2 Installation

For the most up-to-date installation instructions, see the wiki section of the EGSnrc github page:

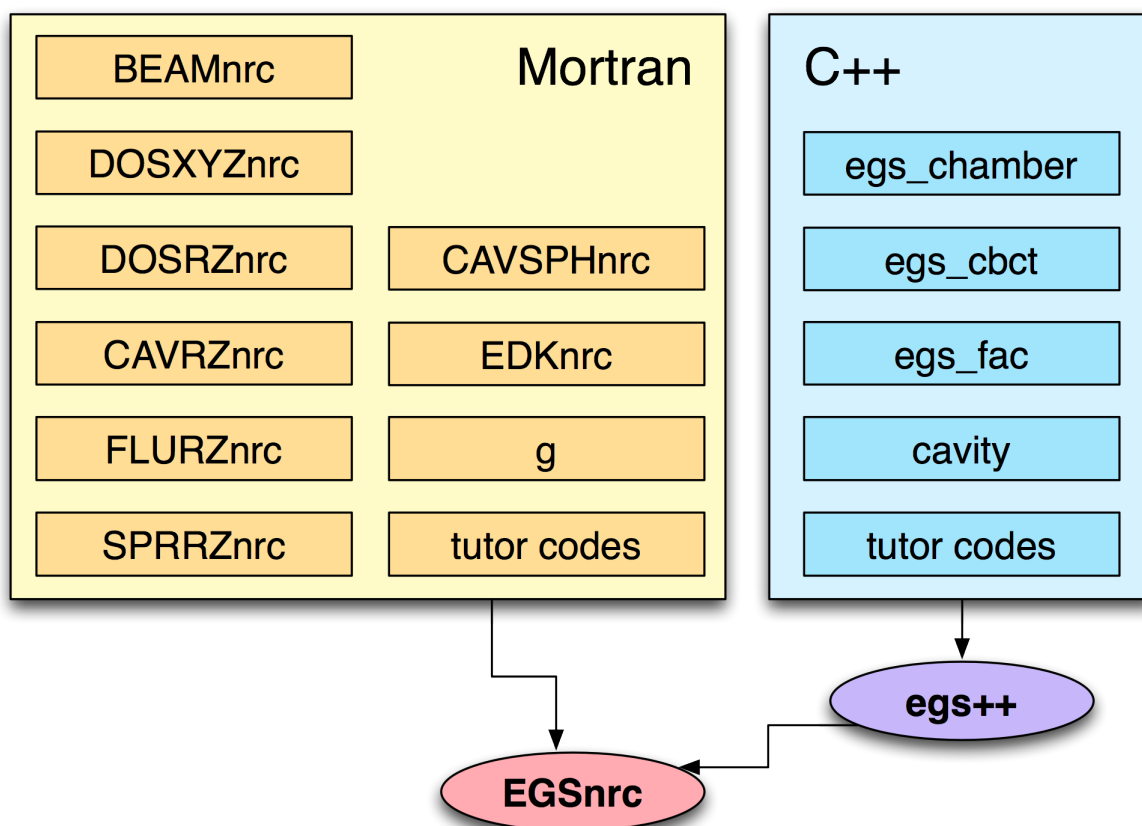
<https://github.com/nrc-cnrc/EGSnrc/wiki/Installation-overview>

If you have any issues with the installation process, the google+ community is a good place to seek support:

<https://plus.google.com/communities/106437507294474212197>

3 EGSnrc overview: the applications

- EGSnrc is a collection of generic routines that simulate the transport and interactions of photons, electrons and positrons in matter.
- The majority of users do not write their own applications - the codes distributed with EGSnrc are useful for a wide range of calculations.
- Users can write their own applications that take advantage of the EGSnrc system, or *toolkit*, by writing a program which specifies their particular problem in the **MAIN** program and communicates with EGSnrc through the user-supplied routines **AUSGAB**, **HOWFAR** and **HOWNEAR** and the EGSnrc routines **HATCH** and **SHOWER**.
- In principle, applications can be written in any programming language as long as it can interface with the Fortran EGSnrc code-base.
- Most commonly, applications are written in **MORTRAN3** (native EGSnrc language) or C++, although they could also be written in **FORTRAN** or C.



EGSnrc source code: [\\$HEN_HOUSE/src/egsnrc.mortran](#)

EGSnrc global variables: [\\$HEN_HOUSE/src/egsnrc.macros](#)

GUI: [~/EGSnrc/HEN_HOUSE/gui/egs_gui](#)

3.1 **BEAMnrc**

- Ideal for linear accelerator modelling and x-ray systems
- Includes geometries (called component modules) that can easily represent flattening filters, collimators, MLCs, etc.
- Accelerators can be compiled as shared libraries to be used as a particle source for other applications
- Includes time-synchronized capabilities (e.g. dynamic jaws and MLCs)
- Written in MORTRAN3

Documentation: PIRS-509a

Source code: [\\$HEN_HOUSE/omega/beamnrc](#)

Global variables: [\\$HEN_HOUSE/omega/beamnrc/beamnrc_user_macros.mortran](#)

GUI (launch with [wish](#)): [\\$HEN_HOUSE/omega/progs/gui/beamnrc/beamnrc_gui.tcl](#)

3.2 **DOSXYZnrc**

- Performs dose calculations in voxelized phantoms
- Includes a variety of source models (e.g. a BEAMnrc shared library)
- Includes time-synchronized capabilities
- Written in MORTRAN3

Documentation: PIRS-794

Source code: [\\$HEN_HOUSE/user_codes/dosxyznrc](#)

Global variables: [\\$HEN_HOUSE/user_codes/dosxyznrc/dosxyznrc_user_macros.mortran](#)

GUI (launch with [wish](#)): [\\$HEN_HOUSE/omega/progs/gui/dosxyznrc/dosxyznrc_gui.tcl](#)

3.3 The RZ applications (cylindrical symmetry)

These applications are based on cylindrically symmetric geometries and are written in [MORTRAN3](#). There is a dedicated GUI to compile these applications and write / edit input files.

Documentation: PIRS-702

GUI documentation: PIRS-801

Source code: [\\$HEN_HOUSE/user_codes](#)

GUI: [\\$HEN_HOUSE/gui/egs_inprz](#)

3.3.1 DOSRZnrc

- Dose and kerma calculations

3.3.2 CAVRZnrc

- Calculates ionization chamber correction factors and relevant quantities

3.3.3 FLURZnrc

- Particle fluence calculations

3.3.4 SPRRZnrc

- Calculates Spencer-Attix spectrum averaged stopping-power ratios for arbitrary media

3.4 The SPH applications (spherical symmetry)

These applications are based on spherically symmetric geometries and are written in the fortran preprocessor language [MORTRAN3](#).

Documentation: PIRS-702

Source code: [\\$HEN_HOUSE/user_codes](#)

3.4.1 CAVSPHnrc

- Identical to CAVRZnrc but for spherical geometries

3.4.2 EDKnrc

- Calculates energy deposition kernels for photons or electrons forced to interact at the centre of a spherical phantom
- Also calculates dose distributions in the entire phantom or the dose to specific regions defined as the cavity of a spherical ion chamber

3.5 g

- Calculates the energy fraction lost to radiation when electrons slow down (if the incident beam is photons), or the radiative yield (if the incident beam is electrons)
- Calculates quantities such as `mu_tr`, `mu_en` and `g-bar` (the average fraction of energy lost to radiation needed for the calculation of `mu_en`)
- Written in [MORTRAN3](#)

Source code: [\\$HEN_HOUSE/user_codes/g](#)

3.6 egs_chamber

- Calculates dose and energy deposited in regions/media
- Optimized for ionization chamber calculations
- Many relevant variance reduction techniques
- Written in c++ (egs++ application)

Documentation: PIRS-898/egs_chamber

Source code: [\\$HEN_HOUSE/user_codes/egs_chamber](#)

3.7 egs_cbct

- Main goal is the fast estimation of the scatter contribution to an ideal detector in a cone-beam CT (CBCT) setup by means of sophisticated variance reduction techniques and a smoothing algorithm
- Can also be used for estimating the total signal to the detector and its individual components: transmitted and scattered
- Initially designed for the purpose of simulating a CBCT setup, but can be equally used for modelling conventional CT scanner setups
- Written in c++ (egs++ application)

Documentation: PIRS-898/egs_cbct

Source code: [\\$HEN_HOUSE/user_codes/egs_cbct](#)

3.8 **egs_fac**

- Designed for the purpose of calculating free air chamber (FAC) correction factors
- Written in c++ (egs++ application)

Documentation: PIRS-898/*egs_fac*

Source code: [\\$HEN_HOUSE/user_codes/egs_fac](#)

3.9 **cavity**

- Calculates dose in ionization chamber and correction factors
- The predecessor of *egs_chamber* and *egs_fac*
- Written in c++ (egs++ application)

Documentation: PIRS-898/*cavity*

Source code: [\\$HEN_HOUSE/user_codes/cavity](#)

3.10 **Tutorials (the tutor codes)**

- Many tutorial applications are distributed, written both in [MORTRAN3](#) and c++
- The codes are heavily documented and intended to teach the user how to write EGSnrc applications

Source code: [\\$HEN_HOUSE/user_codes](#)

4 Getting dirty: writing your first EGSnrc application

Before approaching the more formal descriptions, let's dive right in to actually writing a simple application that will utilize the EGSnrc core physics to perform a Monte Carlo simulation. This hands-on approach will hopefully serve as an instructive lesson on what an EGSnrc application really does, how to write input files and perform simulations, and how to decipher the output.

During the installation, two directories were created and assigned to environment variables: `$HEN_HOUSE` and `$EGS_HOME`. To write an application, navigate to `$EGS_HOME` using a file browser, or on the terminal.

If you're working on linux but unfamiliar with environment variables, now might be a good time to do some quick research. Familiarize yourself with basic terminal commands, the shell you are using and the shell configuration (e.g. the `~/.bashrc` file if you are using the `bash` shell).

4.1 Create a new egs++ application

Let's create a new egs++ application called `myapp`. All EGSnrc applications—including the ones you write yourself—must reside in a folder with the name of the application, within `$EGS_HOME`. When we say egs++, we are referring to using the c++ class library that was built to allow for c++ applications to directly interface with the `MORTRAN3` EGSnrc core.

Start by creating a new directory for your application, `myapp`. Then copy the following files from the existing application, `tutor7pp`: `array_sizes.h`, `Makefile`. We will provide the linux commands to do this, using a bash shell (the first `$` on the left represents the `bash` shell command prompt, so don't type it in):

```
$ cd $EGS_HOME
$ mkdir myapp
$ cd myapp
$ cp ../tutor7pp/array_sizes.h ./
$ cp ../tutor7pp/Makefile ./
```

Open the `Makefile` using your text editor of choice, and change instances of “tutor7pp” to “myapp”. There are only two lines that need changing; they should now read:

```
USER_CODE = myapp
```

and

```
EGSPP_USER_MACROS = myapp.macros
```

Create a new file called `myapp.macros`, and just save it as an empty file. This file is just a placeholder for now - it may be used by more advanced applications.

Finally, create a new file named `myapp.cpp`, and save the following source code in it:

```
#include "egs_advanced_application.h"
APP_MAIN (EGS_AdvancedApplication);
```

This is the smallest, fully functional EGSnrc application you can write! You may consider it a bit of a cheat - we are using the defined function macro `APP_MAIN` to write some code for us.

Compile your application with the `make` command, which builds the executable `myapp` and saves it under `$EGS_HOME/bin/`. For a given application, you only need to run `make` once (unless you modify the source code).

```
$ make
```

4.2 Create an egs++ input file

Next we need to create an input file to define the conditions of the simulation. Create a new file named `slab.egsinp` in the application directory, `$EGS_HOME/myapp/`. Copy and paste the following contents:

```
#####
#
# Simple slab simulation
#
#####

#####
### RUN CONTROL
#####
:start run control:
    ncase = 1e3 # The number of histories to simulate
:stop run control:

#####
### GEOMETRY
#####
:start geometry definition: # Only 1 geometry definition block allowed

    ### Define a plate of tantalum
    :start geometry: # Many geometry blocks can be defined
        name = slab # This name can be anything you like
        library = egs_ndgeometry
```

```

    type = EGS_XYZGeometry
    x-planes = -5, 5 # in cm
    y-planes = -5, 5 # in cm
    z-planes = -10, 0, 0.1, 10 # in cm
    :start media input:
        media = vacuum tantalum # Indexed as 0 1
        set medium = 1 1 # Set region 1 to medium 1 (tantalum)
                        # Other regions default to medium 0 (vacuum)
    :stop media input:
:stop geometry:

### Use the geometry with this name for the simulation
simulation geometry = slab

:stop geometry definition:

#####
### MEDIA
#####
:start media definition: # Only 1 media definition block allowed

    # Defining media in the input file like this is called "pegsless" mode

### Cutoff energies, in MeV
ae = 0.521 # lowest energy for electrons (kinetic+0.511)
ap = 0.01 # lowest energy for photons (kinetic)
ue = 50.511 # maximum energy for electrons (kinetic+0.511)
up = 50 # maximum energy for photons (kinetic)

### Tantalum
:start tantalum: # this name can be anything
    density correction file = tantalum # name the file (no ext.)
:stop tantalum:

### Lead
:start lead:
    density correction file = lead
:stop lead:

### Water
:start water:
    density correction file = water_liquid
:stop water:

:stop media definition:

```

```
#####
### SOURCE
#####
:start source definition: # Only 1 source definition block allowed

    ### Pencil beam
    :start source:
        name = pencil_beam # This name can be anything you like
        library = egs_parallel_beam
        charge = -1
        direction = 0 0 1
        :start spectrum:
            type = monoenergetic
            energy = 20 # in MeV
        :stop spectrum:
        :start shape:
            type = point
            position = 0 0 -10 # in cm
        :stop shape:
    :stop source:

    ### Use the source by this name for the simulation
    simulation source = pencil_beam

:stop source definition:

#####
### VIEWER CONTROL
#####
:start view control:
    # Here we are just assigning some colors for nice
    # viewing in the egs_view application
    set color = lead 120 120 200 200
    set color = tantalum 120 255 120 255
    set color = water 0 220 255 200
:stop view control:

#####
### AUSGAB OBJECTS
#####
:start ausgab object definition: # Only 1 ausgab definition block allowed
```



```
### Particle tracks
:start ausgab object:
    name = tracks
    library = egs_track_scoring
:stop ausgab object:

:stop ausgab object definition:
```

We'll get into the meaning of the various input parameters later! For now, let's just run the simulation and see what happens. The input file models **a thousand 20 MeV electrons incident on a 1 mm thick tantalum plate.**

```
$ myapp -i slab.egsinp
```

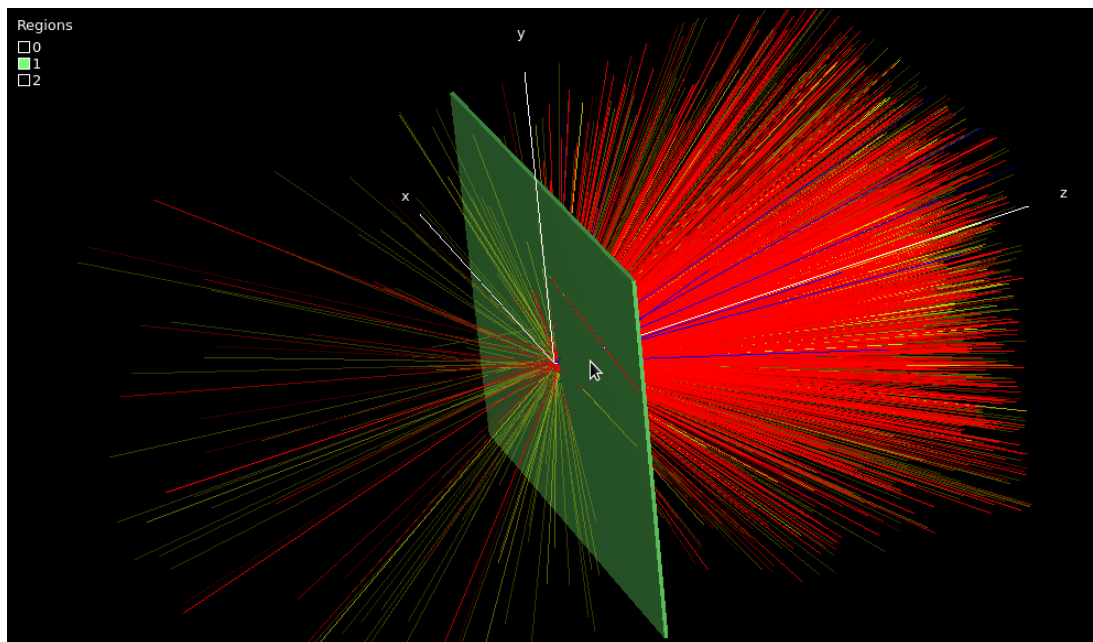
Here `myapp` is the executable name, `-i` is a *flag* to specify the input file `slab.egsinp`. This simulation runs in less than a second and prints results on the terminal. It also saves information about the simulated particle tracks in the file `slab.ptracks`.

This command can actually be executed on the terminal from any directory! The `myapp` executable is found automatically using your `$PATH` environment variable, and the input file is *must* reside in the application directory (`$EGS_HOME/myapp`)

Load the `slab.egsinp` input file and `slab.ptracks` in `egs_view` utility to understand the geometry of the simulation. Click the **Show tracks** checkbox, rotate around the scene (left mouse button in the image window), and zoom in and out (scroll wheel). Pan the view by holding Ctrl and dragging with the left mouse button.

```
$ egs_view slab.egsinp slab.ptracks
```

After turning on particle tracks, `egs_view` should display something like the following image. In the figure, a pencil beam of electrons is impinging from the left (negative z), towards the right (positive z). Notice that by hovering your mouse over the plate, you can investigate the region numbers that were assigned to the geometry. Looking down a ray from your mouse, the numbers are displayed in the top left. In this case, the first region the ray sees is the air above the plate (region 0), then the Ta plate (region 1), then the air below the plate (region 2). These numbers that are assigned to geometrical regions are important, as we will see later!



Congratulations, you created your first *egs++* application and ran your first simulation!

4.2.1 Questions

- 1 Aside from particle tracks, does the *myapp* application provide any useful information when it runs, such as energy deposited, dose, spectrum, etc.?

Not really. By default when an EGSnrc application runs it echoes to the terminal some information about the simulation configuration, parameters, data initialization, media definitions, etc. Finally, the simulation reports progress and some final timing statistics. But in this first example the result is always zero (and the uncertainty 100%) because no physical quantity has been *scored* yet:

```
Running 1000 histories

  Batch CPU time Result Uncertainty(%)
=====
  1 0.08 0 100.00
  2 0.16 0 100.00
  3 0.23 0 100.00
  4 0.31 0 100.00
  5 0.40 0 100.00
  6 0.49 0 100.00
  7 0.56 0 100.00
  8 0.64 0 100.00
  9 0.72 0 100.00
 10 0.80 0 100.00

Finished simulation

Total cpu time for this run: 0.80 (sec.) 0.0002(hours)
Histories per hour: 4.5e+06
Number of random numbers used: 13938479
Number of electron CH steps: 345747
Number of all electron steps: 1.2252e+06
```

4.3 Compile and run using `egs_gui`

All EGSnrc applications can be compiled and run using the GUI, `egs_gui`. Let's rewind, and repeat the `myapp` example using `egs_gui` instead of the command line.

1. Open `egs_gui`, the GUI for compiling and executing any EGSnrc application.
2. Notice that the `Compile` tab is selected by default. Use the selection box in the bottom left of the GUI, and select `myapp`.
3. Click the `Go` button. It will take some time to compile. If you see `ExitCode = 0` in the output, the application has compiled successfully! If you are *recompiling* an application, make sure to `clean` first.
4. Switch to the `Execute` tab.
5. In the `PEGS file` section, check the `pegsless` checkbox. This means media are defined in the input file.
6. In the `Input file` section, click the `...` button, and select the input file we created earlier: `slab.egsinp`.
7. To run the simulation, just click `Start`.
8. The output of the simulation is displayed in the `egs_gui` output textbox after executing the simulation. Depending on the application, there may be additional output in the corresponding application directory in `$EGS_HOME`.

4.4 Add a dose scoring object

The little application `myapp` does not really report any quantity of interest: by default the `egs++` application just transports the particles in the geometry. It is up to the user to implement methods to extract information from the simulation.

The easiest way to get results is by adding a dose scoring objects in your input file. For example, to get a report of the dose deposited in every region of the geometry at the end of the simulation, **add** the following input block **inside** the existing `ausgab object definition` delimiters at the end of your input file (you will learn more about the `egs++` syntax later). After this, you should **only have a single `ausgab object definition`**, containing two `ausgab object` blocks.

If your text editor supports syntax highlighting, try setting the language to *ruby*. This does not match our syntax exactly but still helps the eye.

```
:start ausgab object definition:

# ... there is an existing object here for ptracks
```

```

: start ausgab object:
  name = dose
  library = egs_dose_scoring
  dose regions = 1
  volume = 10
: stop ausgab object:

: stop ausgab object definition:

```

Run the simulation again, and find the “Summary of region dosimetry” in the output:

```
$ myapp -i slab.egsinp
```

4.4.1 Questions

1 How much energy is deposited inside the tantalum plate? What is the dose?

With a dose scoring object, the application echoes additional data to the screen at the end of the simulation. The energy deposited in the tantalum plate is $(2.5506 \pm 1.828\%)$ MeV. The dose is $(2.4535 \times 10^{-12} \pm 1.828\%)$ Gy.

```

==> Summary of region dosimetry (per particle)
ir medium rho/[g/cm3] V/cm3 Edep/[MeV] D/[Gy]
-----
1 tantalum 16.654 10.0000 2.5506e+00 +/- 1.828 % 2.4535e-12 +/- 1.828 %
-----

```

2 Can you manually convert the value of deposited energy to dose ?

Dose is defined as the energy deposited per unit mass, usually reported in Gray (J/kg). The density of tantalum here is 16.654 g/cm³, and the volume of the plate is 10 cm³. Therefore, the mass of the plate is 166.54 g, and the dose is:

$$D = \frac{2.5506}{166.54} = 0.015315 \text{ MeV/g} = 2.4538 \times 10^{-12} \text{ J/kg}.$$

3 What is the relative uncertainty on energy and dose, and why is it the same for both ?

The relative uncertainty on the deposited energy *and* the dose is 1.828%. It is exactly the same in both cases because dose is simply the energy divided the mass. The mass is derived from simulation inputs which are considered *exact* (the density and the volume), hence it does not contribute to the relative uncertainty.

4 Figure out how to increase the number of electrons by a factor 10 in the input file and run the simulation again. Why has the deposited energy not increased by a factor 10 ? What happened to the uncertainty ?

Changing the run control input to `ncase = 1e4` simulates 10⁴ incident electrons. The deposited energy is now $(2.5893 \pm 0.572\%)$ MeV. It has not increased by a factor 10 because Monte Carlo results are reported *per incident particle*. The relative uncertainty has however decreased by a factor of about 3, which is typical of a random sampling process, for which the uncertainty scales like $1/\sqrt{N}$.

4.5 Change the type of incident particles

Figure out how to change the incident particle type to *photons* instead of electrons in `slab.egsinp`. Save and run the simulation again for 10⁴ incident photons. Load the particle tracks in `egs_view`.

4.5.1 Questions

1 Is the simulation faster with electrons or with photons ?

Changing the charge of the source (`charge = 0`) runs the simulation for incident photons instead of electrons. The simulation is about 10 times faster with photons because photons interact with matter much less than do electrons.

2 How did the dose change compared to electrons ?

The dose decreased by a factor of about 10 compared to electrons, again indicating that photons interact less than do electrons. Correspondingly, the uncertainty is larger than with electrons, because fewer events are sampled inside the plate.

3 Are positrons generated in this simulation? Is this expected?

Yes, positrons are generated: they show up as blue tracks in the viewer. This is expected because the incident photons have an energy of 20 MeV, well above the pair production threshold of 1.022 MeV.

4.6 Change the energy of incident particles

Figure out how to change the incident particle energy to 1 MeV in `slab.egsinp`. Save and run the simulation again for 10^4 incident photons. Load the particle tracks in `egs_view`.

4.6.1 Questions

1 What is the biggest qualitative difference with the 20 MeV scenario?

The most striking difference is that there are much fewer secondary charge particles. Note also that there are no positrons since the incident energy is now below the pair production threshold of 1.022 MeV.

2 Did the dose increase or decrease?

The dose is further decreased to $(5.5088 \times 10^{-14} \pm 3.613\%)$ MeV per incident photon.

3 How about simulation time?

The simulation time is reduced. As a general rule, lower energy simulations run faster because there are fewer secondary particles to transport.

4.7 Change the material of the plate

Figure out how to change the material of the plate to `lead` in `slab.egsinp`. Use **100 keV electrons** and run the simulation again for 10^4 incident electrons. Load the particle tracks in `egs_view`. *Hint:* the materials are set in the `geometry definition`. The `media definition` defines the list of media that may (or may not) be used by geometries.

4.7.1 Questions

1 What is happening to the electrons hitting the lead plate?

The electrons are either absorbed in the lead or backscattered in the $-z$ direction.

2 Is that consistent with the deposited energy?

Yes, indeed. The energy deposited in the lead plate by 100 keV electrons is found to be

$(0.058803 \pm 0.709\%)$ MeV per incident electron. So on average only 60% of the incident energy is deposited in the lead plate, owing to the significant backscatter.

3 What is the file size of `slab.ptracks` on disk?

About 5.3 MB. Track files grow very quickly, but by default the number of events saved to file is limited to 1024, even if you run more incident particles.

4 Run 20 MeV electrons striking a `water` plate and interpret the results. Is deposited energy consistent with the known stopping power value of 2.45 MeV cm²/g for 20 MeV electrons in liquid water?

Most electrons go through the water plate, depositing on average 175 keV in the process. This is certainly *less* than the energy loss of 245 keV expected from the nominal electron stopping power, because a lot of energy is carried away from the plate by secondary particles, mostly bremsstrahlung photons.

5 Navigating egs++ input files

EGSnrc now includes egs++, a C++ class library that interfaces with the EGSnrc core [MORTRAN3](#) code. This allows for applications to be written in C++. To expand on what was covered in the previous section, we will play with one of the egs++ tutorial applications, [tutor7pp](#), and experiment with making changes to the input file.

There is an example input file included with [tutor7pp](#), [test1.egsinp](#), found in the application directory (`$EGS_HOME/tutor7pp`). Let's run [tutor7pp](#) with this example. If you look inside the [test1.egsinp](#) file, the geometry is defined as an infinite slab of tantalum, 1 mm thick. A parallel beam of 20 MeV electrons is directed at the slab, and the simulation runs 1000000 particles. Additionally, [tutor7pp](#) has special scoring options that are not available to other applications. Each application may define its own input parameters, but generally nearly all of the inputs (e.g. geometries, sources) are the same. For [tutor7pp](#), the application requires some additional inputs that define regions in which to score pulse height distributions (this is at the bottom of the example input file).

Compile and run [tutor7pp](#) using [egs_gui](#), or on the command line.

If you choose to use the command line, the following commands are an example for linux operating systems:

```
$ cd $EGS_HOME/tutor7pp
$ make clean
$ make
$ tutor7pp -i test1 -p tutor_data | tee test1.output
```

Notice that we have modified the execution line above to pipe the output to [tee](#) in order to both display it in the terminal as well as save it in an output file. If you look in the application directory after the simulation, it will contain some new files! Namely:

```
test1.egsdat
test1.output
```

The [.egsdat](#) file can generally be ignored - it is used by EGSnrc to restart jobs that crashed, or combine parallel runs. The [.output](#) file is where we manually redirected the output from the simulation (if you ran the simulation using the GUI, this file was not created). Looking at the output, we can observe the reflected, deposited and transmitted energy fractions for the simulation of electrons through a single slab. Below this, is the pulse height distribution for region 1 (the slab itself).

5.1 Write a new input file

When writing an egs++ input file, you will find yourself frequently referencing the full online documentation, [PIRS-898](#):

<http://nrc-cnrc.github.io/EGSnrc/doc/pirs898/index.html>

If you would prefer to have a local copy, download all of the manuals ([EGSnrc-manuals.zip](#)) from the release page:

<https://github.com/nrc-cnrc/EGSnrc/releases>

All egs++ applications share the same common input format, though each application may add new input blocks or new input parameters. It is strongly recommended to browse the [PIRS-898](#) documentation on *common inputs*. This will provide an overview of what to expect in an input file:

<http://nrc-cnrc.github.io/EGSnrc/doc/pirs898/common.html>

To get started writing your first input file from scratch:

- Open your favourite text editor! *On linux, some editor suggestions are kate, gedit, vim, emacs, etc. On Windows, perhaps notepad++.*
- There is not a standard syntax highlighter for [.egsinp](#) files. However, you may want to try highlighting for the [Ruby](#) language - it does not match EGSnrc syntax exactly but does do a reasonable job of helping the eye.
- Save a new file with the [.egsinp](#) extension, in the [\\$EGS_HOME/tutor7pp](#) directory.

To get started with egs++ input files, let's work through an example for the [tutor7pp](#) application. Of course, this example will largely translate to any egs++ application, with the exception of the scoring options, which are specific to [tutor7pp](#). If you read through the [PIRS-898](#) documentation on common input blocks, you will recognize that input files contain:

- Geometries
- Particle sources
- Run control
- MC transport parameters
- Media (optional)
- Ausgab output (optional)
- Scoring options (application specific)

5.1.1 Define a cylindrical water phantom

The first step is to define the geometry, which must be placed between the [geometry definition](#) start and stop delimiters in the input file.

First define a cylinder with axis parallel to the z -axis and 5 cm radius. All distance units in `.egsinp` files are in cm, and energy units in MeV.

```
:start geometry definition:

  :start geometry:
    library = egs_cylinders
    type = EGS_ZCylinders
    name = the_cylinder
    midpoint = 0 0 0 # in cm
    radii = 5 # in cm
  :stop geometry:

:stop geometry definition:
```

How did we figure out what to type? By looking at the documentation for `egs_cylinders`, of course.

The above input block defines an *infinitely long* cylinder of 5 cm radius. To produce a finite cylinder of 10 cm height with its front face at $z = 0$, first define bounding planes and then put everything together using an ND geometry. Add these geometries to the geometry definition block:

```
:start geometry:
  library = egs_planes
  type = EGS_Zplanes
  name = the_planes
  positions = 0 10 # in cm
:stop geometry:

:start geometry:
  library = egs_ndgeometry
  name = phantom
  dimensions = the_planes the_cylinder
  :start media input:
    media = H20521ICRU
  :stop media input:
:stop geometry:
```

This is one way of creating a *composite geometry*. As described in the `egs_ndgeometry` documentation, the `egs_ndgeometry` creates regions where the listed geometries (`dimensions`) intersect. Since `the_planes` contains only one region between $z = 0$ and $z = 10$, the intersection with `the_cylinder` contains a cylinder that has been cut by those planes.

Instead of defining media in the input file, we will be doing it *the old way*. The media used above was [H2O521ICRU](#), which corresponds to the name of a water material defined in an external PEGS data file ([HEN_HOUSE/pegs4/data/521icru.pegs4dat](#)). In order to run the simulation (we'll get to this later), the 521icru PEGS4 data file will need to be included as an argument.

Don't forget to tell the [EGS_Application](#) which geometry is the simulation geometry:

```
simulation geometry = phantom
```

After placing all these blocks into the geometry definition block, visualize the geometry with [egs_view](#) and enjoy the fruit of your hard work (of course, use the filename of the input file you just created):

```
$ egs_view yourFile.egsinp
```

There is some description of how to use [egs_view](#) in the geometry module.

5.1.2 Define an isotropic point source

The next step to define the source. Similar to the geometry definition block, you can define as many sources as desired, create composites, and choose the one to use for your simulation.

Define an isotropic point source of 100 keV photons. This source emits particles uniformly in all directions. Position the source at -100 cm from the origin on the z -axis:

```
:start source definition:

  :start source:
    library = egs_point_source
    name = the_point_source
    position = 0, 0, -100 # in cm
    :start spectrum:
      type = monoenergetic
      energy = 0.1 # in MeV
    :stop spectrum:
    charge = 0
  :stop source:

  simulation source = the_point_source

:stop source definition:
```

5.1.3 Define a dose scoring object

Score dose using the ausgab object `egs_dose_scoring` which must be defined between the `:start ausgab object definitions:` delimiter and corresponding `stop` delimiter:

```
:start ausgab object definition:

  :start ausgab object:
    library = egs_dose_scoring
    volume = 785.398163397 # in cm^3
    region dose = yes
    name = my_dose_scoring
  :stop ausgab object:

:stop ausgab object definition:
```

The volume is provided for the cylindrical phantom to be able to compute the dose. By default volumes are set to 1 cm³, so if you do not define the volume, the dose will be incorrect! However, the energy deposited will still be correct, so it is easy to convert to dose after the simulation is complete.

5.1.4 Define the run control

The last piece of input required for the simulation to run, albeit with many defaults, is the number of histories to run, `ncase`. This value is defined in the `run control` block

```
:start run control:
  ncase = 100000
:stop run control:
```

5.2 Run the simulation

To run the simulation type in the following command. The material data is included using the `-p` flag, followed by the name of the PEGS4 data file.

```
$ tutor7pp -i myinput -p 521icru
```

Since the output goes to the screen by default, you can *pipe* the output to the `tee` command to *simultaneously* save the terminal output to a file, for later analysis of the results:

```
$ tutor7pp -i myinput -p 521icru | tee myoutput.txt
```

5.2.1 Questions

- 1 Compare the energy deposited reported by the `egs_dose_scoring` object and the one corresponding to the fraction of energy deposited reported by `tutor7pp`. Are they the same?

`tutor7pp` reports the total energy entering the geometry, E_{tot} , and the fractions, f_i , of that energy deposited in the different regions i of the geometry. In this case there is only one region. The energy deposited in that region, E_{dep} , is obtained as

$$E_{\text{dep}} = E_{\text{tot}} \cdot f_0 = 10^4 \text{ MeV} \cdot 0.1934056 = 1934.056 \text{ MeV}$$

The `egs_dose_scoring` scoring object reports the dose deposited in each region i divided by the source's normalization which could be the number of particles emitted or the fluence. The isotropic point source uses the number of particles emitted as normalization. Hence to obtain the energy deposited in the geometry, one has to multiply the reported energy deposited per particle $\left(\frac{E_{\text{dep}}}{N}\right)$ by the number of particles emitted, N (`last case`):

$$E_{\text{dep}} = \left(\frac{E_{\text{dep}}}{N}\right) \cdot N = 1.2082 \cdot 10^{-5} \text{ MeV} \cdot 160072124 = 1933.991 \text{ MeV}$$

Note above that 10^5 is the number of particles hitting the geometry!

- 2 What is the solid angle, Ω , subtended by the isotropic source and the front face of the geometry?

The number of particles emitted by the isotropic point source in 4π is 160072124, while only 10^5 of these particles actually hit the front face of the cylinder. Hence Ω can be obtained from

$$\Omega = \frac{10^5}{160072124} \cdot 4\pi = 0.00785 \text{ radians}$$

- 3 Compute the efficiency ε based on the uncertainty in the energy deposited calculated with the `egs_dose_scoring` object and take note of it for a later comparison with the collimated source.

$$\varepsilon = \frac{1}{t_{\text{CPU}} \cdot \sigma^2} = \frac{1}{0.482^2 \cdot 67.67s} = 0.0636 \text{ s}^{-1}$$

5.3 Simulate a collimated source

Run the same calculation using a collimated source. Add the following source definition input to the source definition block

```

:start source:
  library = egs_collimated_source
  name = my_collimated_source
  distance = 100
  :start source shape:
    type = point
    position = 0 0 -100
  :stop source shape:
  :start target shape:
    library = egs_circle
    radius = 5
  :stop target shape:
  :start spectrum:
    type = monoenergetic
    energy = 0.1
  :stop spectrum:
  charge = 0
:stop source:

```

Set the simulation source to be `my_collimated_source`. You may want to make a copy of your input file.

Run the simulation again by issuing the following command (with your choice of names for the input and output files):

```
$ tutor7pp -i myinput -p 521icru | tee myoutput.txt
```

5.3.1 Questions

- 1 Compare the *fraction* of energy deposited with the result obtained for the isotropic point source. Are they the same? Explain!

Although the normalization used by both sources is different, the fraction of energy deposited in the geometry, f_0 , is the same within statistics since as a relative quantity, it should be independent of the normalization.

$$f_0^{\text{isotropic}} = 0.1934, \quad f_0^{\text{collimated}} = 0.1948$$

- 2 Compare the energy deposited reported by the `egs_dose_scoring` object with the same result obtained earlier for the isotropic point source. Are they the same? Explain!

The collimated source uses the fluence F as normalization. For this reason, the values reported by the `egs_dose_scoring` in this case will be numerically different from the

value reported for the isotropic source:

$$\frac{E_{\text{dep}}}{F} = 1.5272 \text{ MeV cm}^2 \text{ rad}.$$

To get the absolute energy deposited in the geometry, E_{dep} , one must use the following expression

$$E_{\text{dep}} = \frac{E_{\text{dep}}}{F} \cdot \frac{N_{\Omega}}{(\Omega \cdot d^2)} = 1.5272 \cdot \frac{10^5}{0.00785 \cdot 10^4} \text{ MeV} = 1945.478 \text{ MeV}$$

This value differs by 0.6 % from the value obtained for the isotropic point source which is within the combined uncertainties of the calculations.

A side note : the value reported in this case for the total energy, called here E'_{tot} , does not correspond to the actual total energy E_{tot} . The quantity E'_{tot} is calculated in [tutor7pp](#) as

$$E'_{\text{tot}} = \sum_{i=1}^{N_{\Omega}} w'_i e_i$$

which for the collimated source actually corresponds to

$$E'_{\text{tot}} = E_{\text{tot}} \cdot \Omega.$$

Since the purpose of [tutor7pp](#) is to calculate the energy fractions, normalization is irrelevant. To get the actual total energy deposited E_{dep} from E'_{tot} , one needs to use the expression

$$E_{\text{dep}} = \frac{E'_{\text{tot}} \cdot f_0}{\Omega} = \frac{78.3926 \text{ MeV rad} \cdot 0.19482}{0.00785 \text{ rad}} = 1945.535 \text{ MeV}$$

This value is in excellent agreement with the value obtained above from the output of [egs_dose_scoring](#).

- 3 Compute the efficiency based on the uncertainty in the energy deposited calculated with the [egs_dose_scoring](#) object and compare it with the efficiency of the isotropic source. Do you see any difference?

$$\varepsilon = \frac{1}{t_{\text{CPU}} \cdot \sigma^2} = \frac{1}{0.362^2 \cdot 3.02s} = 2.5268 \text{ s}^{-1}$$

The collimated source is about **40 times more efficient** than the isotropic point source!

- 4 Can you think of a situation where the collimated source should not be used?

The collimated source does not emit particles in all directions, which may not be a good model of reality. If an isotropic source was used instead, the particles directed away from your region of interest *may still contribute to it* by scatter or secondary particles. By collimating the source, these contributions are lost! Thus it is essential to consider whether scatter from the geometry surrounding the source is important for the situation you are modelling.

5.4 Generate particles tracks

And now comes the fun part: *particle tracks*!

You will be generating particle tracks with the collimated source. To this end, you must define an `egs_track_scoring` object between the `ausgab object definition` delimiters in our input file. Here is an example:

```
:start ausgab object:
  library = egs_track_scoring
  name = tracker
  score electrons = no
  score positrons = no
  score photons = yes
  # start scoring = 0 # default is 0
  # stop scoring = 1024 # default is 1024
  # buffer size = 1024 # default is 1024
:stop ausgab object:
```

Tracks are only *scored* inside the geometry, hence particle tracks are only generated inside the water cylinder. To see the particle track from the source all the way to the phantom, you need to embed the cylinder and the source inside another geometrical object.

Use a large box of air as an envelope geometry. Add the following two geometries in your geometry definition block:

```
:start geometry:
  library = egs_box
  box size = 500 500 500
  name = air_box
  :start media input:
    media = AIR521ICRU
  :stop media input:
:stop geometry:

:start geometry:
  library = egs_genvelope
  name = phantom_in_air
  base geometry = air_box
  inscribed geometries = phantom
:stop geometry:
```

and switch the simulation geometry to be `phantom_in_air`. Run the simulation once more with 10 times fewer particles. This simulation should take a fraction of a second and produce the particle track file `myinput.ptracks`. To visualize it launch `egs_view` with the name of the tracks file:


```
$ egs_view myinput.egsinp myinput.ptracks
```

In the GUI check the box **Show tracks** in the Particle Tracks section to display the tracks. You can also load the track files afterwards as well with the **Load tracks** button. To see tracks, make the air box transparent or define a clipping plane. The effect of the collimated source becomes more apparent as one zooms into the picture.

6 Modelling an ionization chamber

In section 4, we wrote the simplest of egs++ applications and used `egs_view` for the first time. This geometry viewing tool turns out to be essential when writing complex geometries - both to visually confirm the accuracy of the design, and to determine the number assigned to *regions* in the geometry. In the following sections, use this tool frequently!

The `egs_view` tool actually uses the `HOWFAR` function of each geometry to render the view. This means that what you see *is what a particle in your simulation sees*. Note that `egs_view` is only compatible with egs++ input files (not the `MORTRAN3` codes).

For more explanation of `egs_view`, refer to the online documentation for the geometry module.

When trying to simulate a particular experiment, constructing the geometry is half the battle! Take your time to learn what can be done; starting from a similar example and tweaking it for your purposes is a key time-saver. There are geometry examples online, and in the distribution:

[\\$HEN_HOUSE/egs++/geometry/examples](#)

Before building an ionization chamber, we'll lay a bit more foundation in geometry design and use of `egs_view`. In a text editor, create a new document with the following geometry input, describing a cubic box:

```
:start geometry definition:

    ### a simple box definition
    :start geometry:
        name = my_box # this name is up to you
        library = egs_box
        box size = 10 # length units are cm
        :start media input:
            media = my_medium
        :stop media input:
    :stop geometry:

    ### name of the geometry to load in the viewer
    simulation geometry = my_box

:stop geometry definition:
```

Note that white space is irrelevant—except for line returns because egs++ reads one input per line. Comment your code as you go, and remember the `simulation geometry` input to tell the viewer which geometry to load (by **name**).

Media names are up to you: the viewer merely assigns different colors to different media. These names will ultimately have to be defined in the `media definition` input block (or a PEGS4 data file) to run a simulation. However, the viewer displays media even if they are

not yet defined.

For a description of the egs++ input syntax for each geometry, and for input examples, refer to the online manual: <http://nrc-cnrc.github.io/EGSnrc/doc/pirs898/index.html>, in the [Geometries](#) section, e.g., [Geometries > EGS_Box](#).

6.1 Visualize your geometry with egs_view

Save your box geometry input in a file named `simple.egsinp`. From the directory where you saved the file, launch the geometry viewer:

```
$ egs_view simple.egsinp &
```

You should see a red square in the geometry window, along with a **View Control** window. The viewer starts with the camera on the positive z-axis, but you can rotate the geometry in 3D by dragging with the mouse, as shown in figure 1a.

You can pan the view by dragging while pressing the `Ctrl` key, and zoom in and out with the middle mouse button or the mouse wheel. Note that these manipulations do *not* affect your input: `egs_view` *never writes to the input file*.

Spend a few minutes exploring the **Material colors**, **View** and **Look at** inputs to understand what they control. At any time you can export the current view as an image with the **Save image** button, while the **Close** button quits the application altogether.

The **Reload** button reads the input file again, which is useful to update the view upon modifying the input file, without having to launch the viewer again. *Be warned however that reloading is not robust: some changes in the input file may cause the viewer to quit abruptly!*

6.1.1 Question

1 What are the region numbers inside and outside the cube?

When hovering the mouse over the cube, a small red square appears in the top left corner of the image window, indicating that the inside of the cube is **region 0**.

The region *outside* the cube is not displayed in the list. In egs++ everything outside the defined geometry is **region -1**. Particles that reach region -1 in the simulation are immediately discarded: they have reached the “end of the world”!

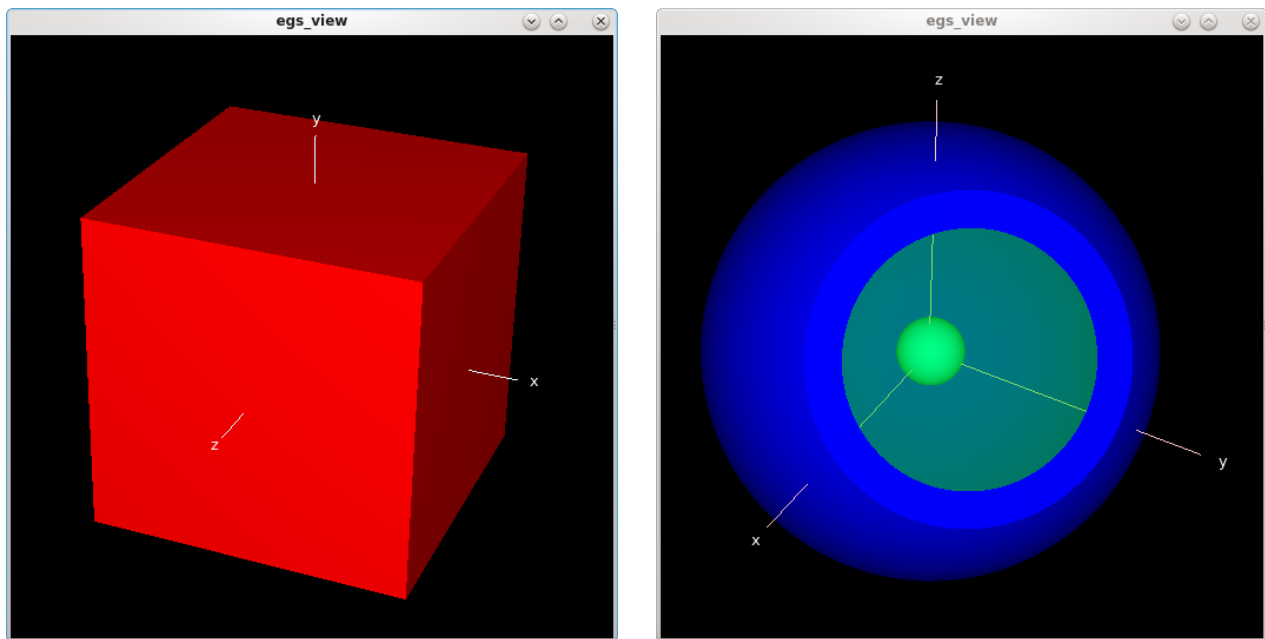


Figure 1: a) snapshot of the cube defined in the `simple.egsinp` input file. b) snapshot of the point of view you are asked to reproduce using viewer clipping planes in Section 6.2.

6.2 Dig in with clipping planes

Edit your input file to add a second geometry (inside the **same geometry definition** input block) defining a set of concentric spheres:

```
:start geometry:
  name = my_spheres # this name is up to you
  library = eggs_spheres
  radii = 0.3 1.8 2 # three concentric spheres
  :start media input:
    media = default shell center # names of media 0 1 2 ...
    set medium = 2 1 # set region 2 to medium 1 (shell)
    set medium = 0 2 # set region 0 to medium 2 (center)
  :stop media input:
:stop geometry:
```

By default the whole geometry is filled with medium 0 (**default**, the first one in the **media** list), so it is redundant to specify **set medium** for regions containing medium 0.

Change the **simulation geometry** to **my_spheres** (or the name you chose) and reload the input in the viewer. You can play with transparency to reveal the internal structure of the geometry, but clipping planes provide a more direct route. Click on the **Clipping planes...** button to access the planes entry table. Apply a clipping plane oriented along the positive z-axis and passing through the origin. Make the **default** medium partially transparent and rotate the geometry around. Apply a clipping plane to get a view similar to the one shown in figure 1b.

For clipping planes, **ax**, **ay** and **az** define the unit normal to a planar surface. The surface can then be offset from the origin by setting **d**. **After typing in the clipping plane, click the “1” to select the row.** Then click **Apply** to update the viewer. It is possible to use many clipping planes at once! It’s a very common mistake to forget to **select the clipping plane row** before clicking **Apply**.

6.2.1 Questions

- 1 From this point of view, what is the list of regions when you hover your mouse over the small central sphere?

When hovering the mouse over the central sphere—with a clipping plane as shown in the figure—the list of regions is **{1, 0, 1, 2}**. These are all the regions *under* the mouse pointer, from the nearest to the farthest. Regions clipped by clipping planes are not included in the list.

- 2 What is the region number *outside* the defined geometry?

As always in **egs++**, the space outside the defined geometry is **region -1**, and it is not included in the list of regions in the image window.

6.3 Build an egs++ model of the 3C chamber

Use your newly acquired expertise in egs++ syntax to model the geometry of NRC's reference ionization chamber "3C", the schematic of which is reproduced in figure 2. This is a cylindrically symmetric geometry with 4 distinct layers along its axis, so the natural option is a **conestack** geometry. Put the conestack along the z-axis, and position it according to the numbers given in the diagram. Below is an input template to get you started. Inspect the geometry with [egs_view](#).

Read the conestack documentation for more information.

You can expect to end up with 4 layers in the conestack. Since there are no sloped surfaces (all of the cones are in fact cylinders), the top and bottom radii will be the same within each layer. Layer 1 requires a single radius, 1.175 for top and bottom. Layer 2 requires two radii for air and graphite, 0.7919 and 1.175, respectively. And so on...

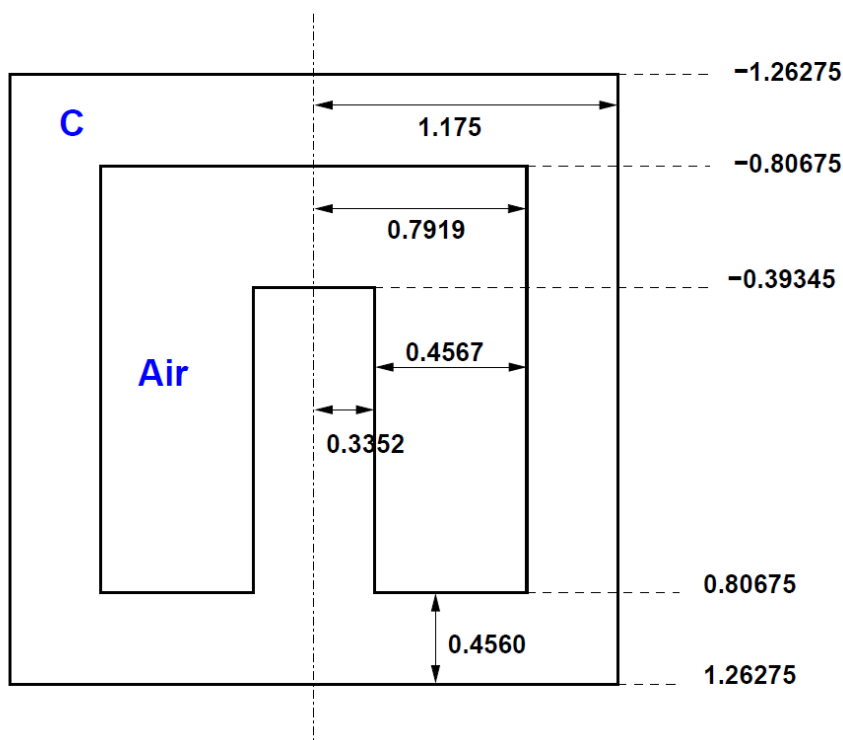


Figure 2: Simplified schematic of NRC's 3C ionization chamber which is Canada's primary standard for air kerma in a ^{60}Co beam. Dimensions are in centimetres. Drawing is not to scale.

```
:start geometry:
  name = 3C
  library = egs_cones
  type = EGS_ConeStack
  axis = # ... figure out the input for 'axis'
```

```

    ### top layer
    :start layer:
        thickness = # ...
        top radii = # ...
        bottom radii = # ...
        media = graphite
    :stop layer:
:stop geometry:

```

6.3.1 Questions

1 What are the region numbers which correspond to the air cavity?

The air cavity comprises **regions 3 and 7**.

2 Where are regions 1 and 2?

Regions 1 and 2 are not visible in this geometry: they are *virtual* regions. In some cases, `egs++` attributes region numbers in a way that is more efficient internally. But the algorithm is deterministic: the same geometry always yields the same region numbers.

3 Can you figure out the conestack region numbering scheme?

In the case of the conestack, `egs++` attributes a fixed number of regions to each layer. This is more efficient because only a single modulo operation is then needed to convert the region number to the layer index.

In the 3C conestack, there are at most three radii per layer, so `egs++` attributes 3 regions per layer, whether or not they are used. The first layer has only one radial dimension—region 0—hence regions 1 and 2 are not used: they become *virtual* regions and don't show up in the viewer. The next layer in the conestack then starts at region 3, and the same logic applies the virtual region 5.

6.4 Build an Exradin A12 chamber

The Exradin A12 is a thimble ionization chamber, shown schematically in figure 3. The basic strategy to model such an instrument is to use a conestack for most of the chamber, which is cylindrically symmetric, except for the spherical tip. Build the chamber tip with **spheres** (as shown by dashed lines in the diagram) and join it with the 7-layer **conestack** chamber body, using a composite **cd geometry**.

6.4.1 Model the spherical chamber tip

Start a new input file for this chamber geometry. Save this input file in the `$EGS_HOME/egs_chamber` directory. It is recommended to use the `egs_chamber` application

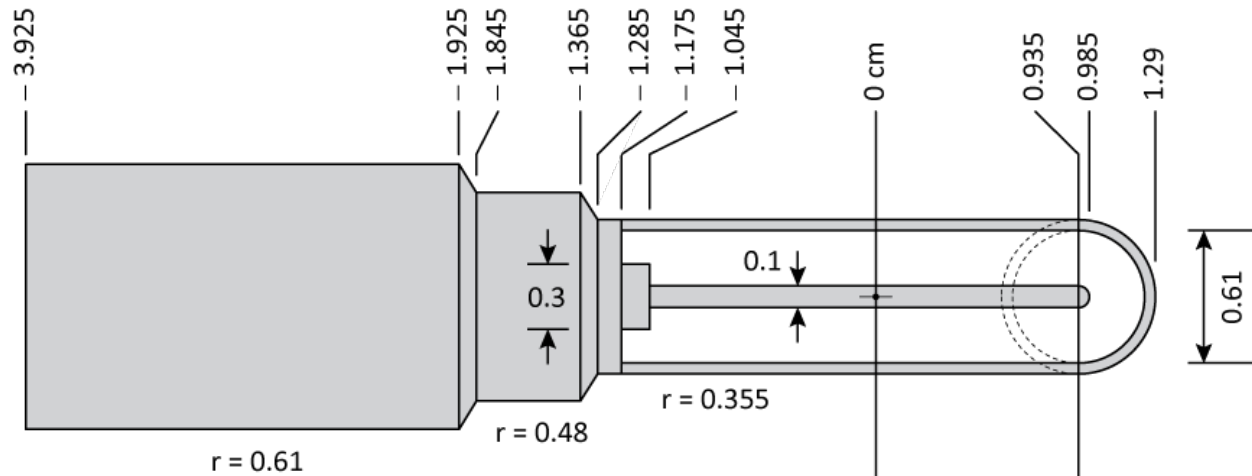


Figure 3: Simplified schematic of an Exradin A12 thimble chamber, based on information from the manufacturer's product brochure. All units are in centimetres. Radial dimensions are given, as well as the position of the various points along the chamber's axis (with the centroid of the collecting volume at 0 cm). The central electrode, the thimble shell and the chamber body are made of C552 plastic.

for ionization chamber simulations due to its useful suite of variance reduction parameters. There are a number of useful features, found in the `egs_chamber` documentation.

Define a geometry for the chamber tip using **2 concentric spheres**, as in Section 6.2. The smallest sphere will become the spherical tip of the central electrode! The outer sphere should be water - the entire chamber will be placed in water and we will end up using a small water shell surrounding the chamber for a variance reduction technique called cross section enhancement. Position the midpoint of the set of spheres on axis in their final location at $x = 0.935$ cm, as shown in figure 3.

```
#-----
# Spherical chamber tip
#-----

:start geometry:
  name = chamber_tip
  library = egs_spheres
  midpoint = 0.935 0 0
  radii = 0.05 0.305 0.355 2.0
  :start media input:
    media = c552 air water # media 0,1,2
    set medium = 1 1 # set region 1 to air (1)
    set medium = 3 2 # set region 3 to water (2)
    # by default the rest of the regions are c552
  :stop media input:
:stop geometry:
```

Include the following bounding box view control input block in your input file (*outside* the geometry definition block) to help the viewer find the spheres. Inspect the geometry in

egs_view.

```
:start view control:
  xmin = -4
  xmax = 4
  ymin = -4
  ymax = 4
  zmin = -4
  zmax = 4
:stop view control:
```

6.4.2 Model the chamber body

Use a **conestack** to define the cylindrically symmetric chamber body. Set the axis of the conestack so that layers are added in the -x direction. In the next step you will join this conestack to the spheres using a cd geometry plane located at $x = 0.935$ cm, so set the top of the conestack at that point. Surround the chamber in water out to 2 cm radius. See if you can design the chamber without copy-pasting the text below!

```
#-----
# Main chamber body (as a conestack)
#-----
:start geometry:
  name = chamber_body
  library = egs_cones
  type = EGS_ConeStack
  axis = 0.985 0 0 -1 0 0 # x-axis

  ### sensitive volume (cylindrical portion)
  :start layer:
    thickness = 2.03
    top radii = 0.05 0.305 0.355 2.0
    bottom radii = 0.05 0.305 0.355 2.0
    media = c552 air c552 water
  :stop layer:

  ### electrode base
  :start layer:
    thickness = 0.13
    top radii = 0.15 0.305 0.355 2.0
    bottom radii = 0.15 0.305 0.355 2.0
    media = c552 air c552 water
  :stop layer:

  ### up to first kink
  :start layer:
    thickness = 0.11
```

```

        top radii = 0.355 2.0
        bottom radii = 0.355 2.0
        media = c552 water
:stop layer:

### first widening
:start layer:
    thickness = 0.08
    bottom radii = 0.48 2.0
    media = c552 water
:stop layer:

### up to second kink
:start layer:
    thickness = 0.48
    bottom radii = 0.48 2.0
    media = c552 water
:stop layer:

### second widening
:start layer:
    thickness = 0.08
    bottom radii = 0.61 2.0
    media = c552 water
:stop layer:

### to end
:start layer:
    thickness = 2.0
    bottom radii = 0.61 2.0
    media = c552 water
:stop layer:

:stop geometry:

```

6.4.3 Join the chamber body to the chamber tip

Create a set of **3 planes** perpendicular to the chamber axis, so as to define two regions, numbered 0 and 1. The middle plane should be located at the joining point $x = 0.935$ cm. The other two planes should be located beyond the chamber on either side.

```

### Planes for cd geometry base
:start geometry:
    name = cd_planes
    library = egs_planes
    type = EGS_Xplanes
    # These 3 plane positions define 2 regions for us to use (0 and 1)

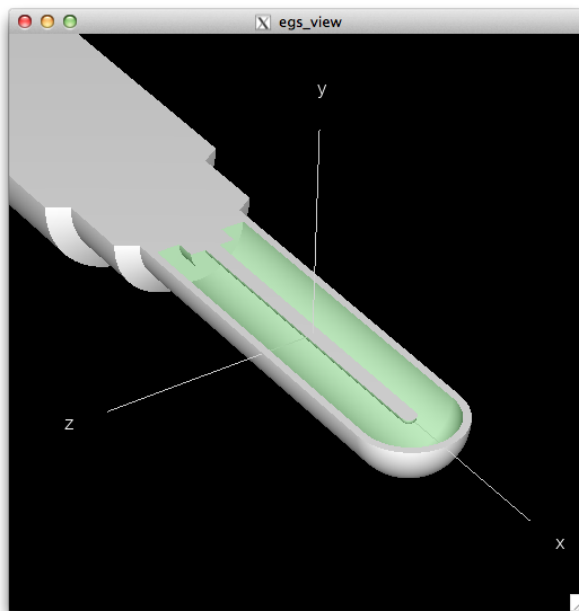
```

```
# the outer planes (-4 and 4) could be anything large
positions = -4 0.935 4
:stop geometry:
```

Finally, use a **cd geometry** to combine the chamber body and the chamber tip, using the planes you just defined as the base geometry. Put the body in region 0, and the tip spheres in region 1. Verify your geometry visually in [egs_view](#). The cd geometry package is very useful for combining and cutting geometries. Check out the relevant documentation for more information.

```
:start geometry:
  name = chamber
  library = egs_cdgeometry
  base geometry = cd_planes
  set geometry = 0 chamber_body
  set geometry = 1 chamber_tip
:stop geometry:
```

It is important to avoid having surfaces perfectly abutting (touching) in egs++, so start your conestack at a location $x > 0.935$ cm to avoid touching the conestack surface to the egs_planes geometry. To do this, make the top layer slightly thicker (say 0.05 cm thicker, making it $1.045 + .935 + .05 = 2.03$ cm), and position the conestack at $0.935 + 0.05 = 0.985$ cm to effectively bleed the conestack into the other geometry. The cd geometry will cut this extra bit off at the plane we define, giving priority to the chamber tip.



6.4.4 Questions

- 1 What are the region numbers that correspond to the air cavity? *Hint: Use a clipping plane (0,1,0,0) in [egs_view](#).*

The air cavity is comprised of **regions 1, 5 and 29**.

- 2 Why is the region number for the spherical tip of the air cavity so large?

The spherical tip of the air cavity is region 29, because the conestack is added first in the `cd` geometry. Therefore, region numbers are attributed first to the conestack (regions 0 to 27, for 7 layers \times 4 regions per layer), and only then to the spherical regions in the chamber tip.

- 3 How many regions are there in total in this model? How many *real* regions?

There are **32 regions** in this chamber model, ranging from 0 to 31. However, there are many *virtual* regions in the chamber stem, where there are only two radii per conestack layer. There are 10 virtual regions, hence there are only **22 real regions** in the geometry.

- 4 What is the largest region number in this geometry?

The largest region number in this geometry is **31** (the chamber tip shell). Keep in mind that region numbering starts from 0, so if a geometry contains n regions, the largest region number is $n - 1$.

6.4.5 Add a region label for the cavity

You have already determined the region numbers for the cavity, but what happens if you add more components to the geometry? The *global* region number might change! For example, if we inscribe the chamber in a box of water and look at the region numbers of the cavity again, they will be different. If you need to know the cavity region numbers for scoring quantities (we will), this can be quite inconvenient. Instead, use region labels. This assigns a series of *local* region numbers within a given geometry to a label that you can reference elsewhere.

Local region numbers are what you find when viewing a particular geometry (e.g. our geometry by the name of `chamber`). To find the local region numbers just set that geometry as the calculation geometry, and open `egs_view`. In contrast, the *global* region numbers are the numbers that get assigned to those same regions when the geometry gets included in a more complex *composite* geometry. For example, the composite geometry of a chamber inscribed in water.

To add a region label, place the following *within a geometry block*. In this example we give the label the name `chamber_cavity_label` (make sure it's unique!) and assign the local regions 1, 5 and 29 (the air cavity regions). Now if we want to reference those regions elsewhere in the input file, we can reference the *label* instead of the global region numbers (which may change if other geometries in the input file are modified).

```
set label = chamber_cavity_label 1 5 29
```

Add region labels both for the air cavity regions, and for the entire chamber. The latter of these will be used for a variance reduction technique.

```
### Join tip and body, attached at the cd_planes
:start geometry:
  name = chamber
  library = egs_cdgeometry
  base geometry = cd_planes
  set geometry = 0 chamber_body
  set geometry = 1 chamber_tip

  # The label for the cavity regions
  # View the chamber geometry alone to find these numbers
  set label = chamber_cavity_label 1 5 29
  set label = chamber_xcse 0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 \\\
    18 19 20 21 22 23 24 25 26 27 28 29 30 31
:stop geometry:
```

6.4.6 Inscribe the chamber in air

Now that the chamber has the sensitive regions labelled, inscribe it in a box of water, then in a larger box of air. There are many ways to do this! Name the new composite geometry `chamber_in_water_in_air`, and set it to be the new simulation geometry (there can only be one).

View the geometry in `egs_view` again, setting air and water transparent and using a clipping plane. Notice how the region numbers of the sensitive volume of the chamber have changed - this is the motivation for region labels.

```
#-----
# A water and air box surrounding the chamber
#-----

# Air outside whole structure
:start geometry:
  name = outer_air_box
  library = egs_box
  box size = 15 15 210
  :start media input:
    media = air
  :stop media input:
:stop geometry:

# Water outside the chamber
```

```

:start geometry:
    name = water_box
    library = egs_box
    box size = 10
    :start media input:
        media = water
    :stop media input:
:stop geometry:

# Inscribe the chamber in the water
:start geometry:
    name = chamber_in_water
    library = egs_genvelope
    base geometry = water_box
    inscribed geometries = chamber
:stop geometry:

# Inscribe the water and chamber in the air
:start geometry:
    name = chamber_in_water_in_air
    library = egs_genvelope
    base geometry = outer_air_box
    inscribed geometries = chamber_in_water
:stop geometry:

simulation geometry = chamber_in_water_in_air

```

6.4.7 Add media definitions

Add media definitions to the input file (pegsless mode). Assign the same names that we used to media in the geometry (air, c552, water), and set the density correction files appropriately. The full set of density correction files distributed with EGSnrc can be found in [\\$HEN_HOUSE/pegs4/density_corrections/](#). If the density correction file you are using resides in that directory, or in [\\$EGS_HOME/pegs4/density_corrections/](#), then the ‘.density’ extension can be omitted. To use a density correction file located elsewhere, type the full path and file extension.

The big benefit of this style of media definition is the ability to set [AE](#) and [AP](#) in the input file. Previously it was necessary to generate a new pegs4 data file each time these parameters were changed (the 521 in [521icru](#) refers to the [AE](#)).

```

#####
### Media definition
#####
:start media definition:

    ### Cutoff energies

```

```

ae = 0.521 # lowest energy for electrons (kinetic+0.511)
ap = 0.01 # lowest energy for photons (kinetic)
ue = 50.511 # maximum energy for electrons (kinetic+0.511)
up = 50 # maximum energy for photons (kinetic)

# I chose the name 'air', use something easy to type
:start air:
    # .density extension omitted for density correction files located in
    # $HEN_HOUSE/pegs4/density_corrections or $EGS_HOME/pegs4/
    ↪ density_corrections
    density correction file = air_dry_nearsealevel
:stop air:

:start c552:
    density correction file = c-552air-equivalentplastic
:stop c552:

:start water:
    density correction file = water_liquid
:stop water:

:stop media definition:

```

6.4.8 Add a particle source

Add a simple collimated 10 MeV photon source. The source shape should be a point, positioned at -100 cm on the z-axis. Set the target shape to be a rectangle at the origin, $10 \times 10 \text{ cm}^2$.

To model a linear accelerator instead, you will need to replace this with a BEAMnrc shared library source, or a phase-space source.

```

#####
### Source definition
#####
:start source definition:

    :start source:
        name = photons
        library = egs_collimated_source
        charge = 0
        :start source shape:
            type = point
            position = 0 0 -100
        :stop source shape:
        :start target shape:
            library = egs_rectangle
            rectangle = -5 -5 5 5

```

```

:stop target shape:
:start spectrum:
    type = monoenergetic
    energy = 10
:stop spectrum:
:stop source:

simulation source = photons

:stop source definition:

```

6.4.9 Add transport parameters and run control

For ionization chamber simulation, it is OK to use most of the EGSnrc default settings. There are two parameters that you should always set manually: the total energy cut-offs for electrons and photons, [ECUT](#) and [PCUT](#), respectively.

Why are these parameters so important? In short, they constitute a stage in the simulation where approximations may occur. Particles whose total energy fall below the cut-off are absorbed in the current region, depositing all energy and no longer being transported. This means that secondary particles will no longer be produced, so even if the energy of the particle was so low that it would not leave the current region, there is a loss of secondary production. For this reason, it is essential to set [ECUT](#) and [PCUT](#) sufficiently low. However, smaller cut-off values means longer simulation times, and in some cases result in negligible changes to quantities of interest.

For now, use [ECUT=0.521](#) and [PCUT=0.01](#).

```

:start MC transport parameter:
    Global Ecut = 0.521
    Global Pcut = 0.01
:stop MC transport parameter:

:start run control:
    ncase = 1e5 # simulate 10^5 source particles
:stop run control:

```

6.4.10 Add egs_chamber scoring options

The scoring options in this section can be used only in input files for the [egs_chamber](#) application.

Calculate the cavity mass (in grams) by determining the volume of each air cavity region, then multiplying by the density of air. To be consistent with the simulation, use the density that is the 3rd value in the air density correction file.

To use the variance reduction technique photon cross-section enhancement, set the `enhance regions` to all of the chamber regions, and the value of `enhancement` to `-128`. The negative value of `enhancement` sets the same value to all of the regions. This will cause photons to interact with 128 times higher probability within the chamber, resulting in more electron production and energy deposition in the cavity region. This is a true variance reduction technique and does not degrade accuracy in any way.

```
#####
### Scoring options
#####
:start scoring options:
  :start calculation geometry:
    geometry name = chamber_in_water_in_air # The simulation geometry
    cavity regions = chamber_cavity_label # Air cavity regions
    cavity mass = 0.00080152267 # g, used density = 1.20478997E-03 g/cm^3
    cavity geometry = chamber # For russian roulette
    enhance regions = chamber_xcse # All the chamber regions
    enhancement = -128 # Use negative to set the same for all regions
  :stop calculation geometry:
:stop scoring options:
```

6.4.11 Add variance reduction

Turn on photon cross section enhancement using the variance reduction input block. These variance reduction parameters are valid *only for egs_chamber*, and are designed specifically for high efficiency in ionization chamber simulations.

Since a cross-section enhancement of 128 was set in the scoring options, turn on Russian Roulette using the `rejection` input, setting it to 128. Then 1/128 will be the probability for survival in a Russian Roulette game played with electrons that cannot reach the cavity.

We will not be using electron range rejection, so set `Esave` to the same value as `ECUT`.

```
#####
### Variance reduction
#####
:start variance reduction:

  # Turn on photon cross-section enhancement (xcse)
  cs enhancement = 1

  # Turn on Russian Roulette for electrons that cannot reach the cavity
  :start range rejection:
    rejection = 128 # 1/128 is the Russian Roulette survival probability
    Esave = 0.521 # The energy below which electrons that are in the
                  # cavity and cannot escape the current region
                  # are immediate discarded and energy deposited
```

```

                                # 0.521 (same as ecut) implies no range rejection
cavity geometry = chamber
rejection range medium = air # Not using range rejection so this
                                # doesn't matter

:stop range rejection:

:stop variance reduction:

```

6.4.12 Add ausgab (output) options

Of course the most useful part of a simulation is the output! This is done using [ausgab](#) objects. Include one for track scoring and one for dose scoring.

Important note: currently, track scoring is incorrect when used with variance reduction techniques turned on. The simulation results are unchanged, but the tracks paths are incorrect. In order to view tracks and avoid this problem, comment out the variance reduction input block before running a very short simulation just for track generation. Comment out the tracks ausgab object for long simulations.

To list the dose in each of the cavity regions, use the label that was defined as [chamber_cavity_label](#). Then, for the regions in order of increasing number, calculate and provide the [volume](#) for each region. This is used to calculate dose from the energy deposited in each region. If you are only interested in the energy deposited, you can neglect this input.

In the example below, notice that particle track scoring is commented out. It is possible to comment out an entire input block by just commenting out the [:start](#) line.

```

#####
### Ausgab
#####
:start ausgab object definition:
  #:start ausgab object:
    name = tracks
    library = egs_track_scoring
    stop scoring = 1000
  :stop ausgab object:
  :start ausgab object:
    library = egs_dose_scoring
    name = doseScoring
    medium dose = yes
    # List of individual regions
    dose regions = chamber_cavity_label
    # List of each region volume (in same order), cm^3
    volume = 0.02880 0.57732 0.05916
  :stop ausgab object:
:stop ausgab object definition:

```

6.4.13 Run the simulation

Run the simulation! Make sure that your input file resides in the `$EGS_HOME/egs_chamber` directory, and has the `.egsinp` extension. It is only necessary to compile `egs_chamber` if you have not done so previously. On linux systems, run `egs_chamber` using the following command (replacing the input/output file names accordingly). Alternatively, use `egs_gui`.

```
$ egs_chamber -i yourInput.egsinp | tee yourOutput.txt
```

Once the simulation is complete, the output near the end should look something like the following (the example simulated 10^6 histories).

```
last case = 1000000 fluence = 1e+06

Geometry Cavity dose
-----
chamber_in_water_in_air 2.1911e-15 +/- 1.081 %

=====
Dose Scoring Object(doseScoring)
=====
=> last case = 1000000 fluence = 1e+06

==> Summary of region dosimetry (per particle)
ir medium rho/[g/cm3] V/cm3 Edep/[MeV] D/[Gy]
-----
 3 air 0.001 0.5773 9.4809e-09 +/- 1.125 % 2.1836e-15 +/- 1.125 %
 7 air 0.001 0.0288 4.7904e-10 +/- 2.005 % 2.2117e-15 +/- 2.005 %
31 air 0.001 0.0592 1.0028e-09 +/- 1.921 % 2.2538e-15 +/- 1.921 %
-----

==> Summary of media dosimetry (per particle)
medium Edep/[MeV] D/[Gy]
-----
c552 6.5399e-05 +/- 3.936 % 9.3737e-16 +/- 3.936 %
air 3.1250e-05 +/- 1.086 % 3.3441e-12 +/- 1.086 %
water 1.1111e-02 +/- 0.477 % 3.4258e-13 +/- 0.477 %
-----

=====
```

6.4.14 Questions

- 1 What is the total dose deposited to the ionization chamber cavity? Why is this different from the total dose to air?

The total dose to the air cavity is the sum of the dose to regions 3, 7 and 31: $6.6491 \times 10^{-15} \pm 1.002\%$.

This is smaller than the total dose to air under *Summary of media dosimetry*, which includes the dose to air regions outside the chamber.

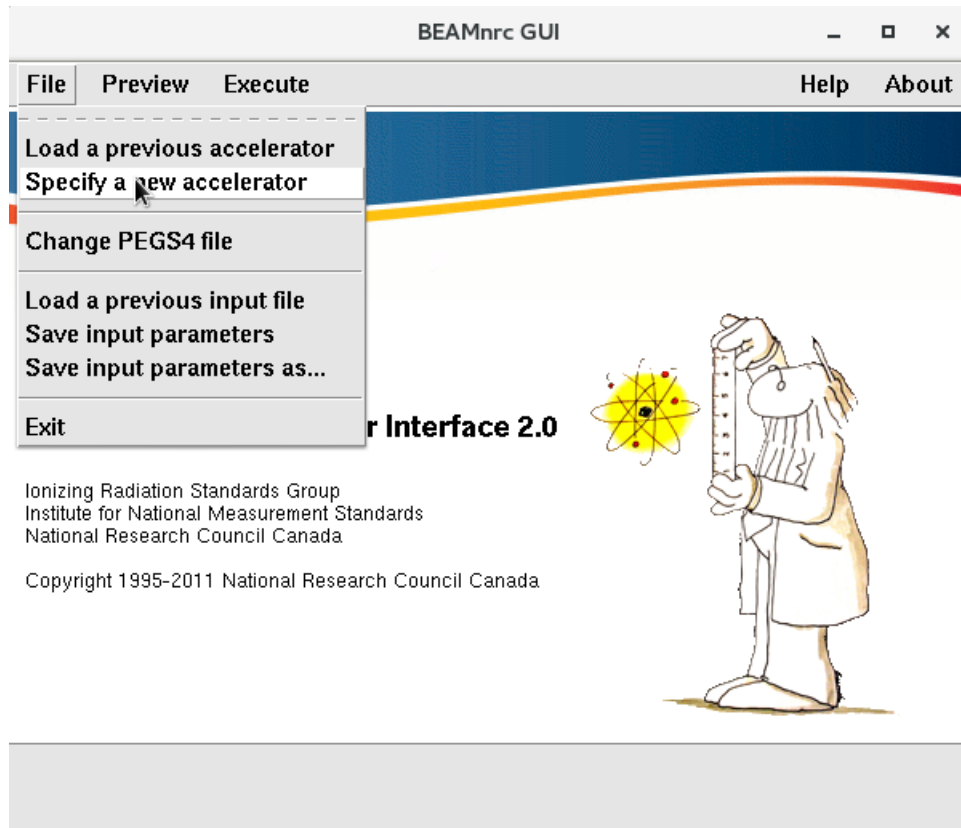


Figure 4: BEAMnrc: Creating a new accelerator

7 Linear accelerator simulations

The BEAMnrc application has long been the workhorse for linear accelerator (LINAC) simulations using EGSnrc. It has been used by many clinical groups to build radiotherapy modelling systems. In such situations BEAMnrc is typically coupled with a second application to perform voxelized transport (e.g. patient modelling), like DOSXYZnrc, vmc++ or any egs++ application.

BEAMnrc is distributed with a powerful GUI that enables the user to easily create new accelerators, compile them, design input files and run simulations. It requires the tcl/tk packages to be installed, as mentioned in the installation instructions. To launch the GUI, either find the shortcut on your desktop or in a linux terminal type:

```
$ beamnrc_gui
```

7.1 Build a new BEAMnrc accelerator

In the BEAMnrc GUI, click **File -> Specify a new accelerator** (figure 4). In the window that pops up (figure 5), select **SYNCHRO** and click **>>**. Type in any name for the component module under **CM Identifier**, and click **Add** then **Save & close**.

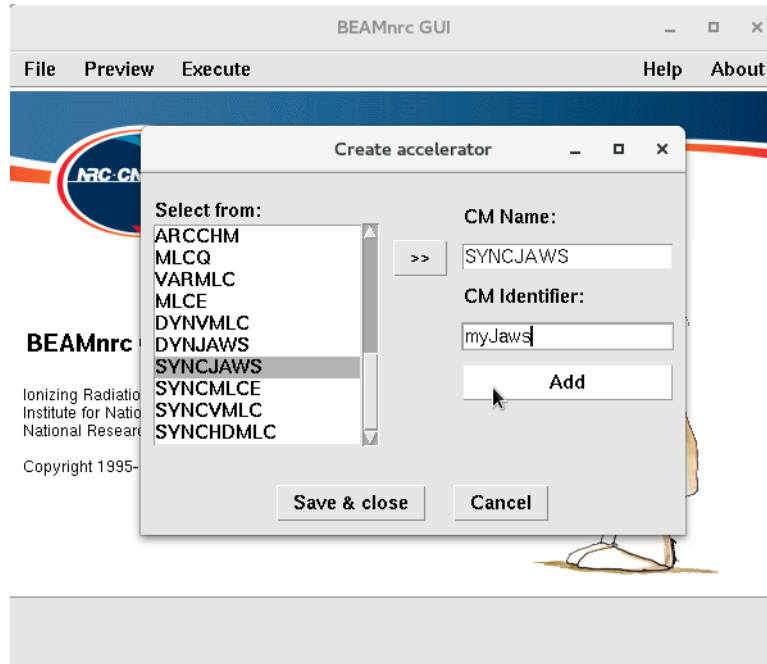


Figure 5: BEAMnrc: Selecting components

Save the accelerator module file as `syncjaws.module` - this will provide the name of the accelerator. The next window (figure 6) allows you to select the material data to use. Click [Browse HEN_HOUSE](#) and select `700icru.pegs4dat`.

Compile the accelerator! In the main BEAMnrc GUI window, go to [Execute -> Compile](#) and click [Build & Compile](#), as in figure 7.

Now you are ready to start editing a new input file. To do this, click on [Edit main input parameters](#) in the [Selected components](#) window. Set a title, and change the parameters to match figure 8. When selecting source 1, set the parameters as in figure 9. For scoring options, set a plane to be scored after the jaws (component 1) as in figure 10.

Close the main input parameters, and instead click the [Edit...](#) button next to [SYNCJAWS](#) to edit the input parameters for the jaws. Enter the parameters suggested in figure 11, clicking the [Define jaw orientation/media](#) button to fill out the jaw settings. Set the [ECUT](#) and [PCUT](#) of the jaw material to 10 MeV. This is an approximation, but will greatly speed up the simulation for demonstration purposes.

For now, leave [File containing jaw opening data:](#) empty. We are setting up the component module (CM) for dynamic mode - motion of the jaws that can be synchronized with other components. Now would be a good time to save our progress ([File -> Save input parameters as...](#)) giving the file a name like `example.egsinp`.

Create a new text file, that will provide the dynamic motion of the jaws. There are examples for all of the synchronized CMs in `$HEN_HOUSE/omega/beamnrc/CMs/sample_sequences/`. Copy the text at the end of this section and save the file as `example.sequence` in `$EGS_HOME/BEAM_syncjaws`. Now go back to the settings of the SYNCJAWS CM, click the

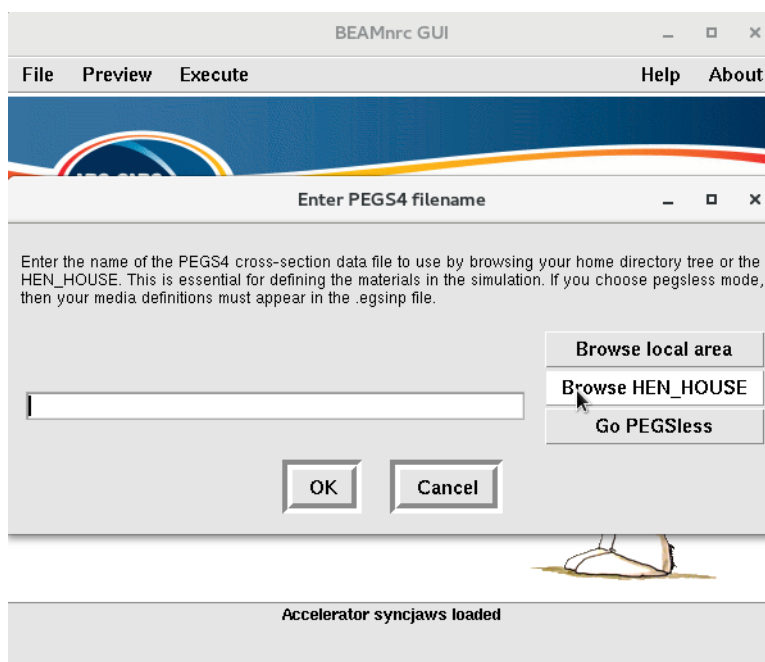


Figure 6: BEAMnrc: Selecting materials



Figure 7: BEAMnrc: Compiling an accelerator

Main Inputs

? Title	A linac with only jaws	? Bremsstrahlung Splitting	none
? Medium	AIR700ICRU	? Brems cross-section enhancement	off
? IWATCH Output	none	? Split electrons or photons at CM	none
? RNG Seed Options	store RNG at start of each batch	? Incident particle	photon
? Run option	first time	? Source number	1 - Point source on axis incident from front
? Output Options	phase space at each scoring plane	? Global electron cutoff energy - ECUT (MeV)	0.7
? Store Data Arrays	no	? Global photon cutoff energy - PCUT (MeV)	0.01
? LATCH option	inherited latch - set by passage	? Electron range rejection	off
? Score Last Z	no	? Global electron cutoff (ESAVE_GLOBAL, range rejection, MeV)	
? Number of histories	100000	? Photon forcing	off
? Initial RNG seed 1	33	? Number of scoring planes	1
? Initial RNG seed 2	97	? Dose calculation	Only total dose
? Maximum CPU hours allowed	0.99	? Z of front of 1st CM to reference plane (cm)	39.0

Define Media Edit EGSnrc Parameters Close

Figure 8: BEAMnrc: Main input parameters

Set source options

Source 1 - Point source on axis incident from front

Distance of point source to front of first CM (cm) 40

Rectangular or circular beam?

☒ rectangular

☐ circular

Beam radius at front of first CM (cm) 10

1/2 angle about Z-axis of source (degrees) (only applies if beam radius is 0)

Min. X of beam (cm)

Max. X of beam (cm)

Min. Y of beam (cm)

Max. Y of beam (cm)

Specify source beam energy or spectrum filename

☒ monoenergetic

Kinetic energy of beam (MeV) 10

☒ spectrum

Spectrum filename (complete):

Browse current

Browse generic

Output spectrum listing file?

OK Help

Figure 9: BEAMnrc: Source 1 options

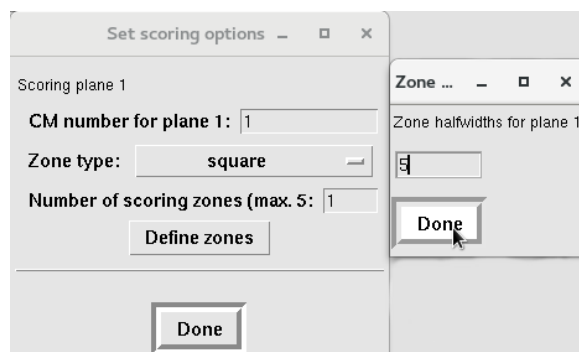


Figure 10: BEAMnrc: Scoring options

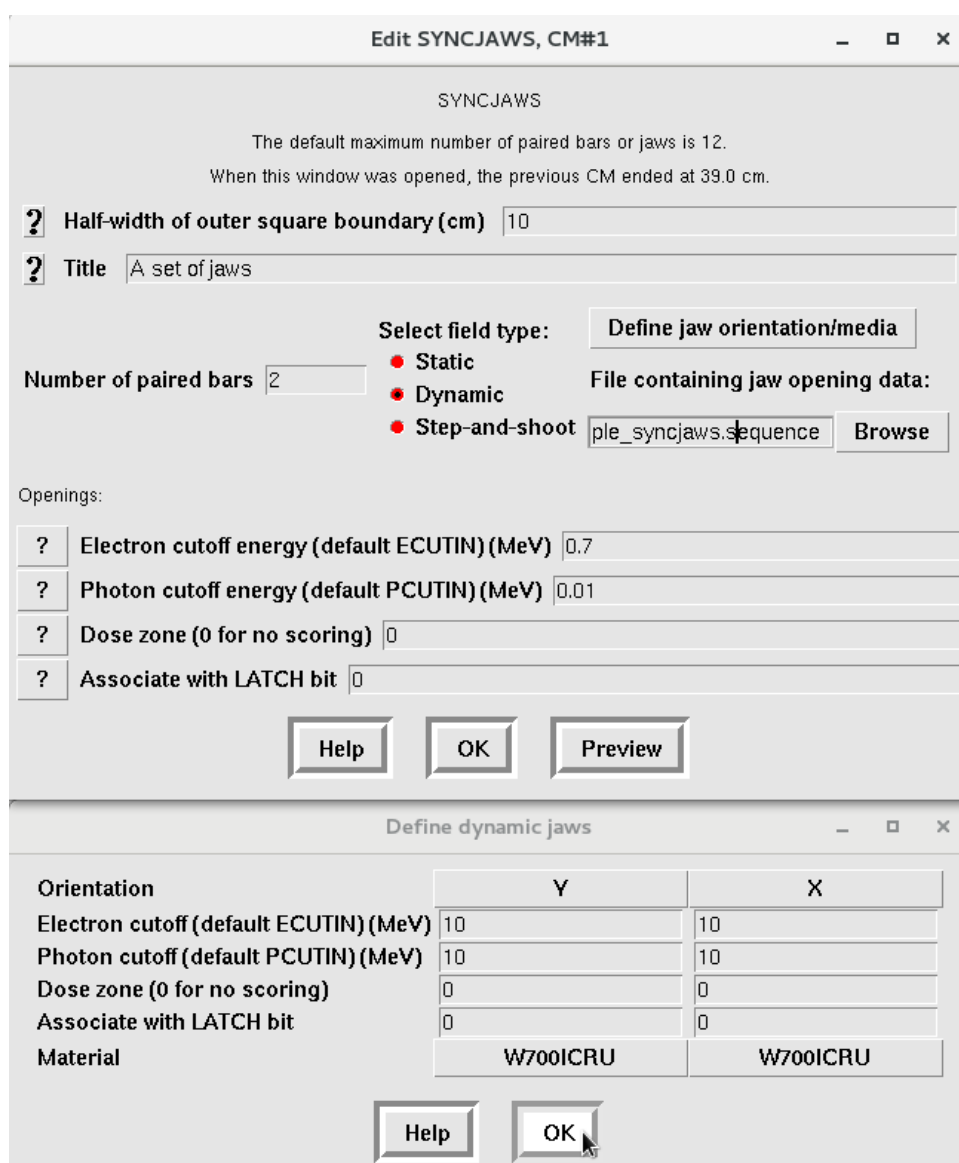


Figure 11: BEAMnrc: Syncjaws definition

[Browse](#) button and select the file (make sure the mode is set to [Dynamic](#)). Click [Preview](#) to view the first position of the jaws, before motion begins.

The format of the sequence file is described in section 15.3.8 of the BEAMnrc Users Manual (PIRS509a). Using this file, it is possible to model motion like jaw tracking, based on the *index* parameter (i.e. fractional MU or cumulative meterset weight). If multiple SYNC CMs are included, the index is used to synchronize motion. Use a repeated index to simulate motion with the beam off.

In the following, we define 2 static fields and 1 dynamic field in 6 steps. The upper y-jaws are at $40 \leq z \leq 50$ and x-jaws at $51 \leq z \leq 61$. From index 0.0 to 3.0 (30% of the simulation), the jaws are statically positioned. Similarly from 0.3 to 0.6. The repeated indices, 0.3 and 0.6, result in simulated collimator shifts while the beam is off. Finally, from index 0.6 to 1.0 (40%) there is a motion of the x-jaws from a nearly-closed position to a 2×2 opening.

```
Ex: 2 static fields, 1 dynamic, for 2 jaws
6
0.0
40, 50, -1, -1, -2, -2
51, 61, -1, -1, -2, -2
0.3
40, 50, -1, -1, -2, -2
51, 61, -1, -1, -2, -2
0.3
40, 50, 2, 2, 1, 1
51, 61, 2, 2, 1, 1
0.6
40, 50, 2, 2, 1, 1
51, 61, 2, 2, 1, 1
0.6
40, 50, 1, 1, -1, -1
51, 61, 0.05, 0.05, -0.05, -0.05
1.0
40, 50, 1, 1, -1, -1
51, 61, 1, 1, -1, -1
```

7.2 Run a BEAMnrc simulation

In the BEAMnrc GUI, go to [Execute -> Run](#) and click the [EXECUTE](#) button. Since we only set 100 histories to simulate, this will complete very quickly. Check the accelerator directory to find the phase-space that was saved as output ([\\$EGS_HOME/BEAM_syncjaws/example.egspsphsp1](#)). The 1 in [*.egspsphsp1](#) refers to the scoring plane number of the phase-space (we scored after CM 1, but if you had more CMs you could score multiple phase-spaces in the same simulation). Additional output is recorded in the [*.egslst](#) file.

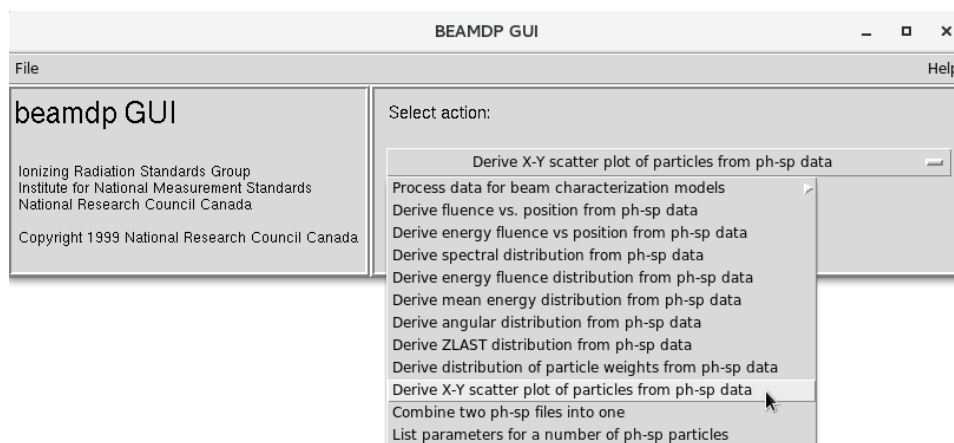


Figure 12: BEAMDP: Mode selection

7.3 Analyze the phase-space with beamdp

Let's do a sanity check on the output of the BEAMnrc simulation by analyzing the phase-space using beamdp. Open the beamdp GUI. On linux this should be aliased:

```
$ beamdp_gui
```

Perform a scatter plot of the phase-space to observe how the jaws have collimated the source (figure 12). Suggested settings are shown in figure 13, and the results in figure 14. As expected, there are two off-axis fields, and one in the centre. Since the phase-space is scored immediately below the second set of jaws, it is easy to confirm that the field openings appear to match the collimator openings that were specified.

Parameters for option 9

Derive X-Y scatter plot of particles from ph-sp data

Particle type photons

☒ Rectangular region anywhere on the scoring plane
 OR
 ☐ Annular region centred at the z-axis

Xmin for the rectangular region (default -15) -15
 Rmin for the annular region (default 0) -15

Xmax for the rectangular region (default 15) 15
 Rmax for the annular region (default 15) 15

Ymin for the rectangular region (default -15) -15

Ymax for the rectangular region (default 15) 15

? Minimum energy (MeV) 0

? Maximum energy (MeV) 10

? LATCH option None

Name of phase space file: example.egsphsp1 Browse

Name of file to save data for graph plotting: example.egsplot Browse

? Maximum number of particles to plot: 10000

Execute beamdp
Close

Figure 13: BEAMDP: Plotting options

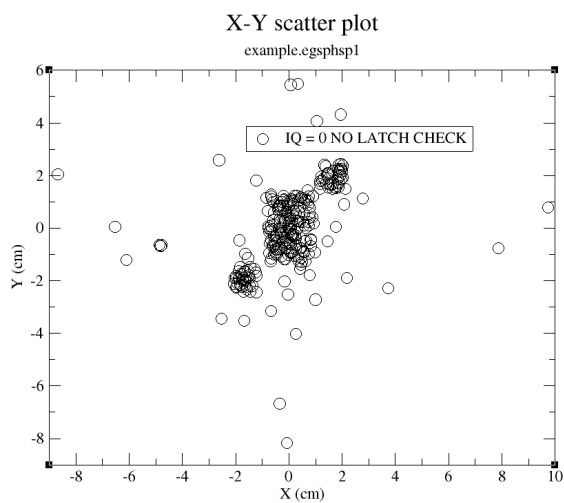


Figure 14: BEAMDP: Example scatter plot

7.4 Convert CT DICOM data to egspphant

In order to perform a MC simulation on a patient phantom, measured images need to be converted to the [egspphant](#) format. This is a rectilinear voxel format in a plain text file. There is a tool distributed with EGSnrc called [ctcreate](#) that can be used to convert CT DICOM files to the [egspphant](#) format.

A set of example CT DICOM files is provided in the [EGSnrc-sample-data.zip](#) file on the release page. Extract the zip file, and navigate to the [CT/DICOM](#) directory.

Edit the [CT_create_DICOM.inp](#) file so that the path to the [slice_files](#) file is correct (on line 2). Then open the [slice_files](#) file, and edit the paths to the ct files similarly. Also adjust the input parameters so that a higher resolution phantom is created. Note that this will use a default CT ramp that should be modified to agree with your imager (see the [ctcreate](#) section in the DOSXYZnrc Users Manual).

```
DICOM
/yourPath/EGSnrc-sample-data/CT/DICOM/slice_files
-150.0, 150.0, -150.0, 150.0, -45.0, 45.0
0.4, 0.4, 0.1
0,0
```

There is no GUI for [ctcreate](#), so it has to be run from the command line (even on windows). From the directory containing the CT files, execute the command:

```
$ ctcreate CT_create_DICOM.inp
```

This should create a new file in the directory, named [slice_names.egspphant](#). Open this in a text editor to observe CT slices represented as 2D blocks of numbers. Each number corresponds to the material index (in the order they are listed at the top of the file). Following these blocks of integers are similar 2D blocks of floating point values - the density of each voxel. This format is described in the DOSXYZnrc Users Manual.

Move the [egspphant](#) file into the [\\$EGS_HOME/dosxyznrc](#) directory for later use.

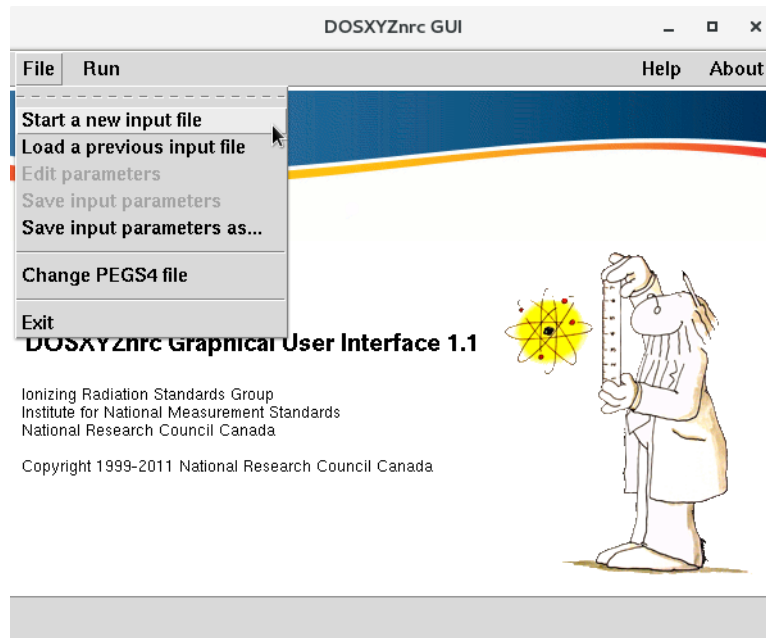


Figure 15: DOSXYZnrc: Start a new input file

7.5 Calculate dose with DOSXYZnrc

The next task in the radiotherapy simulation process is to perform the dose calculation. To do this, create an input file for DOSXYZnrc and direct it to use a BEAMnrc source model and the `egsphant` file that was just created.

In order to use a BEAMnrc accelerator as a dynamic shared library source in DOSXYZnrc, the accelerator must be compiled as a shared library. This is an extra compilation step that is not performed by default. This must be done via command line (there is not an option to compile libraries in `egs_gui`). Navigate to the accelerator directory on the command line and type:

```
$ make library
```

Open the DOSXYZnrc GUI, and start a new input file (figure 15). Make sure to select the same material data file that was used in BEAMnrc (i.e. `700icru` from the `HEN_HOUSE`).

```
$ dosxyznrc_gui
```

Set the input parameters as shown in figure 16. Select source 21, which can be a dynamically synchronized source of particles from BEAMnrc. As long as the accelerator is compiled as a shared library, each particle that DOSXYZnrc needs will be requested from BEAMnrc. The accelerator model then simulates source particles until a particle reaches the scoring plane, at which point the DOSXYZnrc simulation can continue. Each particle is assigned a monitor unit value, to allow for the synchronization of time-dependent components. Notice that large values of `ECUT` and `PCUT` were chosen - this is just to speed up the simulation

The screenshot shows the 'Inputs' dialog box for DOSXYZnrc. It is organized into several sections:

- Title:** A text field containing 'Example dose calculation'.
- Phantom definition:**
 - A note: 'If you are using source 2 or 4, you must define the materials in the phantom here first before defining the source.'
 - 'Define phantom using ...': Radio buttons for 'non-CT data input' (unselected) and 'phantom created from CT data' (selected).
 - 'Global electron cutoff energy - ECUT (MeV)': Text field with '5'.
 - 'Global photon cutoff energy - PCUT (MeV)': Text field with '0.1'.
 - 'Print summary of highest 20 doses': Dropdown menu with 'no' selected.
- Source parameters:**
 - 'Incident particle': Dropdown menu with 'photon' selected.
 - 'Source type': Dropdown menu with '21 - Dynamic BEAM simulation source with multiple variable geometry settings' selected.
- Simulation parameters:**
 - 'Number of histories': Text field with '1000000'.
 - 'IWATCH Output': Dropdown menu with 'none'.
 - 'Maximum CPU time (hours)': Text field with '500.0'.
 - 'RNG seed 1': Text field with '33'.
 - 'RNG seed 2': Text field with '97'.
 - 'Incident beam size (source 2, 4 or 8)': Text field with '100.0'.
 - 'Run option': Dropdown menu with 'first time'.
 - 'HOWFARLESS': Dropdown menu with 'off'.
 - 'Thickness of region outside phantom is': Dropdown menu with 'uniform'.
 - 'Medium of region outside phantom': Dropdown menu with 'VACUUM'.
 - 'Phase space output on exiting phantom': Dropdown menu with 'none'.
 - 'Output restart data': Dropdown menu with 'after every batch'.
 - 'Range rejection': Dropdown menu with 'on'.
 - 'ESAVE: range rejection done only below this energy (MeV)': Text field with '2'.
 - 'Photon splitting number': Text field with '1'.
 - '# times to recycle each particle in phase space source': Text field with '0'.
 - 'Run job in parallel': Dropdown menu with 'no'.

At the bottom, there are three buttons: 'Define Media', 'Edit EGSnrc Parameters', and 'Close'.

Figure 16: DOSXYZnrc: Set the main input parameters

while we test parameters! If you would like an accurate simulation, reduce them to [ECUT=0.7](#) and [PCUT=0.01](#).

For the synchronized source parameters, set two static source positions (later, play around with motion). From index 0.0 to 0.6, set the source to be incident along the negative z-axis. From 0.6 to 1.0, set the source to be incident along the positive z-axis.

Run the simulation! The output file of interest is the [example.3ddose](#) file - this contains a 3D map of dose on the same grid as the [*.egsphant](#) file that we created from the CT data. To view the dose distribution, use the [dosxyz_show](#) tool.

```
$ cd $EGS_HOME/dosxyznrc
$ dosxyz_show slice_files.egsphant example.3ddose
```

Set source options

Source 21 - Dynamic BEAM simulation source with multiple variable geometry settings

define settings

☒ BEAM simulation input (with no LATCH filter)
☐ BEAM simulation + dose component (with LATCH filter)

? LATCH Bit filter

☐ Exclude fat photons from DBS ?

? no. of times to split e+/e- 0

? ☐ Include fractional MU in any phase space output

BEAM simulation: BEAM_syncjaws Browse EGS_HOME/bin/test

input file (no ext.): example Browse EGS_HOME/BEAM_syncjaws

pegs data (no ext.): 700icru Browse HEN_HOUSE pegs4 area
Browse users pegs4 area

☐ using external MLC library

MLC library: 0 Browse EGS_HOME/bin/test

full path to MLC input file: 0 Browse EGS_HOME

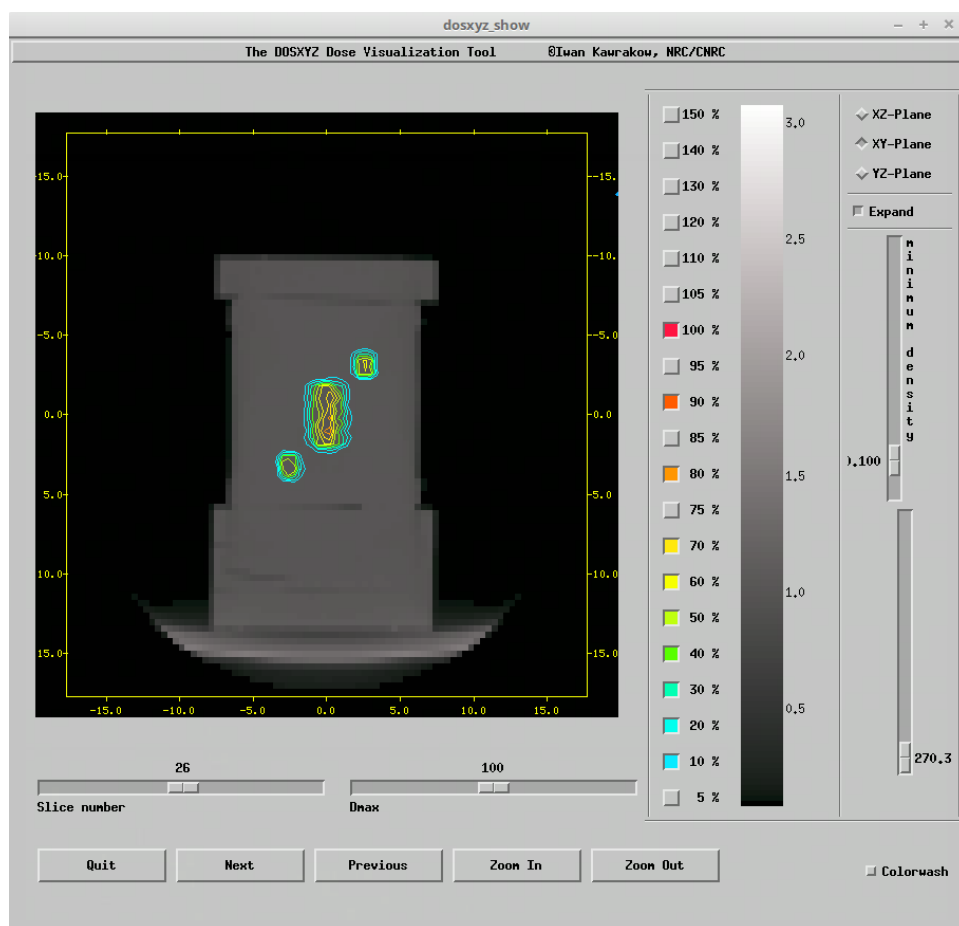
OK Help

Define settings (1)

	xiso (cm)	yiso (cm)	ziso (cm)	theta (degrees)	phi (degrees)	phicol (degrees)	dsource (cm)	MU Index (0-1)
Setting 1	0	0	0	0	0	0	39	0.0
Setting 2	0	0	0	0	0	0	39	0.6
Setting 3	0	0	0	180	0	0	39	0.6
Setting 4	0	0	0	180	0	0	39	1.0

Add setting Delete last setting OK

Figure 17: DOSXYZnrc: Set the synchronized source input parameters

Figure 18: `dosxyz_show`: View the dose distribution

7.6 Extract dose profiles from 3ddose

It is commonly of interest to obtain profile and depth-dose curves from dose files. For the 3ddose format, there is a tool distributed with EGSnrc that can do this called `statdose` (though many users design their own analysis software).

First, it may be necessary to adjust the `statdose` maximum array dimensions. Navigate to `$HEN_HOUSE/omega/progs/statdose` and open `statdose.mortran`. Change the `$MAXVOX` values to 300. The `$MAXVOXEL` parameter should be set to the maximum value of `$MAXVOXX`, `$MAXVOXY` and `$MAXVOXZ`. Then save the file and recompile `statdose` (`make clean; make`).

```
REPLACE {$MAXVOXX} WITH {200};
REPLACE {$MAXVOXY} WITH {200};
REPLACE {$MAXVOXZ} WITH {200};
REPLACE {$MAXVOXEL} WITH {200};
```

The `statdose` code does not have a GUI, so must be run from the command line. Navigate to `$EGS_HOME/dosxyznrc`, and type `statdose` to launch the program. This software uses text input - an example of both the output and input is shown below. Start with **Selection: 1** to read in dose distributions. In our directory we had 9 `*.3ddose` files listed, but you will probably just have one to choose. The number to the left of **example** is the file number to select (7 for us). Assign this file an internal reference number (use 1).

```
*****
      STATDOSE.MORTRAN
      Max array dimensions: 200 200 200
      Max # data sets: 15
*****

MAIN MENU
-----
0 - Exit
1 - Read dose distributions
2 - Statistical analysis
3 - Normalization
4 - Rebinning
5 - Plot
6 - Save

Selection: 1

READ DOSE DISTRIBUTIONS
-----
 1 16MVp_h2o_phantom_beamsource_example
 2 16MVp_h2o_phantom_phsp_example
 3 5MeV-2
 4 5MeV
 5 CT_example_electron
 6 edge_rot
 7 example
 8 tg195_test
 9 tg195_validation

Input file number to Read in: (1- 9 or 0-Main Menu): 7
File number for temporary storage: (1-* or 0-Main Menu): 1

Number of voxels in X,Y,Z directions: 88 88 50
Voxel-boundary values in X-direction: -17.66 - 17.66
Voxel-boundary values in Y-direction: -17.66 - 17.66
Voxel-boundary values in Z-direction: -0.70 - 4.30
Reading dose distribution...
```

Reading in the uncertainties on DOSE Distribution...

After this, type 0 to return to the main menu, then select 5 to begin plotting. Plot profiles of 1 file, and select the x-axis to plot 3 curves at different y-positions. If you have xmgrace installed, it will launch automatically at this point to display the profiles. However, the data is also saved to the output file selected (e.g. [example.agr](#)). This is a plain text file, so even if you do not have xmgrace it is easy to load into a spreadsheet for data analysis and plotting.

Input file number to Read in: (1- 9 or 0-Main Menu): 0

MAIN MENU

0 - Exit
1 - Read dose distributions
2 - Statistical analysis
3 - Normalization
4 - Rebinning
5 - Plot
6 - Save

Selection: 5

PLOT MENU

0 - Main menu
1 - Plot profiles
2 - Comparison plot

Selection: 1

Data for dose plot

Files currently loaded

1 - example

Number of file to Plot (0-PlotMenu): 1

Axis for Profile (0-PlotMenu,1-X,2-Y,3-Z): 1

Graph Title (default=Profile for example):

Output Filename (default=example):

Number of curves to Plot: 3

Generate Automatic Offset? (y/n) [n]:

Coordinates of Axis (y,z): 4,3

3D-dose distribution 1: example

Coordinates of Axis (y,z): 0,3

3D-dose distribution 1: example

Coordinates of Axis (y,z): -4,3

3D-dose distribution 1: example

Calling xmgrace...Please be patient!

If you have xmgrace installed, load the [*.agr](#) file later using the command:

```
$ xmgrace example.agr
```

8 Online resources

8.1 EGSnrc on github

- EGSnrc can be downloaded (or “cloned”) from the [github](#) page. Make sure to read the installation instructions carefully!

<https://github.com/nrc-cnrc/EGSnrc>

- The annual releases can be found on the github release page. Additionally, pre-compiled GUIs, manuals and sample data are available:

<https://github.com/nrc-cnrc/EGSnrc/releases>

- See the [wiki](#) page for release notes, technical instructions and links to online documentation:

<https://github.com/nrc-cnrc/EGSnrc/wiki>

8.2 How to get support

If you have a question, or encounter problems while using EGSnrc, seek support on the EGSnrc google+ community! This is the best place to get answers to questions ranging from the very simple to quite complex. There are 800+ members who regularly answer questions, and the EGSnrc developers also contribute frequently.

<https://plus.google.com/communities/106437507294474212197>

To increase your chances of getting an answer, make sure to include as much information as possible in your original post. This should include:

- The EGSnrc application and version (e.g. [egs_chamber](#), EGSnrc 2018 master branch)
- Operating system
- A clear description of the problem
- Copy/pasted output including any errors
- Optionally: compiler version, screenshots, input files and figures

8.3 Bug reports

Have you found a bug? Report it on the github [issues](#) page! Make sure to include sufficient information for us to reproduce the bug.

<https://github.com/nrc-cnrc/EGSnrc/issues>

8.4 Submitting your code

Have you added a feature or fixed a bug? Submit the changes as a [pull request](#)! This requires learning the basics of “git” - if that is not feasible, you can also submit the changes as an [issue](#) and allow one of us to create the pull request.

<https://github.com/nrc-cnrc/EGSnrc/pulls>

8.5 Documentation

Documentation for EGSnrc is available online! However, keep in mind that the documentation applies to the most recent [release](#) of EGSnrc. In other words, the documentation is generated annually, at the same time as the release of the master branch (usually near the start of the year).

The online documentation:

<http://nrc-cnrc.github.io/EGSnrc/>

Each release is posted with the full set of manuals in a zip file. Find this on the release page:

<https://github.com/nrc-cnrc/EGSnrc/releases>

If you are using an older version of EGSnrc, or if you are working on the [develop](#) experimental branch, you may not find the matching documentation on the release page. To obtain the documentation that exactly matches your installation, find the documentation source code within your installation:

[\\$HEN_HOUSE/docs/src](#)

Some additional steps are required to compile the documentation locally.

8.5.1 Compiling documentation locally

1. Install a suite that includes [pdflatex](#), [bibtex](#), [texmf](#) and [makeindex](#) (e.g. the [texlive](#) suite includes all of these)
2. Install [doxygen](#) (for the egs++ documentation)
3. `cd $HEN_HOUSE/docs/src` (linux instructions)
4. `./makedoc.sh all` (linux instructions)

9 EGSnrc file structure

- `$HEN_HOUSE`: Points to location of the EGSnrc system (**user-defined**)
- `$EGS_HOME`: Points to user's working directory, where applications are copied (**user-defined**)

BEWARE: Path to these folders cannot have spaces! EGSnrc relies on GNU `make` which doesn't handle blank spaces in names. Also avoid special characters like `&`.

9.1 The `$HEN_HOUSE`

The `$HEN_HOUSE` directory contains all source code and data files for the EGSnrc system. After installation, the contents of this directory generally remain unchanged.

Table 1: The folder structure and contents of the HEN_HOUSE.

Category	Folder	Description
EGSnrc core	omega	- BEAMnrc source code - addphsp, beamdp, ctcreate, dosxyz_show, readphsp, and statdose - tcl GUIs for beamdp, BEAMnrc and DOSXYZnrc
	previewRZ	- tcl GUI for viewing geometries from DOSRZnrc, CAVRZnrc, FLURZnrc and SPRRZnrc
	src	- EGSnrc core MORTRAN source code
	user_codes	- Application source code (C++ and MORTRAN)
	utils	- Various MORTRAN tools
egs++ and Qt	cutils	- C tools for parallel jobs and BEAMnrc sources
	egs++	- C++ class library - ausgab objects, geometries and sources
	gui	- Qt GUIs: egs_configure, egs_gui and egs_inprz
	iaea_phsp	- C++ tools for IAEA phase-spaces
	interface	- The code that connects C++ and MORTRAN
data	data	- photon cross section data - bremsstrahlung cross section corrections - eii cross sections - molecular form factors
	pegs4	- PEGS4 mortran code - PEGS4 data files - density correction files
	spectra	- various energy spectrum data - radionuclide decay data
configuration	makefiles	- template application makefiles
	mortran3	- the MORTRAN core code
	pieces	- system test code
	scripts	- configuration and support scripts
	specs	- configuration specifications
documentation	docs	- documentation source code

Medium composition, electron stopping powers, and electron interaction cross sections, for media including with the EGSnrc distribution are contained in the [pegs4](#) directory. The density correction files for [pegsless](#) mode are found in [pegs4/density_corrections](#) and the commonly used [521icru](#) and [700icru](#) pegs4 material data files are in [pegs4/data](#).

There are a few example energy spectrum files contained in [spectra/egsnrc](#). Radionuclide decay data in ensdf format for the egs++ radionuclide source are found in [spectra/lnhb](#).

There are some scripts that help with configuration and parallel runs in [scripts](#).

Compiled utility executables are found in [bin](#), such as [ctcreate](#), [beamdp](#), etc.

9.1.1 Configuring for different machines

To compile EGSnrc on linux, there is a command line script that can be used instead of the installation GUI. This script is convenient to create different configurations that can be switched between - for example changing compile flags, or compiling for different machines.

`$HEN_HOUSE/scripts/configure`

The configure script does not create the `$EGS_HOME` directory, unless you opt to run the `finalize_egs_foruser` option at the end. Run the finalize script to create `$EGS_HOME`, copy over application source and compile applications (this would overwrite an existing `$EGS_HOME`). At the end of this script, the suggested environment variables are printed as output to the terminal (to be copy/pasted into your shell environment, such as `~/.bashrc`).

`$HEN_HOUSE/scripts/finalize_egs_foruser`

It is possible to have EGSnrc configured for execution on several servers, but sharing the same installation. This means that the configuration script is run once on each machine, and set to use a different configuration file (the `*.conf` file) each time. To do this, watch for the prompt to set the configuration name and make sure to give it a unique identifier (usually the default is `linux.conf`). On each server, the `$EGS_CONFIG` environment variable must be set uniquely, and should point to the main configuration file created by the installation script (but type it in as an absolute path, don't use environment variables):

`/full_path_to/HEN_HOUSE/specs/your_machine.conf`

On linux systems, the `$HEN_HOUSE` and `$my_machine` environment variables are automatically set using the `$EGS_CONFIG` environment variable that you provide.

9.2 The \$EGS_HOME

The `$EGS_HOME` directory contains all of the applications and data created or provided by the user. Each application (previously called a `user_code`) has its own directory which contains the source code, a Makefile, and the user's input files. Output files from simulations are also written to the application directories.

The user may store their own set of material data in `$EGS_HOME/pegs4`, which will automatically supplement the data that is available in `$HEN_HOUSE/pegs4`.

`BEAMnrc` has a folder structure different from other applications. Each accelerator that is built for `BEAMnrc` is assigned its own directory with the name `BEAM_yourAccel`, where `yourAccel` is the name of the accelerator. In other words, each accelerator is compiled as an independent EGSnrc application. The component module specifications for all accelerators are contained in `$EGS_HOME/beamnrc/spec_modules`.

The compiled executables for applications reside in `$EGS_HOME/bin/$my_machine`.

