

EPOCH

An open source PIC code for high energy density physics



THE UNIVERSITY OF
WARWICK



UNIVERSITY OF WARWICK

Users Manual for the EPOCH PIC codes

Last manual revision by:

Keith BENNETT

EPOCH written by:

Chris BRADY

Keith BENNETT

Holger SCHMITZ

Christopher RIDGERS

Project coordinators:

Tony ARBER

Roger EVANS

Tony BELL

Contents

1	FAQs	4
1.1	Is this manual up to date?	4
1.2	What is EPOCH ?	4
1.2.1	Features of EPOCH	4
1.3	The origins of the code	4
1.4	What normalisations are used in EPOCH ?	4
1.5	What are those <code>num</code> things doing everywhere?	5
1.6	What is an input deck?	5
1.7	I just want to use the code as a black box, or I'm just starting. How do I do that?	5
1.8	What is the auto-loader?	5
1.9	What is a maths parser?	5
1.10	I am an advanced user, but I want to set up the code so that less experienced users can use it. How do I do that?	5
1.11	I want to develop an addition to EPOCH . How do I do that?	6
1.12	I want to have a full understanding of how EPOCH works. How do I do that?	6
1.13	How do I acknowledge use of the code?	6
2	EPOCH for end users	7
2.1	Structure of the EPOCH codes	7
2.2	Libraries and requirements	8
2.3	Compiling and running EPOCH	8
2.4	Compiler flags and preprocessor defines	9
2.5	Running EPOCH and basic control of EPOCH1D	12
3	The EPOCH input deck	12
3.1	control block	13
3.1.1	Maxwell Solvers	18
3.1.2	Strided Current Filtering	19
3.1.3	Dynamic Load Balancing	19
3.1.4	Automatic halting of a simulation	20
3.1.5	Requesting output dumps at run time	20
3.2	stencil block	20
3.3	boundaries block	21
3.3.1	CPML boundary conditions	23
3.3.2	Thermal boundary conditions	23
3.4	species block	24
3.4.1	Particle migration between species	28
3.4.2	Ionisation	28
3.4.3	Species Boundaries	30
3.4.4	Maxwell-Jüttner distribution function	30
3.4.5	Arbitrary distribution functions	31
3.5	laser block	32
3.6	fields block	34
3.7	particles_from_file block	35
3.8	window block	36
3.9	output block	37
3.9.1	Dumpmask	39
3.9.2	Directives	40
3.9.3	Particle Variables	42
3.9.4	Grid Variables	43
3.9.5	Derived Variables	43

3.9.6	Other Variables	44
3.9.7	Data Averaging	45
3.9.8	Single-precision output	45
3.9.9	Multiple output blocks	46
3.10	output_global block	48
3.11	dist_fn block	50
3.12	probe block	51
3.13	collisions block	52
3.14	qed block	53
3.15	bremsstrahlung block	55
3.16	injector block	57
3.16.1	Concepts	57
3.16.2	Boundary Conditions	57
3.16.3	Moving Window	57
3.16.4	Keys	58
3.17	subset block	59
3.17.1	Subsets of fields	60
3.17.2	Persistent subsets	61
3.18	constant block	61
3.19	The maths parser	62
3.19.1	Constants	62
3.19.2	Functions	64
3.19.3	Operators	65
4	EPOCH use in practice	66
4.1	Specifying initial conditions for particles using the input deck	66
4.1.1	Setting autoloader properties from the input deck	66
4.2	Manually overriding particle parameters set by the autoloader	67
4.2.1	EPOCH internal representation of particles	67
4.2.2	Setting the particle properties	71
4.2.3	Grid coordinates used in EPOCH	72
4.2.4	Loading a separable non-thermal particle distribution.	72
4.3	Lasers	73
4.4	laser blocks in multiple dimensions.	73
4.5	Restarting EPOCH from previous output dumps	76
4.6	Parameterising input decks	76
4.7	Using spatially varying functions to further parameterise initial conditions	78
4.8	Delta-f method in EPOCH	80
5	Basic examples of using EPOCH	81
5.1	Electron two stream instability	81
5.2	Structured density profile in EPOCH2D	83
5.3	A hollow cone in 3D	85
5.4	Focussing a Gaussian Beam	86
6	Using IDL to visualise data	90
6.1	Inspecting Data	93
6.2	Getting Help in IDL	93
6.3	Manipulating And Plotting Data	94
6.4	1D Plotting in IDL	94
6.5	Postscript Plots	96
6.6	Contour Plots in IDL	96
6.7	Shaded Surface Plots in IDL	97

6.8	Interactive Plotting	98
7	Using VisIt to visualise data	99
7.1	LLNL VisIt	99
7.2	Obtaining And Installing VisIt	99
7.3	Compiling The Reader Plugin	101
7.4	Loading Data Into VisIt	101
7.5	Contour Plots in VisIt	103
7.6	1D Plotting in VisIt	104
7.7	Shaded Surface Plots in VisIt	105
7.8	Creating User-Defined Expressions	106
7.9	Creating Movies	107
7.10	Remote Visualisation	108
7.11	Parallel Visualisation	110
A	Changes between versions	111
A.1	Changes between version 3.1 and 4.0	111
A.1.1	Changes to the Makefile	111
A.1.2	Major features and new blocks added to the input deck	111
A.1.3	Additional output block parameters	112
A.1.4	Other additions to the input deck	112
A.2	Changes between version 4.0 and 4.3	112
A.2.1	Changes to the Makefile	112
A.2.2	Additions to the input deck	112
A.2.3	Changes in behaviour which are not due to changes in the input deck	114
A.3	Changes between version 4.3 and 4.8	115
A.3.1	Changes to the Makefile	115
A.3.2	Additions to the input deck	115
A.4	Changes between version 4.8 and 4.9	116
A.4.1	New capabilities	116
A.4.2	Changes to the Makefile	116
A.4.3	Additions to the input deck	116
A.4.4	Changes not resulting from changes to the deck	117
A.5	Changes between version 4.9 and 4.10	117
A.6	Changes between version 4.10 and 4.11	117
A.6.1	New capabilities	117
A.6.2	Corrections	118
A.7	Changes between version 4.11 and 4.12	118
A.8	Changes between version 4.12 and 4.14	118
A.9	Changes between version 4.14 and 4.15	119
A.10	Changes between version 4.15 and 4.16	119
A.11	Changes between version 4.16 and 4.17	120
B	Simple binary files	121
C	Slides from Delta-f Talk	122
D	References	144

1 FAQs

1.1 Is this manual up to date?

Whenever a new milestone version of **EPOCH** is finalised, the version number is changed and this manual is updated accordingly. The version number of the manual should match the first two digits for that of the **EPOCH** source code. This version number is printed to screen when you run the code. The line looks something like the following:

```
Welcome to EPOCH2D version 4.17.0 (commit v4.17.0-0-g1f6d56a22-clean)
```

Here, only the number “4.17” is important.

Since version 3.1 of the manual, new additions and changes are mentioned in the appendix.

1.2 What is **EPOCH** ?

EPOCH is a plasma physics simulation code which uses the Particle in Cell (PIC) method. In this method, collections of physical particles are represented using a smaller number of pseudoparticles, and the fields generated by the motion of these pseudoparticles are calculated using a finite difference time domain technique on an underlying grid of fixed spatial resolution. The forces on the pseudoparticles due to the calculated fields are then used to update the pseudoparticle velocities, and these velocities are then used to update the pseudoparticle positions. This leads to a scheme which can reproduce the full range of classical micro-scale behaviour of a collection of charged particles.

1.2.1 Features of **EPOCH**

- MPI parallelised, explicit, second-order, relativistic PIC code.
- Dynamic load balancing option for making optimal use of all processors when run in parallel.
- MPI-IO based output, allowing restart on an arbitrary number of processors.
- Data analysis and visualisation options include ITT IDL, LLNL VisIt, Mathworks MatLab and matplotlib in Python.
- Control of setup and runs of **EPOCH** through a customisable input deck.

1.3 The origins of the code

The **EPOCH** family of PIC codes is based on the older PSC code written by Hartmut Ruhl and retains almost the same core algorithm for the field updates and particle push routines. **EPOCH** was written to add more modern features and to structure the code in such a way that future expansion of the code is made as easy as possible.

1.4 What normalisations are used in **EPOCH** ?

Since the idea from the start was that **EPOCH** would be used by a large number of different users and that it should be as easy as possible to “plug in” different modules from different people into a given copy of the code, it was decided to write **EPOCH** in SI units. There are a few places in the code where some quantities are given in other units for convenience (for example charges are specified in multiples of the electron charge), but the entire core of the code is written in SI units.

1.5 What are those `_num` things doing everywhere?

Historically using the compiler auto-promotion of `REAL` to `DOUBLE PRECISION` was unreliable, so **EPOCH** uses “kind” tags to specify the precision of the code. The `_num` suffixes and the associated definition of `REALs` as `REAL(num)` are these “kind” tags in operation. The `_num` tags force numerical constants to match the precision of the code, preventing errors due to precision conversion. The important thing is that all numerical constants should be tagged with an `_num` tag and all `REALs` should be defined as `REAL(num)`.

1.6 What is an input deck?

An input deck is text file which can be used to set simulation parameters for **EPOCH** without needing to edit or recompile the source code. It consists of a list of blocks which start as `begin:blockname` and end with `end:blockname`. Within the body of each block is a list of key/value pairs, one per line, with key and value separated by an equals sign. Most aspects of a simulation can be controlled using an input deck, such as the number of grid points in the simulation domain, the initial distribution of particles and initial electromagnetic field configuration. It is designed to be relatively easy to read and edit. For most projects it should be possible to set up a simulation without editing the source code at all. For more details, read “**The EPOCH input deck**” (Section 3).

1.7 I just want to use the code as a black box, or I’m just starting. How do I do that?

Begin by reading “**Basic examples of using EPOCH**” (Section 5). There’s quite a lot to learn in order to get started, so you should plan to read through all of this section. You will also need to refer to “**The EPOCH input deck**” (Section 3). Next, look at the code and have a play with some test problems. After that re-read this section. This should be enough for testing simple problems.

1.8 What is the auto-loader?

Throughout this document we will often refer to the “auto-loader” when setting up the initial particle distribution. In the input deck it is possible to specify a functional form for the density and temperature of a particle species. **EPOCH** will then place the particles to match the density function and set the velocities of the particles so that they match the Maxwellian thermal distribution for the temperature. The code which performs this particle set up is called the “auto-loader”.

At present, there is no way to specify a non-Maxwellian particle distribution from within the input deck. In such cases, it is necessary to edit and recompile the **EPOCH** source code. The recommended method for setting the initial particle properties is to use the “`manual_load`” function as described in Section 4.2.

1.9 What is a maths parser?

As previously mentioned, the behaviour of **EPOCH** is controlled using an input deck which contains a list of key/value pairs. The value part of the pair is not restricted to simple constants but can be a complex mathematical expression. It is evaluated at run time using a section of code called the “maths parser”. There is no need for the end user to know anything about this code. It is just there to enable the use of mathematical expressions in the input deck. Further information about this facility can be found in Section 3.19.

1.10 I am an advanced user, but I want to set up the code so that less experienced users can use it. How do I do that?

See “**Parameterising input decks**” (Section 4.6).

1.11 I want to develop an addition to **EPOCH**. How do I do that?

A slightly outdated developers manual exists which should be sufficient to cover most aspects of the code functionality. However, the code is written in a fairly modular and consistent manner, so reading through that is the best source of information. If you get stuck then you can post questions on the CCPForge forums.

1.12 I want to have a full understanding of how **EPOCH** works. How do I do that?

If you really want to understand **EPOCH** in full, the only way is to read all of this manual and then read through the code. Most of it is commented.

1.13 How do I acknowledge use of the code?

There is a paper which details many aspects of the **EPOCH** implementation and also includes useful information on current PIC codes. This paper is OpenAccess so freely available to all. If using EPOCH in your research output please use this as the reference for EPOCH and ideally also acknowledge the UK grant which funded this work. The BibTeX entry for this paper is as follows.

```
@article{Arber:2015hc,
  author = {Arber, T D and Bennett, K and Brady, C S and Lawrence-Douglas,
            A and Ramsay, M G and Sircombe, N J and Gillies, P and Evans,
            R G and Schmitz, H and Bell, A R and Ridgers, C P},
  title = {{Contemporary particle-in-cell approach to laser-plasma modelling}},
  journal = {Plasma Physics and Controlled Fusion},
  year = {2015},
  volume = {57},
  number = {11},
  pages = {1--26},
  month = nov
}
```

Acknowledgement: “This work was in part funded by the UK EPSRC grants EP/G054950/1, EP/G056803/1, EP/G055165/1 and EP/M022463/1.”

2 EPOCH for end users

This manual aims to give a complete description of how to set up and run **EPOCH** as an end user. Further details on the design and implementation of the code may be found in Arber et al. [1] and Ridgers et al. [11].

We begin by giving a brief overview of the **EPOCH** code-base, how to compile and run the code. Note that throughout this user manual, instructions are given assuming that you are typing commands at a UNIX terminal.

2.1 Structure of the **EPOCH** codes

When obtained, the **EPOCH** codes all have a similar structure. If the tarred and gzipped archive (commonly referred to as a tarball) is downloaded and unpacked into the user's `$HOME` directory, then the extracted contents will consist of a directory named "`$HOME/epoch-4.17.0`" (with "4.17.0" substituted by the current version number) and the subdirectories and files listed below.

Alternatively, if the code is checked out from the GitHub git repository with the command

```
git clone --recursive https://github.com/Warwick-Plasma/epoch.git
```

then the directory will be "`$HOME/epoch`".

Once the code has been obtained, the top-level directory will contain the following 4 directories and several files

- epoch1d - Source code and other files required for the 1D version of **EPOCH**.
- epoch2d - Source code and other files required for the 2D version of **EPOCH**.
- epoch3d - Source code and other files required for the 3D version of **EPOCH**.
- SDF - Source code for the SDF file format which is used to generate output for **EPOCH** runs. This directory also includes various tools and readers for working with SDF files.
- CHANGELOG - A brief overview of the change history for each released version of **EPOCH**.
- CODING_STYLE - This document contains the conventions which must be used for any code being submitted for inclusion in the **EPOCH** project.
- LICENSE - A copy of the GPLv3 license which is used by the **EPOCH** project.
- README.md - A brief overview of obtaining and using the **EPOCH** code.
- make_tarball.sh - This is a shell script which is used for creating the tarred and gzipped archives of **EPOCH** which are posted to the GitHub server each time a new release is made.
- test_all.sh - A regression test script used when testing the code.

The three **EPOCH** subdirectories all have a similar structure. Inside each of the `epoch{1,2,3}d` directories, there are 3 sub-directories:

- src - The **EPOCH** source code.
- example_decks - A sample data directory containing example input deck files.
- Data - This is an empty directory to use for running simulations.

There are also 3 files:

- Makefile - A standard makefile.
- Start.pro - An IDL script which starts the IDL visualisation routines. Execute it using "idl Start".

- `unpack_source_from_restart` - Restart dumps can be written to contain a copy of the input decks and source code used to generate them. This script can be used to unpack that information from a given restart dump. It is run from the command line and must be passed the name of the restart dump file.

2.2 Libraries and requirements

The **EPOCH** codes are written using MPI for parallelism, but have no other libraries or dependencies. Currently, the codes are written to only require MPI1.2 compatible libraries, although this may change to require full MPI2 compliance in the future. Current versions of both MPICH and OpenMPI implement the MPI2 standard and are known to work with this code. The SCALI MPI implementation is only compliant with the MPI1.2 specification and may loose support soon. There are no plans to write a version of **EPOCH** which does not require the MPI libraries.

The code is supplied with a standard GNU make Makefile, which is also compatible with most other forms of the **make** utility. In theory it is possible to compile the code without a **make** utility, but it is much easier to compile the code using the supplied makefile.

2.3 Compiling and running **EPOCH**

To compile **EPOCH** in the supplied state, you must first change to the correct working directory. As explained in Section 2.1, the root directory for **EPOCH** contains several subdirectories, including separate directories for each of the 1D, 2D and 3D versions of the code. To compile the 2D version of the code, you first switch to the “epoch2d” directory using the command

```
cd $HOME/epoch/epoch2d
```

and then type

```
make
```

and the code will compile. There are certain options within the code which are controlled by compiler preprocessors and are described in the next section. When the code is compiled, it creates a new directory called “bin” containing the compiled binary which will be called **epoch1d**, **epoch2d** or **epoch3d**. To run the code, just execute the binary file by typing:

```
./bin/epoch2d
```

or whatever the correct binary is for the dimensionality of the code that you have. You should be given a screen which begins with the **EPOCH** logo, and then reads:

```
Welcome to EPOCH2D version 4.17.0  (commit v4.17.0-0-g1f6d56a22-clean)

*****
The code was compiled with no compile time options
*****
Code is running on 1 processing elements

Specify output directory
```

At this point, the user simply types in the name of the (already existing) output directory and the code will read the input deck files inside the specified directory and start running. To run the code in parallel, just use the normal `mpirun` or `mpiexec` scripts supplied by your MPI implementation. If you want the code to run unattended, then you will need to pipe in the output directory name to be used. The method for doing this varies between MPI implementations. For many MPI implementations (such as recent versions of OpenMPI) this can be achieved with the following:

```
echo Data | mpirun -np 2 ./bin/epoch2d
```

Some cluster setups accept the following instead:

```
mpirun -np 2 ./bin/epoch2d < deck.file
```

where “deck.file” is a file containing the name of the output directory. Some cluster queueing systems do not allow the use of input pipes to mpirun. In this case, there is usually a “-stdin” command line option to specify an input file. See your cluster documentation for more details.

As of version 4.2.12, **EPOCH** now checks for the existence of a file named “USE_DATA_DIRECTORY” in the current working directory before it prompts the user for a Data directory. If such a file exists, it reads it to obtain the name of the data directory to use and does not prompt the user. If no such file exists, it prompts for a data directory name as before. This is useful for cluster setups in which it is difficult or impossible to pipe in the directory name using a job script.

The “Makefile” supplied with **EPOCH** is setup to use the Intel compiler by default. However, it also contains configurations for gfortran, pgi, g95, archer and ibm (the compiler suite used on IBM’s BlueGene machines). In order to compile using one of the listed configurations, add the “COMPILER=” option to the “make” command. For example

```
make COMPILER=gfortran
```

will compile the code using the gfortran compiler and appropriate compiler flags. As of version 4.11, it is now possible for the build system to automatically detect the correct compiler to use. Typing `make COMPILER=auto`

will cause the build system to guess which compiler is in use. Note that this might not always work, so it is better to use the correct value for “COMPILER” if it is already known. You can also compile the code with debugging flags by adding “MODE=debug” and can compile using more than one processor by using “-j<n>”, where “<n>” is the number of processors to use. Note that this is just to speed up the compilation process; the resulting binary can be run on any number of processors.

2.4 Compiler flags and preprocessor defines

As already stated, some features of the code are controlled by compiler preprocessor directives. The flags for these preprocessor directives are specified in “Makefile” and are placed on lines which look like the following:

```
DEFINES += $(D)PER_SPECIES_WEIGHT
```

On most machines “\$(D)” just means “-D” but the variable is required to accommodate more exotic setups.

Most of the flags provided in the “Makefile” are commented out by prepending them with a “#” symbol (the “make” system’s comment character). To turn on the effect controlled by a given preprocessor directive, just uncomment the appropriate “DEFINES” line by deleting this “#” symbol. The options currently controlled by the preprocessor are:

- PER_SPECIES_WEIGHT - By default, each pseudoparticle in the code can represent a different number of real particles. Memory can be saved by disabling this feature and have all of the pseudoparticles in a species use the same particle weight. Many of the codes more advanced features require per-particle weighting so it is enabled by default. Use this flag to disable per-particle weighting if you need to save on memory, but it this option is recommended only for advanced users.
- NO_TRACER_PARTICLES - This flag will disable the option to specify one or more species as zero-current particles. Zero-current particles move about as would a normal particle with the same charge and mass, but they do not generate any current and are therefore passive elements in the simulation. Zero-current particles should be included in collisions to ensure they move identically to ordinary particles. The implementation of zero-current particles requires an additional “IF” clause in the particle push, so it has a slight performance impact. If you do not require the feature then setting this flag will give a slight performance improvement. **WARNING:** Since the particles effectively have zero weight in terms of their numerical heating properties, they do not

always behave in the same way that an ordinary particle with weight would behave and this can sometimes lead to unexpected behaviour. If the purpose is merely to track a subset of a particle species to use as output then a better mechanism to use is “persistent subsets” (see Section 3.17.2). In version 5.0, this flag will be and replaced with “ZERO_CURRENT_PARTICLES”.

- NO_PARTICLE_PROBES - For laser plasma interaction studies it can sometimes be useful to be able to record information about particles which cross a plane in the simulation. Since this requires the code to check whether each particles has crossed the plane in the particles pusher and also to store copies of particles until the next output dump, it is a heavyweight diagnostic. If you don’t require the diagnostic you can set this flag to disable it.
- PARTICLE_SHAPE_TOPHAT - By default, the code uses a first order b-spline (triangle) shape function to represent particles giving third order particle weighting. Using this flag changes the particle representation to that of a top-hat function (0th order b-spline yielding a second order weighting).
- PARTICLE_SHAPE_BSPLINE3 - This flag changes the particle representation to that of a 3rd order b-spline shape function (5th order weighting).
- PARTICLE_ID - When this option is enabled, all particles are assigned a unique identification number when writing particle data to file. This number can then be used to track the progress of a particle during the simulation.
- PARTICLE_ID4 - This does the same as the previous option except it uses a 4-byte integer instead of an 8-byte one. Whilst this saves storage space, care must be taken that the number does not overflow.
- PHOTONS - This enables support for photon particle types in the code. These are a pre-requisite for modelling synchrotron emission, radiation reaction and pair production (see Section 3.14).
- TRIDENT_PHOTONS - This enables support for virtual photons which are used by the Trident process for pair production.
- PREFETCH - This enables an Intel-specific code optimisation.
- PARSER_DEBUG - The code outputs more detailed information whilst parsing the input deck. This is a debug mode for code development.
- PARTICLE_DEBUG - Each particle is additionally tagged with information about which processor it is currently on, and which processor it started on. This is a debug mode for code development.
- MPI_DEBUG - This option installs an error handler for MPI calls which should aid tracking down some MPI related errors.
- SIMPLIFY_DEBUG - This option enables debugging code related to the deck parser simplification routine.
- NO_IO - This option disables all file I/O which can be useful when doing benchmarking.
- COLLISIONS_TEST - This enables some routines for debugging the collision routines. It completely alters the behaviour of the code. This flag should never be enabled by the end user.
- PER_PARTICLE_CHARGE_MASS - By default, the particle charge and mass are specified on a per-species basis. With this flag enabled, charge and mass become a per-particle property. This is a legacy flag which will be removed soon.

- PARSER_CHECKING - Setting this flag adds code which checks for valid values on evaluated deck expressions. This slows down the code but may be required if floating point exceptions are enabled.
- WORK_DONE_INTEGRATED - This enables support for tracking the work done on each particle by the electric field. Note that this increases the size of each particle by 48 bytes. The information gathered can be output to file using the “work_{x,y,z}” and “work_{x,y,z}_total” dumpmasks. See Section 3.9.2.
- HC_PUSH - Use the push from Higuera and Cary [6] rather than the Boris push. This is slower than the Boris push but gives the correct $E \times B$ velocity, which can be important for highly relativistic simulations.
- DELTAF_METHOD - Compile the code to use the delta-f method to represent particles rather than standard PIC. Note that this completely changes the code behaviour and should not be enabled for normal use. See Section 4.8.
- DELTAF_DEBUG - Add debug code for the delta-f method.
- NO_USE_ISATTY - When printing the initial welcome message, EPOCH makes use of the C-library’s isatty function. This requires Fortran2003 features that might not be available on all platforms. The flag allows this functionality to be disabled on platforms that don’t support it.
- NO_MPI3 - This compiler flag allows the user to disable MPI-3 features such as the “MPI_TYPE_SIZE_X” routine. This allows the code to be compiled against older versions of the MPI library. The flag should only be enabled if the code fails to compile without it.
- DECK_DEBUG - If this flag is enabled then the code will exit if any warnings are encountered during deck parsing. If the flag is not enabled then the same behaviour can be obtained by using the “deck_warnings_fatal” flag in the control block of the input deck. See Section 3.1.
- BREMSSTRAHLUNG - This flag enables the model for simulating bremsstrahlung radiation. See Section 3.15 for details.

If a user requests an option which the code has not been compiled to support then the code will give an error or warning message as follows:

```
*** ERROR ***
Unable to set "use_qed=T" in the "qed" block.
Please recompile with the -DPHOTONS preprocessor flag.
```

It is also possible to pass other flags to the compiler. In “Makefile” there is a line which reads
FFLAGS = -O3 -fast

The two commands to the right are compiler flags and are passed unaltered to the FORTRAN compiler. Change this line to add any additional flags required by your compiler.

By default, **EPOCH** will write a copy of the source code and input decks into each restart dump. This can be very useful since a restart dump contains an exact copy of the code which was used to generate it, ensuring that you can always regenerate the data or continue running from a restart. The output can be prevented by using “dump_source_code = F” and “dump_input_deck = F” in the output block. However, the functionality is difficult to build on some platforms so the Makefile contains a line for bypassing this section of the build process. Just below all the DEFINE flags there is the following line:

```
#ENCODED_SOURCE = epoch_source_info_dummy.o
```

Just uncomment this line and source code in restart dumps will be permanently disabled.

2.5 Running **EPOCH** and basic control of EPOCH1D

When the code is run, the output is

```
Command line output

d#####P d#####b .#####b d##### d##P d##P
d#####P d##### d##### .##### d##P d##P
---- ---- ---- ---- ---- ---- -- P
d#####P d####,,###P ####. .#### d###P d#####P#####
d#####P d#####P ####. ####P ####. d#####P#####
d##P d##P #### d#### ####. d##P d##P
d#####P d##P #####P #####P d##P d##P d##P
d#####P d##P d#####P #####P d##P d##P d##P

Welcome to EPOCH2D version 4.17.0 (commit v4.17.0-0-g1f6d56a22-clean)

*****
The code was compiled with no compile time options
*****
Code is running on 1 processing elements

Specify output directory
```

At which point the end user should simply type in the name of the directory where the code output is to be placed. This directory must also include the file “`input.deck`” which controls the code setup, specifies how to set the initial conditions and controls the I/O. Writing an input deck for **EPOCH** is fairly time consuming and so the code is supplied with some example input decks which include all the necessary sections for the code to run. Alternately, the code checks for the Data directory in a file named “`USE_DATA_DIRECTORY`” before prompting at the command-line. This allows the code to be run without waiting for input at the command-line.

3 The **EPOCH** input deck

Most of the control of **EPOCH** is through a text file called `input.deck`. The input deck file must be in the output directory which is passed to the code at runtime and contains all the basic information which is needed to set up the code, including the size and subdivision of the domain, the boundary conditions, the species of particles to simulate and the output settings for the code. For most users this will be capable of specifying all the initial conditions and output options they need. More complicated initial conditions will be handled in later sections.

The input deck is a structured file which is split into separate blocks, with each block containing several “parameter” = “value” pairs. The pairs can be present in any order, and not all possible pairs must be present in any given input deck. If a required pair is missing the code will exit with an error message. The blocks themselves can also appear in any order. The input deck is case sensitive, so true is always “T”, false is always “F” and the names of the parameters are always lower case. Parameter values are evaluated using a maths parser which is described in Section 3.19.

If the deck contains a “\” character then the rest of the line is ignored and the next line becomes a continuation of the current one. Also, the comment character is “#”; if the “#” character is used anywhere on a line then the remainder of that line is ignored.

There are three *input deck directive* commands, which are:

- begin:*block* - Begin the block named *block*.
- end:*block* - Ends the block named *block*.
- import:*filename* - Includes another file (called *filename*) into the input deck at the point where the directive is encountered. The input deck parser reads the included file exactly as if the contents of the included file were pasted directly at the position of the import directive.

Each block must be surrounded by valid *begin:* and *end:* directives or the input deck will fail. There are currently fourteen valid blocks hard coded into the input deck reader, but it is possible for end users to extend the input deck. The fourteen built in blocks are:

- control - Contains information about the general code setup.
- boundaries - Contains information about the boundary conditions for this run.
- species - Contains information about the species of particles which are used in the code. Also details of how these are initialised.
- laser - Contains information about laser boundary sources.
- fields - Contains information about the EM fields specified at the start of the simulation.
- window - Contains information about the moving window if the code is used in that fashion.
- output - Contains information about when and how to dump output files.
- output_global - Contains parameters which should be applied to all output blocks.
- dist_fn - Contains information about distribution functions that should be calculated for output.
- probe - Contains information about particle probes used for output.
- collisions - Contains information about particle collisions.
- qed - Contains information about QED pair production.
- subset - Contains configuration for filters which can be used to modify the data to be output.
- constant - Contains information about user defined constants and expressions. These are designed to simplify the initial condition setup.

Note that it is possible for an end user to add new blocks to the input deck. As a result, a version of the code which you have obtained from a source other than the GitHub repository may include additional input deck blocks that are not documented in this manual.

3.1 control block

The **control** block sets up the basic code properties for the domain, the end time of the code, the load balancer and the types of initial conditions to use.

The control block of a valid input deck for EPOCH2D reads as follows:

```

control block
begin:control
# Global number of gridpoints
nx = 512 # in x
ny = 512 # in y
# Global number of particles
nparticles = 10 * nx * ny

# Final time of simulation
t_end = 1.0e-12
# nsteps = -1

# Size of domain
x_min = -0.1e-6
x_max = 400.0e-6
y_min = -400.0e-6
y_max = 400.0e-6

# dt_multiplier = 0.95
# dlb_threshold = 0.8

# restart_snapshot = 98

# field_order = 2
# maxwell_solver = yee

# stdout_frequency = 10
end:control

```

As illustrated in the above code block, the “#” symbol is treated as a comment character and the code ignores everything on a line following this character.

The allowed entries are as follows:

nx, ny, nz - Number of grid points in the x,y,z direction. This parameter is mandatory.

nparticles - The global number of pseudoparticles in the simulation. This parameter does not need to be given if a specific number of particles is supplied for each particle species by using the “nparticles” directive in each **species** block (see Section 3.4). If both are given then the value in the **control** block will be ignored. “npart” is accepted as an alias.

nsteps - The number of iterations of the core solver before the code terminates. Negative numbers instruct the code to only terminate at **t_end**. If **nsteps** is not specified then **t_end** must be given.

t_end - The final simulation time in simulation seconds before the code terminates. If **t_end** is not specified then **nsteps** must be given. If they are both specified then the first time restriction to be satisfied takes precedence. Sometimes it is more useful to specify the time in picoseconds or femtoseconds. To accomplish this, just append the appropriate multiplication factor. For example, “t_end = 3 * femto” specifies 3 femtoseconds. A list of multiplication factors is supplied in Section 3.19.1.

{x,y,z}_min - Minimum grid position of the domain in metres. These are required parameters. Can be negative. “{x,y,z}_start” is accepted as a synonym. In a similar manner to that described above, distances can be specified in microns using a multiplication constant. eg. “x_min = 4 * micron”

specifies a distance of $4\mu\text{m}$.

{x,y,z}_max - Maximum grid position of the domain in metres. These are required parameters. Must be greater than **{x,y,z}_min**. “**{x,y,z}_end**” is accepted as a synonym.

dt_multiplier - Factor by which the timestep is multiplied before it is applied in the code, i.e. a multiplying factor applied to the CFL condition on the timestep. Must be less than one. If no value is given then the default of 0.95 is used. If **maxwell_solver** is different from “yee” (the default) this parameter becomes increasingly relevant.

dlb_threshold - The minimum ratio of the load on the least loaded processor to that on the most loaded processor allowed before the code load balances. Set to 1 means always balance, set to 0 means never balance. If this parameter is not specified then the code will only be load balanced at initialisation time.

restart_snapshot - The number of a previously written restart dump to restart the code from. If not specified then the initial conditions from the input deck are used.

Note that as of version 4.2.5, this parameter can now also accept a filename in place of a number. If you want to restart from “0012.sdf” then it can either be specified using “**restart_snapshot = 12**”, or alternatively it can be specified using “**restart_snapshot = 0012.sdf**”. This syntax is required if output file prefixes have been used (see Section 3.9).

field_order - Order of the finite difference scheme used for solving Maxwell’s equations. Can be 2, 4 or 6. If not specified, the default is to use a second order scheme.

maxwell_solver - Choose a Maxwell solver scheme with an extended stencil. This option is only active if **field_order** is set to 2. Possible options are “yee”, “lehe_{x,y,z}”, “pukhov”, “cowan” and “custom”. Note that not all options are available in 1d and 2d. The default is “yee” which is the default second order scheme.

stdout_frequency - If specified then the code will print a one line status message to stdout after every given number of timesteps. The default is to print nothing to screen (ie. “**stdout_frequency = 0**”).

use_random_seed - The initial particle distribution is generated using a random number generator. By default, **EPOCH** uses a fixed value for the random generator seed so that results are repeatable. If this flag is set to “T” then the seed will be generated using the system clock.

nproc{x,y,z} - Number of processes in the **x,y,z** directions. By default, **EPOCH** will try to pick the best method of splitting the domain amongst the available processors but occasionally the user may wish to override this choice.

smooth_currents - This is a logical flag. If set to “T” then a smoothing function is applied to the current generated during the particle push. This can help to reduce noise and self-heating in a simulation. The smoothing function used is the same as that outlined in Buneman [3]. The default value is “F”.

field_ionisation - Logical flag which turns on field ionisation. See Section 3.4.2.

use_bsi - Logical flag which turns on barrier suppression ionisation correction to the tunnelling ionisation model for high intensity lasers. See Section 3.4.2. This flag should always be enabled when using field ionisation and is only supplied for testing purposes. The default is “T”.

use_multiphoton - Logical flag which turns on modelling ionisation by multiple photon absorption. This should be set to “F” if there is no laser attached to a boundary as it relies on laser frequency. See Section 3.4.2. This flag should always be enabled when using field ionisation and is only supplied for testing purposes. The default is “T”.

particle_tstart - Specifies the time at which to start pushing particles. This allows the field to evolve using the Maxwell solver for a specified time before beginning to move the particles.

use_exact_restart - Logical flag which makes a simulation restart using exactly the same configuration as the original simulation. If set to “T” then the domain split amongst processors will be identical along with the seeds for the random number generators. Note that the flag will be ignored if the number of processors does not match that used in the original run. The default value is “T”.

allow_cpu_reduce - Logical flag which allows the number of CPUs used to be reduced from the number specified. In some situations it may not be possible to divide the simulation amongst all the processors requested. If this flag is set to “T” then **EPOCH** will continue to run and leave some of the requested CPUs idle. If set to “F” then code will exit if all CPUs cannot be utilised. The default value is “T”.

check_stop_file_frequency - Integer parameter controlling automatic halting of the code. The frequency is specified as number of simulation cycles. Refer to description later in this section. The default value is 10.

stop_at_walltime - Floating point parameter controlling automatic halting of the code. Refer to description later in this section. The default value is -1.0.

stop_at_walltime_file - String parameter controlling automatic halting of the code. Refer to description later in this section. The default value is an empty string.

simplify_deck - If this logical flag is set to “T” then the deck parser will attempt to simplify the maths expressions encountered after the first pass. This can significantly improve the speed of evaluation for some input deck blocks. The default value is “F”.

print_constants - If this logical flag is set to “T”, deck constants are printed to the “deck.status” and “const.status” files as they are parsed. The default value is “F”.

use_migration - Logical flag which determines whether or not to use particle migration. The default is “F”. See Section 3.4.1.

migration_interval - The number of timesteps between each migration event. The default is 1 (migrate at every timestep). See Section 3.4.1.

allow_missing_restart - Logical flag to allow code to run when a restart dump is absent. When “restart_snapshot” is specified then the simulation first checks that the specified restart dump is valid. If the restart dump exists and is valid then it is used to provide initial conditions for the simulation. However, if the restart dump does not exist or is not usable for some reason then by default the simulation will abort. If “allow_missing_restart” is set to “T” then the simulation will not abort but will continue to run and use the initial conditions contained in the input deck to initialise the simulation. The default value is “F”.

print_eta_string - If this logical flag is set to “T” then the current estimated time to completion will be appended to the status updates. The default value is “T”.

n_zeros - Integer flag which specifies the number of digits to use for the output file numbers. (eg. “0012.sdf”). By default, the code tries to calculate the number of digits required by dividing t_end by dt_snapshot. Note that the minimum number of digits is 4.

use_accurate_n_zeros - If this logical flag is set to “T” then the code performs a more rigorous test to determine the number of digits required to accommodate all outputs that are to be generated by a run. Since this can be time consuming and is overkill for most cases, it is disabled by default. The default value is “F”.

use_particle_count_update - If this logical flag is set to “T” then the code keeps global particle counts for each species on each processor. This information isn’t needed by the core algorithm, but can be useful for developing some types of additional physics packages. It does require one additional MPI_ALL_REDUCE per species per timestep, so it is not activated by default. The default value is “F”.

reset_walltime - When restarting from a dump file, the current walltime displayed will include the elapsed walltime recorded in the restart dump. The user can request that this time is ignored by setting the **reset_walltime** flag to “T”. The default value is “F”.

dlb_maximum_interval - This integer parameter determines the maximum number of timesteps to allow between load balancing checks. Each time that the load balancing sweep is unable to improve the load balance of the simulation, it doubles the number of steps before the next check will occur. It will keep increasing the check interval until it reaches the value given by **dlb_maximum_interval**. If the value of **dlb_maximum_interval** is negative then the check interval will increase indefinitely. When the load balancing sweep finds an improvement to the load balance of the simulation, the check interval is reset to one. The default value is 500.

dlb_force_interval - This integer parameter determines the maximum number of timesteps to allow between forcing a full load balance sweep. If the current load balance is greater than the value of **dlb_threshold** then the load balancer exits before attempting to improve the loading. If **dlb_force_interval** is greater than zero, then the full load balancer will be run at the requested interval of timesteps, regardless of the value of **dlb_threshold**. Note that the simulation will only be redistributed if this would result in an improved load balance. The default value is 2000.

balance_first - This logical flag determines whether a load balance will be attempted on the first call of the load balancer. The load balancer performs two functions: first it attempts to find a domain decomposition that balances the load evenly amongst processors. Next, it redistributes the domain and particles onto the new layout (if required). This latter step is always required when setting up the simulation, so the load balancer is always called once during set-up. This flag controls whether or not a load balance is attempted during this call, regardless of the value of **dlb_threshold**. The default value is “T”.

use_pre_balance - This logical flag determines whether a load balance will be attempted before the particle load occurs. If this flag is set to “T” then the particle auto-loader will be called at setup time, but instead of creating particles it will just populate a particle-per-cell field array. This will then be used to calculate the optimal domain decomposition and all field arrays will be redistributed to use the new layout. Finally, after all of this has been done, the auto-loader will be called again and create just the particles that are present on their optimally load-balanced domains. In contrast, if the flag is set to “F” then the domain is just divided evenly amongst processors and the particles are loaded on this domain decomposition. Balancing is then carried out on to redistribute the work load. For heavily imbalanced problems, this can lead to situations in which there is insufficient memory to setup a simulation, despite there being sufficient resources for the final load-balanced conditions. The default

value is “T”.

use_optimal_layout - This logical flag determines whether the load balancer attempts to find an optimal processor split before loading the particles. The initial domain split is chosen in such a way as to minimize the total surface area of the resulting domains in 3D, or edge lengths in 2D. For example, if a 2D square domain is run on 16 CPUs then the domain will be divided by 4 in the x-direction and 4 in the y-direction. The other possible splits (1x16, 2x8, 8x2, 16x1) are rejected because they all yield rectangular subdomains whose total edge length is greater than the 4x4 edge length. For some problems (eg. a density ramp or thin foil) this is a poor choice and a better load balance would be obtained by a less even split. It is always possible to specify such a split by using `nproc{x,y,z}` flags but enabling the **use_optimal_layout** flag will automatically determine the best split for you. Future versions of the code will also allow the split to be changed dynamically at run time. The default value is “T”.

use_more_setup_memory - This logical flag determines whether the extra memory will be used during the initial setup of particle species. If set to false then only one set of arrays will be used for storing temperature, density and drift during species loading. This can be a significant memory saving but comes at the expense of recalculating grid quantities multiple times. Setting the flag to true enables one set of arrays per species. The default value is “F”.

deck_warnings_fatal - This logical flag controls the behaviour of the deck parser when a warning is encountered. Usually the code will just print a warning message and continue running. Setting this flag to “T” will force the code to abort. The default value is “F”.

3.1.1 Maxwell Solvers

With the default settings “`field_order=2`”, “`maxwell_solver=yee`” EPOCH will use the standard second order Yee scheme for solving Maxwell’s equations. This scheme has a grid dispersion relation with phase velocities smaller than c , especially for large spatial frequencies. It is possible to introduce extended stencils into the update step of the Maxwell-Faraday equation which will help improving the dispersion relation. All of the following extended stencils are only available when “`field_order=2`”. Please note that you will also need to choose an appropriate `dt_multiplier`, according to the selected scheme. A `dt_multiplier` equal to unity would result in using the largest time-step allowed by the CFL condition for any of the implemented schemes. This time-step is said to be marginally stable. While, in general, the marginally stable time-step has the best dispersion properties, simulations may suffer from numerical problems such as exponentially growing noise. Choosing smaller values for the `dt_multiplier` tend to improve on this, while adversely affecting the dispersion relation. The implemented solvers behave differently in this regard. Different options are available as follows:

maxwell_solver = lehe_{x,y,z} - This setting will enable an extended stencil proposed by Lehe et al. [7]. This stencil focusses on improving the dispersion relation along a specific direction. The “lehe_x” stencil optimises for waves travelling along the x -axis. Similarly “lehe_{y,z}” are tuned for the y and z axis. please take this into account when defining your laser input. It is available in EPOCH1D, EPOCH2D and EPOCH3D. While it is not technically required to use a `dt_multiplier` smaller than unity, the value proposed by Lehe et al. [7] is “`dt_multiplier=0.96`”.

maxwell_solver = pukhov - This setting will enable an extended stencil proposed by Pukhov [10] under the name of NDFX. It is available in EPOCH2D and EPOCH3D. In EPOCH1D, setting “`maxwell_solver = pukhov`” will make the code fall back silently to Yee’s scheme. Pukhov’s NDFX scheme aims at improving the numerical dispersion relation by allowing to choose “`dt_multiplier = 1.0`”, while smaller values are also valid. The resulting dispersion relation is best along the axis with the smallest grid spacing.

maxwell_solver = cowan - This setting will enable an extended stencil proposed by Cowan et al. [4]. It is available only in EPOCH3D. In EPOCH1D and EPOCH2D, setting “**maxwell_solver = cowan**” will make the code fall back silently to Yee’s scheme. Cowan et al. [4] propose to numerically calculate a time step that has the correct group velocity for the input laser. Typically these time steps are only slightly below the CFL condition, e.g. “**dt_multiplier = 0.999**”. When Cowan’s scheme is reduced to 2D it is the same as Pokhov’s scheme with **dt_multiplier < 1.0**. The resulting dispersion relation is best along the axis with the smallest grid spacing.

maxwell_solver = custom - This setting will enable full user control over the extended stencil coefficients. This allows for the specification of optimised coefficients as outlined in Blinne et al. [2]. This option must be accompanied by a “stencil” block. See Section 3.2

3.1.2 Strided Current Filtering

EPOCH can perform strided multipass digital current filtering as described and benchmarked in the review by Vay and Godfrey [15]. This can be tuned to substantially damp high frequencies in the currents and can be used to reduce the effect of numerical Cherenkov radiation. Once you turn on current filtering by specifying **smooth_currents=T** you can then set the following keys.

smooth_iterations - Integer number of iterations of the smoothing function to be performed. If not present defaults to one iteration. More iterations will produce smoother results but will be slower.

smooth_compensation - Logical flag. If this flag is set to true then perform a compensation step (see Vay and Godfrey [15]) after the smoothing steps are performed. The default value is false.

smooth_strides - Either a comma separated list of integers or “auto” (without quote marks). This specifies the strides (in number of grid cells) to use when performing strided filtering. Specifying “1, 3” will smooth each point with the points immediately adjacent and with the points 3 cells away on each side of the current cell. Setting this key to “auto” uses a “1, 2, 3, 4” set of strides as a “good” starting point for strided filtering.

It should be stressed that there is no set of values that is guaranteed to give any given result from filtering while not affecting the physical correctness of your simulation. Current filtering should be tuned to match the problem that you want to work on and should always be carefully tested to ensure that it doesn’t produce unphysical results.

3.1.3 Dynamic Load Balancing

EPOCH can be configured to balance the workload across processors at run-time via the **dlb_threshold** flag. “dlb” stands for Dynamic Load Balancing and, when turned on, it allows the code to rearrange the internal domain boundaries to try and balance the workload on each processor. This rearrangement is an expensive operation, so it is only performed when the maximum load imbalance reaches a given critical point. This critical point is given by the parameter “**dlb_threshold**” which is the ratio of the workload on the least loaded processor to the most loaded processor. When the calculated load imbalance is less than “**dlb_threshold**” the code performs a re-balancing sweep, so if “**dlb_threshold = 1.0**” is set then the code will keep trying to re-balance the workload at almost every timestep. At present the workload on each processor is simply calculated from the number of particles on each processor, but this will probably change in future. If the “**dlb_threshold**” parameter is not specified then the code will only be load balanced at initialisation time.

3.1.4 Automatic halting of a simulation

It is sometimes useful to be able to halt an **EPOCH** simulation midway through execution and generate a restart dump. Two methods have been implemented to enable this.

The first method is to check for the existence of a “STOP” file. Throughout execution, **EPOCH** will check for the existence of a file named either “STOP” or “STOP_NODUMP” in the simulation output directory. The check is performed at regular intervals and if such a file is found then the code exits immediately. If “STOP” is found then a restart dump is written before exiting. If “STOP_NODUMP” is found then no I/O is performed.

The interval between checks is controlled by the integer parameter “check_stop_frequency” which can be specified in the “control” block of the input deck. If it is less than or equal to zero then the check is never performed.

The next method for automatically halting the code is to stop execution after a given elapsed wall-time. If a positive value for “stop_at_walltime” is specified in the control block of an input deck then the code will halt once this time is exceeded and write a restart dump. The parameter takes a real argument which is the time in seconds since the start of the simulation.

An alternative method of specifying this time is to write it into a separate text file. “stop_at_walltime_file” is the filename from which to read the value for “stop_at_walltime”. Since the walltime will often be found by querying the queueing system in a job script, it may be more convenient to pipe this value into a text file rather than modifying the input deck.

3.1.5 Requesting output dumps at run time

In addition to polling for the existence of a “STOP” file, **EPOCH** also periodically checks the output directory for a file named “DUMP”. If such a file is found then **EPOCH** will immediately create an output dump and remove the “DUMP” file. By default, the file written will be a restart dump but if the “DUMP” file contains the name of an output block then this will be used instead.

3.2 stencil block

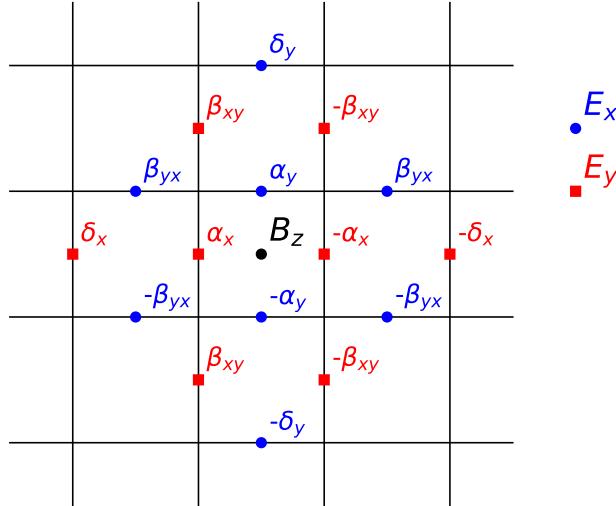


Figure 1: Coefficient locations for the B_z field computational stencil in 2D.

The extended stencil Maxwell solvers described above all operate by including points in neighbouring cells with a carefully chosen weighting. These weightings are determined by adjusting the coefficients shown in Figure 1. Full control over these coefficients can be achieved by specifying “custom” for the “maxwell_solver” parameter in the control block and then supplying a “stencil” block to provide the desired coefficient values.

This option allows the user to specify an extended stencil scheme that has been specifically optimised for the simulation grid spacing and timestep. See Blinne et al. [2] for further details.

Note that there is no option for changing the value of $\alpha_{x,y,z}$ since these are calculated using the following equations:

$$\begin{aligned}\alpha_x &= 1 - 2\beta_{xy} - 2\beta{xz} - 3\delta_x, \\ \alpha_y &= 1 - 2\beta_{yx} - 2\beta{yz} - 3\delta_y, \\ \alpha_z &= 1 - 2\beta_{zx} - 2\beta{zy} - 3\delta_z.\end{aligned}$$

delta{x,y,z}, gamma{x,y,z}, beta{xy,xz,yx,yz,zx,zy} - The coefficients to use for the extended stencil points as shown in Figure 1. See Blinne et al. [2] for further details. These coefficients are specified as floating point numbers. The default values are to set all coefficients to zero which results in $\alpha_{x,y,z}$ having values of unity. This corresponds to the standard Yee scheme.

dt - The timestep restriction to use for the field solver

3.3 boundaries block

The **boundaries** block sets the boundary conditions of each boundary of the domain. Some types of boundaries allow EM wave sources (lasers) to be attached to a boundary. Lasers are attached at the initial conditions stage.

An example boundary block for EPOCH2D is as follows:

```
----- boundaries block -----
begin:boundaries
  bc_x_min = simple_laser
  bc_x_max_field = simple_outflow
  bc_x_max_particle = simple_outflow
  bc_y_min = periodic
  bc_y_max = periodic
end:boundaries
```

The **boundaries** accepts the following parameters:

bc_{x,y,z}_min - The condition for the lower boundary for both fields and particles. “xbc_left”, “ybc_down” and “zbc_back” are accepted as a synonyms.

bc_{x,y,z}_min_{field,particle} - The condition for the lower boundary for {fields,particles}. “xbc_left_{field,particle}”, “ybc_down_{field,particle}” and “zbc_back_{field,particle}” are accepted as a synonyms.

bc_{x,y,z}_max - The condition for the upper boundary for both fields and particles. “xbc_right”, “ybc_up” and “zbc_front” are accepted as a synonyms.

bc_{x,y,z}_max_{field,particle} - The condition for the upper boundary for {fields,particles}. “xbc_right_{field,particle}”, “ybc_up_{field,particle}” and “zbc_front_{field,particle}” are accepted as a synonyms.

cpml_thickness - The thickness of the CPML boundary in terms of the number of grid cells (see Section 3.3.1). The default value is 6.

cpml_kappa_max - A tunable CPML parameter (see Section 3.3.1).

cpml_a_max - A tunable CPML parameter (see Section 3.3.1).

cpml_sigma_max - A tunable CPML parameter (see Section 3.3.1).

There are ten boundary types in **EPOCH** and each boundary of the domain can have one and only one of these boundaries attached to it. These boundary types are:

periodic - A simple periodic boundary condition. Fields and/or particles reaching one edge of the domain are wrapped round to the opposite boundary. If either boundary condition is set to periodic then the boundary condition on the matching boundary at the other side of the box is also assumed periodic.

simple_laser - A characteristic based boundary condition to which one or more EM wave sources can be attached. EM waves impinging on a **simple_laser** boundary are transmitted with as little reflection as possible. Particles are fully transmitted. The field boundary condition works by allowing outflowing characteristics to propagate through the boundary while using the attached lasers to specify the inflowing characteristics. The particles are simply removed from the simulation when they reach the boundary.

simple_outflow - A simplified version of **simple_laser** which has the same properties of transmitting incident waves and particles, but which cannot have EM wave sources attached to it. These boundaries are about 5% more computationally efficient than **simple_laser boundaries** with no attached sources. This boundary condition again allows outflowing characteristics to flow unchanged, but this time the inflowing characteristics are set to zero. The particles are again simply removed from the simulation when they reach the boundary.

reflect - This applies reflecting boundary conditions to particles. When specified for fields, all field components are clamped to zero.

conduct - This applies perfectly conducting boundary conditions to the field. When specified for particles, the particles are reflected.

open - When applied to fields, EM waves outflowing characteristics propagate through the boundary. Particles are transmitted through the boundary and removed from the system.

heat_bath - Applies to particles only. See Section 3.3.2.

cpml_laser - See Section 3.3.1.

cpml_outflow - See Section 3.3.1.

thermal - Applies to particles only. See Section 3.3.2.

NOTE: If simple_laser, simple_outflow, cpml_laser, cpml_outflow or open are specified on one or more boundaries then the code will no longer necessarily conserve mass.

Note also that it is possible for the user to specify contradictory, unphysical boundary conditions. It is the users responsibility that these flags are set correctly.

3.3.1 CPML boundary conditions

There are now Convolutional Perfectly Matched Layer boundary conditions in **EPOCH**. The implementation closely follows that outlined in the book “Computational Electrodynamics: The Finite-Difference Time-Domain Method” by Taflove and Hagness [14]. See also Roden and Gedney [12].

CPML boundaries are specified in the input deck by specifying either “cpml_outflow” or “cpml_laser” in the boundaries block. “cpml_outflow” specifies an absorbing boundary condition whereas “cpml_laser” is used to attach a laser to an otherwise absorbing boundary condition.

There are also four configurable parameters:

cpml_thickness - The thickness of the CPML boundary in terms of the number of grid cells. The default value is 6.

cpml_kappa_max, **cpml_a_max**, **cpml_sigma_max** - These are tunable parameters which affect the behaviour of the absorbing media. The notation follows that used in the two references quoted above. Note that the “cpml_sigma_max” parameter is normalised by σ_{opt} which is taken to be $3.2/dx$ (see Taflove and Hagness [14] for details). These are real valued parameters which take the following default values: cpml_kappa_max=20, cpml_a_max=0.15, cpml_sigma_max=0.7

An example usage is as follows:

```
begin:boundaries
    cpml_thickness = 16
    cpml_kappa_max = 20
    cpml_a_max = 0.2
    cpml_sigma_max = 0.7
    bc_x_min = cpml_laser
    bc_x_max = cpml_outflow
    bc_y_min = cpml_outflow
    bc_y_max = cpml_outflow
end:boundaries
```

3.3.2 Thermal boundary conditions

The “heat_bath” particle boundary condition simulates the existence of a “thermal bath” of particles in the domain adjacent to the boundary. This is achieved by setting up particle injectors whose properties match those of each species. These injectors will then steadily add particles to the simulation as if the plasma continued for an infinite extent beyond the simulation boundaries. Note that the injectors do **not** attempt to match the species as they evolve. The injector properties are only set to match the properties specified at the start of the simulation. However, these properties can vary both temporally and spatially. Time or space varying initial conditions will be evaluated at the current simulation time and at the position of the specified boundary. All restrictions on particle injectors apply (see 3.16).

Historically, thermal boundary conditions in **EPOCH** were specified using the “thermal” boundary condition for particles in the “boundaries” block. These work by replacing each particles that leaves the simulation with an identical incoming particle whose momentum has sampled from a Maxwellian of a temperature corresponding to that of the initial conditions. In most situations, the “heat_bath” boundary condition will give better results, but the latter method has been kept for backwards compatibility.

For example:

```

begin:boundaries
  bc_x_min = laser
  bc_x_max_field = open
  bc_x_max_particle = heat_bath
end:boundaries

```

3.4 species block

The next section of the input deck describes the particle species used in the code. An example species block for any **EPOCH** code is given below.

```

species block
begin:species
  name = Electron
  charge = -1.0
  mass = 1.0
  fraction = 0.5
  # nparticles = 2000 * 100
  number_density = 1.e4
  temperature = 1e6

  temperature_x = 0.0
  temperature_y = temperature_x(Electron)
  number_density_min = 0.1 * den_max
  number_density = if(abs(x) lt thick, den_max, 0.0)
  number_density = if((x gt -thick) and (abs(y) gt 2e-6), \
                      0.0, number_density(Carbon))
end:species

begin:species
  name = Carbon
  charge = 4.0
  mass = 1836.0 * 12
  fraction = 0.5

  number_density = 0.25 * number_density(Electron)
  temperature_x = temperature_x(Electron)
  temperature_y = temperature_x(Electron)

  dumpmask = full
end:species

```

Each species block accepts the following parameters:

name - This specifies the name of the particle species defined in the current block. This name can include any alphanumeric characters in the basic ASCII set. The name is used to identify the species in any consequent input block and is also used for labelling species data in any output dumps. It is a mandatory parameter.

NOTE: IT IS IMPOSSIBLE TO SET TWO SPECIES WITH THE SAME NAME!

charge - This sets the charge of the species in multiples of the electron charge. Negative numbers are used for negatively charged particles. This is a mandatory parameter.

mass - This sets the mass of the species in multiples of the electron mass. Cannot be negative. This is a mandatory parameter.

nparticles - This specifies the number of pseudoparticles which should be loaded into the simulation domain for this species block. Using this parameter is the most convenient way of loading particles for simulations which contain multiple species with different number densities. If **nparticles** is specified in a species block then any value given for **nparticles** in the **control** block is ignored. **nparticles** should not be specified at the same time as **fraction** within a **species** block. “npart” is accepted as an alias.

fraction - This specifies what fraction of **nparticles** (the global number of particles specified in the control block) should be assigned to the species. “frac” is accepted as an alias.

NOTE: “fraction” should not be specified at the same time as “nparticles” for a given species.

nparticles_per_cell - Integer parameter which specifies the number of particles per cell to use for the initial particle loading. At a later stage this may be extended to allow “nparticles_per_cell” to be a spatially varying function.

If per-species weighting is used then the value of “nparticles_per_cell” will be the average number of particles per cell. If “nparticles” or “fraction” have also been specified for a species, then they will be ignored.

To avoid confusion, there is no globally used “nparticles_per_species”. If you want to have a single value to change in the input deck then this can be achieved using a constant block. “npart_per_cell” is accepted as an alias.

dumpmask - Determines which output dumps will include this particle species. The dumpmask has the same semantics as those used by variables in the “output” block, described in Section 3.9. The actual dumpmask from the output block is applied first and then this one is applied afterwards. For example, if the species block contains “dumpmask = full” and the output block contains “vx = always” then the particle velocity will be only be dumped at full dumps for this particle species. The default dumpmask is “always”.

dump - This logical flag is provided for backwards compatibility. If set to “F” it has the same meaning as “dumpmask = never”. If set to “T” it has the same meaning as “dumpmask = always”.

zero_current - Logical flag switching the particle species into zero-current particles. Zero-current particles are enabled if the if the “NO_TRACER_PARTICLES” precompiler option has not been used and the “zero_current” flag is set to true for a given species. When set, the species will move correctly for its charge and mass, but contribute no current. This means that these particles are passive elements in the simulation. In all other respects they are designed to behave identically to ordinary particles, so they do take part in collisions by default. This can be prevented using the collision matrices in “**collisions block**” (Section 3.13). **WARNING:** Since the particles effectively have zero weight in terms of their numerical heating properties, they do not always behave in the same way that an ordinary particle with weight would behave and this can sometimes lead to unexpected behaviour. If the purpose is merely to track a subset of a particle species to use as output then a better mechanism to use is “persistent subsets” (see Section 3.17.2). “tracer” is currently accepted as an alias but this will be removed in version 5.0. “zero_current = F” is the default value.

identify - Used to identify the type of particle. Currently this is used primarily by the QED and bremsstrahlung routines. See Section 3.14 and Section 3.15 for details.

atomic_number - Integer parameter specifying the atomic number of a species. This is used by the bremsstrahlung radiation model. See Section 3.15 for details.

immobile - Logical flag. If this parameter is set to “T” then the species will be ignored during the particle push. The default value is “F”.

background_species - Logical flag. If set to “T” the species will be treated as a non evolving continuum background. No particles are loaded. Any particle-like specifications will be ignored. Background species are currently only used by the bremsstrahlung radiation model. See Section 3.15 for details. Default value is “F”. “background” is accepted as an alias.

The species blocks are also used for specifying initial conditions for the particle species. The initial conditions in **EPOCH** can be specified in various ways, but the easiest way is to specify the initial conditions in the input deck file. This allows any initial condition which can be specified everywhere in space by a number density and a drifting Maxwellian distribution function. These are built up using the normal maths expressions, by setting the density and temperature for each species which is then used by the autoloader to actually position the particles.

The elements of the species block used for setting initial conditions are:

number_density - Particle number density in m^{-3} . As soon as a `number_density =` line has been read, the values are calculated for the whole domain and are available for reuse on the right hand side of an expression. This is seen in the above example in the first two lines for the Electron species, where the number density is first set and then corrected. If you wish to specify the number density in parts per cubic metre then you can divide by the “cc” constant (see Section 3.19.1). This parameter is mandatory. “density” is accepted as an alias.

number_density_min - Minimum particle number density in m^{-3} . When the number density in a cell falls below `number_density_min` the autoloader does not load any pseudoparticles into that cell to minimise the number of low weight, unimportant particles. If set to 0 then all cells are loaded with particles. This is the default. “density_min” is accepted as an alias.

number_density_max - Maximum particle number density in m^{-3} . When the number density in a cell rises above `density_max` the autoloader clips the density to `number_density_max` allowing easy implementation of exponential rises to plateaus. If it is a negative value then no clipping is performed. This is the default. “density_max” is accepted as an alias.

mass_density - Particle mass density in $kg\ m^{-3}$. The same as “number_density” but multiplied by the particle mass. If you wish to use units of $g\ cm^{-3}$ then append the appropriate multiplication factor. For example: “`mass_density = 2 * 1e3 / cc`”.

temperature_{x,y,z} - The temperature in each direction for a thermal distribution specified in Kelvin. “`temp_{x,y,z}`” can be used as aliases.

temperature - Sets an isotropic temperature distribution in Kelvin. If both “temperature” and a specific “temperature_x”, “temperature_y” or “temperature_z” parameter is specified then the last to appear in the deck has precedence. If neither are given then the species will have a default temperature of zero Kelvin. “temp” is accepted as an alias.

temperature_{x,y,z}_ev, temperature_ev - These are the same as the “tempera-

ture” parameters described above except the units are given in electronvolts rather than Kelvin. “temp_{x,y,z}_ev” and “temp_ev” are accepted as aliases.

drift_p{x,y,z} - Specifies a momentum space offset in $kg\ ms^{-1}$ to the distribution function for this species. By default, the drift is zero. “drift_{x,y,z}” are also accepted as aliases.

offset - File offset. See below for details.

It is also possible to set initial conditions for a particle species using an external file. Instead of specifying the initial conditions mathematically in the input deck, you specify in quotation marks the filename of a simple binary file containing the information required. For more information on what is meant by a “simple binary file”, see “**Simple binary files**” (Section B).

external initial conditions

```
begin:species
    name = Electron
    number_density = 'Data/ic.dat'
    offset = 80000
    temperature_x = 'Data/ic.dat'
end:species
```

The sizes of the variables to be filled do not need to be provided: the code will continue reading until the given variable is filled. Note that ghost or guard cells should not be included in the file as they cannot be set this way.

An additional element is also introduced, the offset element. This is the offset in bytes from the start of the file to where the data should be read from. As a given line in the block executes, the file is opened, the file handle is moved to the point specified by the offset parameter, the data is read and the file is then closed. Therefore, unless the offset value is changed between data reading lines the same data will be read into all the variables. The data is read in as soon as a line is executed, and so it is perfectly possible to load data from a file and then modify the data using a mathematical expression.

The example block above is for 10,000 values at double precision, i.e. 8-bytes each. The density data is the first 80,000 bytes of “ic.dat”. Bytes 80,000 to 160,000 are the temperature_x data.

The file should be a simple binary file consisting of floating point numbers of the same precision as **num** in the core **EPOCH** code. For multidimensional arrays, the data is assumed to be written according to FORTRAN array ordering rules (ie. column-major order).

NOTE: The files that are expected by this block are SIMPLE BINARY files, NOT FORTRAN unformatted files. It is possible to read FORTRAN unformatted files using the offset element, but care must be taken!

The following entries are used for configuring the Delta-f method (see Section 4.8).

number_density_background, drift_p{x,y,z}_background,
temperature_{x,y,z}_background, temperature_{x,y,z}_background_ev,
temperature_background, temperature_background_ev - These all have the same meanings as the parameters listed above that don’t include the “_background” text, except that they specify the values to use for the background distribution function.

3.4.1 Particle migration between species

It is sometimes useful to separate particle species into separate energy bands and to migrate particles between species when they become more or less energetic. A method to achieve this functionality has been implemented. It is specified using two parameters to the “control” block:

use_migration - Logical flag which determines whether or not to use particle migration. The default is “F”.

migration_interval - The number of timesteps between each migration event. The default is 1 (migrate at every timestep).

The following parameters are added to the “species” block:

migrate - Logical flag which determines whether or not to consider this species for migration. The default is “F”.

promote_to - The name of the species to promote particles to.

demote_to - The name of the species to demote particles to.

promote_multiplier - The particle is promoted when its energy is greater than “promote_multiplier” times the local average. The default value is 1.

demote_multiplier - The particle is demoted when its energy is less than “demote_multiplier” times the local average. The default value is 1.

promote_number_density - The particle is only considered for promotion when the local number density is less than “promote_number_density”. The default value is the largest floating point number.

demote_number_density - The particle is only considered for demotion when the local number density is greater than “demote_number_density”. The default value is 0.

3.4.2 Ionisation

EPOCH now includes field ionisation which can be activated by defining ionisation energies and an electron for the ionising species. This is done via the species block using the following parameters:

ionisation_energies - This is an array of ionisation energies (in Joules) starting from the outermost shell. It expects to be given all energies down to the fully ionised ion; if the user wishes to exclude some inner shell ionisation for some reason they need to give this a very large number. Note that the ionisation model assumes that the outermost electron ionises first always, and that the orbitals are filled assuming ground state. When this parameter is specified it turns on ionisation modelling. If you wish to specify the values in Electron-Volts, add the “ev” multiplication factor (see Section 3.19.1).

ionisation_electron_species - Name of the electron species. This can be specified as an array in the event that the user wishes some levels to have a different electron species which can be handy for monitoring ionisation at specific levels. “electron” and “electron_species” are accepted as synonyms. Either one species for all levels, or one species for each species should be specified.

For example, ionising carbon species might appear in the input deck as:

```
begin:species
    charge = 0.0
    mass = 1837.2
    name = carbon
    ionisation_energies = (11.26*ev,24.38*ev,47.89*ev,64.49*ev,392.1*ev,490.0*ev)
    ionisation_electron_species = \
        (electron,electron,electron,fourth,electron,electron)
    number_density = den_gas
end:species

begin:species
    charge = -1.0
    mass = 1.0
    name = electron
    number_density = 0.0
end:species

begin:species
    charge = -1.0
    mass = 1.0
    name = fourth
    number_density = 0.0
end:species
```

Ionised states are created automatically and are named according to the ionising species name with a number appended. For example

```
begin:species
    name = Helium
    ionisation_energies = (24.6*ev,54.4*ev)
    dump = F
end:species
```

With this species block, the species named “Helium1” and “Helium2” are automatically created. These species will also inherit the “dump” parameter from their parent species, so in this example they will both have “dump = F” set. This behaviour can be overridden by explicitly adding a species block of the same name with a differing dumpmask. eg.

```
begin:species
    name = Helium1
    dump = T
end:species
```

Field ionisation consists of three distinct regimes; multiphoton in which ionisation is best described as absorption of multiple photons, tunnelling in which deformation of the atomic Coulomb potential is the dominant factor, and barrier suppression ionisation in which the electric field is strong enough for an electron to escape classically. It is possible to turn off multiphoton or barrier suppression ionisation through the input deck using the following control block parameters:

use_multiphoton - Logical flag which turns on modelling ionisation by multiple photon absorption. This should be set to “F” if there is no laser attached to a boundary as it relies on laser frequency. The default is “T”.

use_bsi - Logical flag which turns on barrier suppression ionisation correction to the tunnelling ionisation model for high intensity lasers. The default is “T”.

3.4.3 Species Boundaries

bc_x_min - Boundary condition to be applied to this species only on the lower x boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

bc_x_max - Boundary condition to be applied to this species only on the upper x boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

bc_y_min - Boundary condition to be applied to this species only on the lower y boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

bc_y_max - Boundary condition to be applied to this species only on the upper y boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

bc_z_min - Boundary condition to be applied to this species only on the lower z boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

bc_z_max - Boundary condition to be applied to this species only on the upper z boundary. Can be any normal boundary condition apart from periodic. If not specified then the global boundary condition is applied.

meet_injectors - Logical flag determining whether the background plasma should be extended to meet the point where particle injectors operate from. This means that plasma is loaded one particle shape function length outside the boundary. This means that it is possible to use an injector to “continue” an existing drifting plasma. **NOT COMPATIBLE WITH PERIODIC BOUNDARY CONDITIONS!**

3.4.4 Maxwell-Jüttner distribution function

As of version 4.15, **EPOCH** allows the user to request a Maxwell-Jüttner distribution rather than a Maxwellian distribution when sampling the particle momentum for a species.

use_maxwell_juttner - Logical flag determining whether to sample from the Maxwell-Jüttner distribution when loading the particle species. If “T” then Maxwell-Jüttner is used and if “F” Maxwellian is used. The default value is “F”.

fractional_tail_cutoff - The sampling is carried out using a rejection method with an arbitrary cut-off. This parameter takes a floating-point argument which specifies the fraction of maximum value at which the sampling should be cut off. Smaller values lead to distortion nearer the peak of the distribution but are faster to sample. Larger values lead to a better approximation of the distribution

function but are slower to sample. The default value is 0.0001.

If drifts are specified with the Maxwell-Jüttner distribution then the distribution is calculated in the rest frame and then Lorentz transformed to the specified drifting frame.

3.4.5 Arbitrary distribution functions

As of version 4.15, **EPOCH** also allows the user to request an arbitrary non-Maxwellian distribution function to use when sampling the particle momentum for a species. If combined with a specified drift then the distribution function is calculated first and the drift is applied to the resulting particles by Lorentz transform.

This feature does not at present work with the delta_f loader and is not available for particle injectors. It does work correctly with the moving window.

dist_fn - Specifies the functional form of the distribution function, normalised to have a maximum value of 1. The variables “px”, “py” and “pz” should be used to parameterise the x , y and z components of momentum. This may freely vary in space but temporal variation will be ignored since this is only evaluated at the start of the simulation.

dist_fn_p{x,y,z}_range - Comma separated pair of numbers to specify the range of momentum for $p_{\{x,y,z\}}$ in SI units. Should be of the form “⟨lower_range⟩, ⟨upper_range⟩”

If a range for a momentum direction is not specified then that momentum is assumed to be zero. It is up to the user to ensure that the range is large enough to correctly capture their desired distribution function. Sampling is by a simple rejection sampling and may be much slower than the existing Maxwellian sampler. **EPOCH** will print a warning if a large number of samples are needed to complete the sampling. If this occurs then you might need to reduce the range of momentum over which sampling is considered.

If the “dist_fn” key is supplied then any supplied temperature keys are ignored. An example of setting up a truncated power law distribution in p_x would be

```
begin:constant
dens = 10
v0 = 0.05 * c
vmax = 0.5 * c
p0 = v0 * me * (1.0 + 4.0 * x / x_max)
pmax = vmax * me
alpha = -2.0
end:constant

begin:species
name = Electron_pl
charge = -1
mass = 1.0
fraction = 0.5
number_density = dens
# Truncated power law distribution in px
dist_fn = exp(-p0 / px) * (px / p0)^alpha
dist_fn_px_range = (0, pmax)
end:species
```

3.5 laser block

Laser blocks attach an EM wave source to a boundary which is set as `simple_laser`.

```
laser block
begin:laser
    boundary = x_min
    id = 1
    intensity_w_cm2 = 1.0e15
    lambda = 1.06 * micron
    pol_angle = 0.0
    phase = 0.0
    t_profile = gauss(time, 40.0e-15, 40.0e-15)
    t_start = 0.0
    t_end = 80.0e-15
end:laser
```

As already mentioned in the discussion of laser boundaries in the boundaries block, lasers are attached to compatible boundaries here in the initial conditions deck.

boundary - The boundary on which to attach the laser. In 1D, the directions can be either `x_min` or `x_max`. “left” and “right” are accepted as synonyms. In 2D, `y_min` and `y_max` may also be specified. These have synonyms of “down” and “up”. Finally, 3D adds `z_min` and `z_max` with synonyms of “back” and “front”.

amp - The amplitude of the E field of the laser in V/m .

intensity - The intensity of the laser in W/m^2 . There is no need to specify both intensity and amp and the last specified in the block is the value used. It is mandatory to specify at least one. The amplitude of the laser is calculated from intensity using the formula `amp = sqrt(2*intensity*c/epsilon0)`. “irradiance” is accepted as a synonym.

intensity_w_cm2 - This is identical to the **intensity** parameter described above, except that the units are specified in W/cm^2 .

id - An id code for the laser. Used if you specify the laser time profile in the **EPOCH** source rather than in the input deck. Does not have to be unique, but all lasers with the same id will have the same time profile. This parameter is optional and is not used under normal conditions.

omega - Angular frequency (rad/s not Hz) for the laser.

frequency - Ordinary frequency (Hz not rad/s) for the laser.

lambda - Wavelength in a vacuum for the laser specified in m . If you want to specify in μm then you can multiply by the constant “micron”. One of **lambda** or **omega** (or **frequency**) is a required parameter.

polarisation_angle - Polarisation angle for the electric field of the laser in radians. This parameter is optional and has a value of zero by default. The angle is measured with respect to the right-hand triad of propagation direction, electric and magnetic fields. Although the 1D code has no y or z spatial axis, the fields still have y and z components. If the laser is on `x_min` then the default E field is in the y -direction and the B field is the z -direction. The polarisation angle is measured clockwise about the x -axis with zero along the E_y direction. If the laser is on `x_max` then the angle is

anti-clockwise. “pol_angle” is accepted as an alias.

Similarly, for propagation directions:

y_min - angle about y -axis, zero along z -axis

z_min - angle about z -axis, zero along x -axis

y_max - angle anti-clockwise about y -axis, zero along z -axis

z_max - angle anti-clockwise about z -axis, zero along x -axis

polarisation - This is identical to **polarisation_angle** with the angle specified in degrees rather than radians. If both are specified then the last one is used. “pol” is accepted as an alias.

phase - The phase profile of the laser wavefront given in radians. Phase may be a function of both space and time. The laser is driven using $\sin(\omega t + \phi)$ and **phase** is the ϕ parameter. There is zero phase shift applied by default.

profile - The spatial profile of the laser. This should be a spatial function not including any values in the direction normal to the boundary on which the laser is attached, and the expression will be evaluated at the boundary. It may also be time-dependant. The laser field is multiplied by the profile to give its final amplitude so the intention is to use a value between zero and one. By default it is a unit constant and therefore has no affect on the laser amplitude. This parameter is redundant in 1D and is only included for consistency with 2D and 3D versions of the code.

t_profile - Used to specify the time profile for the laser amplitude. Like **profile** the laser field is multiplied by this parameter but it is only a function of time and not space. In a similar manner to **profile**, it is best to use a value between zero and one. Setting values greater than one is possible but will cause the maximum laser intensity to grow beyond **amp**. In previous versions of **EPOCH**, the **profile** parameter was only a function of space and this parameter was used to impose time-dependance. Since **profile** can now vary in time, **t_profile** is no longer needed but it has been kept to facilitate backwards compatibility. It can also make input decks clearer if the time dependance is given separately. The default value of **t_profile** is just the real constant value of 1.0.

t_start - Start time for the laser in seconds. Can be set to the string “start” to start at the beginning of the simulation. This is the default value. When using this parameter, the laser start is hard. To get a soft start use the **t_profile** parameter to ramp the laser up to full strength.

t_end - End time for the laser in seconds, can be set to the string “end” to end at the end of the simulation. This is the default value. When using this parameter, the laser end is clipped straight to zero at $t > \text{t_end}$. To get a soft end use the **t_profile** parameter to ramp the laser down to zero.

If you add multiple laser blocks to the initial conditions file then the multiple lasers will be additively combined on the boundary.

In theory, any laser time profile required is possible, but the core FDTD solver for the EM fields in **EPOCH** produces spurious results if sudden changes in the field intensity occur. This is shown in Figures 2 and 3. The pulse shown in Figure 2 used a constant **t_profile** and used **t_end** to stop the laser after 8fs. Since the stopping time was not an exact multiple of the period, the result was to introduce spurious oscillations behind the pulse. If the laser had a finite phase shift so that the amplitude did not start at zero, a similar effect would be observed on the front of the pulse.

Figure 3 instead used a Gaussian window function with a characteristic width of 8fs as well as using **t_end** to introduce a hard cutoff. It can clearly be seen that there are no spurious oscillations and the wave packet propagates correctly, showing only some dispersive features.

There is no hard and fast rule as to how rapid the rise or fall for a laser can be, and the best advice is to simply test the problem and see whether any problems occur. If they do then there are various solutions. Essentially, the timestep must be reduced to the point where the sharp change in amplitude

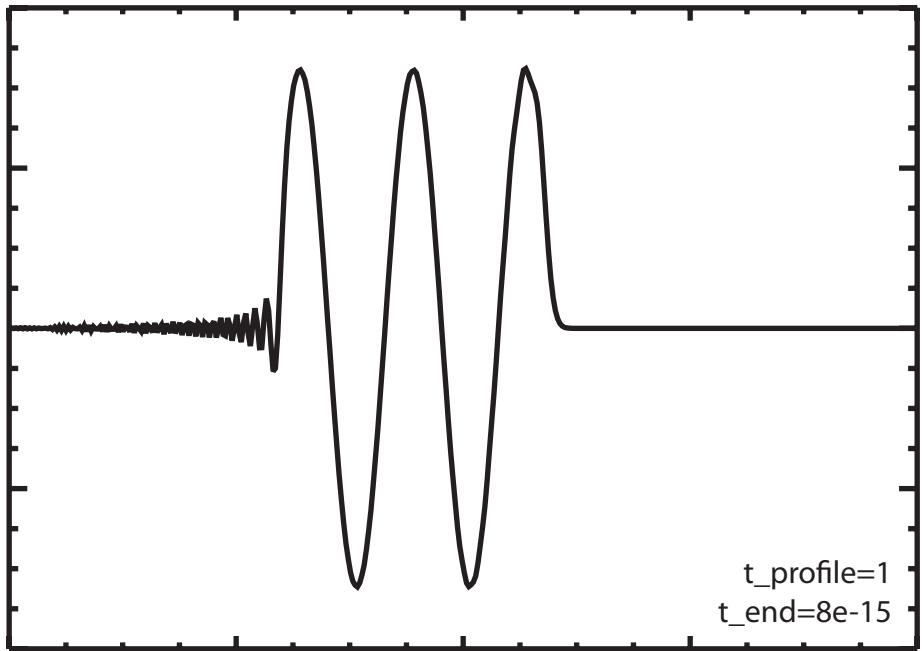


Figure 2: A laser pulse with a sharp cutoff shows numerical artefacts behind the pulse.

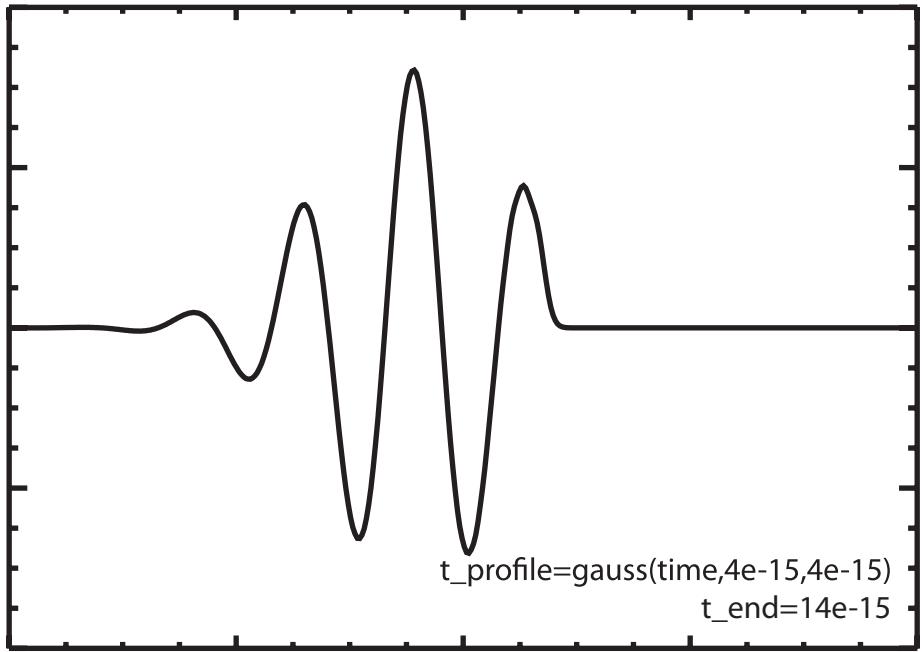


Figure 3: A laser pulse with a smooth temporal profile shows no artefacts.

can be accommodated. The best solution for this is to increase the spatial resolution (with a comparable increase in the number of pseudoparticles), thus causing the timestep to drop via the CFL condition.

This is computationally expensive, and so a cheaper option is simply to decrease the input.deck option `dt_multiplier`. This artificially decreases the timestep below the timestep calculated from the internal stability criteria and allows the resolution of sharp temporal gradients. This is an inferior solution since the FDTD scheme has increased error as the timestep is reduced from that for EM waves. **EPOCH** includes a high order field solver to attempt to reduce this.

3.6 **fields** block

The next type of block in the **EPOCH** input deck is the fields block. This allows you to specify the electric and magnetic fields at any point in the domain. An example block is shown below:

```

begin:fields
    ex = sin(pi * x / length_x)
    ey = cos(pi * x / length_x)
    ez = 0
    bx = 1.0
    by = -1.0
    bz = 0
end:fields

```

fields block

Once again, this is a very simple block needing only limited explanation. All field variables are accessible by name and can be read back using the appropriate commands from the maths parser (see Section 3.19). The possible parameters are as follows:

ex, ey, ez - The electric field vectors pointing in all three directions. The default value is zero.

bx, by, bz - The magnetic field vectors pointing in all three directions. The default value is zero.

offset - File offset. The field values may also be specified using a binary file in a similar way to that used for species variables. See Section 3.4 for more details.

Any valid maths parser expression can be used to set up the fields, and no check is made to ensure that the $\nabla \cdot B = 0$ is satisfied.

3.7 particles_from_file block

The **particles_from_file block** is similar in function to the **fields** block. It allows the loading of custom particle data from raw binary data files. An example usage of the block is shown below

```

begin:particles_from_file
    species = "electron"

    # Load mandatory data for 3D simulation
    x_data = "xdata.dat"
    y_data = "ydata.dat"
    z_data = "zdata.dat"
    w_data = "wdata.dat"

    # Load particle ids in 4 byte int format,
    # ignoring first 8 bytes of file
    # offset = 8
    # id4_data = "iddata.dat"
end:particles_from_file

```

particles_from_file block

Specifying a **particles_from_file** block for a species causes **EPOCH** to load the per-particle data from the specified files. Data files are assumed to be in order such that the first variable in each file will be attributed to one particle, the second variable in each file to a second electron, and so on. A target species to load to, as well as particle position and weight data (unless **-DPER_SPECIES_WEIGHT** has been set) must be supplied. With the exception of particle ID, any optional parameters which are left unspecified will be initialised to zero.

If the code has been compiled with **-DPARTICLE_ID** or **-DPARTICLE_ID4** then particle IDs may be loaded from a raw binary file of integers of either size 4 or size 8 regardless of the compile time flag

choice. If no particle ID data is supplied, IDs will be generated sequentially from 1.

All other data should be in the form of floating point numbers of the same precision as `num` in the core `EPOCH` code.

A `particles_from_file` block accepts the following parameters:

species - Name of the species to which the particles will be loaded. This is a mandatory parameter and the corresponding species block must be defined.

{x,y,z}_data - File containing particle position data in m . This data must be supplied, up to the dimensionality of the simulation.

w_data - File containing pseudoparticle weight, this is the number of real particles the pseudoparticle represents. This data must be supplied.

{px,py,pz}_data - File containing particle momentum data in $kg\ ms^{-1}$. The default value is zero.

id{4,8}_data - File containing particle IDs in either 4 or 8 byte unsigned integer representation.

offset - File offset. Number of bytes at the head of the file to be ignored, may be specified multiple times. see Section 3.4 for more details of behaviour.

3.8 window block

`EPOCH` can include an optional block which causes the simulation domain to operate as a moving window. At present, it is only possible to have the window moving at a speed parallel to the x direction, although the window does not have to start moving at $t = 0$. When the window moves, the code removes particles from the left hand edge of the domain and introduces new particles at the right hand edge. The new particles are placed by re-evaluating the species density, temperature and drift using the new time and spatial coordinates. The block looks like:

```
begin:window
  move_window = T
  window_v_x = 3.0e8
  window_start_time = 7.0e-13
  bc_x_min_after_move = simple_outflow
  bc_x_max_after_move = simple_outflow
end:window
```

move_window - Logical flag determining whether or not to move the window. If the window block is absent then this is the same as setting `move_window` to “F”.

window_v_x - The speed in m/s of the window. This may be a function of time.

window_start_time - The time in seconds at which the window should start moving.

window_stop_time - The time in seconds at which the window should stop moving.

bc_x_min_after_move - The boundary condition which should apply to the left boundary after the window has started moving. This is to allow the swapping of a laser boundary to a simple outflow boundary. Boundary codes are the same as when just specifying normal boundaries. If a boundary value isn't specified then it is assumed that the boundary isn't changed when the window starts moving. “xbc_left_after_move” is accepted as a synonym.

bc_x_max_after_move - The boundary condition which should apply to the right boundary after the window has started moving. “xbc_right_after_move” is accepted as a synonym.

bc_{y,z}_{min,max}_after_move - “y” and “z” versions of the previous two parameters. “ybc_down_after_move”, “ybc_up_after_move”, “zbc_back_after_move” and “zbc_front_after_move” are accepted as synonyms.

Because of how the moving window must work, there are some compatibility issues with certain features. In particular:

- Lasers attached to an X boundary which remain in place after the window moves, or attached to Y or Z boundaries:

The laser will behave as though it is attached to the window itself: for Y or Z boundaries with spatial variations this may not give the expected result.

For X boundaries, the moving emitter will result in a form of numerical Doppler shifting. In addition to this the boundary used to drive the field will shift discontinuously, yielding noisy and erratic changes in the electromagnetic field.

- Injectors attached to an X boundary will not work. Those on a Y or Z boundary may appear to work, but the rates will be incorrect.
- CPML boundary conditions:

In X these cannot work as they rely on time-history which is simply missing.

On Y or Z boundaries they will approximately work, but the history will be truncated and so they will generally require more tuning. We can't help with this in general.

- Load of particles from file is not supported since it can't be made to work in general.

3.9 **output** block

Output in **EPOCH** is handled using the custom designed SDF file format ([Self Describing Format](#)). A detailed specification of this format is available elsewhere, although this is only of interest to developers wishing to write new libraries. **EPOCH** comes with readers for ITT IDL, LLNL VisIt, Mathworks MatLab and Python. The IDL reader is also compatible with the open source GDL tool.

There are two styles of output block supported by **EPOCH**. The first style, which will be referred to as the “traditional” style, is the method that has been supported by **EPOCH** since its inception. With this method, a single output block governs all the output dumps which are to be performed. There are a few levels of output which give some small amount of flexibility over what gets dumped but these do not allow for a very fine-grained control.

In version 4.0 of **EPOCH**, a new style was introduced in which multiple named output blocks may be specified allowing for much greater flexibility. The existence of a “name” parameter is what determines that an output block is the new style rather than the traditional style.

Most of the parameters are shared by both styles. The following sections document the traditional style of output block and any differences between the two styles are described in Section 3.9.9.

What the code should output and when it should output it is specified in the “output” block of the input deck. An example output block is shown below:

```

begin:output
  # If use_offset_grid is true then the code dumps a grid which
  # displays positions relative to the left hand edge of the window
  use_offset_grid = F
  # Number of timesteps between output dumps
  dt_snapshot = 1.0e-14
  # Number of dt_snapshot between full dumps
  full_dump_every = 10
  restart_dump_every = -1
  force_final_to_be_restartable = T

  # Properties at particle positions
  particles = never
  px = never
  py = never
  pz = never
  vx = never
  vy = never
  vz = never
  charge = never
  mass = never
  particle_weight = never

  # Properties on grid
  grid = always
  ex = always
  ey = always
  ez = always
  bx = always
  by = always
  bz = always
  jx = always
  jy = always
  average_particle_energy = always + species
  mass_density = never + species
  charge_density = always
  number_density = always + species
  temperature = always + species

  distribution_functions = always
  particle_probes = never
end:output

```

There are three types of output dump in **EPOCH** which are used for different purposes. These types are:

- normal - The most frequent type of output dump in **EPOCH** is a normal dump.
- full - A full dump is usually written every 10 or so normal dumps. A full dump contains all the data that a normal dump contains and should also contain any information which is needed only infrequently, whether this is the full particle information or a large distribution function. It is possible to turn off full dumps completely.

- **restart** - A restart dump is a dump where the code guarantees to write enough data to allow the code to restart from the output. Output dumps are guaranteed to contain all the information in a normal dump and, if they coincide with the timing for a full dump, will also contain the full dump information.

Information will never be written into a file twice, even if two conditions for it being written are satisfied (i.e even if px should be dumped both because it is a full dump and a restart dump, px will only be written once).

Note that these dump levels only really make sense for the traditional style of output block and are not really required when the new style is used.

3.9.1 Dumpmask

When specifying which type of output dump to write a variable to there are eight options which can be specified for each variable and can be combined by addition. Some combinations make no sense but are formally valid. The first four options specify at which output types the variable is to be dumped:

never - If the variable is not a required restart variable then it will never be written. If it is a required restart variable then it will be written only at restart dumps.

full - This variable will be written at full dumps only.

always - This variable will be written at full, normal and restart dumps.

restart - This variable will be written at restart dumps only. Note that variables required for restarting the code are always written to restart dumps. This flag is to enable the writing of additional variables into such dump files.

For grid variables derived from summing over particles (ie. “average_particle_energy”, “mass_density”, “charge_density”, “number_density”, “temperature”) the following two parameters also apply.

species - The derived variable should be output on a species by species basis. It is combined with a dumpmask code by addition as in:

`charge_density = always + species.`

no_sum - The output for this derived variable should not be summed over all species. By default, derived variables are summed over all species. If you don’t want to include this sum, you must use the “no_sum” flag. It is combined with a dumpmask code by addition as in:

`charge_density = always + species + no_sum.`

Most grid variables may be averaged over time. A more detailed description of this is given in Section 3.9.7. Data averaging is specified using the following dumpmask parameters.

average - The output for this variable should be averaged over time. The time span over which the variable will be averaged is controlled using flags described in Section 3.9.2.

snapshot - By default, the “average” parameter replaces the variable with an averaged version of the data. Adding this flag specifies that the non-averaged variable should also be dumped to file.

When applied to a variable, these codes are referred to as a **dumpmask**.

3.9.2 Directives

The first set of options control the type and frequency of output dumps. They are used as follows

disabled - Logical flag. If this is set to “T” then the block is ignored and never generates any output. The default value is “F”.

dt_snapshot - Sets the interval between normal output dumps in simulation seconds. Setting zero or negative means that the code will not output based on this condition. The code does NOT guarantee that outputs will be exactly **dt_snapshot** apart, what is guaranteed is that the next output will be after the first iteration which takes the simulation to a time $\geq \text{dt_snapshot}$ from the last output. As with other variables which specify a unit of time, it can be specified in more convenient unit by using a multiplication factor (see Section 3.19.1). For example, “**dt_snapshot = 5 * femto**” will set it to be 5 femtoseconds. The default value is a large number which will never trigger an output.

nstep_snapshot - Sets the number of timesteps between normal output dumps. Setting zero or negative means that the code will not output based on this condition. If **dt_snapshot** is also specified then both conditions are considered and output will be generated when either condition is met. The default value is a large integer which will never trigger an output.

full_dump_every - The number of normal output dumps between full output dumps. Setting to zero makes every dump a full dump. Setting to a negative number stops the code from producing any full dumps. This is the default.

restart_dump_every - The number of normal output dumps between restart dumps. Setting to zero makes every dump a restart dump. Setting to a negative number stops the code from producing any restart dumps. This is the default.

force_first_to_be_restartable - Logical flag which determines whether the file written at time zero is a restart dump. The default value is “F”.

force_last_to_be_restartable - Force the code to override other output settings and make the last output dump it writes be a restart dump. Any internal condition which causes the code to terminate will make the code write a restart dump, but code crashes or scheduler terminations will not cause the code to write a restart dump. “**force_final_to_be_restartable**” is accepted as a synonym. The default value is “T”.

dump_first - Logical flag which determines whether to write an output file immediately after initialising the simulation. The default is “T”.

dump_last - Logical flag which determines whether to write an output file just before ending the simulation. The default is “T” if an output block exists in the input deck and “F” otherwise. “**dump_final**” is accepted as a synonym.

time_start - Floating point parameter which specifies the simulation time at which to start considering output for the block. Note that if “**dump_first**” or “**dump_last**” are set to true for this block then dumps will occur at the first or last timestep regardless of the value of the **time_start** parameter. This also applies to the three following parameters. The default value is 0.

time_stop - Floating point parameter which specifies the simulation time at which to stop considering output for the block. The default value is the largest possible float.

nstep_start - Integer parameter which specifies the step number at which to start considering output for the block. The default value is 0.

nstep_stop - Integer parameter which specifies the step number at which to stop considering output for the block. The default value is the largest possible integer.

walltime_start - Floating point parameter which specifies the elapsed walltime in seconds at which to start considering output for the block. Note that if “dump_first” or “dump_last” are set to true for this block then dumps will occur at the first or last timestep regardless of the value of the **walltime_start** parameter. The default value is 0.

walltime_stop - Floating point parameter which specifies the elapsed walltime in seconds at which to stop considering output for the block. The default value is the largest possible float.

dump_cycle - If this is set to a positive integer then the output file number will be reset to zero after the specified cycle number is reached. eg. if “dump_cycle = 2” then the sequence of output dumps will be 0000.sdf, 0001.sdf, 0002.sdf, 0000.sdf, 0001.sdf, etc. The default is 0, so dump cycling never occurs.

dump_cycle_first_index - If this is set to a positive integer then the value is used as the first index to use when cycling output dumps due to the “dump_cycle” parameter. For example, if “dump_cycle = 2” and “dump_cycle_first_index = 1” then the sequence of output dumps will be 0000.sdf, 0001.sdf, 0002.sdf, 0001.sdf, 0002.sdf, 0001.sdf, etc. The default is 0.

dump_source_code - **EPOCH** has the ability to write its own source code into restart dumps. This is generated at compile time and embedded into the binary and so is guaranteed to match that corresponding to the running code. **EPOCH** comes with a script called **unpack_source_from_restart** which can be used to unpack the source code from a restart dump. To use this script, just type

unpack_source_from_restart <sdf_filename>

at the command-line. If this logical flag is set to false then the feature will be disabled. The default value is “T”.

dump_input_decks - If this logical flag is set to true then a copy of the input decks for the currently running simulation is written into the restart dumps. The default value is “T”.

dt_average - When averaged variables are being output to file, this parameter specifies the simulation time period over which averaging is to occur. “averaging_period” is accepted as a synonym.

nstep_average - When averaged variables are being output to file, this parameter specifies the number of time steps over which averaging is to occur. “min_cycles_per_average” is accepted as a synonym. If both **dt_average** and **nstep_average** are specified, the code will use the one which gives the longest simulation time-span.

use_offset_grid - When using moving windows some visualisation programs (notably VisIt) show the motion of the window by moving the visualisation window rather than by changing the x-axis. Setting this option to “T” causes the code to write another grid which always gives the offset relative to the left hand edge of the window rather than the true origin. Performs no function when not using the moving window. The default value is “F”.

filesystem - String parameter. Some filesystems can be unreliable when performing parallel I/O. Often this is fixable by prefixing the filename with ‘ufs’ or ‘nfs’. This parameter supplies the prefix to be used. The default value is an empty string.

file_prefix - Although this parameter is supported by the traditional style of output block, its primary purpose is for use with multiple output blocks so it is documented in Section 3.9.9.

A few additional parameters have been added for use with the new style of output block. These are documented in Section 3.9.9.

3.9.3 Particle Variables

The next set are per particle properties. If you wish to plot these according to their spatial positions, you must include the “particle_grid” in your output variables. All entries have a default dumpmask of “never”.

particle_grid - Requests the output of particle positions. This is a restart variable. No particle variables can be plotted in VisIt unless this is dumped. If any particle variables are written then the “particle_grid” is automatically written unless “particle_grid = never” is specified. The synonym “particles” may also be used.

px, py, pz - The dumpmasks for the particle momenta. Restart variable.

vx, vy, vz - The dumpmasks for the particle velocities.

charge - The dumpmask for the charge of a given particle. This has no effect if the code is not compiled with the flag “-DPER_PARTICLE_CHARGE_MASS” (see Section 2.4).

mass - The dumpmask for the mass of a given particles. This has no effect if the code is not compiled with the flag “-DPER_PARTICLE_CHARGE_MASS” (see Section 2.4). The synonym “rest_mass” may also be used.

particle_weight - The dumpmask for the weighting function which describes how many real particles each pseudoparticle represents. Restart variable. The synonym “weight” may also be used.

ejected_particles - If requested then all the particles which have left the simulation domain since the last output dump of this type are included in the output. The list of ejected particles is treated as if it were a separate species and the particle variables which get written are requested using the other particle variable flags (ie. “particle_grid”, etc). Once the data has been written, the ejected particle lists are reset and will accumulate particles until the next requested output dump.

particle_energy - The dumpmask for per-particle kinetic energy.

relativistic_mass - The dumpmask for per-particle relativistic mass (ie. not rest mass).

gamma - The dumpmask for per-particle relativistic gamma (ie. $[1 - (v/c)^2]^{-1/2}$).

optical_depth - The dumpmask for per-particle optical depth. Restart variable. This option is only supplied for debugging purposes and should not be required by most users.

trident_optical_depth - The dumpmask for per-particle optical depth used by the Trident model. Restart variable. This option is only supplied for debugging purposes and should not be required by most users.

bremsstrahlung_optical_depth - The dumpmask for per-particle optical depth used by the bremsstrahlung radiation model. Restart variable. This option is only supplied for debugging purposes and should not be required by most users.

qed_energy - The dumpmask for per-particle QED-related particle energy. Restart variable. This option is only supplied for debugging purposes and should not be required by most users.

work_{x,y,z} - The dumpmask for the work exerted by the fields on each particle during the last time step. The work is divided into its three spatial components. The output is in numbers of mc^2 corresponding to the particle's γ -factor. Requires compiler flag "WORK_DONE_INTEGRATED".

work_{x,y,z}_total - Same as above, but the work is integrated over the entire simulation duration. The sum of all three components equals the particle's γ -factor. Requires compiler flag "WORK_DONE_INTEGRATED".

id - Global particle ID. See below for details.

Particle IDs are useful if you want to track the progress of each particle throughout the simulation. Since they increase the size of each particle data structure, they are disabled by default and must be enabled using a compiler flag. The "PARTICLE_ID" flag will use an 8-byte integer to represent the ID and "PARTICLE_ID4" uses a 4-byte integer. They are written to file using the "id" flag.

Note: In the current implementation, the particle IDs are passed between processors and written to file using REAL numbers. This means that in double precision the maximum particle ID is $2^{53} \sim 10^{16}$. This should be ample for the foreseeable future. However, if the code is compiled for single precision then the maximum ID is $2^{24} = 16777216$. Probably not big enough.

3.9.4 Grid Variables

The next set of parameters specify properties which are defined on a regular cartesian mesh. All entries have a default dumpmask of "never".

grid - The dumpmask for the Cartesian grid which defines the locations of the grid variables. No grid variables can be plotted in VisIt unless this variable is output. If any grid variables are written then the "grid" is automatically written unless "grid = never" is specified. The synonym "field_grid" may also be used.

ex, ey, ez - The electric field vectors pointing in all three directions. Restart variables.

bx, by, bz - The magnetic field vectors pointing in all three directions. Restart variables. In 1D bx is a trivial variable because of the Solenoidal condition. It is included simply for symmetry with higher dimension codes.

jx, jy, jz - The current densities pointing in all three directions. Restart variables.

3.9.5 Derived Variables

The final set of parameters specify properties which are not variables used in the code but are derived from them. The first six variables are derived by summing properties of all the particles in each grid cell. The resulting quantities are defined on the regular cartesian mesh used for grid variables. All entries have a default dumpmask of "never".

average_particle_energy - Mean kinetic energy on grid in J . Can have species dumpmask. “ekbar” is accepted as an alias.

particle_energy_flux - Mean kinetic energy flux in each direction on the grid in W/m^2 . Can have species dumpmask. “ekflux” is accepted as an alias.

mass_density - Mass density on grid in kg/m^3 . Can have species dumpmask.

charge_density - Charge density on grid in C/m^3 . Can have species dumpmask.

number_density - Number density on grid in m^{-3} . Can have species dumpmask.

particles_per_cell - Number of particles per cell. Can have species dumpmask. The synonym “ppc” may also be used.

average_weight - Average of weight of the particles in each cell. Can have species dumpmask.

average_p{x,y,z} - Average momentum of the particles in each cell. Can have species dumpmask.

temperature - Isotropic temperature on grid in K . The synonym “temp” may also be used. Can have species dumpmask.

temperature_{x,y,z} - The temperature in each of the {x,y,z} directions, respectively, in K . The synonyms “temp_{x,y,z}” and “t{x,y,z}” may also be used. Can have species dumpmask.

poynting_flux - Poynting flux in each direction in W/m^2 . “poynt_flux” is accepted as an alias.

3.9.6 Other Variables

distribution_functions - Dumpmask for outputting distribution functions specified in the input deck. Each individual distribution function can have its own dumpmask and these will be applied after the value of “distribution_functions” has been considered. For example, if the output block contains “distribution_functions = full” and the dist_fn block (see Section 3.11) contains “dumpmask = always” then the distribution function will only be output at full dumps.

particle_probes - Dumpmask for outputting particle probes specified in the input deck. Each individual particle probe can have its own dumpmask and these will be applied after the value of “particle_probes” has been considered. For example, if the output block contains “particle_probes = always” and the dist_fn block contains “dumpmask = full” then the particle probe will only be output at full dumps.

absorption - This is a two-valued output variable. It accepts a dumpmask in the same manner as other output variables. When selected, two numbers will be calculated and written to file:

1. “Absorption/Laser_enTotal” - The total amount of energy injected into the simulation by laser boundaries.
2. “Absorption/Abs_frac” - The fraction of the total laser energy being absorbed by the open boundaries.

total_energy_sum - This is also a two-valued output variable. It accepts a dumpmask in the same manner as other output variables. When selected, the following two numbers will be calculated and written to file:

1. “Total Particle Energy in Simulation (J)”
2. “Total Field Energy in Simulation (J)”

3.9.7 Data Averaging

EPOCH can accumulate an average value for field variables to be written to output dumps. These may be requested by using the “average” keyword when specifying a dump variable. The non-averaged variable will still be written to restart dumps where required for restarting the code but not full or normal dumps. If you also want the non-averaged variable to be written then you can add the “snapshot” option.

The period of time over which averaging occurs can be specified using the “dt_average” keyword. Alternatively, you may specify the number of cycles over which to perform the averaging using the “nstep_average” keyword. If both “dt_average” and “nstep_average” are specified then the averaging will be performed over the longest of the two intervals.

Note that previous versions of the code would alter the time step to ensure that there were enough cycles between output dumps to satisfy the “nstep_average” parameter. However, since it affects the accuracy of the result, this is no longer the case and only a warning message is issued.

The following shows an example use of averaging in the output block.

```
begin:output
  dt_snapshot = 1.0e-15
  full_dump_every = 10
  dt_average = 1.0e-17

  charge_density = always + average + snapshot
  mass_density = full + average + snapshot
  average_particle_energy = full + average
end:output
```

With this configuration, “charge_density” will be written in both normal and averaged form at normal, full and restart dumps. “mass_density” will be written in both forms at full dumps. Only the average value of “average_particle_energy” will be written at full dumps.

Only field and derived variables can be averaged currently in **EPOCH**. Particle properties, distribution functions and particle probes cannot currently be averaged.

3.9.8 Single-precision output

By default, **EPOCH** is compiled and run using double precision arithmetic. This is the only method which has been fully tested and the method that we recommend to other users of the code. However, this also means that data files can get very large.

To avoid this problem, it is possible to run the code in double precision but convert the data to single precision when writing to disk. This is done by adding the “single” field to the dumpmask of an output variable. It can be specified on a per-variable basis.

```
begin:output
  dt_snapshot = 8 * femto

  grid = always
  ex = always
  ey = always + single
end:output
```

In this example, the grid variable “ex” will be written as a double precision array and “ey” will be converted to single precision.

Dumping variable averages adds an extra field variable for each average requested. These take up memory during runtime but do not influence the simulation behaviour in any way. For this reason, if the average is to be written out in single precision then it may as well be stored in a single precision variable. This behaviour can be requested using the “average_single” dumpmask flag.

3.9.9 Multiple output blocks

In more recent versions of **EPOCH**, it is now possible to have multiple “output” blocks in the input deck, each with their own “dt_snapshot” or “nstep_snapshot” and their own set of output variables.

The syntax remains the same as the original “output” block syntax with the addition of “name” and “restartable” fields.

The “name” field specifies the file name to use for the output list. Each time **EPOCH** generates an output dump, it writes an entry into the file “`<name>.visit`”. This can be used to find all the output dumps of a specific output block. It is named with a “.visit” suffix to enable its use as a file grouping list in the VisIt data analysis tool, but it is just a plain text file so it can equally be used by any other program.

If two output blocks are written at the same time, the output will be combined into a single file.

The “restartable” field specifies that the output block should generate output dumps containing all the information necessary to restart a simulation.

The “full” and “restart” dumpmask flags do not make much sense in the new style of output block and these have been deprecated.

The following parameters are supported by the new style of output block in addition to those for the traditional style:

name - Identifies the output block with a name which is required when multiple output blocks are used.

restartable - Specifies whether or not the output for this block is a restartable dump.

dump_at_times - Floating point parameter which specifies a set of simulation times at which to write the current output block. This can only be used with named output blocks. The values are given as a comma separated list. eg. “`dump_at_times = 0, 0.15, 1.1`”. The name “times_dump” is accepted as a synonym. By default the list is empty.

dump_at_nsteps - Integer parameter which specifies a set of step numbers at which to write the current output block. This can only be used with named output blocks. The values are given as a comma separated list. eg. “`dump_at_nsteps = 5, 11, 15`”. The name “nsteps_dump” is accepted as a synonym. By default the list is empty.

dump_at_walltimes - Floating point parameter which specifies a set of elapsed walltimes at which to write the current output block. This can only be used with named output blocks. The values are given as a comma separated list. eg. “`dump_at_walltimes = 10, 100.1, 250.5`”. These times are the total elapsed time in seconds since the start of the simulation. Note that if the simulation has been restarted then the total elapsed time will include the accumulated walltime of all previous runs that were used to produce the restart dump. The name “walltimes_dump” is accepted as a synonym. By default the list is empty.

walltime_interval - Floating point parameter which specifies the interval between output dumps in elapsed walltime seconds. Setting zero or negative means that the code will not output based on this condition. The default value is -1.0.

file_prefix - String parameter. It is sometimes useful to distinguish between dumps generated by the different output blocks. This parameter allows the user to supply a file prefix to be prepended to all dumps generated by the current output block. See below for further details. The default value is an empty string.

rolling_restart - Logical flag. If set to “T”, this sets the parameters required for performing rolling restarts on the current block. It is a shorthand for setting the following flags: “dump_cycle = 1”, “restartable = T” and “file_prefix = roll”. With rolling restarts enabled the first file will be named “roll0000.sdf” and the second will be “roll0001.sdf”. The third dump will again be named “roll0000.sdf”, overwriting the first one. In this way, restart dumps can be generated throughout the duration of the simulation whilst limiting the amount of disk space used.

The following parameters cannot be used in conjunction with the new style of output block:

- full_dump_every
- restart_dump_every
- force_first_to_be_restartable
- force_last_to_be_restartable
- use_offset_grid

The “file_prefix” parameter warrants some further discussion. This parameter prepends the given prefix to all files generated by the output block in which it is specified. For example, if “file_prefix = aa” is set then files generated by the output block will be named “aa0000.sdf”, etc. instead of just “0000.sdf”.

This also allows different variables to different files at the same time step. For example, here are two output blocks which do not use file prefixes:

```
begin:output
  name = o1
  nstep_snapshot = 1
  charge_density = always
end:output

begin:output
  name = o2
  dump_at_nsteps = 10
  restartable = T
end:output
```

With this input deck, we want to have the “charge_density” derived variable at every snapshot and then periodically write a restart dump. The problem is that the dump file “0010.sdf” contains both the restart information and the “charge_density” variable. At the end of the run we can’t just delete the large restart dumps without losing the smaller variables at that time step.

With the new version we would add a prefix to one or both blocks:

```
begin:output
  name = o1
  file_prefix = small
  nstep_snapshot = 1
```

```

charge_density = always
end:output

begin:output
  name = o2
  nstep_snapshot = 10
  restartable = T
end:output

```

Now the “charge_density” will be written to “small0000.sdf”, etc. At step 10, two files will be written: “small0010.sdf” containing just the charge_density and “0000.sdf” containing all the restart variables.

Note that some care must be taken, since if the same variable is in the output block for multiple file prefixes then multiple copies will be written to file. This obviously uses more disk space and is more time consuming than necessary.

It should also be noted that if multiple output blocks use the same file stem then their output will be combined. eg:

```

begin:output
  name = o1
  file_prefix = a
  dump_at_nsteps = 2,4
  ex = always
end:output

begin:output
  name = o2
  file_prefix = a
  dump_at_nsteps = 3,4
  ey = always
end:output

begin:output
  name = o3
  file_prefix = b
  dump_at_nsteps = 4
  ez = always
end:output

```

In this example, at step 2 a0000.sdf contains ex, step 3 a0001.sdf contains ey, step 4 a0002.sdf contains ex, ey and b0000.sdf contains ez.

3.10 **output_global** block

With the introduction of multiple output blocks, there are now a few parameters that only make sense to be applied globally across all output blocks. To accommodate this, a new block named “output_global” has been added. Most of the parameters accepted by this block have the same meaning as those in the “output” block except that they are applied to all “output” blocks.

The parameters that can be specified in the “output_global” block are as follows:

force_first_to_be_restartable - Logical flag which determines whether the file written at time zero is a restart dump. The default value is “F”.

force_last_to_be_restartable - Force the code to override other output settings and make the last output dump it writes be a restart dump. Any internal condition which causes the code to terminate will make the code write a restart dump, but code crashes or scheduler terminations will not cause the code to write a restart dump. “force_final_to_be_restartable” is accepted as a synonym. The default value is “T”.

dump_first - Logical flag which determines whether to write an output file immediately after initialising the simulation. The default is “F”.

dump_last - Logical flag which determines whether to write an output file just before ending the simulation. The default is “T” if an output block exists in the input deck and “F” otherwise. “dump_final” is accepted as a synonym.

time_start - Floating point parameter which specifies the simulation time at which to start considering output for all output blocks. Note that if “dump_first” or “dump_last” are set to true for any block then dumps will occur at the first or last timestep regardless of the value of this parameter. This also applies to the three following parameters. The default value is 0.

time_stop - Floating point parameter which specifies the simulation time at which to stop considering output for all output blocks. The default value is the largest possible float.

nstep_start - Integer parameter which specifies the step number at which to start considering output for the block. The default value is 0.

nstep_stop - Integer parameter which specifies the step number at which to stop considering output for the block. The default value is the largest possible integer.

walltime_start - Floating point parameter which specifies the elapsed walltime in seconds at which to start considering output for all output blocks. Note that if “dump_first” or “dump_last” are set to true for any blocks then dumps will occur at the first or last timestep regardless of the value of the **walltime_start** parameter. The default value is 0.

walltime_stop - Floating point parameter which specifies the elapsed walltime in seconds at which to stop considering output all output blocks. The default value is the largest possible float.

sdf_buffer_size - Integer parameter. When writing particle data to an SDF file, the data is first transferred into an output buffer. The size of this buffer can have a big impact on the overall speed of writing dump files. This parameter allows the size of the buffer to be specified in bytes. The default value is 67108864 (64 MB).

filesystem - String parameter. Some filesystems can be unreliable when performing parallel I/O. Often this is fixable by prefixing the filename with ‘ufs’ or ‘nfs’. This parameter supplies the prefix to be used. The default value is an empty string.

use_offset_grid - When using moving windows some visualisation programs (notably VisIt) show the motion of the window by moving the visualisation window rather than by changing the x-axis. Setting this option to “T” causes the code to write another grid which always gives the offset relative to the left hand edge of the window rather than the true origin. Performs no function when not using the moving window. The default value is “F”.

dump_first_after_restart - Logical flag to enable a dump to occur immediately after restart. In the past, a “dump_first” flag in the output block would cause an output dump immediately after

restarting. Since this is rarely the desired behaviour, the flag is now ignored when restarting. To force a dump to occur immediately after restart, set “dump_first_after_restart = T” in the output block. The default value is “F”.

3.11 dist_fn block

Sometimes it is useful to reconstruct part of the full phase space for one or more particle species. This functionality is provided through a **dist_fn** block. The distribution function is integrated over all dimensions which are not axes of the distribution function.

Calculating distribution functions requires some degree of integration of data leading to various possible ways of normalising the resulting distribution function. In **EPOCH**, distribution functions are normalised so that the value at every point of the distribution function is the number of particles within that cell of the distribution function, ignoring all phase space directions which are not considered as an axis of the distribution function. Summing the distribution function should give the total number of real particles (as opposed to computational pseudoparticles) in the simulation.

An example **dist_fn** block is given below:

```
----- dist_fn block -----
begin:dist_fn
    name = x_px
    ndims = 2
    dumpmask = always

    direction1 = dir_x
    direction2 = dir_px

    # Range is ignored for spatial coordinates
    range1 = (1, 1)
    range2 = (-50.0e-20, 50.0e-20)

    # Resolution is ignored for spatial coordinates
    resolution1 = 1
    resolution2 = 5000

    restrict_py = (-3.0e-20, 3.0e-20)

    include_species:Electron
    include_species:Carbon
end:dist_fn
```

name - The name of the distribution function when it is output. This name is appended with the name of each species for which the data is output and so, for example, when applied to a species named carbon the output is called **x_px_Carbon**. The Cartesian grid which describes the axes of the distribution function would then be called **grid_x_px_Carbon**.

ndims - The number of dimensions in this phase space reconstruction. Due to difficulties in visualising data in more than three dimensions, this is restricted to being 1, 2 or 3.

dumpmask - Determines which output dumps will include this distribution function. The dumpmask has the same semantics as those used by variables in the “output” block, described in Section 3.9. The dumpmask from “distribution_functions” in the output block is applied first and then this one is applied afterwards. For example, if the dist_fn block contains “dumpmask = full” and the output block contains “distribution_functions = always” then this distribution function will be only be

dumped at full dumps. The default dumpmask is “always”.

directionn - This is the phase space to sample along axis **n**. This can be any one of: dir_x, dir_y, dir_z, dir_px, dir_py, dir_pz, dir_energy, dir_gamma_minus_one, dir_xy_angle, dir_yz_angle, dir_zx_angle, dir_mod_p with spatial codes only being available in dimensionalities of the code which have that direction. Therefore dir_z does not exist in EPOCH1D or EPOCH2D and dir_y does not exist in EPOCH1D. “dir_en” and “dir_gamma_m1” are accepted as aliases for “dir_energy” and “dir_gamma_minus_one” respectively.

The flags “dir_xy_angle”, “dir_yz_angle” and “dir_zx_angle” calculate the distribution of particle momentum directions in the X-Y, Y-Z and Z-X planes.

rangen - The range between which this axis should run. This is in the form of (minimum, maximum). Any particle which exceeds the range is ignored. For momentum directions this parameter is specified in $kg\ ms^{-1}$. If the range of a momentum direction is set so that the maximum and the minimum are equal then the code will automatically set the range to exactly span the range of particle momenta at the point of writing the dump.

NOTE: Currently the range parameters have to be simple floating point numbers and NOT maths parser expressions.

resolutionn - The number of gridpoints in a given direction. This is ignored for spatial dimensions where the resolution is always the same as the resolution of the underlying simulation.

include_species - Specifies a species which should be included in the output. This is useful since it is rare that momentum limits are appropriate for both electrons and ions, so usually for a given dist_fn block only electrons or ions are considered. It is possible to have two dist_fn blocks with the same name but different ranges and different include_species settings produce the effect of a single diagnostic for all species in the output file.

output_deltaf - If set to “T”, the particle weights used in calculating the distribution function is adjusted by subtracting the Delta-f distribution function for the particle species. The default value is “F”.

restrict_<direction> - “<direction>” can be any one of “x, y, z, px, py, pz, energy, gamma_minus_one, xy_angle, yz_angle, zx_angle, mod_p”. Restrictions are specified in the same way as ranges, but have a subtly different behaviour. Ranges specify the range of a visible axis on the resulting distribution function, whereas restrictions allow you to specify minimum and maximum values for each spatial and momentum direction and use only particles which fall within this range when calculating the distribution function. Restrictions can be specified even for properties which are not being used as axes. It is possible to set a restriction that is more restrictive than the range applied. This is not trapped as an error and such parts of the distribution function are guaranteed to be empty. The available spatial restrictions depend on the dimensionality of the code. Therefore, attempting to set restrict_z in EPOCH1D will produce a warning.

At present, the code to calculate the distribution functions has one limitation: it ignores particle shape functions when calculating properties on the spatial axis, meaning that the result is less smooth than normal properties from the code.

3.12 probe block

Sometimes it is useful to consider all the properties of particle which pass through a point/line/plane (depending on dimension) in the simulation. To allow this, it is possible to specify one or more *Particle*

Probe blocks in the input deck. These record copies of all particles which cross a point/line/plane in a given direction which meet minimum and maximum kinetic energy criteria and output the particle properties into the normal output files. Particle probes record the positions, momenta and weight of all particles passing through the plane. To use particle probes, the code must not have been compiled with the **-DNO_PARTICLE_PROBES** compiler option. This is a fairly heavyweight diagnostic since each particle position must be tested from within the particle push. The code will run faster if it is not compiled in.

The probe is specified in terms of a point in the plane and the normal vector to the plane which is to be monitored. Particles are only recorded if they cross the plane in the direction given by the normal vector. If you want to record particles travelling in both directions then use two particle probes, one with an opposite signed normal vector to the other.

```
begin:probe
    name = electron_back_probe

    point = (50.0e-6, -50.0e-6)
    normal = (1.0, 0.0)

    ek_min = 0.0
    ek_max = -1.0
    include_species:s1

    dumpmask = always
end:probe
```

name - The name that the probe should have in output dumps. Output variables are then named this as a prefix. For example, the block shown above will result in the name **electron_back_probe_px** for the x momentum. The particle positions would just be called **electron_back_probe**.

point - An arbitrary point in the plane of the probe.

normal - A vector normal to the plane of the probe, in the direction of crossings you wish to monitor.

include_species - The species to which this probe should be applied. To probe several species, use several probe blocks in the input deck. “probe_species” is accepted as a synonym.

ek_min - The minimum kinetic energy of particles to store information about. Set to 0 for no minimum kinetic energy.

ek_max - The maximum kinetic energy of particles to store information about. Set to -1 for no maximum kinetic energy.

dumpmask - The dump code for this particle probe. This is the same as that for the main output controls in **input.deck**. Note that the code has to store copies of particles which pass through the probe until a dump occurs. This means that the code’s memory requirements can increase drastically if this code only dumps probe information infrequently. If this is set to **never** then the code effectively never uses the probe.

3.13 collisions block

EPOCH has a particle collision routine with scattering algorithms based on the model presented by Sentoku and Kemp [13] or the model presented by Pérez et al. [9], which in turn was based on the work of Nanbu and Yonemura [8]. This adds a new output block named “collisions” which accepts the following four parameters.

use_collisions - This is a logical flag which determines whether or not to call the collision routine. If omitted, the default is “T” if any of the frequency factors are non-zero (see below) and “F” otherwise.

use_nanbu - This logical flag determines whether the scattering angle of Pérez/Nanbu will be used. If “F”, the Sentoku-Kemp algorithm will be used. The default is “T”.

coulomb_log - This may either be set to a real value, specifying the Coulomb logarithm to use when scattering the particles or to the special value “auto”. If “auto” is used then the routine will calculate a value based on the local temperature and density of the particle species being scattered, along with the two particle charges. If omitted, the default value is “auto”.

collide - This sets up a symmetric square matrix of size nspecies \times nspecies containing the collision frequency factors to use between particle species. The element (s1,s2) gives the frequency factor used when colliding species s1 with species s2. If the factor is less than zero, no collisions are performed. If it is equal to one, collisions are performed normally. For any value between zero and one, the collisions are performed using a frequency multiplied by the given factor.

If “collide” has a value of “all” then all elements of the matrix are set to one. If it has a value of “none” then all elements are set to minus one.

If the syntax “species1 species2 <value>” is used, then the (species1,species2) element of the matrix is set to the factor “<value>”. This may either be a real number, or the special value “on” or “off”. The “collide” parameter may be used multiple times.

The default value is “all” (ie. all elements of the matrix are set to one).

collisional_ionisation - If this logical flag is set to “T” then the collisional ionisation model is enabled. This process is independent of **field_ionisation** (Section 3.4.2). However, in order to set up **collisional_ionisation** you must also specify ionisation energies and electrons in a **species** block (see Section 3.4.2). The default value is “F”.

For example:

```
begin:collisions
    use_collisions = T
    use_nanbu = T
    coulomb_log = auto
    collide = all
    collide = spec1 spec2 off
    collide = spec2 spec3 0.1
end:collisions
```

With this block, collisions are turned on, the Nanbu-Pérez scattering algorithm is used and the Coulomb logarithm is automatically calculated. All values of the frequency array are set to one except (spec1,spec2) is set to minus one (and also (spec2,spec1)) and (spec2,spec3) is set to 0.1

3.14 qed block

EPOCH can model QED pair production, synchrotron emission and radiation reaction as described in Duclous et al. [5]. It is enabled using the compiler flag “-DPHOTONS”. Additionally, the Trident process is enabled using “-DTRIDENT_PHOTONS”.

A new input deck block named “qed” has been added which accepts the following parameters:

use_qed - Logical flag which turns QED on or off. The default is “F”.

qed_start_time - Floating point value specifying the time after which QED effects should be turned on. The default is 0.

produce_photons - Logical flag which specifies whether to track the photons generated by synchrotron emission. If this is “F” then the radiation reaction force is calculated but the properties of the emitted photons are not tracked. The default is “F”.

photon_energy_min - Minimum energy of produced photons. Radiation reaction is calculated for photons of all energies, but photons with energy below this cutoff are not tracked. The default is 0.

photon_dynamics - Logical flag which specifies whether to push photons. If “F” then photons are generated, but their motion through the domain is not simulated and they stay where they were generated. The default is “F”.

produce_pairs - Logical flag which determines whether or not to simulate the process of pair generation from gamma ray photons. Both produce_photons and photon_dynamics must be “T” for this to work. The default is “F”.

qed_table_location - **EPOCH**’s QED routines use lookup tables to calculate gamma ray emission and pair production. If you want to use tables in a different location from the default, specify the new location using this parameter. The default is “src/physics_packages/TABLES”.

use_radiation_reaction - Logical flag which determines whether or not to calculate the radiation reaction force. If set to “F” then the force is not calculated. This should nearly always be enabled when using the QED model. It is only provided for testing purposes. The default value is “T”.

QED also requires that the code now know which species are electrons, positrons and photons. The species type is specified using a single “identify” tag in a species block. To specify an electron the block in the deck would look like

```
begin:species
    name = electron
    fraction = 0.5
    number_density = 7.7e29
    identify:electron
end:species
```

Once the identity of a species is set then the code automatically assigns mass and charge states for the species. Possible identities are:

electron - A normal electron species. All species of electrons in the simulation must be identified in this way or they will not generate photons.

positron - A normal positron species. All species of positron in the simulation must be identified in this way or they will not generate photons.

photon - A normal photon species. One species of this type is needed for photon production to work. If multiple species are present then generated photons will appear in the first species of this type.

breit_wheeler_electron - The electron species for pair production by the Breit-Wheeler process. If a species of this type exists then electrons from the pair production module will be created in this species. If no species of this type is specified then pair electrons will be generated in the first electron species. “bw_electron” is accepted as an alias.

breit_wheeler_positron - As above but for positrons. “bw_proton” is accepted as an alias.

trident_electron - The electron species for pair production by the Trident process. If a species of this type exists then electrons from the pair production module will be created in this species. If no species of this type is specified then pair electrons will be generated in the first electron species.

trident_positron - As above but for positrons.

proton - A normal proton species. This is for convenience only and is not required by the pair production routines.

brem_photon - A bremsstrahlung photon species. This is used by the bremsstrahlung radiation model. See Section 3.15 for details.

A species should be identified only once, so a “breit_wheeler_electron” species does not need to also be identified as an “electron” species. If the code is running with “produce_photons=T” then a photon species must be created by the user and identified. If the code is running with “produce_pairs=T” then the code must specify at least one electron (or breit_wheeler_electron) species and one positron (or breit_wheeler_positron) species. These species will usually be defined with zero particles from the start of the simulation and will accumulate particles as the simulation progresses. The code will fail to run if the needed species are not specified.

3.15 bremsstrahlung block

EPOCH is capable of simulating bremsstrahlung radiation using a combination of the techniques described by Vyskočil et al. [16] and Wu et al. [17]. In order to run this module, the compiler flag “-DBREMSSTRAHLUNG” must be switched on in the Makefile. There should also be a corresponding “bremsstrahlung” block for the input deck which uses a similar format to the “qed” block.

An example bremsstrahlung block is shown below:

```
begin:bremsstrahlung
    enable = T
    start_time = 0
    produce_photons = T
    photon_energy_min = 1 * kev
    photon_weight = 1.0
    photon_dynamics = F
    use_plasma_screening = F
end:bremsstrahlung
```

enable - Logical flag to turn bremsstrahlung on or off. “use_bremsstrahlung” is accepted as an alias. The default is “F”.

start_time - Floating point value specifying the time after which bremsstrahlung radiation is modelled. The default is 0.

produce_photons - Logical flag to allow population of a photon species. If “F”, then only the energy loss of the electrons will be simulated. The photon species can be specified by including the line **identify:brem_photon** in the corresponding species block. If the compiler flag “-DPHOTONS” is active, QED and bremsstrahlung will both populate the first species with **identify:photon** if no brem_photon species is specified. The default is “F”.

photon_energy_min - Floating point value specifying the minimum energy of produced photons. Electron energy loss is still calculated for lower energy photon emissions, but these photons are not added to the photon species. The default is 0.

photon_weight - Floating point value which applies a multiplier to the weight of produced macro-photons in order to increase the number of overall emissions and obtain better spectra. Must be less than or equal to 1 and greater than 0. For example, 0.1 would make emission 10 times more likely, but for macro-photons only 10% the weight of the generating macro-electrons. The default is 1. Note that only one emission is possible per macro-electron per timestep, so setting this too low will saturate emissions.

photon_dynamics - Logical flag to specify whether or not to push photons. If “F”, then the generated photons are immobilised at the point of emission. The default is “F”.

use_plasma_screening - Logical flag to specify whether a cross section enhancement due to heated ionised targets is considered, based on theory described by Wu et al. [17]. It is expected that for high energy electrons passing through low density, ionised plasmas with electron temperatures over ~ 100 eV ($\sim 8 \times 10^5$ K), the bremsstrahlung emission rate could increase by a factor of 2-3. This has not been tested experimentally, and so the default value is set to “F”.

use_radiation_reaction - Logical flag to specify whether the electrons experience energy loss when emitting photons or not. “use_bremsstrahlung_recoil” is accepted as an alias. Debugging flag, default “T”.

table_location - String specifying the location of the emission look-up tables for bremsstrahlung. The default path is set to **src/physics_packages/TABLES/br**.

Bremsstrahlung, like QED, requires the code to know which species are electrons and which are photons, so uses the same identify system (with **identify:brem_photon** for a bremsstrahlung-only photon species). Additionally, the atomic numbers of the atom/ion species are required in the species block. For example, atomic aluminium (charge = 0) could be specified as:

```
begin:species
    name = Aluminium
    atomic_number = 13
    charge = 0

    mass = 49218
    number_density = 6.022e28
    fraction = 0.5
    dump = T
end:species
```

If the atomic number is not specified then it will be assumed that the ion is fully ionised and the atomic number would be set to the charge (the nearest integer to the ion charge when expressed in units of elementary charge). If ionisation is considered, the atomic number must be specified for the first ion species and all child species will retain the same atomic number.

The direction of emitted macro-photons will always be in the direction of the generating macro-electron trajectory. This assumes emission from relativistic electrons only - this code will not accurately reproduce the angular distribution of bremsstrahlung radiation from low energy electrons.

3.16 injector block

Everything in this Subsection requires EPOCH Version 4.10 or later.

The **injector** block specifies a particle source to be introduced through a simulation boundary. Each injector block specifies a source of a single species of particle defined by a density, centre of mass drift momentum, temperature and number of simulation particles per cell. The current version of the injectors is incompatible with the **PER_SPECIES_WEIGHT** compiler flag, and attempting to use an injector with a version of **EPOCH** compiled with this flag will fail.

3.16.1 Concepts

EPOCH can inject particles through any of the simulation boundaries. This plasma is either a drifting Maxwellian corresponding to a collisionally thermalized beam or a “flux Maxwellian” corresponding to a Maxwellian source accelerated by an electrostatic accelerator. It can have any temporal or transverse spatial profile of density, temperature or drift that you wish to specify.

EPOCH does not automatically make any assumption about the plasma that you wish to inject and does not correct for currents injected into the domain. Current due to an injected beam will be smoothly created as the particles enter the domain. If you wish to inject a neutral beam, you will have to use multiple injectors to inject electrons and ions so as to produce a neutral beam. Great care must be taken when introducing relativistic beams since the current due to a highly relativistic beam will not be the current due to the centre of mass velocity since **EPOCH** does not use the Maxwell-Jüttner distribution for loading particles.

3.16.2 Boundary Conditions

The injectors only work properly with certain boundary conditions. For most purposes the “open” boundary condition is the only one that makes sense with injectors since particles are flowing freely through the boundary. Remember that in any version of **EPOCH** that supports injectors you can also use per species boundary conditions to allow you to have different boundary conditions for injected and bulk particles.

3.16.3 Moving Window

Injectors and moving windows can be tricky to work with, so the default behaviour of **EPOCH** is to stop all injectors when the window starts to move. If you wish to override this behaviour then simply explicitly set **t_end** in the injector block to a value after the window starts to move. Setting

```
t_end = t_end
```

will cause the injectors to continue running until the end of the simulation even with the moving window. You must take great care when specifying injectors for a moving window because you will likely get gaps or bunches in particles injected through the x boundary and there will probably be some shearing of particles introduced through y and z boundaries. It is in general recommended that you specify a velocity profile for the moving window that stops at times when particles are to be injected and then starts again once the injection is complete.

3.16.4 Keys

The “injector” block accepts the following keys:

boundary - specifies which boundary to attach the particle source to. Same specification as Section 3.5, so permitted values are x_min, x_max, y_min, y_max, z_min and z_max.

species - specifies which species should be injected through the boundary. Just specify the name of the species required.

t_start - Time at which to start the injector.

t_end - Time at which to end the injector.

nparticles_per_cell - Target pseudo-particle density for the injector. Average number of particles injected will be this value or slightly higher if very few particles are specified. “npart_per_cell” is accepted as an alias.

number_density - Number density of the particle source in m^{-3} . Can be space varying along the boundary to which the injector is attached and time varying.

number_density_min - Minimum particle number density in m^{-3} below which pseudo particles are not loaded. Use if the density has a profile to avoid injecting low weight particles in low density regions.

number_density_max - Maximum particle number density in m^{-3} . When the number density in a cell rises above density_max the injector clips the density to number_density_max allowing easy implementation of exponential rises to plateaus for time-varying injectors. Note that the number of particles per cell is kept fixed and the number density adjustment is achieved by modifying the particle weight. This flag has no effect for particles with per-species weighting. If the flag has a negative value then no clipping is performed. This is the default.

temperature_{x,y,z} - The temperature in each direction for a thermal distribution specified in Kelvin. “temp_{x,y,z}” can be used as aliases.

temperature - Sets an isotropic temperature distribution in Kelvin. If both “temperature” and a specific “temperature_x”, “temperature_y” or “temperature_z” parameter is specified then the last to appear in the deck has precedence. If neither are given then the injector will have a default temperature of zero Kelvin. “temp” is accepted as an alias.

temperature_{x,y,z}_ev, temperature_ev - These are the same as the “temperature” parameters described above except the units are given in electronvolts rather than Kelvin. “temp_{x,y,z}_ev” and “temp_ev” are accepted as aliases.

drift_p{x,y,z} - Specifies a momentum space offset in $kg\ ms^{-1}$ to the distribution function for this injector. By default, the drift is zero. “drift_{x,y,z}” are also accepted as aliases.

use_flux_maxwellian - Logical flag to determine whether to use a flux Maxwellian rather than a drifting Maxwellian when sampling particles to inject. By default, the injector will sample from a flux Maxwellian for small values of drift and, for computational efficiency, a drifting Maxwellian when the drift velocity is much larger than the thermal velocity. It is possible to force the code to always use a drifting Maxwellian by setting this flag to “F”. The default value is “T”.

An example input deck might look as follows:

```
begin:injector

boundary = x_min
species = Electron
number_density = dens
temperature_x = temp
drift_px = drift_p
nparticles_per_cell = 32

end:injector
```

Injectors are a new feature so please be wary of unusual behaviour, and contact the **EPOCH development team if unexpected results occur**

3.17 subset block

It is possible to restrict the number of particles written to file according to various criteria. For example, you can now output the momentum of all particles which have a gamma lower than 1.8 or the positions of a randomly chosen subset of a given species.

A new input deck block named “subset” is defined which accepts the following parameters:

name - The name given to this subset. This is used to identify the subset in the output block and is also used when labelling the data in the SDF files.

include_species - Add the given particle species to the set of particles that this subset applies to. By default, no particle species are included.

dumpmask - The dumpmask to use when considering this subset in an output block. This takes the same form as the output block dumpmask. The default value is “always”.

random_fraction - Select a random percentage of the particle species. This is a real value between zero and one. If 0 is specified, no particles are selected. If 1 is specified, all the particles are selected. If 0.2 is specified, 20% of the particles are selected.

{px,py,pz,weight,charge,mass,gamma}_min - Select only the particles with momentum, weight, charge, mass or gamma which is greater than the given value.

{px,py,pz,weight,charge,mass,gamma}_max - Select only the particles with momentum, weight, charge, mass or gamma which is less than the given value.

{x,y,z}_min - Select only the particles whose position lies above the given value.

{x,y,z}_max - Select only the particles whose position lies below the given value.

id_min,max - Select only the particles whose “id” is greater than or less than the given values. The “id” field is explained below.

skip,skip_x,y,z - Integer parameter for subsampling the particle list. If set to a positive integer then all grid-based variables using the subset restriction will be reduced when being written to file. This is achieved by skipping by the specified number of cells in each of the specified directions. The “skip” parameter provides a quick method for setting the same number of cells to skip in all directions. This currently only applies to grid-based variables and is ignored for data averages. The default value is “0”.

Once a subset has been defined, the subset name can then be used in place of (or in addition to) the dumpmask in an “output” block. For example:

```

begin:subset
    name = background
    random_fraction = 0.1
    include_species:electron
    include_species:proton
end:subset

begin:subset
    name = high_gamma
    gamma_min = 1.3
    include_species:electron
end:subset

begin:output
    particles = background + high_gamma + always
    px = background + high_gamma
    py = background
    pz = always
end:output

```

In this example, three “px” blocks will be written: “Particles/background/electron/Px”, “Particles/background/proton/Px” and “Particles/high_gamma/electron/Px”. The “background” blocks will contain 10% of the each species, randomly selected. The “high_gamma” block will contain all the electrons with a gamma greater than 1.3.

There will also be “Particles/background/electron/Py” and “Particles/background/proton/Py” block containing y-momentum for the same 10% random subset of particles. Finally, the “Particles/All/electron/Pz” and “Particles/All/proton/Pz” will contain the z-momentum for all particles.

The final selection criteria given in the list above is “id_min” and “id_max”. As of **EPOCH** version 4.0, the code can now assign a unique ID field to every particle in the simulation. This can be useful for tracking specific particles as they move through a simulation. As this field adds extra memory requirements to the particles, it is disabled by default and must be compiled in using the “-DPARTICLE_ID” compiler flag.

Particle IDs can be written to file using the “id” variable name in the “output” block. Eg.

```

begin:output
    particles = always
    id = always
end:output

```

3.17.1 Subsets of fields

Subset blocks can be applied to per-species variables such as current and temperature. Only particles within the given momentum ranges and of the selected species are included in the calculations. In

addition, subset blocks can now be applied to field or grid variables. This allows you to output spatial sections using the **{x,y,z}_max,min** restrictions. The output data will be trimmed to the selected ranges and a corresponding restricted grid included in the output. Note that specifying an empty range will lead to output of the entire domain. For example, the following snippet will output an ex_c_centre variable restricted to the centre 1/3rd of the domain with a corresponding grid grid_centre:

```

begin:subset
    name = centre
    x_min = x_min + (x_max - x_min)/3.0
    x_max = x_min + 2.0*(x_max - x_min)/3.0
end:subset

begin:output
    ...
    ex = always + centre
end:output

```

3.17.2 Persistent subsets

Persistent subsets are subsets that capture a set of particles once, given a specified set of parameters, and then track those particles permanently. Persistent subsets use the same blocks as normal subsets and take the same parameters as normal subsets (except the skip parameters which only apply to fields). Subsets are marked as persistent by setting either

persist_start_time - Time at which to record the list of particles to be tracked. Throughout the rest of the simulation this recorded list will be used whenever requesting output for this subset. “persist_after_time” is accepted as an alias. Set to 0 to record from the start of the simulation.

persist_start_step - Similar to “persist_start_time” except this specifies a simulation step number to use instead of time. “persist_after_step” is accepted as an alias.

If the input deck is edited on restart to add a new persistent subset then it must be added after existing persistent subsets or problems may occur on restart.

3.18 constant block

The **constant** block type helps to make the input deck more flexible and maintainable. It allows you to define constants and maths parser expressions (see Section 3.19) which can be used by name later in the deck.

Constants are simply maths parser expressions which are assigned to a name as shown above. When the name is used on the right hand side of a deck expression it is replaced by the expression it was assigned with. This expression may be a simple numerical constant, a mathematical expression or a function. Constants may contain spatially varying information without having to pre-calculate them at every location in the domain. To those familiar with FORTRAN codes which use statement functions, parameters appearing in the “constant” block are fairly similar.

If a constant name is reused in a constant block then the old constant is deleted and replaced with the new one. This happens without warning.

```

begin:constant
    lambda = 1.06 * micron
    omega = 2.0 * pi * c / lambda

```

```

den_crit = critical(omega)
scale = 3.5 * micron
den_max = 5.0 * den_crit
thick = 300e-9
pplength = 6000e-9
widscale = 5.0e-6

t_wid = (10.0e-6) / c
amax = 1.0
wy = 1e-6
y = 0.0

slope = exp(-2.0 * (y/wy)^2)
blob = gauss(sqrt(x^2+y^2), 0.0, 1.0e-6)
end:constant

```

Using constants can be very helpful when dealing with long, complicated expressions since they allow the expression to be broken down into much simpler parts. They can also be used to get around the FORTRAN string length limitation built into many compilers which prevents deck lines being longer than 512 characters long. As a general rule, it is a good idea to break down complicated expressions using constants or by other means, in order to make the deck look more readable.

Constants are persistent for the entire runtime of the code, allowing them to be used when specifying time profiles for lasers, and also allowing developers to use maths parser expressions for other internal parts of the code where needed.

In the above example, several pre-defined constants have been used ([pi](#) and [c](#)) and also several functions ([critical](#), [exp](#), [gauss](#) and [sqrt](#)). These are described in Section 3.19.1 and Section 3.19.2.

3.19 The maths parser

A discussion of the input deck for [EPOCH](#) would not be complete without consideration of the maths parser. The maths parser is the code which reads the input decks. The parser makes it possible that any parameter taking a numerical value (integer or real) can be input as a mathematical expression rather than as a numerical constant. The maths parser is fairly extensive and includes a range of mathematical functions, physical and simulation constants and appropriately prioritised mathematical operators.

3.19.1 Constants

The maths parser in [EPOCH](#) has the following constants

- pi - The ratio of the circumference of a circle to its diameter.
- kb - Boltzmann's constant.
- me - Mass of an electron.
- qe - Charge of an electron.
- c - Speed of light.
- epsilon0 - Permeability of free space.
- mu0 - Permittivity of free space.

- ev - Electronvolt.
- kev - Kilo-Electronvolt.
- mev - Mega-Electronvolt.
- micron - A convenience symbol for specifying wavelength in microns rather than metres.
- milli - 10^{-3}
- micro - 10^{-6}
- nano - 10^{-9}
- pico - 10^{-12}
- femto - 10^{-15}
- atto - 10^{-18}
- cc - A convenience symbol for converting from cubic metres to cubic centimetres (ie. 10^{-6})
- time - Initial simulation time.
- x,y,z - Cell-centred grid coordinates in the x,y,z direction.
- xb,yb,zb - Node-centred grid coordinates in the x,y,z direction.
- ix,iy,iz - Grid index in the x,y,z direction.
- nx,ny,nz - Number of grid points in the x,y,z direction.
- dx,dy,dz - Grid spacing in the x,y,z direction.
- {x,y,z}_min - Grid coordinate of the minimum x,y,z boundary.
- {x,y,z}_max - Grid coordinate of the maximum x,y,z boundary.
- length_{x,y,z} - The length of the simulation box in the x,y,z direction.
- nproc_{x,y,z} - The number of processes in the x,y,z directions.
- nsteps - The number of timesteps requested.
- t_end - The end time of the simulation.
- p{x,y,z} - Momentum in the x, y, z directions. Used in specifying arbitrary distribution functions.
- r_xyz - Radial distance from the origin
- r_{xy,xz,yz} - Radial distances from the z, y and x axes, respectively

It is also possible for an end user to specify custom constants both within the code and from the input deck. These topics are covered later in this subsection. An example of using a constant would be:

length_x = pi

3.19.2 Functions

The maths parser in **EPOCH** has the following functions

- `abs(a)` - Absolute value.
- `floor(a)` - Convert real to integer rounding down.
- `ceil(a)` - Convert real to integer rounding up.
- `nint(a)` - Convert real to integer rounding to nearest integer.
- `sqrt(a)` - Square root.
- `sin(a)` - Sine.
- `cos(a)` - Cosine.
- `tan(a)` - Tangent.
- `asin(a)` - Arcsine.
- `acos(a)` - Arccosine.
- `atan(a)` - Arctangent.
- `atan2(Y,X)` - Arctangent using the Fortran ATAN2 function. This computes the principal value of the argument function of the complex number $X + iY$. This function can be used to transform from Cartesian into polar coordinates and allows to determine the angle in the correct quadrant.
- `sinh(a)` - Hyperbolic sine.
- `cosh(a)` - Hyperbolic cosine.
- `tanh(a)` - Hyperbolic tangent.
- `exp(a)` - Exponential.
- `loge(a)` - Natural logarithm.
- `log10(a)` - Base-10 logarithm.
- `log_base(a,b)` - Base-b logarithm.
- `gauss(x, x0, w)` - Calculate a Gaussian profile in variable x centred on x_0 with a characteristic width w . $f(x) = \exp(-((x - x_0)/w)^2)$. In this expression the full width at half maximum is given by $fwhm = 2w\sqrt{\ln 2}$
- `supergauss(x, x0, w, n)` - This is identical to “gauss” except it takes a fourth parameter, n , which is the power to raise the exponential argument to.
- `semigauss(t, A, A0, w)` - Calculate a semi Gaussian profile in variable t with maximum amplitude A , amplitude at $t = 0$ A_0 and width w . The parameter A_0 is used to calculate t_0 , the point at which the Gaussian reaches its maximum value. For t less than t_0 the profile is Gaussian and for t greater than t_0 it is the constant A .

$$t_0 = w\sqrt{-\ln(A_0/A)}$$

$$f(t) = \begin{cases} A \exp(-((t - t_0)/w)^2), & t < t_0 \\ A, & \text{otherwise} \end{cases}$$

- `interpolate(interp_var,...,n_pairs)` - Linear interpolation function, explained later.
- `if(a,b,c)` - Conditional function. If $a \neq 0$ the function returns b , otherwise the function returns c .
- `number_density(a)` - Returns the number density for species a .
- `temperature_{x,y,z}(a)` - Returns temperature in the x , y or z direction for species a . “`temp_{x,y,z}(a)`” are accepted as aliases.
- `temperature_{x,y,z}_ev(a)` - Returns temperature in the x , y or z direction measured in electronvolts for species a . “`temp_{x,y,z}_ev(a)`” are accepted as aliases.
- `e{x,y,z}(x,y,z)` - Returns the x , y or z component of the electric field at the specified location.
- `b{x,y,z}(x,y,z)` - Returns the x , y or z component of the magnetic field at the specified location.
- `critical(ω)` - Returns the critical density for the given frequency ω . ie. $n_{crit}(\omega) = \omega^2 m_0 \epsilon_0 / e^2$

It is also possible for an end user to specify custom functions within the code. An example of using a function would be:

```
length_x = exp(pi)
```

The use of most of these functions is fairly simple, but “`interpolate`” requires some additional explanation. This function allows a user to specify a set of position,value pairs and have the code linearly interpolate the values between these control points. This function is mainly intended for ease of converting initial conditions from other existing PIC codes, and the same effect can usually be obtained more elegantly using the “`if`” command. The structure of the “`interpolate`” command is as follows: The first parameter is the variable which is to be used as the axis over which to interpolate the values. This can in general be any valid expression, but will normally just be a coordinate axis. The next $2n$ entries are the position,value pairs and the final parameter is the number of position,value pairs. The slightly clunky syntax of this command is unfortunately necessary to allow it to work with some fairly fundamental features of the maths parser used in [EPOCH](#).

3.19.3 Operators

The maths parser in [EPOCH](#) allows the following operators

- $a + b$ - Addition operator.
- $a - b$ - Subtraction operator or unary negation operator (auto-detected).
- $a * b$ - Multiplication operator.
- a / b - Division operator.
- a^b - Power raise operator.
- $a e b$ - Power of ten operator ($1.0e3 = 1000$).
- $a \text{ lt } b$ - Less than operator. Returns 1 if $a < b$, otherwise returns 0. Intended for use with `if`.
- $a \text{ gt } b$ - Greater than operator. Returns 1 if $a > b$, otherwise returns 0.
- $a \text{ eq } b$ - Equality operator. Returns 1 if $a == b$, otherwise returns 0.
- $a \text{ and } b$ - Logical and operator. Returns 1 if $a != 0$ and $b != 0$, otherwise returns 0.
- $a \text{ or } b$ - Logical or operator. Returns 1 if $a != 0$ or $b != 0$, otherwise returns 0.

These follow the usual rules for operator precedence, although it is best to surround more complex expressions in parenthesis if the precedence is important. It is not possible at this time to specify custom operators without major changes to the code. An example of using an operator would be:

```
length_x = 10.0 + 12.0
```

4 EPOCH use in practice

4.1 Specifying initial conditions for particles using the input deck

If the initial conditions for the plasma you wish to model can be described by a number density and temperature profile on the underlying grid then **EPOCH** can create an appropriate particle distribution for you. The set of routines which accomplish this task are known as the autoloader. For many users, this functionality is sufficient to make use of the code and there is no need to deal with the internal representation of particles in **EPOCH**.

The autoloader randomly loads particles in space to reproduce the number density profile that was requested. It then sets the momentum components of the particles to approximate a Maxwell-Boltzmann distribution corresponding to the temperature profile. Sometimes this is not the desired behaviour, for example you may wish to model a bump-on-tail velocity distribution. It is currently not possible to specify these initial conditions from the input deck and the particles must be setup by modifying the source code.

There are two stages to the particle setup in **EPOCH**

- `auto_load` - This routine is called after reading and parsing the input deck. It takes care of allocating particles and setting up their initial positions and momenta using the initial conditions supplied in deck file. It is not necessary to recompile the code, or even have access to the source to change the initial conditions using this method.
- `manual_load` - Once particles have been allocated they can have their properties altered in this routine. By default it is an empty routine which does nothing.

4.1.1 Setting autoloader properties from the input deck

To illustrate using the autoloader in practice, we present an example for setting up an isolated plasma block in 2D. This would look like:

```
begin:species
  name = s1
  # first set density in the range 0->1
  # cut down density in x direction
  number_density = if ((x gt -1) and (x lt 1), 1.0, 0.2)
  # cut down density in y direction
  number_density = if ((y gt -1) and (y lt 1), number_density(s1), 0.2)

  # multiply density by real particle density
  number_density = number_density(s1) * 100.0

  # Set the temperature to be zero
  temperature_x = 0.0
  temperature_y = temperature_x(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * 100.0
```

```

end:species

begin:species
    # Just copy the density for species s1
    number_density = number_density(s1)

    # Just copy the temperature from species s1
    temperature_x = temperature_x(s1)
    temperature_y = temperature_y(s1)

    # Set the minimum density for this species
    number_density_min = 0.3 * 100.0
end:species

```

An important point to notice is that the two parts of the logical expressions in the input deck are enclosed within their own brackets. This helps to remove some ambiguities in the functioning of the input deck parser. It is hoped that this will soon be fixed, but at present ALWAYS enclose logical expressions in brackets.

4.2 Manually overriding particle parameters set by the autoloader

Since not all problems in plasma physics can be described in terms of an initial distribution of thermal plasma, it is also possible to manually control properties of each individual pseudoparticle for an initial condition. This takes place in the subroutine `manual_load` in the file `user_interaction/ic_module.f90`. Manual loading takes place after all the autoloader phase, to allow manual tweaking of autoloader specified initial conditions.

4.2.1 EPOCH internal representation of particles

Before we can go about manipulating particle properties in `manual_load`, we first need an overview of how the particles are defined in `EPOCH`. Inside the code, particles are represented by a Fortran90 TYPE called “particle”. The current definition of this type (in 2D) is:

```

TYPE particle
    REAL(num), DIMENSION(3) :: part_p
    REAL(num), DIMENSION(c_ndims) :: part_pos
    REAL(num) :: weight
    REAL(num) :: charge
    REAL(num) :: mass
    TYPE(particle), POINTER :: next, prev
#ifndef PARTICLE_DEBUG
    INTEGER :: processor
    INTEGER :: processor_at_t0
#endif
    INTEGER(i8) :: id
    INTEGER :: coll_count
    REAL(num) :: optical_depth
    REAL(num) :: particle_energy
    REAL(num) :: optical_depth_tri
END TYPE particle

```

Here, most of the preprocessing directives have been stripped out for clarity. We have left **#ifdef** **PARTICLE_DEBUG** as an example. Here, the “processor” and “processor_at_t0” variables only exist if the **-DPARTICLE_DEBUG** define was put in the makefile. If you want to access a property that does not seem to be present, check the preprocessor defines.

The “particle” properties can be explained as follows:

- part_p - The momentum in 3 dimensions for the particle. This is always of size 3.
- part_pos - The position of the particle in space. This is of the same size as the dimensionality of the code.
- weight - The weight of this particle. The number of real particles represented by this pseudoparticle.
- charge - The particle charge. If the code was compiled without per particle charge, then the code uses the charge property from TYPE(particle_species).
- mass - The particle rest mass. If the code was compiled without per particle mass, then the code uses the mass property from TYPE(particle_species).
- next, prev - The next and previous particle in the linked list which represents the particles in the current species. This will be explained in more detail later.
- processor - The rank of the processor which currently holds the particle.
- processor_at_t0 - The rank of the processor on which the particle started.
- id - Unique particle ID.
- coll_count - Used for debugging the collision routines.
- optical_depth - Optical depth used in the QED routines.
- particle_energy - Particle energy used in the QED routines.
- optical_depth_tri - Optical depth for the trident process in the QED routines.

Collections of particles are represented by another Fortran TYPE, called **particle_list**. This type represents all the properties of a collection of particles and is used behind the scenes to deal with inter-processor communication of particles. The definition of the type is:

```

TYPE particle_list
  TYPE(particle), POINTER :: head
  TYPE(particle), POINTER :: tail
  INTEGER(i8) :: count
  INTEGER :: id_update
  ! Pointer is safe if the particles in it are all unambiguously linked
  LOGICAL :: safe

  ! Does this partlist hold copies of particles rather than originals
  LOGICAL :: holds_copies

  TYPE(particle_list), POINTER :: next, prev
END TYPE particle_list

```

- head - The first particle in the linked list.
- tail - The last particle in the linked list.

- count - The number of particles in the list. Note that this is NOT MPI aware, so reading count only gives you the number of particles on the local processor.
- safe - Any particle_list which a user should come across will be a safe particle_list. Don't change this property.
- next, prev - For future expansion it is possible to attach particle_lists together in another linked list. This is not currently used anywhere in the code.

An entire species of particles is represented by another Fortran TYPE, this time called **particle_species**. This represents all the properties which are common to all particles in a species. The current definition is:

```

TYPE particle_species
  ! Core properties
  CHARACTER(string_length) :: name
  TYPE(particle_species), POINTER :: next, prev
  INTEGER :: id
  INTEGER :: dumpmask
  INTEGER :: count_update_step

  REAL(num) :: charge
  REAL(num) :: mass
  REAL(num) :: weight
  INTEGER(i8) :: count
  TYPE(particle_list) :: attached_list
  LOGICAL :: immobile
  LOGICAL :: fill_ghosts

  ! Parameters for relativistic and arbitrary particle loader
  INTEGER :: ic_df_type
  REAL(num) :: fractional_tail_cutoff

  TYPE(primitive_stack) :: dist_fn
  TYPE(primitive_stack) :: dist_fn_range(3)

#ifndef NO_TRACER_PARTICLES
  LOGICAL :: zero_current
#endif

  ! ID code which identifies if a species is of a special type
  INTEGER :: species_type

  ! particle cell division
  INTEGER(i8) :: global_count
  LOGICAL :: split
  INTEGER(i8) :: npart_max
  ! Secondary list
  TYPE(particle_list), DIMENSION(:,:,), POINTER :: secondary_list

  ! Injection of particles
  REAL(num) :: npart_per_cell
  TYPE(primitive_stack) :: density_function, temperature_function(3)

```

```

TYPE(primitive_stack) :: drift_function(3)

! Thermal boundaries
REAL(num), DIMENSION(:, :, ), POINTER :: ext_temp_x_min, ext_temp_x_max
REAL(num), DIMENSION(:, :, ), POINTER :: ext_temp_y_min, ext_temp_y_max

! Species_ionisation
LOGICAL :: electron
LOGICAL :: ionise
INTEGER :: ionise_to_species
INTEGER :: release_species
INTEGER :: n
INTEGER :: l
REAL(num) :: ionisation_energy

! Attached probes for this species
#ifndef NO_PARTICLE_PROBES
    TYPE(particle_probe), POINTER :: attached_probes
#endif

! Particle migration
TYPE(particle_species_migration) :: migrate

! Initial conditions
TYPE(initial_condition_block) :: initial_conditions

! Per-species boundary conditions
INTEGER, DIMENSION(2*c_ndims) :: bc_particle
END TYPE particle_species

```

This definition is for the 2D version of the code. The only difference for the other two versions is the number of dimensions in the arrays “secondary_list” and “ext_temp_*”. temperature arrays. There are quite a lot of fields here, so we will just document some of the most important ones.

- name - The name of the particle species, used in the output dumps etc.
- next, prev - Particle species are also linked together in a linked list. This is used internally by the output dump routines, but should not be used by end users.
- id - The species number for this species. This is the same number as is used in the input deck to designate the species.
- dumpmask - Determine when to include this species in output dumps. Note that the flag is ignored for restart dumps.
- charge - The charge in Coulombs. Even if PER_PARTICLE_CHARGE_MASS is specified, this is still populated from the input deck, and now refers to the reference charge for the species.
- mass - The mass in kg.
- weight - The per-species particle weight.
- count - The global number of particles of this species (NOTE may not be accurate). This will only ever be the number of particles on this processor when running on a single processor. While this

property will be accurate when setting up initial conditions, it is only guaranteed to be accurate for the rest of the code if the code is compiled with the correct preprocessor options.

- attached_list - The list of particles which belong to this species.
- immobile - If set to .TRUE. then the species is ignored during the particle push.
- zero_weight - If set to .TRUE. then the current is not updated for this particle species.

4.2.2 Setting the particle properties

The details of exactly what a linked list means in **EPOCH** requires a more in-depth study of the source code. For now, all we really need to know is that each species has a list of particles. A pointer to the first particle in the list is contained in `species_list(ispecies)%attached_list%head`. Once you have a pointer to a particle (eg. `current`), you advance to the next pointer in the list with `current => current%next`.

After all the descriptions of the types, actually setting the properties of the particles is fairly simple. The following is an example which positions the particles uniformly in 1D space, but doesn't set any momentum.

```
SUBROUTINE manual_load

TYPE(particle), POINTER :: current
INTEGER :: ispecies
REAL(num) :: rpos, dx

DO ispecies = 1, n_species
    current => species_list(ispecies)%attached_list%head
    dx = length_x / species_list(ispecies)%attached_list%count
    rpos = x_min
    DO WHILE(ASSOCIATED(current))
        current%part_pos = rpos
        current%weight = 1.0_num
        rpos = rpos + dx
        current => current%next
    ENDDO
    ENDDO

END SUBROUTINE manual_load
```

This will take the particles which have been placed at random positions by the autoloader and repositions them in a uniform manner. In order to adjust the particle positions, you need to know about the grid used in **EPOCH**. In this example we only required the length of the domain, “length_x” and the minimum value of x, “x_min”. A more exhaustive list is given in the following section. Note that I completely ignored the question of domain decomposition when setting up the particles. The code automatically moves the particles onto the correct processor without user interaction.

In the above example, note that particle momentum was not specified and particle weight was set to be a simple constant. Setting particle weight can be very simple if you can get the pseudoparticle distribution to match the real particle distribution, or quite tricky if this isn't possible. The weight of a pseudoparticle is calculated such that the number of pseudoparticles in a cell multiplied by their weights equals the number of physical particles in that cell. This can be quite tricky to get right, so in more complicated cases it is probably better to use the autoloader than to manually set up the number density distribution.

4.2.3 Grid coordinates used in EPOCH .

When setting up initial conditions within the **EPOCH** source (rather than using the input deck) there are several constants that you may need to use. These constants are:

- nx - Number of gridpoints on the local processor in the x direction.
- ny - Number of gridpoints on the local processor in the y direction (2D and 3D).
- nz - Number of gridpoints on the local processor in the z direction (3D).
- length_{x,y,z} - Length of domain in the x, y, z directions.
- {x,y,z}_min - Minimum value of x, y, z for the whole domain.
- {x,y,z}_max - Maximum value of x, y, z for the whole domain.
- n_species - The number of species in the code.

There are also up to three arrays which are available for use.

- x(-2:nx+3) - Position of a given gridpoint in real units in the x direction.
- y(-2:ny+3) - Position of a given gridpoint in real units in the y direction (2D and 3D).
- z(-2:nz+3) - Position of a given gridpoint in read units in the z direction (3D).

4.2.4 Loading a separable non-thermal particle distribution.

It is now possible to specify any arbitrary distribution function to use for setting up the initial particle momenta, as described in Section 3.4.5. However, if the distribution function can be represented as a set of 1D distribution functions in p_x , p_y and p_z separately, then it may be significantly faster to use the method outlined below.

The code includes a helper function to select a point from an arbitrary distribution function. To use the helper function, you need to define two 1D arrays which are the x and y axes for the distribution function as shown in the following example.

```
SUBROUTINE manual_load

TYPE(particle), POINTER :: current
INTEGER, PARAMETER :: np_local = 1000
INTEGER :: ispecies, ip
REAL(num) :: temperature, stdev2, tail_width, tail_height, tail_drift
REAL(num) :: frac, tail_frac, min_p, max_p, dp_local, p2, tail_p2
REAL(num), DIMENSION(np_local) :: p_axis, distfn_axis

temperature = 1e4_num
tail_width = 0.05_num
tail_height = 0.2_num
tail_drift = 0.5_num

DO ispecies = 1, n_species
    stdev2 = kb * temperature * species_list(ispecies)%mass
    frac = 1.0_num / (2.0_num * stdev2)
    tail_frac = 1.0_num / (2.0_num * stdev2 * tail_width)
```

```

max_p = 5.0_num * SQRT(stdev2)
min_p = -max_p

dp_local = (max_p - min_p) / REAL(np_local-1, num)
DO ip = 1, np_local
    p_axis(ip) = min_p + (ip - 1) * dp_local
    p2 = p_axis(ip)**2
    tail_p2 = (p_axis(ip) - tail_drift * max_p)**2
    distfn_axis(ip) = EXP(-p2 * frac) &
        + tail_height * EXP(-tail_p2 * tail_frac)
ENDDO

current=>species_list(ispecies)%attached_list%head
DO WHILE(ASSOCIATED(current))
    current%part_p(1) = sample_dist_function(p_axis, distfn_axis)
    current=>current%next
ENDDO
ENDDO

END SUBROUTINE manual_load

```

This example will set the particles to have a bump-on-tail velocity distribution, a setup which is not possible to do using only the input deck. It is not necessary to normalise the distribution function, as this is done automatically by the **sample_dist_function** function.

4.3 Lasers

EPOCH has the ability to add EM wave sources such as lasers at boundaries. To use lasers, set the boundary that you wish to have a laser on to be of type **simple_laser** and then specify one or more lasers attached to that boundary. Lasers may be specified anywhere initial conditions are specified.

4.4 **laser** blocks in multiple dimensions.

When running **EPOCH** in 2D or 3D, the laser can be modified spatially via the **profile** and **phase** parameters. These are briefly outlined in Section 3.5 but in this section we will describe them in a little more depth.

profile - The spatial profile for the laser. This is essentially an array defined along the edge (or surface) that the laser is attached to. It is clear that the spatial profile is only meaningful perpendicular to the laser's direction of travel and so it is just a single constant in 1D. The laser profile is evaluated as an initial condition and so cannot include any temporal information which must be encoded in **t_profile**. The spatial profile is evaluated at the boundary where the laser is attached and so only spatial information in the plane of the boundary is significant. This is most clearly explained through a couple of examples. In these examples the spatial profile of the laser is set to vary between a flat uniform profile (**profile = 1**) and a Gaussian profile in y (**profile = gauss(y,0,2.5e-6)**). The difference between these profiles is obvious but the important point is that a laser travelling parallel to the x-direction has a profile in the y direction. Similarly a laser propagating in the y-direction has a profile in the x direction. In 3D this is extended so that a laser propagating in a specified direction has a profile in both orthogonal directions. So a laser travelling parallel to the x axis in 3D would have a profile in y and z. Since 3D lasers are very similar to 2D lasers, they will not be considered here in greater detail, but in 3D, it is possible to freely specify the laser profile across the entire face where a

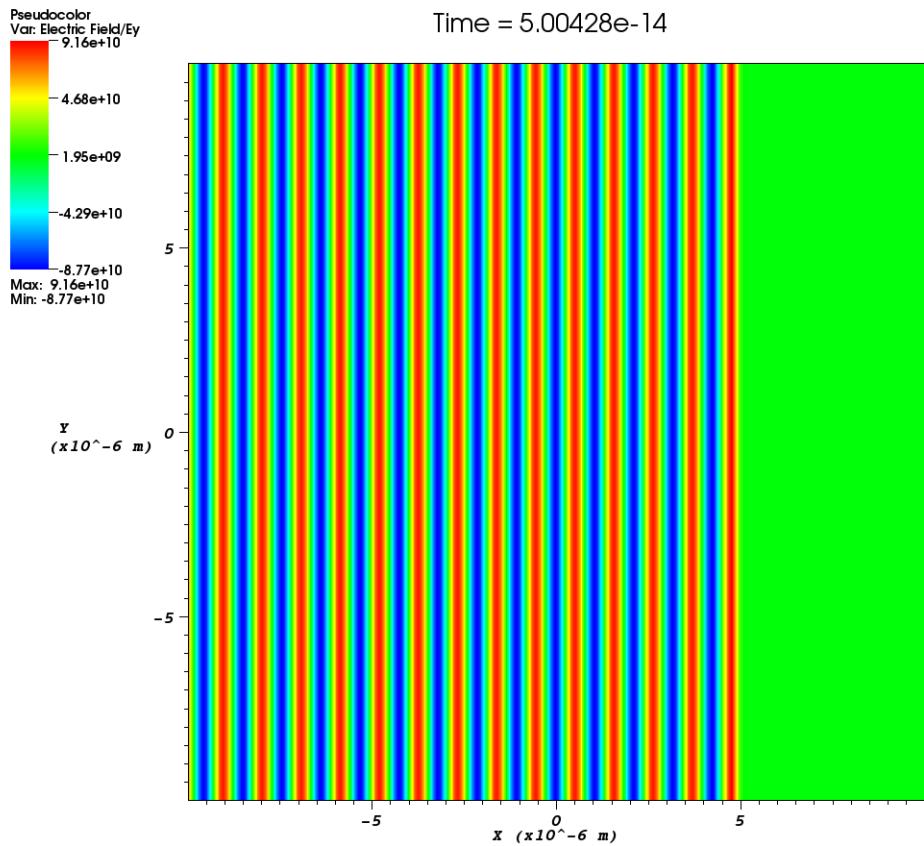


Figure 4: Constant laser profile

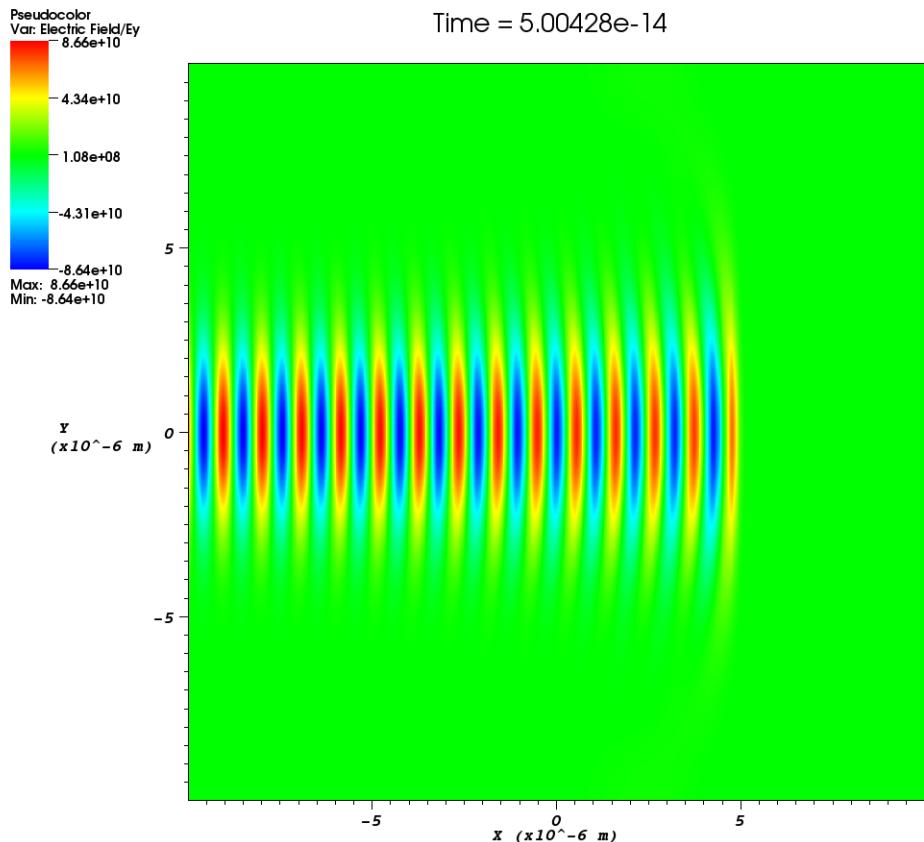


Figure 5: Gaussian laser profile

laser is attached.

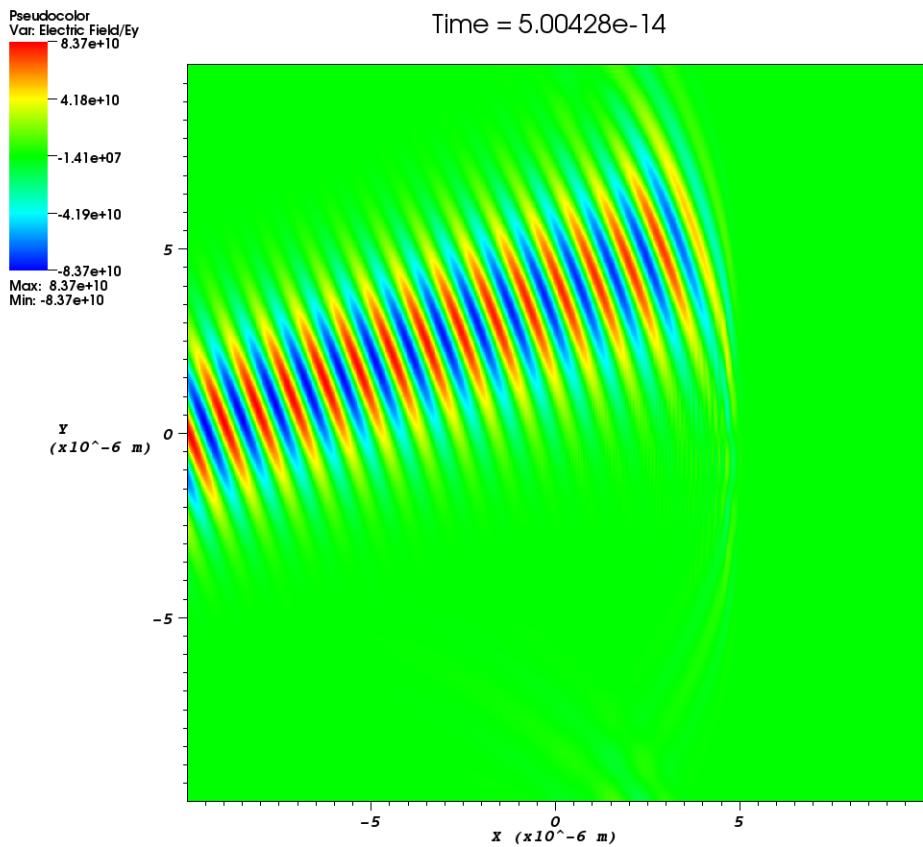


Figure 6: Angled laser profile

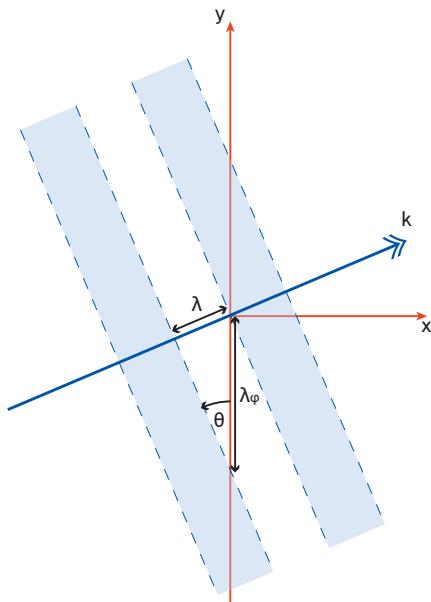


Figure 7: Laser at an angle

phase - Phase shift for the laser in radians. This is a spatial variable which is also defined across the whole of the boundary on which the laser is attached. This allows a user to add a laser travelling at an angle to a boundary as shown in Figure 6. The setup for this is not entirely straightforward and requires a little bit of explanation. Figure 7 illustrates a laser being driven at an angle on the x-min

boundary. Different wave fronts cross the y -axis at different places and this forms a sinusoidal profile along y that represents the phase. The wavelength of this profile is given by $\lambda_\phi = \lambda / \sin \theta$, where λ is the wavelength of the laser and θ is the angle of the propagation direction with respect to the x -axis. The actual phase to use will be $\phi(y) = -k_\phi y = -2\pi y / \lambda_\phi$. It is negative because the phase of the wave is propagating in the positive y direction. It is also necessary to alter the wavelength of the driver since this is given in the direction perpendicular to the boundary. The new wavelength to use will be $\lambda \cos \theta$. Figure 6 shows the resulting E_y field for a laser driven at an angle of $\pi/8$. Note that since the boundary conditions in the code are derived for propagation perpendicular to the boundary, there will be artefacts on the scale of the grid for lasers driven at an angle.

Using this technique it is also possible to focus a laser. This is done by using the same technique as above but making the angle of propagation, θ , a function of y such that the laser is focused to a point along the x -axis. A simple example of this is given in “**Focussing a Gaussian Beam**” (Section 5.4).

4.5 Restarting **EPOCH** from previous output dumps

Another possible way of setting up initial conditions in **EPOCH** is to load in a previous output dump and use it to specify initial conditions for the code. The effect of this is to restart the code from the state that it was in when the dump was made. To do this, you just set the field “restart_snapshot” to the number of the output dump from which you want the code to restart. Because of the way in which the code is written you cannot guarantee that the code will successfully restart from any output dump. To restart properly, the following *must* have been dumped

- Particle positions.
- Particle momenta.
- Particle species.
- Particle weights.
- Relevant parts of the electric field (If for example it is known that $e_z == 0$ then it is not needed).
- Relevant parts of the magnetic field.

It is possible to use the manual particle control part of the initial conditions to make changes to a restarted initial condition after the restart dump is loaded. The output files don’t include all of the information needed to restart the code fully since some of this information is contained in the input deck. However, a restart dump also contains a full copy of the input deck used and this can be unpacked before running the code.

If specific “restart” dumps are specified in the input deck, or the “force_final_to_be_restartable” flag is set then in some cases the output is forced to contain enough information to output all the data. These restart dumps can be very large, and also override the “dumpmask” parameter specified for a species and output the data for that species anyway.

4.6 Parameterising input decks

The simplest way to allow someone to use **EPOCH** as a black box is to give them the input.deck files that control the setup and initial conditions of the code. The input deck is simple enough that a quick read through of the relevant section of the manual should make it fairly easy for a new user to control those features of the code, but the initial conditions can be complex enough to require significant work on the part of an unfamiliar user to understand. In this case, it can be helpful to use the ability to specify constants in an input deck to parameterise the file. So, to go back to a slight variation on an earlier example:

```

begin:species
  name = s1

  # first set density in the range 0->1
  # cut down density in x direction
  number_density = if ((x gt -1) and (x lt 1), 1.0, 0.2)
  # cut down density in y direction
  number_density = if ((y gt -1) and (y lt 1), number_density(s1), 0.2)

  # multiply density by real particle density
  number_density = number_density(s1) * 100.0

  # Set the temperature to be zero
  temperature_x = 0.0
  temperature_y = temperature_x(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * 100.0
end:species

begin:species
  name = s2

  # Just copy the density for species s1
  number_density = number_density(s1)

  # Just copy the temperature from species s1
  temperature_x = temperature_x(s1)
  temperature_y = temperature_y(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * 100.0
end:species

```

The particle density (100.0) is hard coded into the deck file in several places. It would be easier if this was given to a new user as:

```

begin:constant
  particle_density = 100.0 # Particle number density
end:constant

begin:species
  name = s1

  # first set density in the range 0->1
  # cut down density in x direction
  number_density = if ((x gt -1) and (x lt 1), 1.0, 0.2)
  # cut down density in y direction
  number_density = if ((y gt -1) and (y lt 1), number_density(s1), 0.2)

  # multiply density by real particle density

```

```

number_density = number_density(s1) * particle_density

# Set the temperature to be zero
temperature_x = 0.0
temperature_y = temperature_x(s1)

# Set the minimum density for this species
number_density_min = 0.3 * particle_density
end:species

begin:species
  name = s2

  # Just copy the density for species s1
  number_density = number_density(s1)

  # Just copy the temperature from species s1
  temperature_x = temperature_x(s1)
  temperature_y = temperature_y(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * particle_density
end:species

```

It is also possible to parameterise other elements of initial conditions in a similar fashion. This is generally a good idea, since it makes the initial conditions easier to read and maintain.

4.7 Using spatially varying functions to further parameterise initial conditions

Again, this is just a readability change to the normal input.deck file, but it also makes changing and understanding the initial conditions rather simpler. In this case, entire parts of the initial conditions are moved into a spatially varying constant in order to make changing them at a later date easier. For example:

```

begin:constant
  particle_density = 100.0 # Particle number density
  profile_x = if((x gt -1) and (x lt 1),1.0,0.2)
  profile_y = if((y gt -1) and (y lt 1),1.0,0.2)
end:constant

begin:species
  name = s1

  # multiply density by real particle density
  density = particle_density * profile_x * profile_y

  # Set the temperature to be zero
  temperature_x = 0.0
  temperature_y = temperature_x(s1)

```

```

# Set the minimum density for this species
number_density_min = 0.3 * particle_density
end:species

begin:species
  name = s2

  # Just copy the density for species s1
  number_density = number_density(s1)

  # Just copy the temperature from species s1
  temperature_x = temperature_x(s1)
  temperature_y = temperature_y(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * particle_density
end:species

```

This creates the same output as before. It is now trivial to modify the profiles later. For example:

```

begin:constant
  particle_density = 100.0 # Particle number density
  profile_x = gauss(x, 0.0, 1.0)
  profile_y = gauss(y, 0.0, 1.0)
end:constant

begin:species
  name = s1

  # multiply density by real particle density
  density = particle_density * profile_x * profile_y

  # Set the temperature to be zero
  temperature_x = 0.0
  temperature_y = temperature_x(s1)

  # Set the minimum density for this species
  number_density_min = 0.3 * particle_density
end:species

begin:species
  name = s2

  # Just copy the density for species s1
  number_density = number_density(s1)

  # Just copy the temperature from species s1
  temperature_x = temperature_x(s1)
  temperature_y = temperature_y(s1)

  # Set the minimum density for this species

```

```

number_density_min = 0.3 * particle_density
end:species

```

This changes the code to run with a Gaussian density profile rather than a step function. Again, this can be extended as far as required.

4.8 Delta-f method in EPOCH

To help reduce the impact of numerical noise in certain simulations, the delta-f method may be used in **EPOCH** for specified particle species. The delta-f method effectively subtracts a background distribution f_0 when calculating currents due to the motion of markers; when the particle distribution function f is close to f_0 , this substantially reduces the statistical noise in the currents. Only current deposition is implemented differently, and the equations of motion of the markers are not modified in this delta-f approach.

The component of the currents associated with the background f_0 may be in principle be calculated analytically, but the delta-f implementation in **EPOCH** assumes (but does not check) that the total background current is zero.

In order to use the delta-f method **EPOCH** must be compiled with the `#DELTAF_METHOD` precompiler flag enabled. Standard **EPOCH** input simulations are not affected by switching on this flag, but the user may then choose to treat certain species in the plasma using the delta-f method. To enable delta-f calculation for a species the background distribution function f_0 must be defined in the input block. f_0 is specified using variables similar to those used to specify f . The current implementation of delta-f allows only a spatially uniform drifting Maxwellian f_0 , with temperatures T_x , T_y and T_z allowed to differ from each other. In 3D, for the case where the temperature in each direction is equal, we have

$$f_0 = n_0 (2\pi T)^{-3/2} \exp\left(\frac{m(\mathbf{v} - \mathbf{v}_d)^2}{2k_B T}\right).$$

The parameters `number_density_background`, `temperature(x,y,z)_background` and `drift(x,y,z)_background` in each species specification in the input deck are used to set f_0 . `number_density_background = 0` is the default and is equivalent to not using the delta-f method.

For example, the electron species component of an input deck solved using delta-f might be written:

```

begin:species
    name = electron
    charge = -1.0
    mass = 1.0
    fraction = 0.3
    temp = 1e8
    temperature_background = 1e8
    number_density = 1e20
    number_density_background = 1e20
end:species

```

`section of input.deck`

Additional distribution function diagnostic options are supplied for the Delta-f version. Standard diagnostics work as usual based on the total distribution function f but it is also possible to output the Delta-f component of the distribution functions by adding “`output_deltaf=T`” in “`dist_fn`” components of the input deck.

An example input deck is supplied in the 1D **EPOCH** version as `twostream_deltaf.deck`. This uses the delta-f method to solve the weak-beam two stream instability. The bulk plasma species is solved using the delta-f method, since this evolves very little, and mostly supports the Langmuir waves that the weak beam interacts with. The relative change to the beam species is large, and the standard PIC

method, rather than delta-f is used to model this species. A comparison of the electric field diagnostics between standard and delta-f simulations shows a substantial decrease in noise.

Further details of the method are provided in a set of slides included in the appendix of this document. See “Slides from Delta-f Talk” (Section C).

5 Basic examples of using **EPOCH**

In this section we outline a few worked examples of setting up problems using the **EPOCH** input deck.

5.1 Electron two stream instability

An obvious simple test problem to do with **EPOCH** is the electron two stream instability. An example of a nice dramatic two stream instability can be obtained using EPOCH1D by setting the code with the following input deck files.

`input.deck`

```
begin:control
    nx = 400
    nparticles = 20000

    # Final time of simulation
    t_end = 1.5e-1

    # Size of domain
    x_min = 0
    x_max = 5.0e5
end:control

begin:boundaries
    bc_x_min = periodic
    bc_x_max = periodic
end:boundaries

begin:constant
    drift_p = 2.5e-24
    temp = 273
    dens = 10
end:constant

begin:species
    # Rightwards travelling electrons
    name = Right
    charge = -1
    mass = 1.0
    fraction = 0.5
    temperature_x = temp
    drift_px = drift_p
    number_density = dens
end:species
```

```

begin:species
    # Leftwards travelling electrons
    name = Left
    charge = -1
    mass = 1.0
    fraction = 0.5
    temperature_x = temp
    drift_px = -drift_p
    number_density = dens
end:species

begin:output
    # Number of timesteps between output dumps
    dt_snapshot = 1.5e-3
    # Number of dt_snapshot between full dumps
    full_dump_every = 1

    # Properties at particle positions
    particles = always
    px = always

    # Properties on grid
    grid = always
    ex = always
    ey = always
    ez = always
    bx = always
    by = always
    bz = always
    jx = always
    average_particle_energy = always
    mass_density = never + species
    charge_density = always
    number_density = always + species
    temperature = always + species

    # Extended io
    distribution_functions = always
end:output

```

In this example, the constant block sets up constants for the momentum space drift, the temperature and the electron number density. The two species blocks set up the two drifting Maxwellian distributions and the constant density profile. The final output from this simulation is shown in Figure 8.

Time = 0.15 s

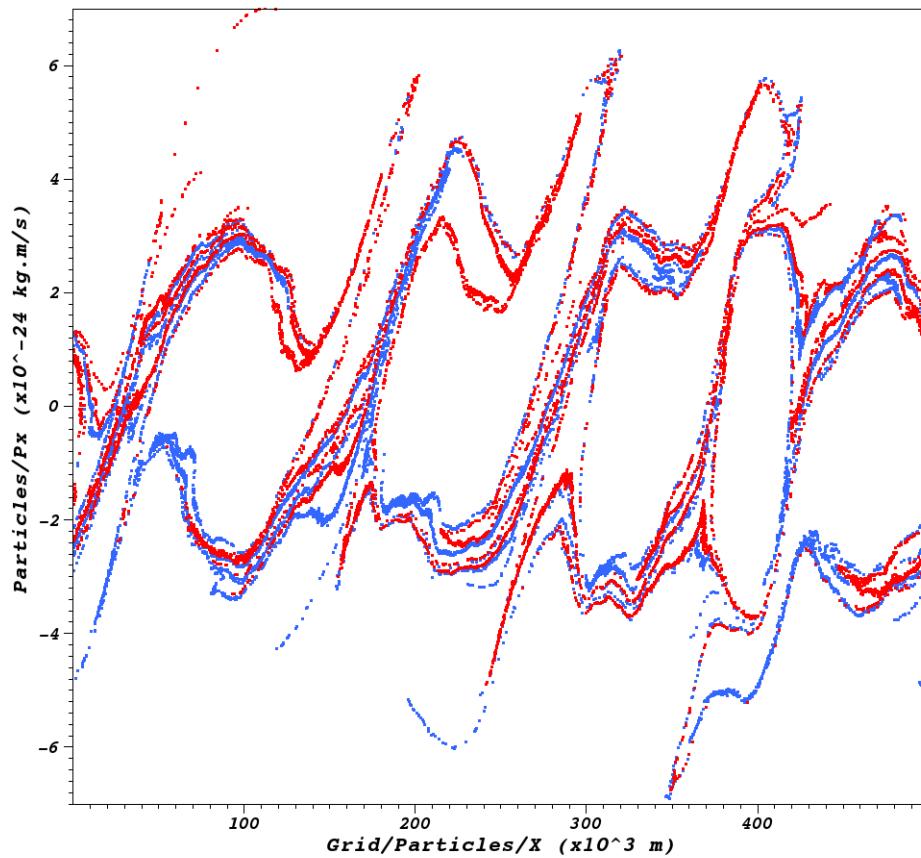


Figure 8: The final state of the electron phase space for the two stream instability example.

5.2 Structured density profile in EPOCH2D

A simple but useful example for EPOCH2D is to have a highly structured initial condition to show that this is still easy to implement in EPOCH . A good example initial condition would be:

```
begin:control
nx = 500
ny = nx
x_min = -10 * micron
x_max = -x_min
y_min = x_min
y_max = x_max
nsteps = 0
nparticles = 40 * nx * ny
end:control
```

```
begin:boundaries
bc_x_min = periodic
bc_x_max = periodic
bc_y_min = periodic
bc_y_max = periodic
end:boundaries
```

```
begin:constant
den_peak = 1.0e19
end:constant
```

```

begin:species
  name = Electron
  number_density = den_peak * sin(4.0 * pi * x / length_x + pi / 4) \
                  * (sin(8.0 * pi * y / length_y) + 1)
  number_density_min = 0.1 * den_peak
  charge = -1.0
  mass = 1.0
  fraction = 0.5
end:species

begin:species
  name = Proton
  number_density = number_density(Electron)
  charge = 1.0
  mass = 1836.2
  fraction = 0.5
end:species

begin:output
  number_density = always + species
end:output

```

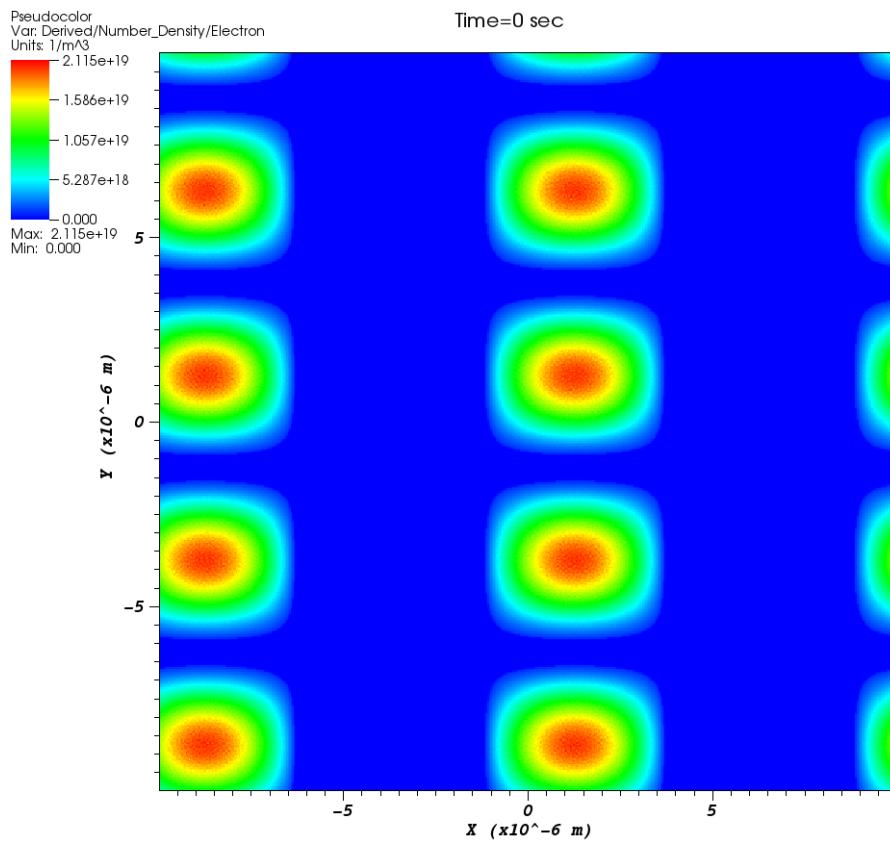


Figure 9: Complex 2D density structure

The species block for **Electron** is specified first, setting up the electron density to be a structured 2D sinusoidal profile. The species block for **Proton** is then set to match the density of **Electron**, enforcing charge neutrality. On its own this initial condition does nothing and so only needs to run for 0 timesteps (**nsteps = 0** in input.deck). The resulting electron number density should look like Figure 9

5.3 A hollow cone in 3D

A more useful example of an initial condition is to create a hollow cone. This is easy to do in both 2D and 3D, but is presented here in 3D form.

input.deck

```
begin:control
    nx = 250
    ny = nx
    nz = nx
    nparticles = 2 * nx * ny * nz
    x_min = -10 * micron
    x_max = -x_min
    y_min = x_min
    y_max = x_max
    z_min = x_min
    z_max = x_max
    nsteps = 0
end:control

begin:boundaries
    bc_x_min = simple_laser
    bc_x_max = simple_outflow
    bc_y_min = periodic
    bc_y_max = periodic
    bc_z_min = periodic
    bc_z_max = periodic
end:boundaries

begin:output
    number_density = always + species
end:output

begin:constant
    den_cone = 1.0e22
    ri = abs(x - 5.0e-6) - 0.5e-6
    ro = abs(x - 5.0e-6) + 0.5e-6
    xi = 3.0e-6 - 0.5e-6
    xo = 3.0e-6 + 0.5e-6
    r = sqrt(y^2 + z^2)
end:constant

begin:species
    name = proton
    charge = 1.0
    mass = 1836.2
    fraction = 0.5
    number_density = if((r gt ri) and (r lt ro), den_cone, 0.0)
    number_density = if((x gt xi) and (x lt xo) and (r lt ri), \
                        den_cone, number_density(proton))
    number_density = if(x gt xo, 0.0, number_density(proton))
end:species

begin:species
```

```

name = electron
charge = -1.0
mass = 1.0
fraction = 0.5
number_density = number_density(proton)
end:species

```

To convert this to 2D, simply replace the line `r = sqrt(y^2+z^2)` with the line `r = abs(y)`. The actual work in these initial conditions is done by the three lines inside the block for the `Proton` species. Each of these lines performs a very specific function:

1. Creates the outer cone. Simply tests whether `r` is within the range of radii which corresponds to the thickness of the cone and if so fills it with the given density. Since the inner radius is `x` dependent this produces a cone rather than a cylinder. On its own, this line produces a pair of cones joined at the tip.
2. Creates the solid tip of the cone. This line just tests whether the point in space is within the outer radius of the cone and within a given range in `x`, and fills it with the given density if true.
3. Cuts off all of the cone beyond the solid tip. Simply tests if `x` is greater than the end of the cone tip and sets the density to zero if so.

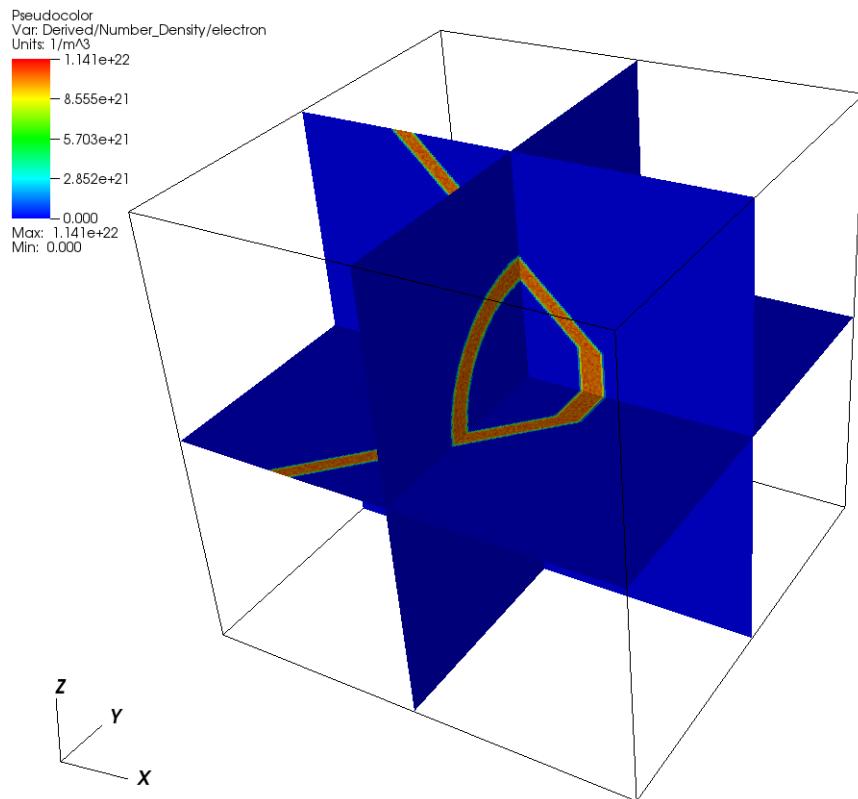


Figure 10: Cone initial conditions in 3D

This deck produces and initial condition which looks like Figure 10 and Figure 11 in 3D and 2D respectively.

5.4 Focussing a Gaussian Beam

A laser can be driven on the boundary so that it focusses on a given spot. Basic details of how to do this are available on wikipedia under “Gaussian Beam”. The following deck gives an example for a laser

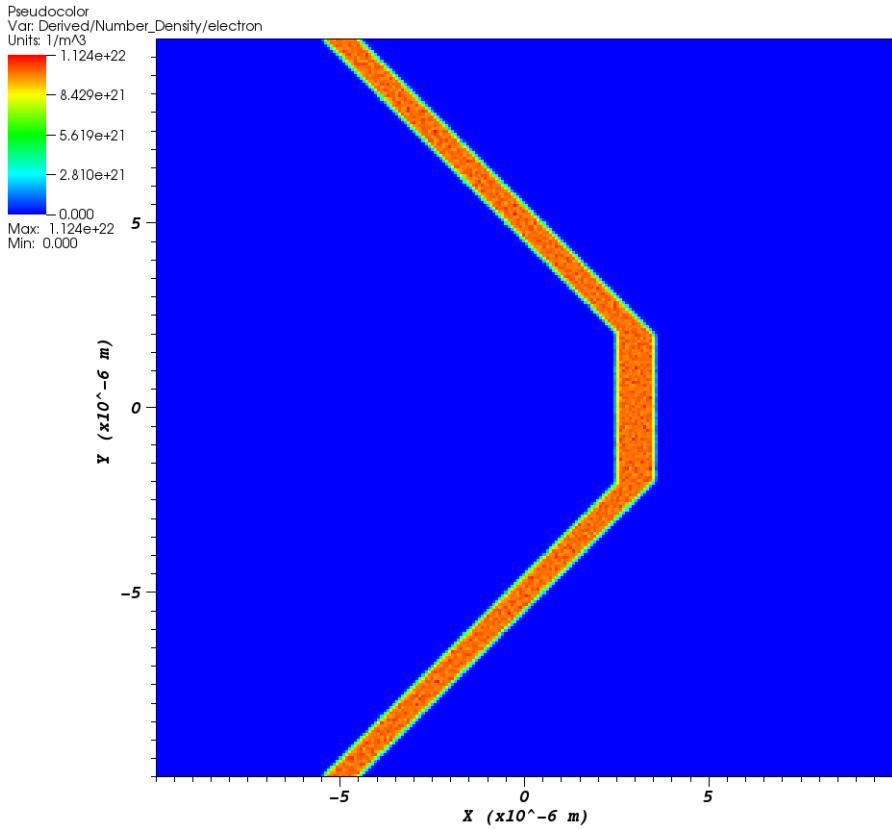


Figure 11: Cone initial conditions in 2D

attached to x_{\min} . The important parameters are the waist size (the size of the spot at the focus point) and the distance from the attached boundary to the focus. In this case the waist size is $0.9\mu\text{m}$, and the distance is 15 micron, meaning the focus is at $x = 5\mu\text{m}$.

[input.deck](#)

```

begin:control
nx = 500
ny = nx

# Final time of simulation
t_end = 100 * femto

# Size of domain
x_min = -10 * micron
x_max = -x_min
y_min = x_min
y_max = x_max

stdout_frequency = 10
end:control

begin:boundaries
bc_x_min = simple_laser
bc_x_max = open
bc_y_min = periodic
bc_y_max = periodic
end:boundaries

begin:constant

```

```

lambda0 = 1.0 * micron

# These two set the beam focus
w_0 = 0.9 * micron #Beam waist size
x_spot = 15 * micron #Distance from x_min to spot

# These are the parameters calculated for driving the laser
# These should not need to be modified
x_R = pi * w_0^2 / lambda0 #Rayleigh range
RC = x_spot * (1.0 + (x_R/x_spot)^2) #Radius of curvature on x_min
w_bnd = w_0 * sqrt( 1.0 + (x_spot/x_R)^2) #Spot size at x_min
gouy = atan(x_spot/x_R) # Gouy phase shift at x_min
end:constant

begin:laser
boundary = x_min
intensity_w_cm2 = 1.0e15
lambda = lambda0

phase = 2.0 * pi/ lambda0 * y^2 / (2.0 * RC) - gouy
profile = gauss(y, 0, w_bnd)
end:laser

begin:output
# Number of timesteps between output dumps
dt_snapshot = 25 * femto

# Properties on grid
grid = always
ey = always
bz = always
end:output

```

The deck is based on the laser test deck supplied with Epoch, with a modified laser and longer runtime. Other classes of beam (Bessel etc) can be created similarly. The results from this deck once the laser is well into the domain are in Figure 12

The details presented above are a first rough guide to using **EPOCH**. To clearly understand **EPOCH** it is best to now try some simple examples. To view the results you will have to jump forward to “**Using IDL to visualise data**” (Section 6).

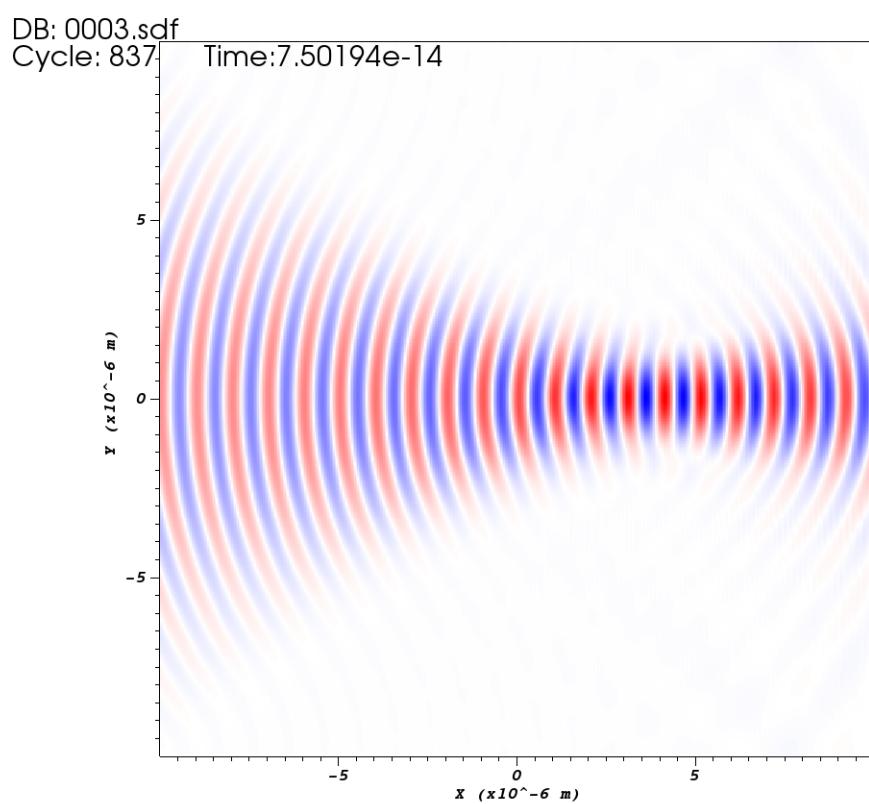


Figure 12: Plot of E_y for a laser focussed to a spot size of 0.9 micron at $x = 5\mu\text{m}$.

6 Using IDL to visualise data

The **EPOCH** distribution comes with procedures for loading and inspecting SDF self-describing data files. The IDL routines are held in the **SDF/IDL/** directory. There is also a procedure named **Start.pro** in each of the **epoch*d/** directories which is used to set up the IDL environment.

To load data into IDL, navigate to one of the base directories (eg. **epoch/epoch2d/** where **epoch/** is the directory in which you have checked out the git repository) and type the following:

```
$> idl Start
IDL Version 8.1 (linux x86_64 m64). (c) 2011, ITT Visual Information Solutions
Installation number: .
Licensed for use by: STAR404570-5University of Warwick

% Compiled module: TRACKEX_EVENT.
% Compiled module: ISOPILOT.
% Compiled module: READVAR.
% Compiled module: LOADSDFFILE.
% Compiled module: SDFHANDLEBLOCK.
% Compiled module: SDFGETPLAINMESH.
% Compiled module: SDFGETLAGRANMESH.
% Compiled module: SDFGETPOINTMESH.
% Compiled module: SDFGETPLAINVAR.
% Compiled module: SDFGETPOINTVAR.
% Compiled module: SDFGETCONSTANT.
% Compiled module: SDFCHECKNAME.
% Compiled module: INIT_SDFHELP.
% Compiled module: SWAPCHR.
% Compiled module: GETDATA.
% Compiled module: GETSTRUCT.
% Compiled module: EXPLORE_DATA.
% Compiled module: EXPLORE_STRUCT.
% Compiled module: LIST_VARIABLES.
% Compiled module: QUICK_VIEW.
% Compiled module: GET_WKDIR.
% Compiled module: SET_WKDIR.
% Compiled module: INIT_STARTPIC.
% Compiled module: INIT_WIDGET.
% Compiled module: GENERATE_FILENAME.
% Compiled module: COUNT_FILES.
% Compiled module: LOAD_RAW.
% Compiled module: GET_SDF_METATEXT.
% Compiled module: VIEWER_EVENT_HANDLER.
% Compiled module: EXPLORER_EVENT_HANDLER.
% Compiled module: XLOADCT_CALLBACK.
% Compiled module: LOAD_DATA.
% Compiled module: DRAW_IMAGE.
% Compiled module: LOAD_META_AND_POPULATE_SDF.
% Compiled module: CLEAR_DRAW_SURFACE.
% Compiled module: SDF_EXPLORER.
% Compiled module: EXPLORER_LOAD_NEW_FILE.
% Compiled module: CREATE_SDF_VISUALIZER.
% Compiled module: VIEWER_LOAD_NEW_FILE.
```

```
% LOADCT: Loading table RED TEMPERATURE  
IDL>
```

This starts up the IDL interpreter and loads in all of the libraries for loading and inspecting SDF files.

We begin by inspecting SDF file contents and finding out what variables it contains. To do this we execute the `list_variables` procedure call which is provided by the `EPOCH` IDL library.

At each timestep for which `EPOCH` is instructed to dump a set of variables a new data file is created. These files take the form `0000.sdf`. For each new dump the number is incremented. The procedure call accepts up to two arguments. The first argument is mandatory and specifies the number of the SDF file to be read in. This argument can be any integer from 0 to 9999. It is padded with zeros and the suffix ‘.sdf’ appended to the end to give the name of the data file. eg. 99 ⇒ ‘0099.sdf’. The next argument is optional. The keyword “`wkdir`” specifies the directory in which the data files are located. If this argument is omitted then the currently defined global default is used. Initially, this takes the value “`Data`” but this can be changed using the `set_wkdir` procedure and queried using the `get_wkdir()` function.

```
IDL> list_variables,0,"Data"  
Available elements are  
1) EX (ELECTRIC_FIELD) : 2D Plain variable  
2) EY (ELECTRIC_FIELD) : 2D Plain variable  
3) EZ (ELECTRIC_FIELD) : 2D Plain variable  
4) BX (MAGNETIC_FIELD) : 2D Plain variable  
5) BY (MAGNETIC_FIELD) : 2D Plain variable  
6) BZ (MAGNETIC_FIELD) : 2D Plain variable  
7) JX (CURRENT) : 2D Plain variable  
8) JY (CURRENT) : 2D Plain variable  
9) JZ (CURRENT) : 2D Plain variable  
10) WEIGHT_ELECTRON (PARTICLES) : 1D Point variable  
11) WEIGHT_PROTON (PARTICLES) : 1D Point variable  
12) PX_ELECTRON (PARTICLES) : 1D Point variable  
13) PX_PROTON (PARTICLES) : 1D Point variable  
14) GRID_ELECTRON (GRID) : 2D Point mesh  
15) GRID_PROTON (GRID) : 2D Point mesh  
16) EKBAR (DERIVED) : 2D Plain variable  
17) EKBAR_ELECTRON (DERIVED) : 2D Plain variable  
18) EKBAR_PROTON (DERIVED) : 2D Plain variable  
19) CHARGE_DENSITY (DERIVED) : 2D Plain variable  
20) NUMBER_DENSITY (DERIVED) : 2D Plain variable  
21) NUMBER_DENSITY_ELECTRON (DERIVED) : 2D Plain variable  
22) NUMBER_DENSITY_PROTON (DERIVED) : 2D Plain variable  
23) GRID (GRID) : 2D Plain mesh  
24) GRID_EN_ELECTRON (GRID) : 1D Plain mesh  
25) EN_ELECTRON (DIST_FN) : 3D Plain variable  
26) GRID_X_EN_ELECTRON (GRID) : 2D Plain mesh  
27) X_EN_ELECTRON (DIST_FN) : 3D Plain variable  
28) GRID_X_PX_ELECTRON (GRID) : 2D Plain mesh  
29) X_PX_ELECTRON (DIST_FN) : 3D Plain variable  
IDL>
```

Each variable in the SDF self-describing file format is assigned a name and a class as well as being

defined by a given variable type. The `list_variables` procedure prints out the variable name followed by the variable's class in parenthesis. Following the colon is a description of the variable type.

To retrieve the data, you must use the `getdata()` function call. The function must be passed a snapshot number, either as the first argument or as a keyword parameter "snapshot". It also accepts the `wkdir` as either the second argument or the keyword parameter "wkdir". If it is omitted altogether, the current global default is used. Finally, it accepts a list of variables or class of variables to load. Since it is a function, the result must be assigned to a variable. The object returned is an IDL data structure containing a list of named variables.

To load either a specific variable or a class of variables, specify the name prefixed by a forward slash. It should be noted here that the IDL scripting language is not case sensitive so P_x can be specified as either "/Px" or "/px".

We will now load and inspect the "Grid" class, this time omitting the optional "wkdir" parameter. This time we will load from the third dump file generated by the `EPOCH` run, which is found in the file `0002.sdf` since the dump files are numbered starting from zero.

6.1 Inspecting Data

```
IDL> gridclass = getdata(1,/grid)

IDL> help,gridclass,/structures
** Structure <22806408>, 11 tags, length=536825024, data length=536825016, refs=1:
  FILENAME      STRING    'Data/0001.sdf'
  TIMESTEP      LONG      43
  TIME          DOUBLE   5.0705572e-15
  GRID_ELECTRON STRUCT   -> <Anonymous> Array[1]
  GRID_PROTON   STRUCT   -> <Anonymous> Array[1]
  GRID          STRUCT   -> <Anonymous> Array[1]
  X             DOUBLE   Array[1024]
  Y             DOUBLE   Array[512]
  GRID_EN_ELECTRON
                STRUCT   -> <Anonymous> Array[1]
  GRID_X_EN_ELECTRON
                STRUCT   -> <Anonymous> Array[1]
  GRID_X_PX_ELECTRON
                STRUCT   -> <Anonymous> Array[1]
IDL> help,gridclass.grid,/structures
** Structure <1701168>, 5 tags, length=12376, data length=12376, refs=2:
  X             DOUBLE   Array[1025]
  Y             DOUBLE   Array[513]
  LABELS        STRING   Array[2]
  UNITS         STRING   Array[2]
  NPTS          LONG     Array[2]
```

Here we have used IDL’s built in “`help`” routine and passed the “`/structures`” keyword which prints information about a structure’s contents rather than just the structure itself.

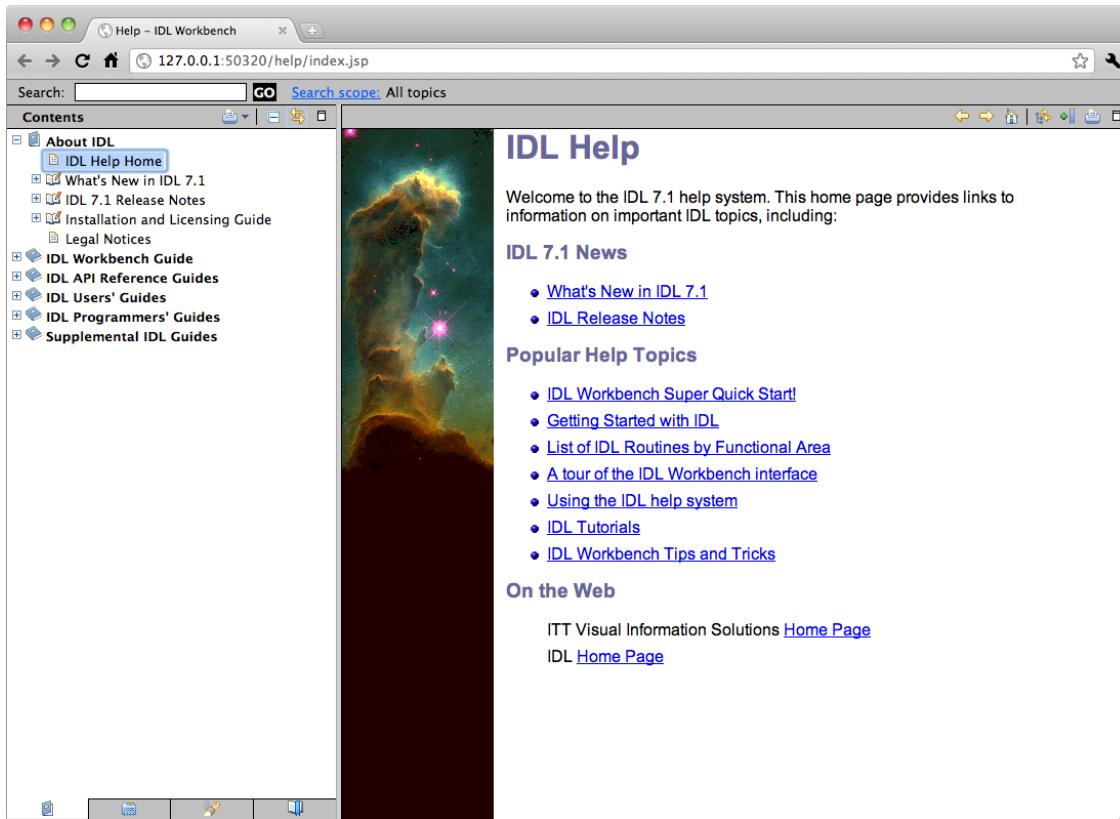
Since “`Grid`” is a class name, all variables of that class have been loaded into the returned data structure. It is a nested type so many of the variables returned are structures themselves and those variables may contain structures of their own.

The “`Grid`” variable itself contains “`x`” and “`y`” arrays containing the x and y coordinates of the 2D cartesian grid. The other variables in the “`Grid`” structure are metadata used to identify the type and properties of the variable. In order to access the “`Grid`” variable contained within the “`gridclass`” data structure we have used the “`.`” operator. In a similar way, we would access the “`x`” array contained within the “`Grid`” variable using the identifier “`gridclass.grid.x`”.

6.2 Getting Help in IDL

IDL is a fairly sophisticated scripting environment with a large library of tools for manipulating data. Fortunately, it comes with a fairly comprehensive array of documentation. This can be accessed by typing `?` at the IDL prompt.

```
IDL> ?
% ONLINE_HELP: Starting the online help browser.
IDL>
```



The documentation is divided into books aimed at users or developers and is fully searchable and cross indexed.

6.3 Manipulating And Plotting Data

Once the data has been loaded from the SDF file we will want to extract the specific data we wish to analyse, perhaps perform some mathematical operations on it and then plot the results.

To do this we must learn a few basic essentials about the IDL scripting language. Since we are all familiar with the basic concepts shared by all computer programming languages, I will just provide a brief overview of the essentials and leave other details to the excellent on-line documentation.

IDL supports multidimensional arrays similar to those found in the FORTRAN programming language. Whole array operations are supported such as “`5*array`” to multiply every element of “`array`” by 5. Also matrix operations such as addition and multiplication are supported.

The preferred method for indexing arrays is to use brackets. It is possible to use parenthesis instead but this usage is deprecated. Column ordering is the same as that used by FORTRAN, so to access the (i, j, k) th element of an array you would use “`array[i, j, k]`”. IDL arrays also support ranges so “`array[5:10, 3, 4]`” will return a one dimensional array with five elements. “`array[5:*`” specifies elements five to n of an n element array. “`array[* , 3]`” picks out the third row of an array.

There are also a wide range of routines for querying and transforming arrays of data. For example, finding minimum and maximum values, performing FFTs, etc. These details can all be found by searching the on-line documentation.

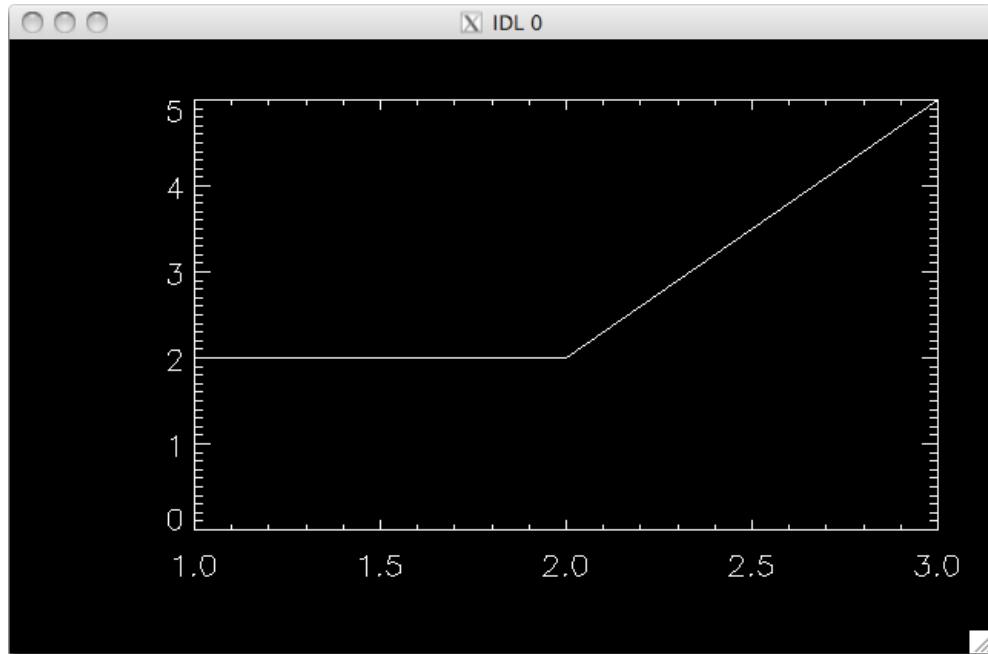
Finally, IDL is a full programming language so you can write your own functions and procedures for processing the data to suit your needs.

6.4 1D Plotting in IDL

The most commonly performed plot and perhaps the most useful data analysis tool is the 1D plot. In IDL, this is performed by issuing the command `plot, x, y` where “`x`” and “`y`” are one dimensional arrays of equal length. For each element “`x[i]`” plotted on the x -axis the corresponding value “`y[i]`” is plotted along the y -axis. As a simple example:

```
IDL> plot,[1,2,3],[2,2,5]
```

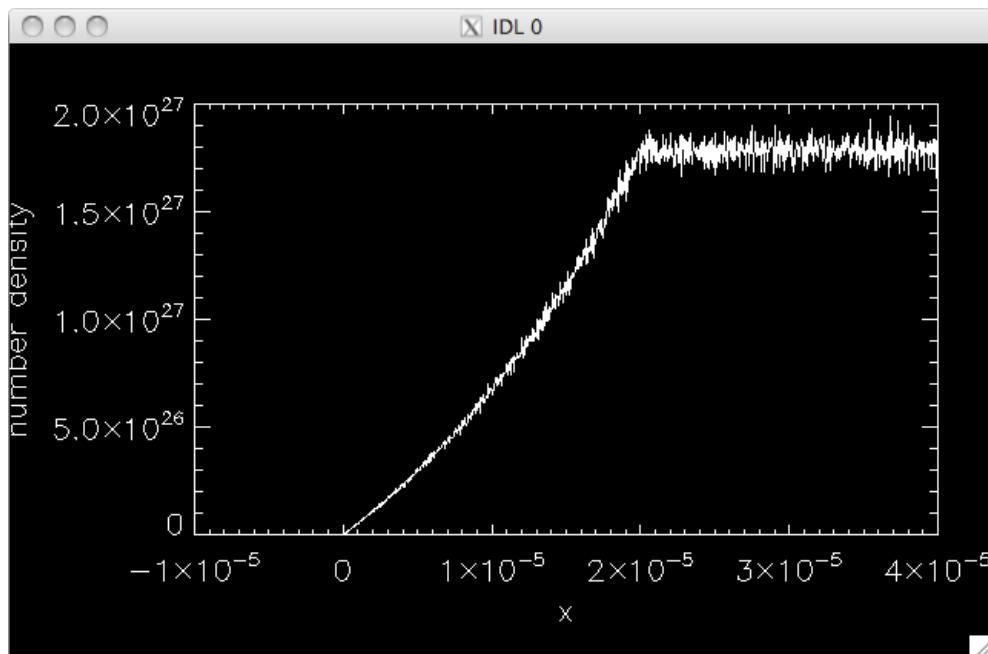
Gives rise to the following plot:



As a more concrete example, we will now take a one-dimensional slice through the 2D “Number_Density” array read in from our SDF data file. In this example we will give the x and y axes labels by passing extra parameters to the “plot” routine. A full list of parameters can be found in the on-line documentation. In this example we also make use of the “\$” symbol which is IDL’s line continuation character.

```
IDL> data = getdata(0)
IDL> plot,data.x,data.number_density[*,256],xtitle='x', $
IDL>      ytitle='number density'
```

This command generates the following plot:



6.5 Postscript Plots

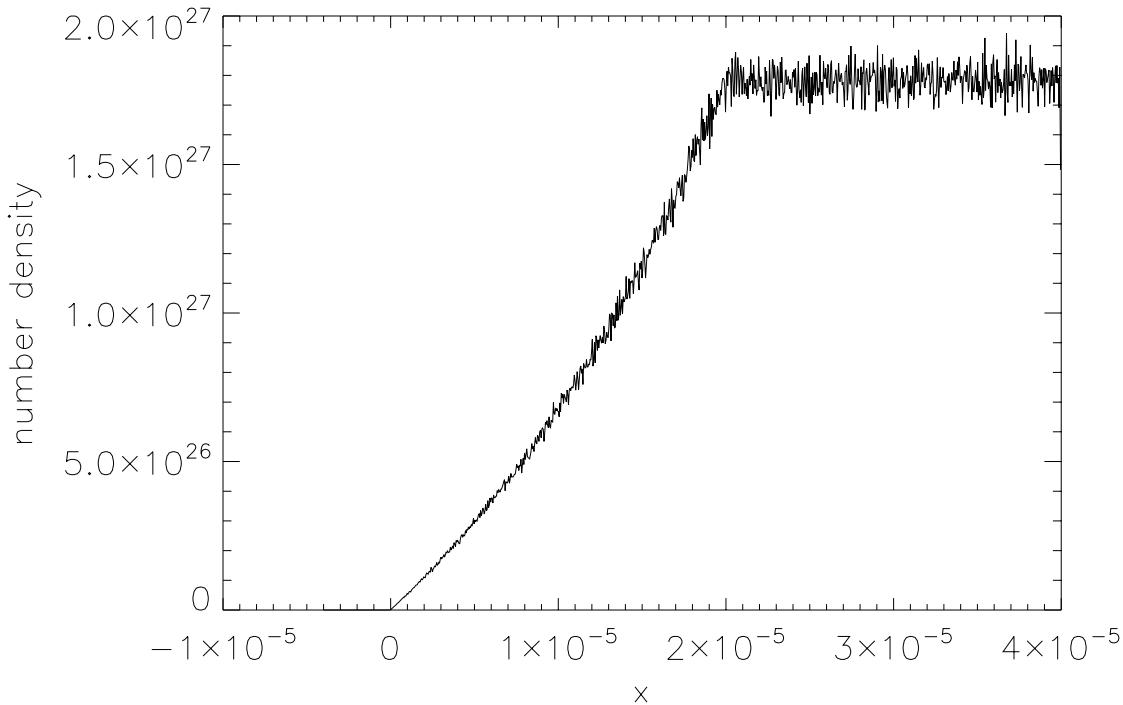
The plots shown so far have just been screen-shots of the interactive IDL plotting window. These are fairly low quality and could included as figures in a paper.

In order to generate publication quality plots, we must output to the postscript device. IDL maintains a graphics context which is set using the `set_plot` command. The two most commonly used output devices are “x” which denotes the X-server and “ps” which is the postscript device. Once the desired device has been selected, various attributes of its behaviour can be altered using the `device` procedure. For example, we can set the output file to use for the postscript plot. By default, a file with the name “`idl.ps`” is used.

Note that this file is not fully written until the postscript device is closed using the `device,/close` command. When we have finished our plot we can resume plotting to screen by setting the device back to “x”.

```
IDL> set_plot,'ps'  
IDL> device,filename='out.ps'  
IDL> plot,data.x,data.number_density[*,256],xtitle='x', $  
IDL>      ytitle='number density',charsize=1.5  
IDL> device,/close  
IDL> set_plot,'x'
```

This set of commands results in the following plot being written to a file named “`out.ps`”.



By default, IDL draws its own set of fonts called “Hershey vector fonts”. Much better looking results can be obtained by using a postscript font instead. These options are passed as parameters to the `device` procedure. More details can be found in the on-line documentation under “Reference Guides ⇒ IDL Reference Guide ⇒ Appendices ⇒ Fonts”.

6.6 Contour Plots in IDL

Whilst 1D plots are excellent tools for quantitative analysis of data, we can often get a better qualitative overview of the data using 2D or 3D plots.

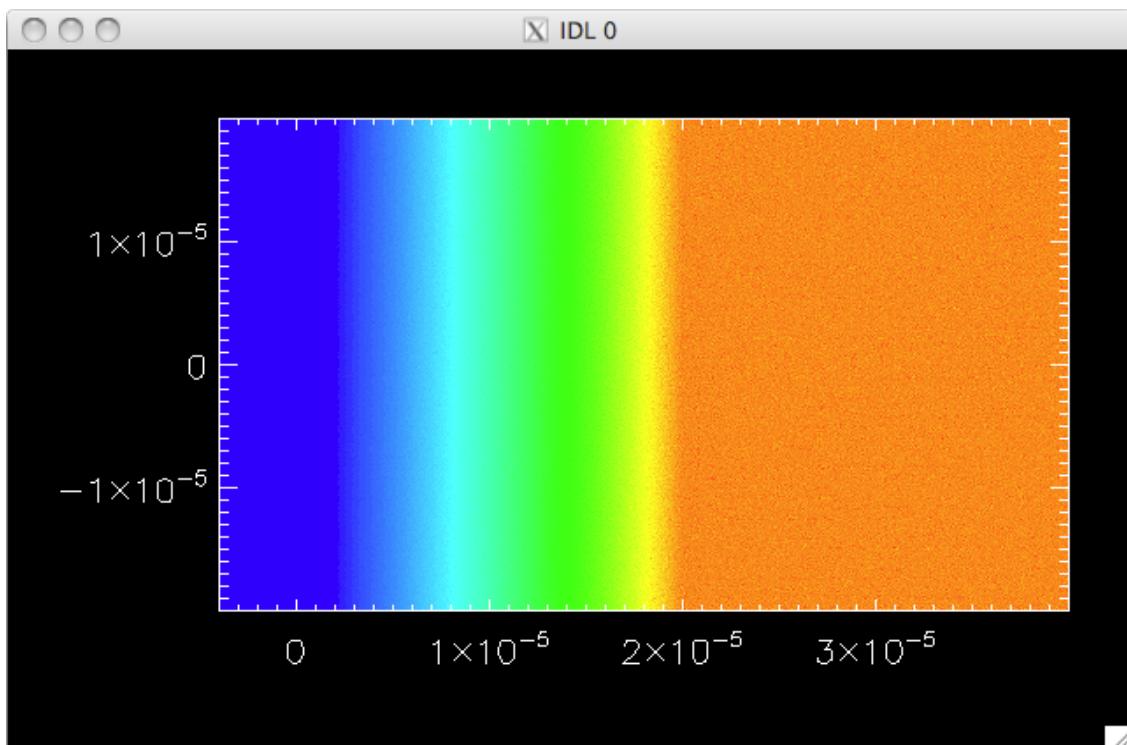
One commonly used plot for 2D is the contour plot. The aptly named `contour,z,x,y` procedure takes a 2D array of data values, “`z`”, and plots them against `x` and `y` axes which are specified in the 1D

“x” and “y” arrays. The number of contour lines to plot is specified by the “nlevels” parameter. If the “/fill” parameter is used then IDL will fill each contour level with a solid colour rather than just drawing a line at the contour value.

The example given below plots a huge number of levels so that a smooth looking plot is produced. “xstyle=1” requests that the x axes drawn exactly matches the data in the “x” variable rather than just using a nearby rounded value and similarly for “ystyle=1”.

```
IDL> n=100
IDL> levels=max(data.number_density)*findgen(n)/(n-1)
IDL> colors=253.*findgen(n)/(n-1)+1
IDL> contour,data.number_density,data.x,data.y,xstyle=1,ystyle=1, $
IDL>      levels=/fill,c_colors=colors
```

Issuing these commands gives us the contour plot shown below. Note that the colour table used is not the default one but has been constructed to be similar to the one used by VisIt.

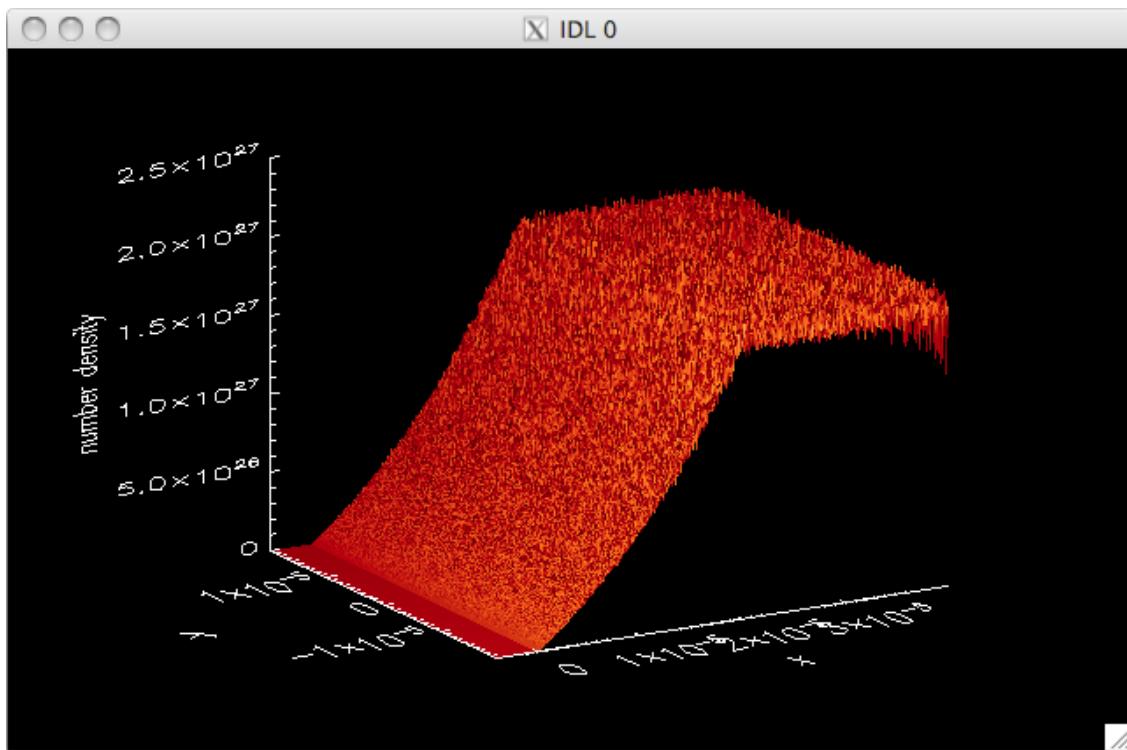


6.7 Shaded Surface Plots in IDL

Another method for visualising 2D datasets is to produce a 3D plot in which the data is elevated in the z direction by a height proportional to its value. IDL has two versions of the surface plot. `surface` produces a wireframe plot and `shade_surf` produces a filled and shaded one. As we can see from the following example, many of IDL’s plotting routines accept the same parameters and keywords.

The first command shown here, `loadct,3`, asks IDL to load the third colour table which is “RED_TEMPERATURE”.

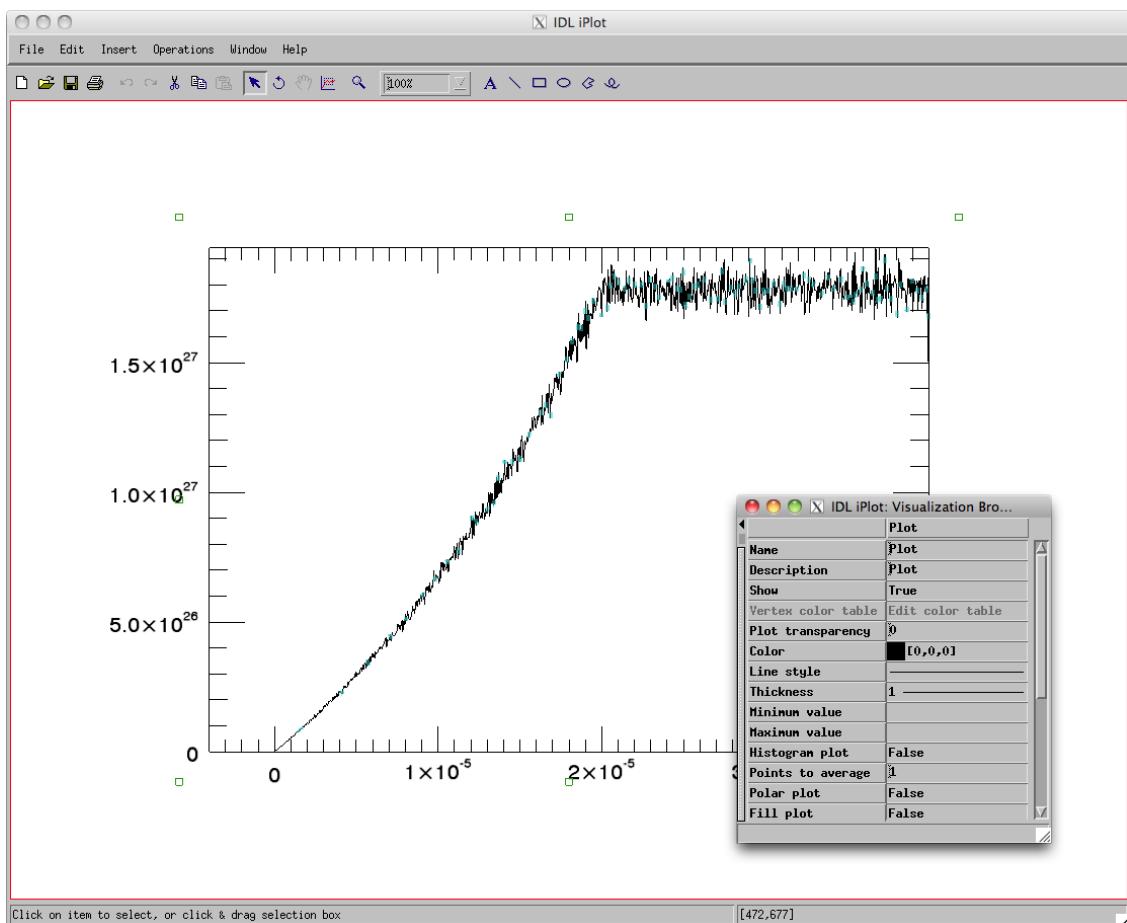
```
IDL> loadct,3
IDL> shade_surf,data.number_density,data.x,data.y,xstyle=1, $
IDL>      ystyle=1,xtitle='x',ytitle='y',ztitle='number density',charsize=3
```



6.8 Interactive Plotting

Finally, in recent versions of IDL it is now possible to perform all of these plot types in an interactive graphical user interface. The corresponding procedures are launched with the commands `iplot`, `icontour` and `isurface`.

```
IDL> iplot,data.x,data.number_density[*,256]
```



IDL is an extremely useful tool but it also comes with a fairly hefty price tag. If you are not part of an organisation that will buy it for you then you may wish to look into a free alternative. It is also a proprietary tool and you may not wish to work within the restrictions that this imposes.

There are a number of free tools available which offer similar functionality to that of IDL, occasionally producing superior results.

For a simple drop-in replacement, the GDL project aims to be fully compatible and works with the existing **EPOCH** IDL libraries after a couple of small changes. Other tools worth investigating are “yorick” and “python” with the “SciPy” libraries. At present there is no SDF reader for either of these utilities but one may be developed if there is sufficient demand.

7 Using VisIt to visualise data

7.1 LLNL VisIt

LLNL’s VisIt software is a parallel data visualisation package (<https://wci.llnl.gov/simulation/computer-codes/visit>). **EPOCH** comes with source code for the plug-in needed to allow VisIt to load the SDF output files which are generated by **EPOCH**. There are full manuals for VisIt which can be downloaded from the above link so no further details will be given here. To build the plug-in, first ensure that the visit binary is in the \$PATH environment variable. Then simply type “make visit” in one of the **epoch{1,2,3}d** directories. For more experienced users of VisIt, the xml file which is used to generate the plug-in is supplied in the VisIt subdirectory, called **SDF2.xml**.

Whilst IDL is an excellent tool for visualising 1D and 2D datasets, it is extremely poor when it comes to dealing with 3D data. For this purpose, we recommend the use of the “VisIt” visualisation tool.

The other great advantage that VisIt has over IDL is the ability to render in parallel, enabling the visualisation of huge datasets which IDL would be incapable of dealing with.

- Initially developed by the Department of Energy (DOE) Advanced Simulation and Computing Initiative (ASCI)
- Now developed and maintained by the Lawrence Livermore National Laboratory along with a group of external contributors
- Written in C++ and supports python and Java interfaces
- Available for UNIX (Irix, Tru64, AIX, Linux, Solaris), Mac OS X (10.3 - 10.9), and Windows platforms
- Open source and freely available under the BSD license
- Plots, operators and database readers are implemented as plugins allowing the VisIt to be dynamically extended at run-time
- Powerful set of tools for manipulating, analysing and visualising 3D datasets
- Parallel and distributed architecture for visualising huge data sets

7.2 Obtaining And Installing VisIt

Both the source code and pre-compiled binaries are available for download from the projects web page which is found at the URL <https://wci.llnl.gov/simulation/computer-codes/visit>

The screenshot shows a web browser window with the URL wci.llnl.gov. The page is titled "Weapons and Complex Integration". A sidebar on the left lists "Computer Codes" (ALE3D, MERCURY, MIRANDA, SPHERAL) and "Visit" (Co-Design, Support Libraries, Basic Science Simulation, Visualization). The main content area shows the "Visit Executables" page with a navigation bar including "Visit Home", "Downloads", "What's New", "Screen Shots", "Gallery", and "FAQs". Below the navigation is a text block about the Visit executables, followed by instructions for parallel support, a note about NERSC's filesystem, and a citation section. At the bottom, it mentions DOE funding and links to Visit 2.10.2 release notes, install script, install notes, and md5 checksums.

There are full instructions for compiling the project from source code along with build scripts written to help ease the process. However, this is not recommended as it is an extremely large tool and the compilation takes hours to complete. It is usually far easier to download a pre-compiled binary which matches your system architecture.

However, occasionally compilation may be a necessary step. Linux in particular is a moving target and it is not always possible to find a binary which matches the particular combination of libraries installed on your system.

The easiest way to install the VisIt tool is to ask the system administrator to do it for you. However, this may not always be the best option. The system in question may be run by someone who is not concerned with your particular software needs or has insufficient skills to deal with the task. In any case, VisIt has a fairly rapid release schedule and you may find that some functionality you need is not present in the version installed on the machine.

Fortunately, for all these scenarios it is usually quite easy to install a copy in your own home directory. Just find a binary on the web page <https://wci.llnl.gov/simulation/computer-codes/visit/executables> which closely matches your machine and download it. This can be unpacked into your home directory with the command `tar xzf visit2_10_2.linux-x86_64-ubuntu14.tar.gz`. The actual name of the file will vary depending on which version you downloaded. This will unpack the VisIt binary into a subdirectory named `visit/`. Now all that is necessary is to add this to your search path. eg. `export PATH=$HOME/visit/bin:$PATH`

These instructions illustrate the steps required for installing your own copy of VisIt when you have no other choice. VisIt is an extremely large program, so if a version is already available then it is usually better to use the installed version.

The machines at Warwick have a recent version of VisIt installed which is available via the “modules” system. To make use of it you must first issue the command `module load visit`.

7.3 Compiling The Reader Plugin

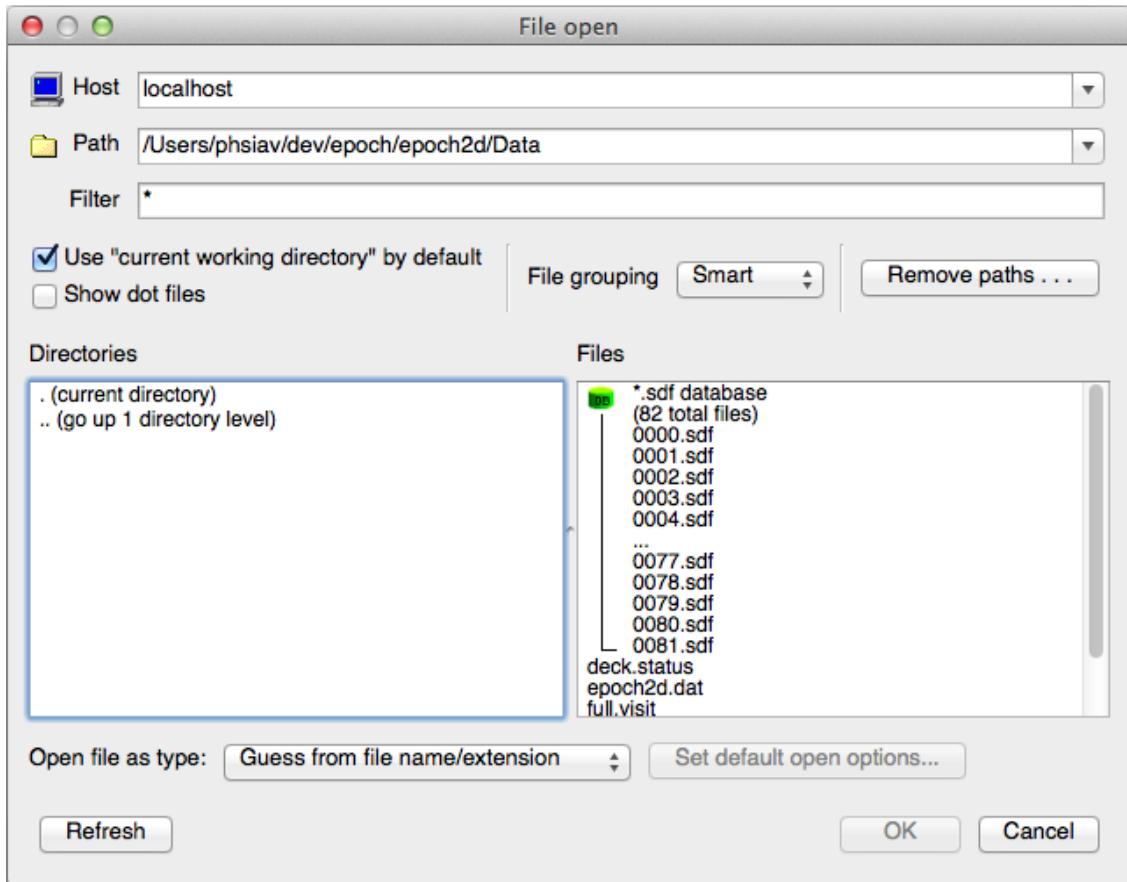
One piece of compilation which is almost always necessary is that of the SDF reader plugin. This is shipped as source code in a subdirectory of the **EPOCH** repository. It is located in the **VisIt/** subdirectory of the main **epoch/** directory. The reader will work for any SDF file generated by any code which uses the SDF I/O routines. You do not need a separate reader for each version of **EPOCH**.

To compile, first navigate to one of the **epoch*2d/** directories in your **EPOCH** repository. Just type “make visit” and the build scripts should take care of the rest. The SDF reader plugin will be installed into the directory **\$HOME/.visit/linux-intel/plugins/databases/** on your system. Note that the **linux-intel/** component will vary depending on your machine operating system and architecture.

Each time you install a new version of VisIt you must recompile the reader to match the new installation. It will also occasionally be necessary to recompile when changes occur to the SDF data format or the reader plugin itself. The developers will notify users if this is the case, although it does no harm to regularly recompile the reader as a matter of course.

We will see later that it is possible to do remote data visualisation with VisIt in which the GUI is launched and interacted with on one machine and the data files are located on a separate machine entirely. In this situation the reader must be installed on the remote machine and must match the setup there. The setup on the local machine is unimportant. In fact it is not even necessary to have the plugin installed on the local machine. This is particularly useful when using a Windows environment to analyse data located on a remote UNIX workstation.

7.4 Loading Data Into VisIt

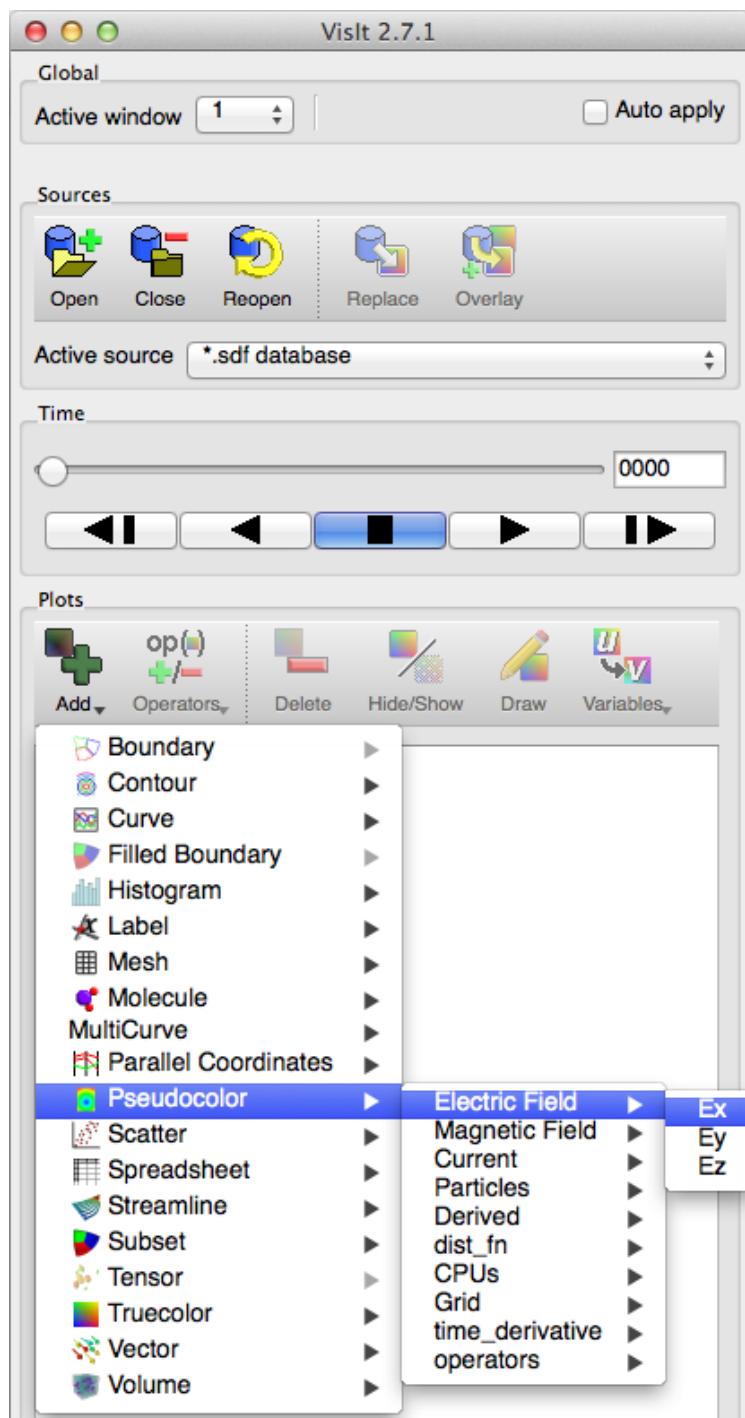


The most straightforward method for loading data into VisIt is to start the application and then browse the filesystem for the dataset you are interested in. This is done by selecting “File ⇒ Open file” from the VisIt menu bar. A file selection dialogue will appear allowing you to browse directories along with the options to filter the results according to a given regular expression and grouping options. By default, VisIt will attempt to group all files containing the same suffix and some kind of numbering system into a sort of virtual database.

The right-hand pane of this window shows a list of selected files which will appear in the main VisIt window when you are finished.

An alternative method of specifying the data file to open is to pass a command line option when the tool is launched. An example of this method is `visit -o Data/0000.sdf`. When the file is specified in this manner the list of files shown in the VisIt window will also include the full list of files in the dataset's subdirectory and all the files in the current working directory. The other SDF files will be grouped together in a virtual database.

Yet another method for selecting the dataset to use is by opening a previously saved session file. We will discuss this further in a later section.



Once an SDF file has been successfully loaded the “Add” menu item will become un-greyed and the cycle numbers for each file in the virtual database will be displayed. If we navigate to one of the plot types we are able to select the variable to plot from a drop-down list.

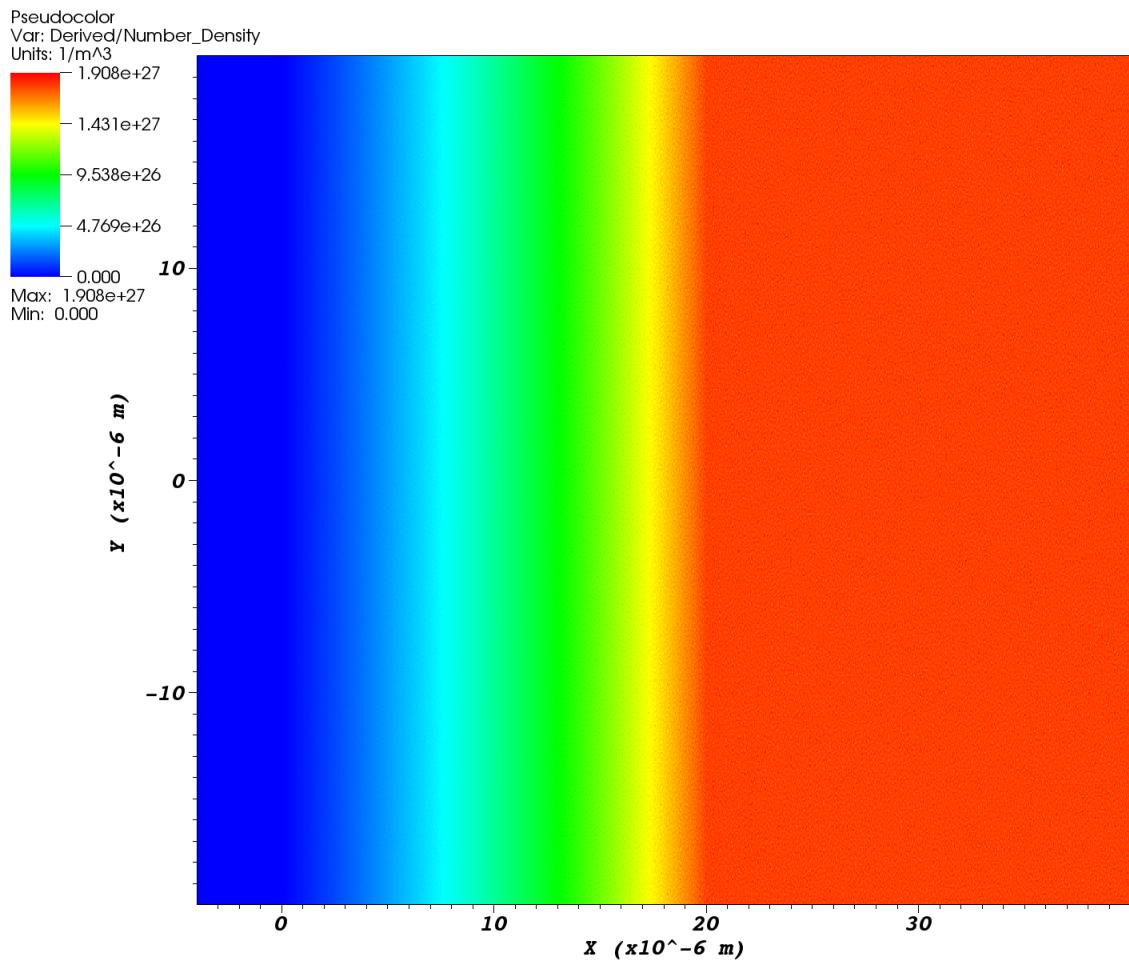
7.5 Contour Plots in VisIt

We will now replicate each of the plots which we generated using IDL in earlier sections. For reasons which will soon become clear we begin with the contour plot and move on to the 1D plot in the next section.

Having opened the same dataset we were using in the IDL discussion we now select the “Add” menu item. Notice that many of the plot types listed here are greyed out and cannot be selected. This is because many of the plots are dependent on the type or dimensionality of the variable to be plotted. If our dataset contains no variables which match the required properties for a plot, the plot menu will be disabled.

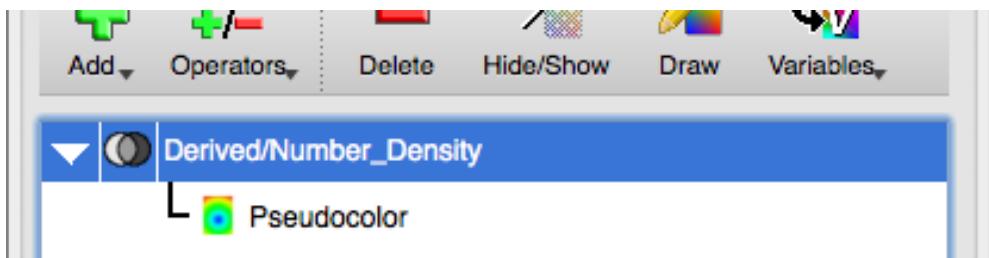
For the current dataset there is no “Boundary” plot available since this requires multi-material data and none of our variables meet that criteria.

The list contains a menu item for a “Contour” plot. We are not going to select this item since it only generates a contour plot with lines indicating each contour level and not a filled version. Instead we choose “Add ⇒ Pseudocolor ⇒ Derived ⇒ Number_Density” and then hit the “Draw” button.

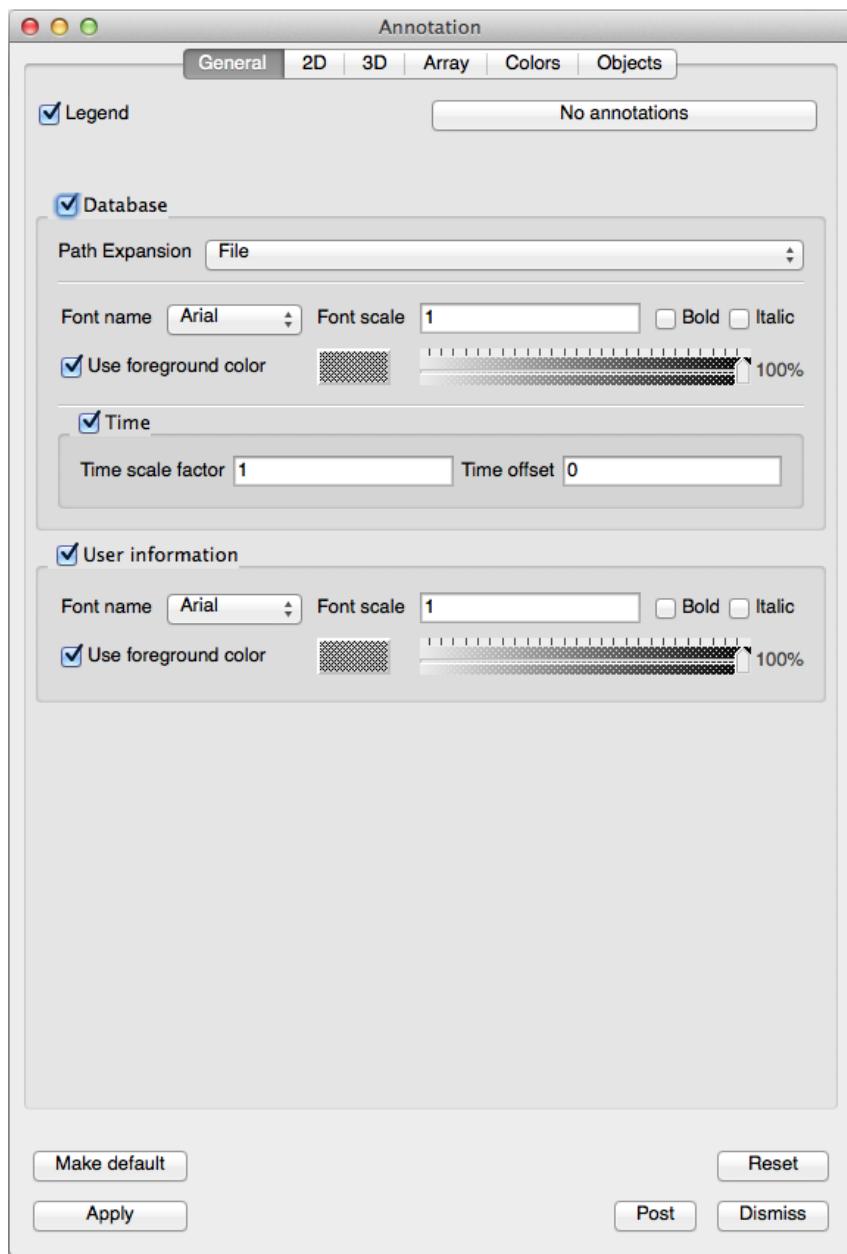


There are many settings which can alter the visual appearance of plots generated by VisIt. The first point of call is usually to open up the “Plot Attributes” or “Operator Attributes” dialogue corresponding to the plot in question. A simpler method for accomplishing this task is to double-click on the plot in the main VisIt menu pane which will launch the corresponding “Plot Attributes” dialogue.

If it is the operator attributes you wish to change, click on the white arrow on the left hand side of the plot in the main VisIt menu pane. This will drop down to reveal a list containing the plot and all operators acting on it. Double-clicking on an operator will launch the corresponding “Operator Attributes” dialogue.



Another important tool for controlling the appearance of plots can be found in “Controls ⇒ Annotation” from the VisIt menu bar. This allows all of the plot annotations to be modified such as the legend, title, axis labels, etc.



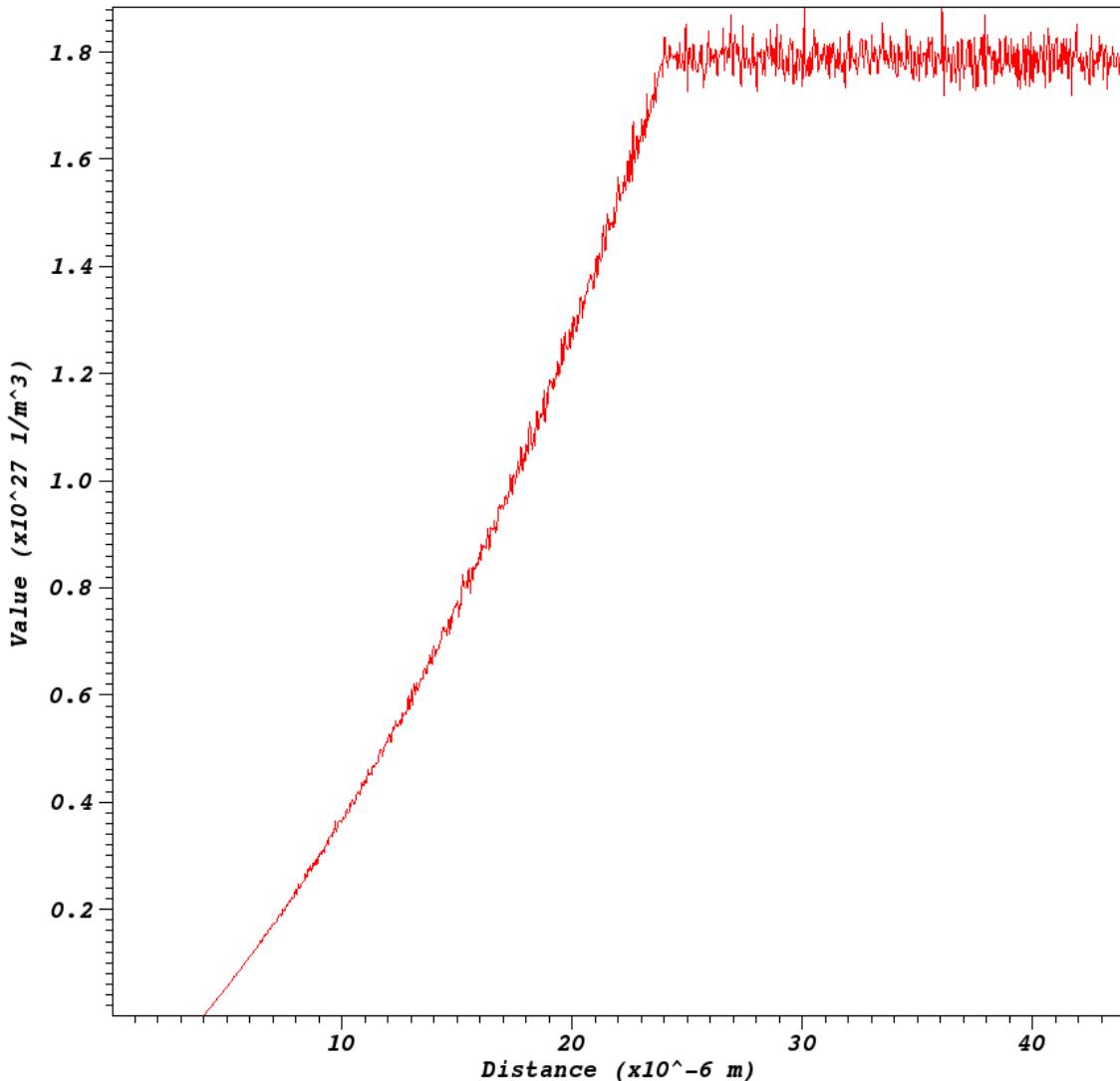
7.6 1D Plotting in VisIt

A 1D plot in VisIt is called a “Curve” plot. We already mentioned that this was greyed out because we have no one dimensional variables in our data file.

The solution to this dilemma is the lineout operator which extracts a one dimensional array from a 2D or 3D variable. This operator is selected by pressing the button with red and blue lines located at the top of the plot window.



Once the button has been pressed, we can click and drag anywhere in the “Pseudocolor” plot window. When we release the mouse button a new plot window pops up containing a “Curve” plot of the data just selected.

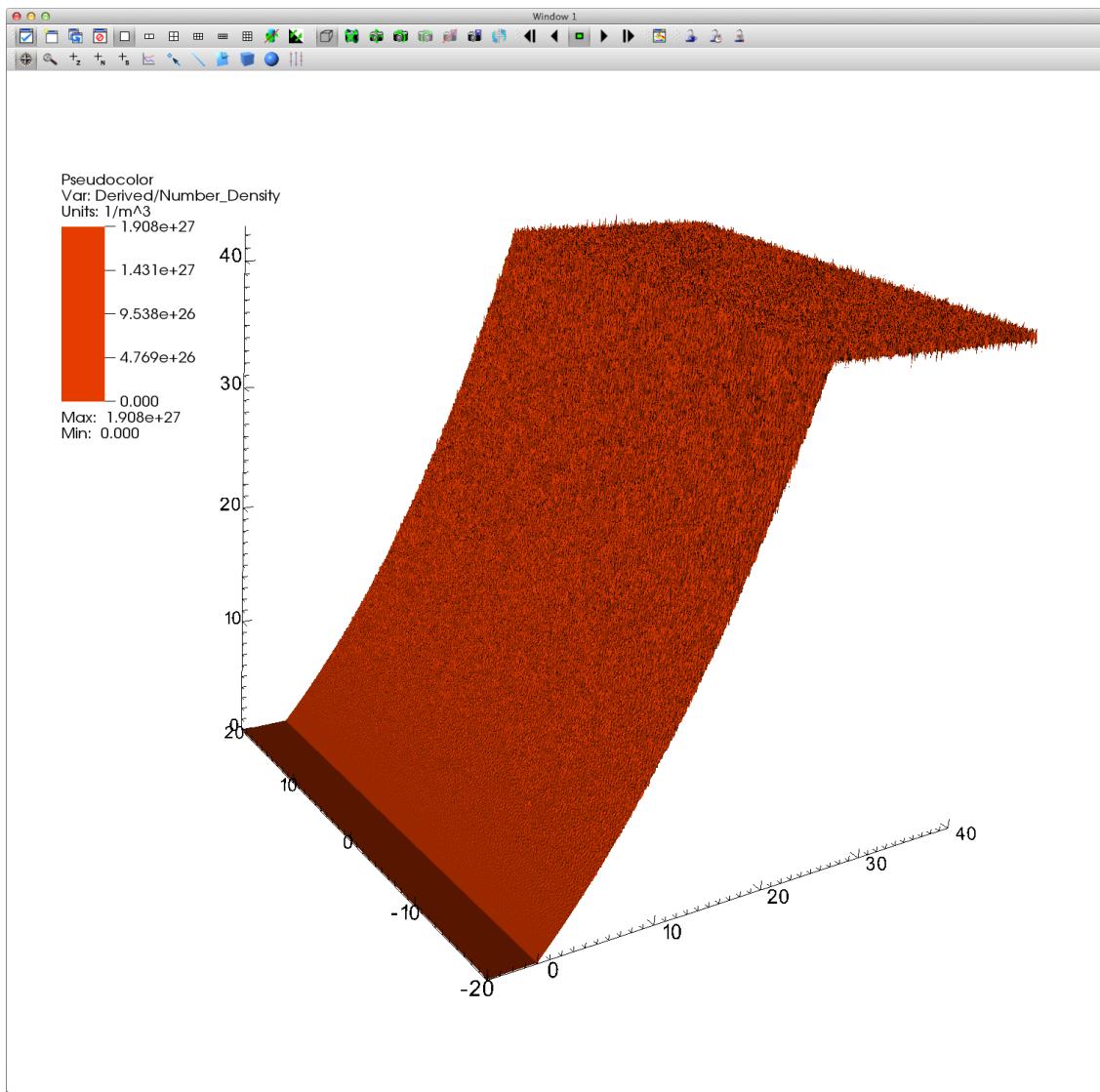


In order to change the attributes for this plot, we must first select “Active window” number 2 in the main VisIt pane.

7.7 Shaded Surface Plots in VisIt

Again, we will confusingly refuse to pick the obvious plot type for this task. There is “Surface” plot listed in the menu. However, most of the time the “Elevator” operator does what we want and also gives us more flexibility.

The first step is to do a “Pseudocolor” plot of “Number_Density” as we did before. Next select the “Operator Attributes ⇒ Transforms ⇒ Elevate” menu item. In the pop up dialogue click on the “Elevation height relative to XY limits?” and then “Apply”. Click “Yes” when the warning dialogue pops up.

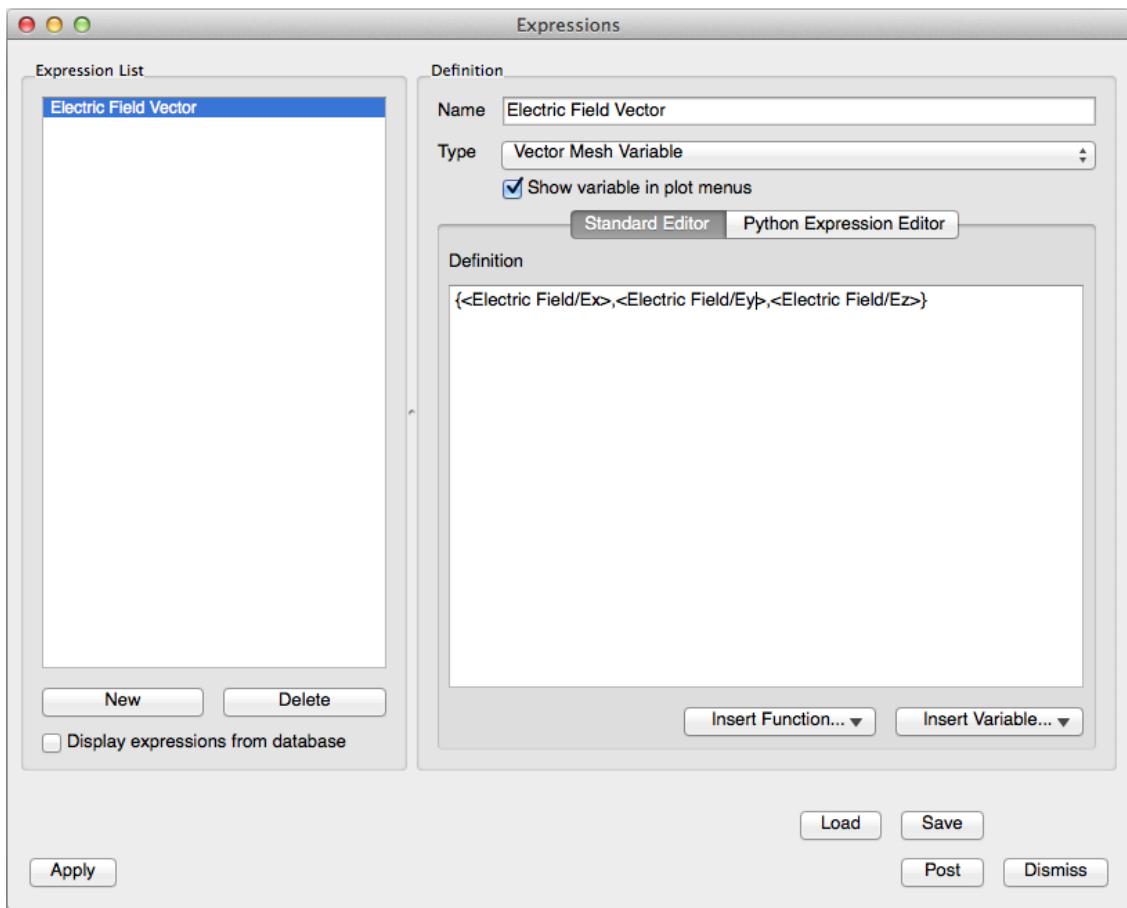


To make this plot look similar to the one generated by IDL, we have changed the colour table using “Controls ⇒ Color table”. We also changed the axis appearance with the annotations menu discussed earlier and changed the height of the elevation using the min and max operator attributes.

7.8 Creating User-Defined Expressions

VisIt comes with an extremely powerful method of manipulating data before visualising the results. The basic idea is that an array is transformed by applying a set of mathematical functions on all its elements and then the result is defined as a new variable. Once defined, this variable behaves in exactly the same way as any of the variables read from the data file.

As an example, we can combine the three components of electric field to generate a single electric field vector.



Now when we return to the “Add” menu we see that the “Vector” and “Streamline” plot types now have an entry for our newly defined vector.

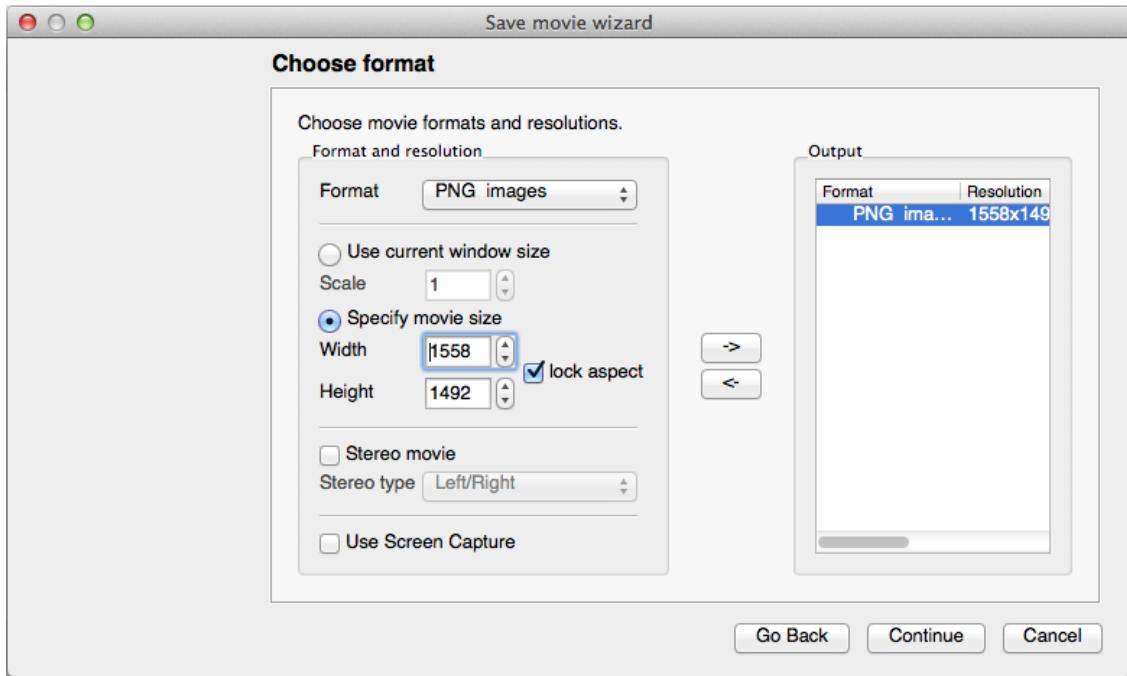
7.9 Creating Movies

A compelling visualisation of numerically generated data is often made by combining a series of images into a movie. This can be an invaluable method for illustrating the basic behaviour of a system as it changes over time. Alternatively rotating around a 3D scene can sometimes give a much better idea of the structure in the model being presented. There can also be much to gain by constructing visual fly-throughs of a scene, dynamically slicing through sets of data or combinations of all these techniques.

VisIt provides several facilities for generating movies from your data. The simplest of these is to select the “File ⇒ Save movie” menu item. This pops up a movie wizard which will walk you through the process of generating a simple linear movie based on the time-advancing snapshots represented by your virtual database of files. Alternatively you can select one of the pre-defined movie templates which manipulate the currently selected plot and create a movie from that.

Creating a simple time advancing movie is as simple as walking through the wizard dialogue and selecting from the self-explanatory options presented to you.

For many uses, the wizard will give exactly the desired results. However it is occasionally useful to have a little more control over how the movie is created. In such cases it can be useful to specify an image format such as “PNG” to save to rather than “MPEG”. VisIt will then generate one image per frame and number them consecutively. At the end of the process the images can be converted into a movie using whatever tool best accomplishes the task.



Another useful tip is to select the “Later, tell me the command to run” radio button. This will output a long command which can run from a UNIX terminal screen. The advantage is that no X session is required so the command can be run in the background. It also becomes a simple task to interrupt the job at any point and resume it from where it left off at a later date. In a similar manner it is easy to resume a job which crashes half way through for any reason.

More complex movies can be created by using VisIt’s keyframing facility which allows you to change animation attributes such as view or plot attributes as the animation progresses. Further information about this somewhat complex task can be found in the on-line help.

Finally, you can use VisIt’s python scripting interface to programmatically describe the details of each frame as the movie progresses. This approach offers far more flexibility in what can be achieved but is also much more involved and time consuming than the previous two methods. Again, further information on this subject can be found in the on-line help system.

7.10 Remote Visualisation

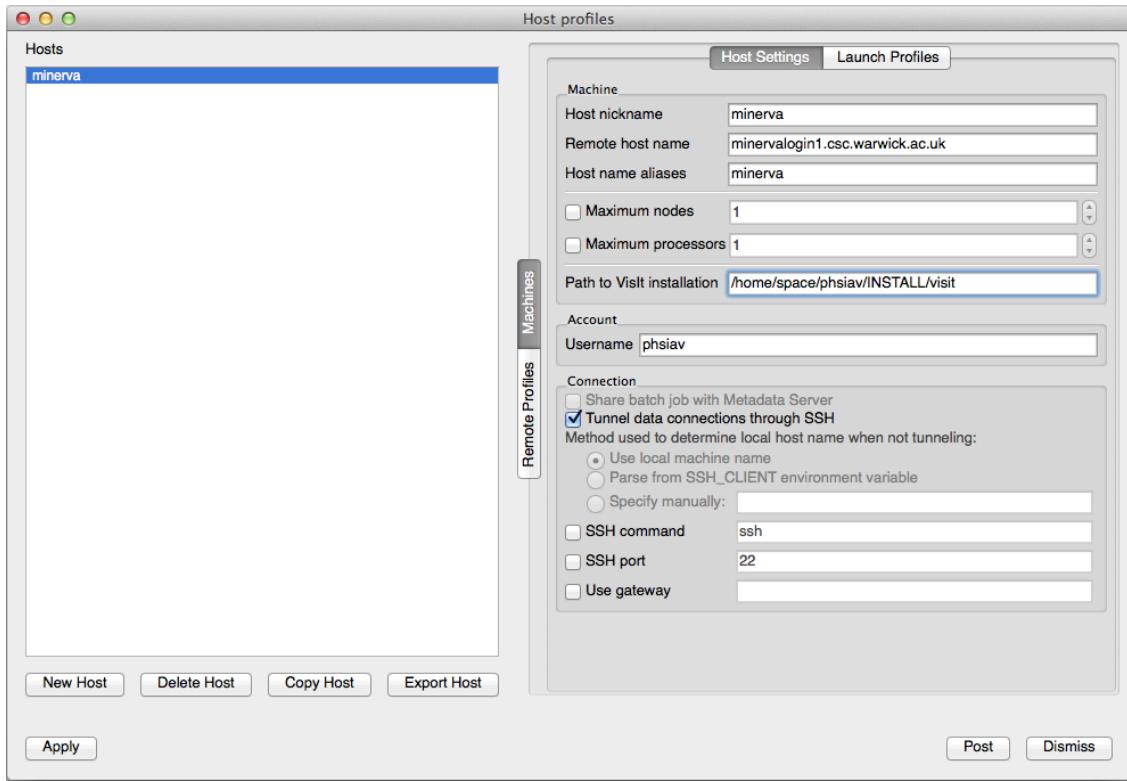
It was mentioned earlier that it is possible to perform remote visualisation using VisIt. This is a process in which the data files being interrogated reside on a different machine to the one on which the VisIt GUI runs and where the results are plotted.

This method of working can be extremely useful when the data is generated on a powerful machine located in an external environment such as a large cluster. Another common use is when **EPOCH** is executed on a UNIX machine and the desktop used for visualisation is running Windows.

It is sometimes possible to run a graphical tool on the remote machine and tunnel the X-server session through to the local machine but this can be quite slow and unstable. When connecting to a remote VisIt instance the only data which needs to be sent between machines is the pre-rendered image and a few simple plotting commands. Naturally, this can be a much faster approach.

Also, as mentioned before, it is possible to use a machine on which the reader plugin is difficult or impossible to compile for and connect to a machine on which the reader is already installed.

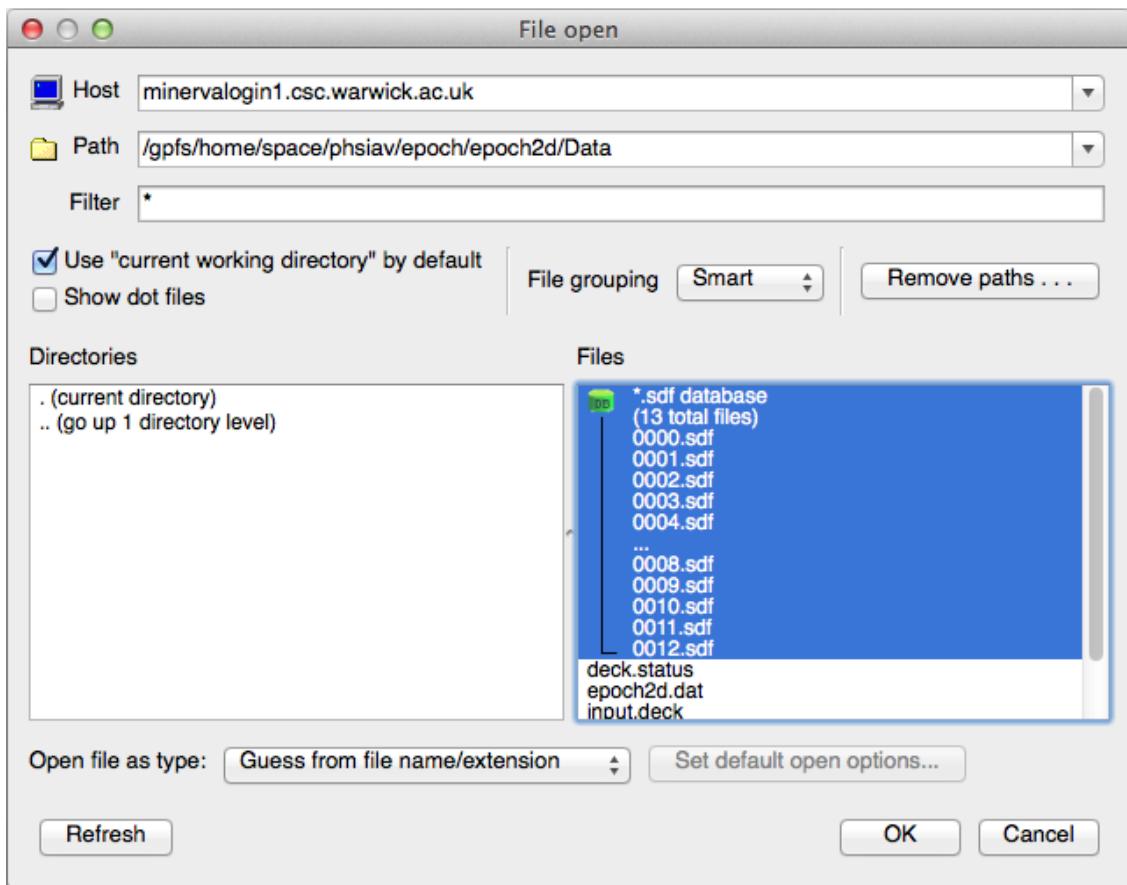
In order to use the remote visualisation facility, you must first set up a “Host profile” for the remote machine using the “Options ⇒ Host profiles” menu item. The pre-compiled binaries are shipped with a long list of pre-defined host profiles. These are unnecessary for anyone not affiliated and can safely be removed by deleting the directory `$HOME/visit/current/.visit/` (assuming you have unpacked the VisIt tarball into your home directory).



Create a new profile by clicking on the “New Host” button and filling out some of the form fields. The important ones to change are “Host nickname”, “Remote host name”, “Host name aliases” and “Username”. If the visit binary is not in your default search path on the remote machine then you must specify its location by filling in the “Path to VisIt installation” field.

Now click “Apply” and “Dismiss” followed by the “Options ⇒ Save Settings” menu item to ensure that the profile is saved for future sessions.

Data on the remote machine can now be loaded by selecting “File ⇒ Open file” and picking the desired host profile from the drop down list of “Hosts”. VisIt will wait for the remote process to launch and then continue with the file selection procedure but now displaying files located on the remote machine rather than the local one. From this point on everything should work as before except you should see the name of the remote machine in the “Selected files” dialogue.



7.11 Parallel Visualisation

Parallel visualisation is performed in almost exactly the same manner as remote visualisation. Again, you must create a host profile for the purpose except this time you need to set up a parallel launch profile in the ‘‘Launch Profiles’’ tab pane. Click the ‘‘New Profile’’ button, give the profile a name and then set the required options in the ‘‘Parallel’’ tab on the bottom section of the page. Selecting the ‘‘Launch parallel engine’’ radio button will allow you to set the various launch options which relate to the cluster on which the job will run.

The major difference now is due to the fact that VisIt must be launched by an external job script which fits in with the queueing system used by the parallel machine. Usually you will need to consult with the system administrator of the cluster to confirm which launch method and arguments to use.

The details of job launch can be better understood by reading through the ‘‘User documentation’’ section provided at <http://www.visitusers.org/>. Of particular help here is the ‘‘Getting VisIt to run in parallel’’ section and the ‘‘How VisIt Launching works’’ entry in the ‘‘Developer documentation’’ section.

A Changes between versions

A.1 Changes between version 3.1 and 4.0

A.1.1 Changes to the Makefile

Some changes have been made to the Makefile. These are documented in Section 2.4.

The following compile-time defines have been added to the Makefile:

- NO_IO
- PARTICLE_ID
- PARTICLE_ID4
- COLLISIONS_TEST
- PHOTONS
- TRIDENT_PHOTONS
- PREFETCH

The following compile-time defines have been removed from the Makefile:

- COLLISIONS
- SPLIT_PARTICLES_AFTER_PUSH
- PARTICLE_IONISE

A.1.2 Major features and new blocks added to the input deck

- CPML boundary conditions - See Section 3.3.1
- Thermal boundary conditions - See Section 3.3.2
- Collisions - See Section 3.13
- QED - See Section 3.14
- Subsets - See Section 3.17
- Ionisation - See Section 3.4.2
- Single-precision output - See Section 3.9.8
- Multiple output blocks - See Section 3.9.9
- Particle migration - See Section 3.4.1

A.1.3 Additional output block parameters

The following parameters have also been added to the “output” block (see Section 3.9.2):

- dump_first
- dump_last
- force_first_to_be_restartable
- ejected_particles
- absorption
- id
- name
- restartable

A.1.4 Other additions to the input deck

- npart_per_cell - See Section 3.4
- dir_{xy,yz,zx}_angle - See Section 3.11
- particle_tstart - See Section 3.1
- identify - See Section 3.4

Finally, the input deck now has a method for writing continuation lines. If the deck contains a “\” character then the rest of the line is ignored and the next line becomes a continuation of the current one.

A.2 Changes between version 4.0 and 4.3

A.2.1 Changes to the Makefile

Some changes have been made to the Makefile. These are documented in Section 2.4.

The following compile-time define has been added to the Makefile:

- MPI_DEBUG

The following compile-time define has been removed from the Makefile:

- FIELD_DEBUG

A.2.2 Additions to the input deck

The following parameters have been added to the “control” block of the input deck (see Section 3.1):

- nproc{x,y,z}
- smooth_currents
- field_ionisation
- use_exact_restart

- allow_cpu_reduce
- check_stop_file_frequency
- stop_at_walltime
- stop_at_walltime_file
- simplify_deck
- print_constants
- The “restart_snapshot” parameter now accepts filenames

The following parameters have been added to the “output” block of the input deck (see Section 3.9):

- disabled
- time_start
- time_stop
- nstep_start
- nstep_stop
- dump_at_times
- dump_at_nsteps
- dump_cycle
- dump_cycle_first_index
- filesystem
- file_prefix
- rolling_restart
- particle_energy
- relativistic_mass
- gamma
- total_energy_sum
- optical_depth
- qed_energy
- trident_optical_depth
- The default value of “dump_first” is now “T”

The following parameter has been added to the “collisions” block of the input deck (see Section 3.13):

- collisional_ionisation

The following parameter has been added to the “qed” block of the input deck (see Section 3.14):

- use_radiation_reaction

The following parameter has been added to the “species” block of the input deck (see Section 3.4):

- immobile

The following parameters were changed in the “laser” block of the input deck (see Section 3.5):

- The “phase” parameter can now be time varying
- The “profile” parameter can now be time varying

The following parameters have been added to the list of pre-defined constants (see Section 3.19.1).

- nproc_{x,y,z}
- nsteps
- t_end
- cc

There has also been a new “output_global” block added to the input deck. This is documented in Section 3.10.

A.2.3 Changes in behaviour which are not due to changes in the input deck

- The species “drift” property is now applied to particles whilst the moving window model is active. In previous versions of the code, this property was ignored once the moving window began.
- Ionisation species now inherit their “dumpmask”. See Section 3.4.2 for details.
- Default values for ignorable directions were added. This change allows submitting 3D or 2D input decks to a 1D version of **EPOCH** and 3D input decks to a 2D version of **EPOCH**. Any references to y/z will be set equal to zero unless overridden by a deck constant. Other y/z values also assume sensible defaults, eg. 1 grid cell, 1 metre thick, etc.
- Automatic byte swapping is carried out by the SDF library. The library now checks the endianness of the SDF file and byte-swaps the data if required.
- “qed” blocks may now be present even if the code was not compiled using the “-DPHOTONS” flag. The code will only halt if “use_qed=T” inside the “qed” block.
- The code now checks for the Data directory in a file named “USE_DATA_DIRECTORY” before prompting at the command-line. This allows the code to be run without waiting for input at the command-line.
- The field and particle grids are now automatically written to SDF output files if they are needed.
- The Data directory may now contain a ‘/’ character.

A.3 Changes between version 4.3 and 4.8

A.3.1 Changes to the Makefile

Some changes have been made to the Makefile. These are documented in Section 2.4.

The following compile-time define has been added to the Makefile:

- PER_SPECIES_WEIGHT
- NO_TRACER_PARTICLES
- NO_PARTICLE_PROBES
- PARSER_CHECKING

The following compile-time define has been removed from the Makefile:

- PER_PARTICLE_WEIGHT
- TRACER_PARTICLES
- PARTICLE_PROBES

A.3.2 Additions to the input deck

The following parameters have been added to the “control” block of the input deck (see Section 3.1):

- allow_missing_restart
- print_eta_string
- n_zeros

The following parameters have been added to the “output” block of the input deck (see Section 3.9):

- weight (synonym for particle_weight)

The following parameters have been added to the “output_global” block of the input deck (see Section 3.10):

- dump_first_after_restart

The following parameters have been added to the “subset” block of the input deck (see Section 3.17):

- skip, skip_x,y,z

The following parameters have been added to the “dist_fn” block of the input deck (see Section 3.11):

- restrict_en
- restrict_gamma_m1
- restrict_xy_angle
- restrict_yz_angle
- restrict_zx_angle

A.4 Changes between version 4.8 and 4.9

A.4.1 New capabilities

Version 4.9 adds significant new capabilities as follows:

- delta-f version: particle distributions can be expressed as $f_0 + f_1$ where f_0 is a specified background plasma and all simulation particles are used to describe the f_1 component, documented in Section 4.8.
- selectable field solvers: 3 new solvers have been added for fields, fully documented in Section 3.1.1.

A.4.2 Changes to the Makefile

Some changes have been made to the Makefile. These are documented in Section 2.4.

The following compile-time define has been added to the Makefile:

- DELTAF_METHOD
- DELTAF_DEBUG
- USE_ISATTY

A.4.3 Additions to the input deck

The following alterations were made to the input deck:

- ioniz* (with a “z”) aliases have been added for ionis* keywords.
- y and z parameters can now appear in the input deck in EPOCH 1D and 2D.

A new deck block has been added. The particles_from_file block allows loading of custom particle data from raw binary data files. See Section 3.7 for details. This block accepts the following parameters:

- species
- {xyz}_data
- w_data
- {xyz}_data
- id{4,8}_data
- offset

The following parameters have been added to the “control” block of the input deck (see Section 3.1):

- maxwell_solver
- use_current_correction

The following parameters have been added to the “species” block of the input deck (see Section 3.4):

- maxwell_solver
- density_back
- drift_{x,y,z}_back
- temp_{x,y,z}_back
- temp_{x,y,z}_back_ev
- temp_back
- temp_back_ev

The following parameters have been added to the “dist_fn” block of the input deck (see Section 3.11):

- dir may now take the value mod_p
- restrict_mod_p

A.4.4 Changes not resulting from changes to the deck

- Lasers can be specified with time-varying frequency profile.
- The existing subset blocks can now be applied to field and derived grid variables. If spatial restrictions are used, subsections will be output, along with a corresponding grid. Note that these are not compatible with the “skip” parameter to subset blocks.
- The dist_fn block “range” keyword is now respected for spatial directions, allowing a spatial subset of the distribution function to be output directly.
- Some corrections were applied to calculation of thermal boundary conditions for particles.
- The load balancer may now be disabled by setting a 0 or negative threshold.

A.5 Changes between version 4.9 and 4.10

- Time varying particle injectors. See Section 3.16
- Per-species particle boundaries. You can now specify bc_x_min and bc_x_max to a species block. This overrides the global boundaries for that species. See Section 3.4.3
- Added “particles_per_cell” output diagnostic. See Section 3.9.2

A.6 Changes between version 4.10 and 4.11

A.6.1 New capabilities

Version 4.11 adds the following new capabilities:

- Added time dependent moving window. No new input deck parameters have been added, but it is now possible to specify “window_v_x” to be a function that varies in time. See Section 3.8
- If “print_constants=T” in the control block (see Section 3.1) deck constants are now output to a separate file named “const.status”. This allows for easier post-processing.
- Added COMPILER=auto option to automatically detect compiler. See Section 2.4

A.6.2 Corrections

The following correction has been made:

- Fractional numbers of particles-per-cell now function as expected when used in conjunction with the moving window.

A.7 Changes between version 4.11 and 4.12

- Added “average_weight” output diagnostic. See Section 3.9.2
- Removed the “PARTICLE_COUNT_UPDATE” Makefile flag and replaced it with a “use_particle_count_update” parameter in the control block. See Section 3.1
- Added “use_flux_maxwellian” option to the “injector” block. See Section 3.16
- Added “lehe_{x,y,z}” flags to the “maxwell_solver” option in the control block. See Section 3.1
- Added “custom” flag to the “maxwell_solver” option in the control block. See Section 3.1 and Section 3.2
- Added the “WORK_DONE_INTEGRATED” Makefile flag and corresponding dumpmask directives “work_{x,y,z}” and “work_{x,y,z}_total”. These add a diagnostic for the work done on a particle by the electric field. See Section 2.4 and Section 3.9.2.
- Added the “use_accurate_n_zeros” flag to the control block. See Section 3.1

A.8 Changes between version 4.12 and 4.14

- Added the “reset_walltime” flag to the control block. See Section 3.1
- Changed the default value of “print_eta_string” to “T” in the control block.
- Added the ability to request an output dump at run time. See Section 3.1.5
- Added the “window_stop_time” parameter to the window block. See Section 3.8
- Added the “atan2” function to the maths parser. See Section 3.19.2
- Added “dlb_maximum_interval” parameter to the control block. See Section 3.1
- Added “dlb_force_interval” parameter to the control block. See Section 3.1
- Added “balance_first” parameter to the control block. See Section 3.1
- Added y and z versions of the “bc_x_min_after_move” and “bc_x_max_after_move” parameters to the window block. See Section 3.8
- Added a “dump_at_walltimes” parameter to the output block. See Section 3.9.9
- Added “walltime_start” and “walltime_stop” parameters to the output block and output_global block. See Section 3.9.2 and Section 3.10
- Added “walltime_interval” parameter to the output block. See Section 3.9.9
- Added the Higuera-Cary particle push. This can be enabled using the “HC_PUSH” Makefile flag. See Section 2.4.

A.9 Changes between version 4.14 and 4.15

- Added averaging of “poynt_flux” and “ekflux” variables.
- The initial problem setup can now be load-balanced before any particles are loaded. This enables some heavily imbalanced setups to be run that were not previously possible.
 - Added the “use_pre_balance” flag to the control block. See Section 3.1
- Allow the load balancer to adjust the processor topology
 - Added the “use_optimal_layout” flag to the control block. See Section 3.1
- Added control block option “use_more_setup_memory” for controlling the way that species are setup. See Section 3.1
- Added strided multipass digital current filtering (See Section 3.1.2). This adds the following flags to the control block.
 - smooth_iterations
 - smooth_compensation
 - smooth_strides
- Added persistent subsets. See Section 3.17. This adds the following flags to the subset block
 - persist_start_time
 - persist_start_step
- Added loading of relativistic particle species using the Maxwell-Jüttner distribution. See Section 3.4.4. This adds the following flags to the species block
 - use_maxwell_juttner
 - fractional_tail_cutoff
- Added loading of particle species using an arbitrary distribution function for sampling the momentum components. See Section 3.4.5. This adds the following flags to the species block
 - dist_fn
 - dist_fn_p{x,y,z}_range
- Added “temperature_{x,y,z}” derived output variables to the output block. See Section 3.9.5.

A.10 Changes between version 4.15 and 4.16

- Added “number_density” aliases for “density” in the species and injector blocks (see Section 3.4 and Section 3.16). These aliases include:
 - number_density for density
 - promote_number_density for promote_density
 - demote_number_density for demote_density
 - number_density_min for density_min
 - number_density_max for density_max
 - number_density_back for density_back

- Replaced “USE_ISATTY” Makefile flag with “NO_USE_ISATTY”. See Section 2.4.
- Added “NO_MPI3” Makefile flag. See Section 2.4.
- Added a “zero_current” alias for “tracer” in the species blocks. See Section 3.4. The use of “tracer” has now been deprecated and will be removed in version 5.0. At that time, the compiler flag will also be renamed.
- Added collisions block option “use_nanbu” for enabling the scattering algorithm of Nanbu, with relativistic corrections by Pérez. Due to known issues in the Sentoku-Kemp method for some test problems, the Nanbu method is now the default. See Section 3.13.

A.11 Changes between version 4.16 and 4.17

- Added “average_p{x,y,z}” derived output variables to the output block. See Section 3.9.5.
- Added “deck_warnings_fatal” flag to the control block for controlling the way that deck parser warnings are handled. See Section 3.1
- Added “DECK_DEBUG” Makefile flag. See Section 2.4.
- Added “number_density_max” parameter to the injector block. This has similar functionality to the species block parameter of the same name. See Section 3.16.
- Added “temp_{x,y,z}_ev” and “temp_ev” parameters to the injector block. These have the same functionality as the species block parameters of the same name. See Section 3.16.
- Added multiple aliases for input deck parameters:
 - “drift_p{x,y,z}” and “drift_p{x,y,z}_back” aliases for “drift_{x,y,z}” and “drift_{x,y,z}_back”. See Section 3.4 and Section 3.16.
 - “temperature” aliases for the various “temp” parameters. See Section 3.4, Section 3.16 and Section 3.19.2.
 - “background” aliases for “number_density_back”, “drift_p{x,y,z}_back”, “temperature_{x,y,z}_back”, “temperature_{x,y,z}_back_ev”, “temperature_back” and “temperature_back_ev”. See Section 4.8.
 - “nparticles” and “nparticles_per_cell” aliases for “npart” and “npart_per_cell”. See Section 3.1, Section 3.4 and Section 3.16.
 - “fraction” alias for “frac” in the control block. See Section 3.4.
 - “energy” alias for “en” in the dist_fn block. See Section 3.11.
 - “gamma_minus_one” alias for “gamma_m1” in the dist_fn block. See Section 3.11.
 - “average_particle_energy” alias for “ekbar” in the output block. See Section 3.9.5.
 - “particle_energy_flux” alias for “ekflux” in the output block. See Section 3.9.5.
 - “poynting_flux” alias for “poynt_flux” in the output block. See Section 3.9.5.
 - “polarisation” and “polarisation_angle” aliases for “pol” and “pol_angle” in the laser block. See Section 3.5.
 - “breit_wheeler_{electron,proton}” aliases for “bw_{electron,proton}” in the qed block. See Section 3.14.
- Added “heat_bath” particle boundary conditions. These use the particle injectors to create an improved form of thermal boundary conditions. See Section 3.3.

- Changed the default value for the “use_exact_restart” flag to “T”. See Section 3.1.
- Add model for simulating bremsstrahlung radiation. This is disabled by default and can be enabled by specifying the “-DBREMSSTRAHLUNG” flag at compile time. The model is configured using a new “bremsstrahlung” block which takes the following parameters:
 - enable
 - start_time
 - produce_photons
 - photon_energy_min
 - photon_weight
 - photon_dynamics
 - use_plasma_screening
 - use_radiation_reaction
 - table_location

See Section 3.15 for details. It also adds the “bremsstrahlung_optical_depth” output block parameter and “atomic_number”, “background_species” and “identify:brem_photon” species parameters. See Section 3.9 and Section 3.4.

B Simple binary files

There are several input deck blocks which can read conditions directly from a user-specified file. These include the “**fields block**” (Section 3.6), “**species block**” (Section 3.4) and “**particles_from_file block**” (Section 3.7). In all such cases, the files specified must be in a simple binary format, often referred to as “raw” binary files.

Binary files are machine readable, but not human readable. If you try opening a binary file in a text editor then you will see incomprehensible characters and some text editors might even crash. Most languages can write binary files, see “writeu” (in IDL/GDL), “fwrite” in MatLab, the “b” parameter to “open” in Python and “form=’UNFORMATTED’ ” in Fortran, so please see the documentation for those languages. Note that standard unformatted output in Fortran also writes some additional hidden output to the file that alters the offset of the actual binary array data within the file. It is therefore recommended that you always use the “access=’STREAM’ ” modifier whenever writing such files from Fortran programs.

For illustration purposes, here is a simple example of writing a 2D array to file using Fortran:

```
PROGRAM output_array

INTEGER :: iu, istat
INTEGER, PARAMETER :: nx = 10, ny = 20
DOUBLE PRECISION :: array(nx,ny)
CHARACTER(LEN=*), PARAMETER :: filename = 'array.dat'

array = 2.0d0

OPEN(newunit=iu, file=filename, status='NEW', form='UNFORMATTED', &
      access='STREAM', iostat=istat)
```

```

IF (istat == 0) THEN
    WRITE(iu) array
    CLOSE(iu, iostat=istat)
ELSE
    PRINT*, 'ERROR: failed to open file ', '"' // filename // '", &
        ' for writing'
END IF

END PROGRAM output_array

```

In this example, there are 200 array elements written to file ($10 * 20$). Each element is a double-precision number which is 8 bytes. Therefore, the total file size will be 1600 bytes. Note that for Fortran, arrays are indexed using “column-major order”. This means that in the file, the first array element “array(1,1)” will be followed by “array(2,1)” and so on up to “array(10,1)”. After this, the second index will be incremented and the array element “array(1,2)” will be output, followed by “array(2,2)”, etc. In contrast, languages such as C and C++ use row-major order. For these languages the array output is transposed, so the array elements are output in the order: “array[0][0], array[0][1], .. array[0][19], array[1][0], ..”

Simple binary files merely contain a long sequence of real numbers and do not contain any information about the shape of the arrays that have been written. This information must be supplied using the input deck. These should correspond to the values of “nx”, “ny”, etc. For example, to use the array generated by the Fortran code shown above, the input deck must specify “nx = 10” and “ny = 20”.

It is possible to write multiple arrays into the same binary file and use the “offset” command in the input deck to specify where the next array in the file is to be located. This can be tricky to work with and it is therefore recommended to write each separate array to its own file.

C Slides from Delta-f Talk

δf Particle-In-Cell simulation.

Ben F McMillan,

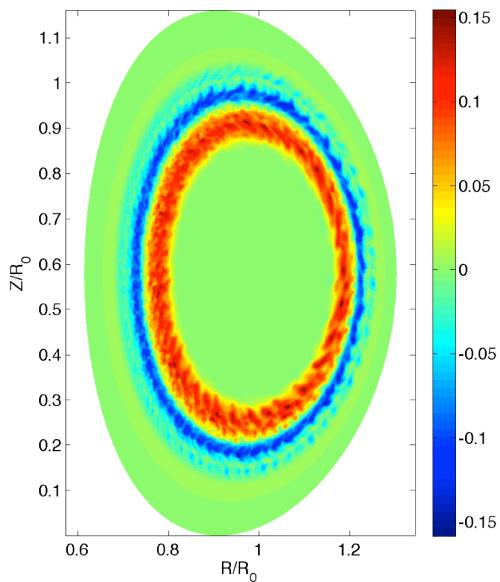
Centre for Fusion, Space and Astrophysics,
Physics Department, University of Warwick, UK

Acknowledgements to the EPOCH team.

Contents

- Numerical simulations for collisionless plasma simulation
- The PIC (Particle in cell) method
 - Noise in the PIC method
- The ‘low noise’ δf PIC method
 - Formulation
 - Some comparisons with standard PIC

The problem I'm usually solving:



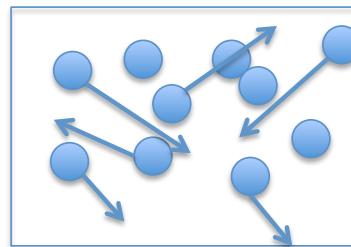
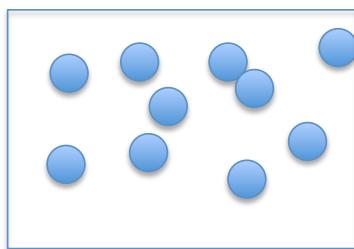
- Simulating turbulence in tokamaks.
- Low-collisionality particles interacting with an electromagnetic field.
- Small fluctuations (1%), large system size to turbulent eddy scale (1000), 5D problem (position and velocity), large range of timescales.

Numerical methods for Vlasov-Maxwell equations.

- Kinetic simulations of plasmas require solution of the Vlasov equation $df(x,v,t)/dt = 0$, with the convective derivative along the particle trajectory.
- Two basic methods:
 1. Fixed discretisation in (x,v) (Eulerian)
 2. Follow characteristics/particle trajectories in (x,v) (Lagrangian/PIC)
- Eulerian generally too hard with 6 phase space directions.

Particle in Cell (PIC) loop

Load particles in a box

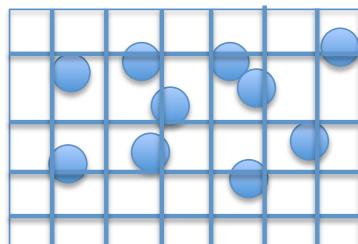
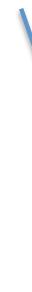


Solve equations of motion for a short time
 $\text{particle } (x_i, v_i) \rightarrow (x'_i, v'_i)$ using fields E, B.

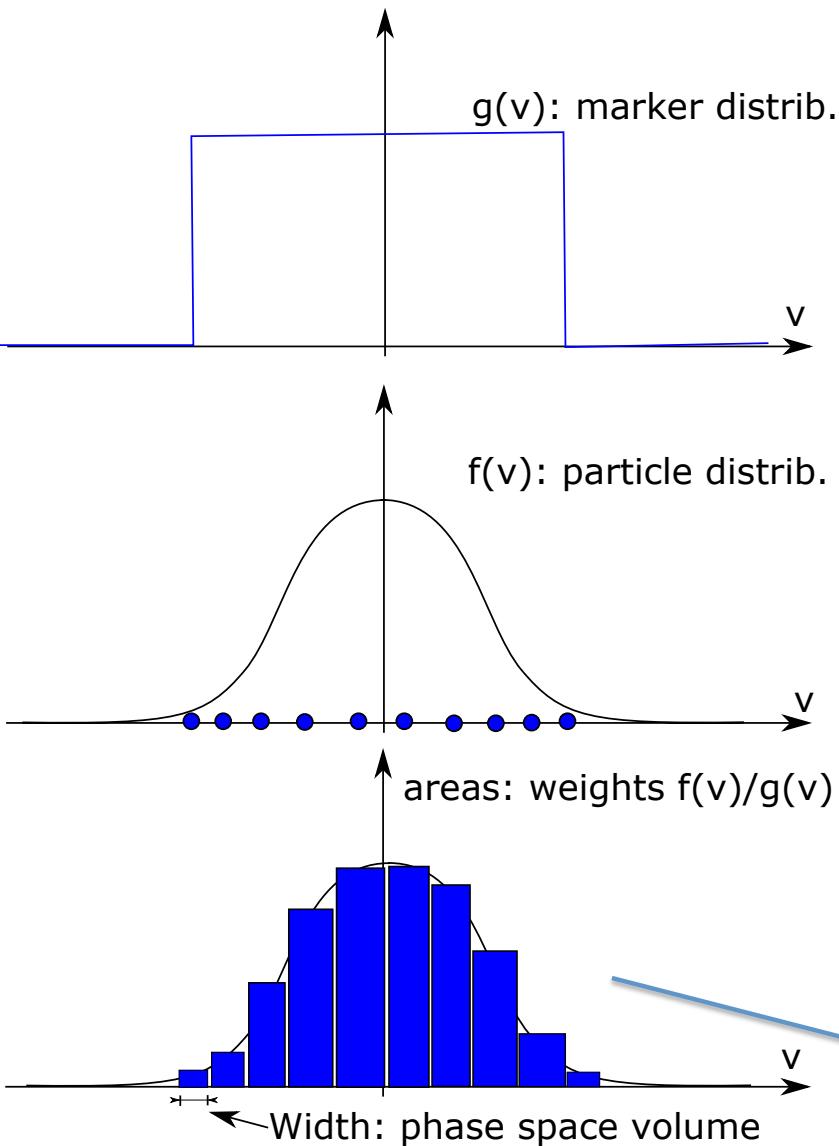
Find the new fields E, B.



Find densities and/or
currents on grid associated
with particles at (x'_i, v'_i)



PIC methods and Monte-Carlo



- The markers in a PIC simulation sample the distribution function: densities/ currents are Monte-Carlo integrals.
- In real simulations, markers are randomly distributed.

Integral->Charge density->Fields

PIC errors.

- Moments have errors scaling like $1/N^{1/2}$
- In a spatial grid cell, usually $N=100$ for typical simulations.
- Why does this work at all -> 10% density fluctuations are similar/comparable to cases of interest.
 - Fluid properties conserved (roughly). PIC simulations act like a coarse-grained, highly collisional plasma.
 - For some problems, noise mostly at different phase speeds.

Does noise matter?

NOT REALLY?

- Particle simulation can be interpreted as real plasma system with different parameters.
- Simulation results look ‘similar’ under convergence.
- Can use filtering to remove some of it.

YES?

- Much easier to interpret low-noise simulations.
- Noise introduces unphysical diffusion and unphysical collisions.
- This can prevent the real physics from manifesting.

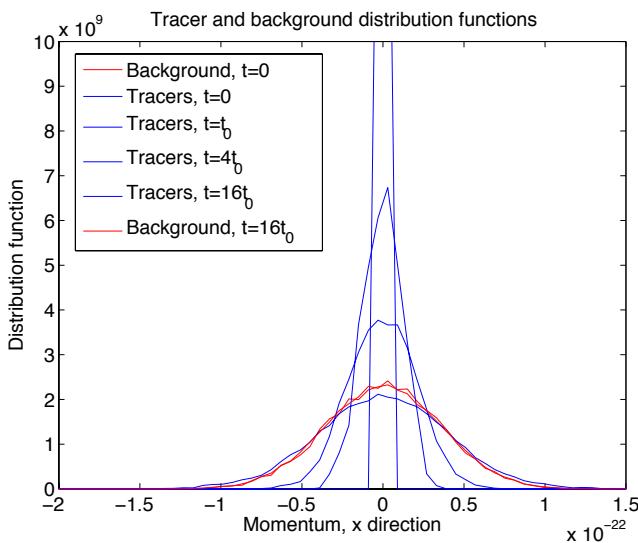
Noise disasters in tokamak simulation.

- PIC simulations predicted lower levels of transport in tokamaks than Eulerian simulations.
- Seemed to be converged with particle number (but weren't!). (See Hammett et. al., 2006)
- It was the diffusion due to noise that killed them.
- Lessons:
 - Understand how much noise is in the system and its effect.
 - Convergence scans necessary but liable to misinterpretation.

Computational noise versus thermal noise.

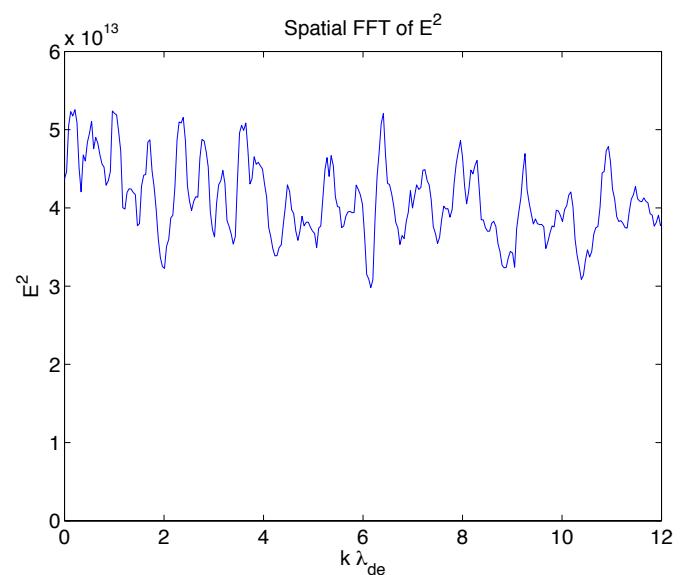
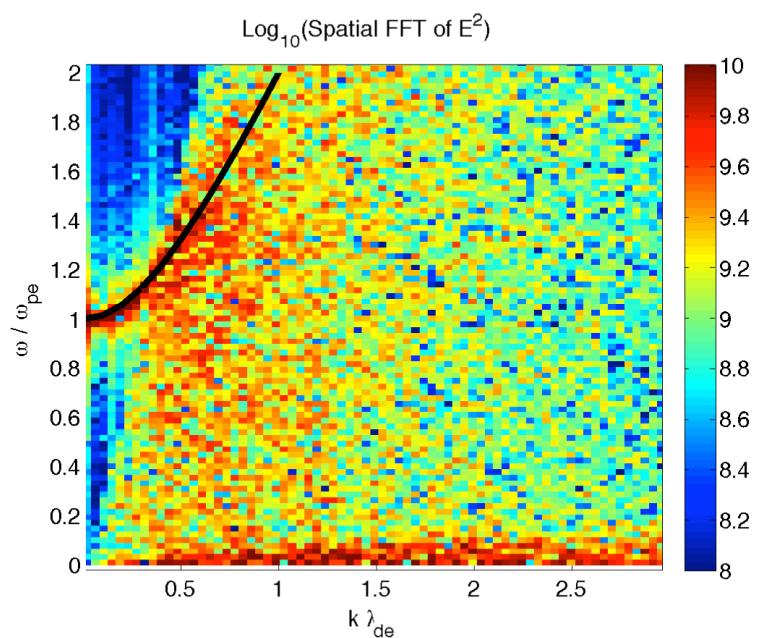
- Point particle versus smoothed ‘cloud’.
- Fluctuation/Dissipation theorem and BBGKY approaches can be used (with reservations) for *computational* gas like a real gas. (A.B.Langdon, Physics of Fluids 22, 163, 1979)
- Very few markers in ‘Debye Sphere’-> numerical collisionality.

Velocity diffusion.



- Tracers with initially zero velocity in background thermal plasma.
- 300 markers per cell.
- t_0 is $100\omega_{pe}^{-1}$.
- ‘Collisional thermalisation’ after $\sim 1000\omega_{pe}^{-1}$.

Spectrum of noise.



The δf method

- Noise is mostly a problem when the signal is small: systems close to stability.
- Used widely in gyrokinetics (tokamak turbulence): for standard PIC, see Sydora, JCAM 109, (1999) 243-259.
- Define a ‘background function’ $f_0(x,v,t)$ which has analytically known moments, and is close to f , and split $f = f_0 + \delta f$.

Choose an f_0 with analytically known moments.

$$f = f_0 + \delta f \quad (1)$$

$$0 = \frac{df}{dt} = \frac{df_0}{dt} + \frac{d\delta f}{dt} \quad (2)$$

As in standard PIC, define N markers which follow particle equations of motion, with phase space position $\mathbf{z} = (\mathbf{x}, \mathbf{v})$.

Analogous to standard PIC, the distribution function is written in terms of markers

$$\delta f = \sum w_i(t) \delta(\mathbf{z} - \mathbf{z}_i). \quad (3)$$

The distribution of markers $g(\mathbf{z}, t)$ also satisfies the Vlasov equation.

$$0 = \frac{dg}{dt} \quad (4)$$

Integrate equation 3 over a small volume of phase space to find

$$\delta f = \langle w_i(t) \rangle g(\mathbf{z}, t) = \langle w_i(t) \rangle g(\mathbf{z}_0, 0) \quad (5)$$

The marker density at the initial position $g_i = g(\mathbf{z}_{i0}, 0)$ can be interpreted as the inverse of the phase volume represented by each marker.

We solve to find

$$w_i = \frac{1}{g_i} [f(\mathbf{z}_{i0}, 0) - f_0(\mathbf{z}, t)]. \quad (6)$$

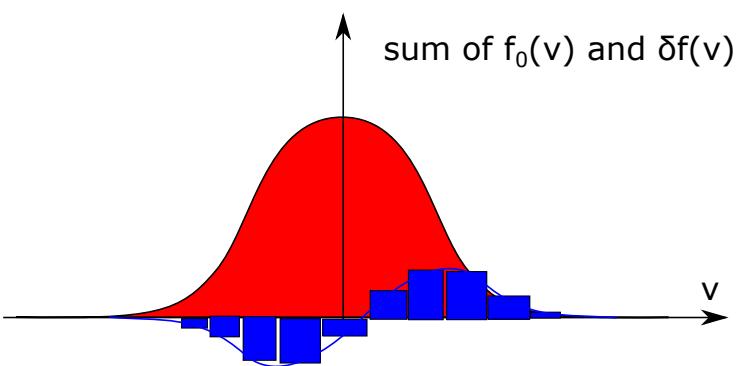
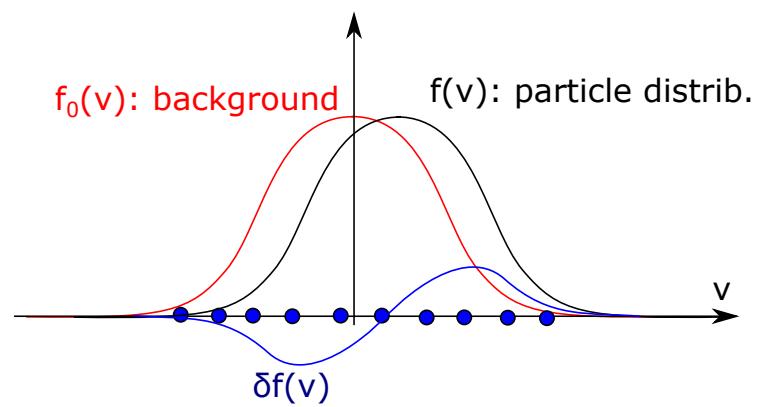
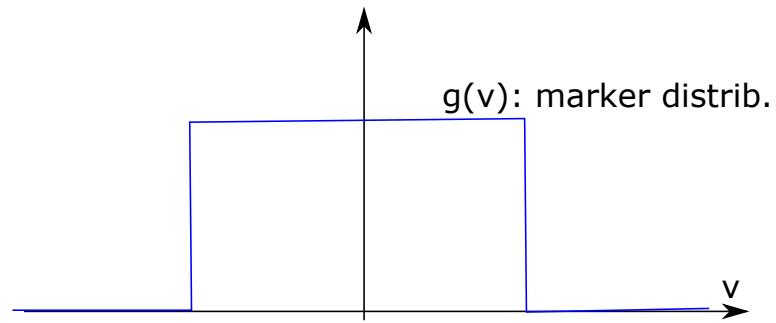
We can also find an equation for the time derivative of $w_i(t)$:

$$\frac{dw_i}{dt} = \frac{1}{g_i} \frac{d\delta f}{dt} = -\frac{1}{g_i} \frac{df_0}{dt} = -\frac{1}{g_i} \left[\frac{\partial f_0}{\partial t} + \dot{\mathbf{z}} \cdot \frac{\partial f_0}{\partial \mathbf{z}} \right]. \quad (7)$$

Currents and densities are defined as

$$\mathbf{J}(\mathbf{x}) = \int d\mathbf{v} f \mathbf{v} = \int d\mathbf{v} f_0 \mathbf{v} + \sum w_i(t) \mathbf{v}_i \delta(\mathbf{x} - \mathbf{x}_i) \quad (8)$$

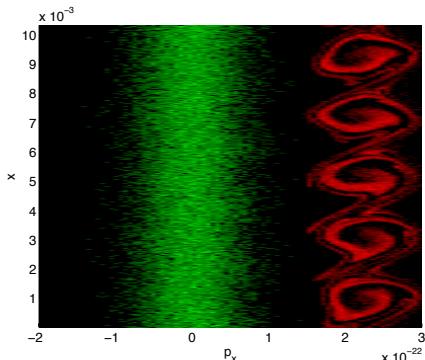
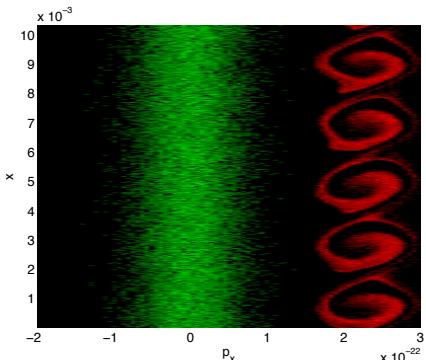
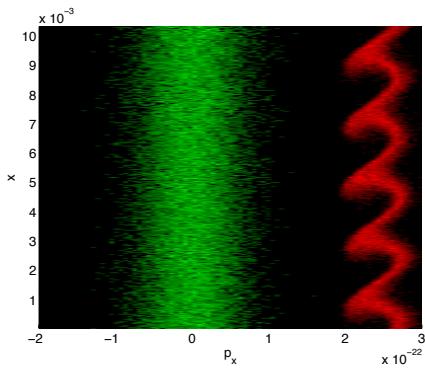
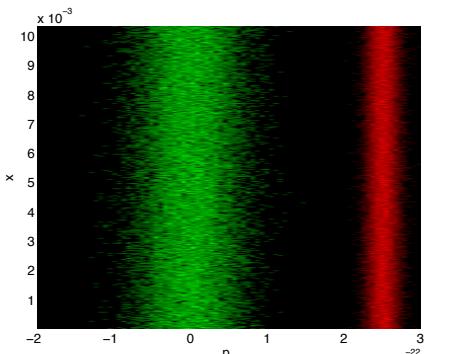
where the background terms often sum to zero (current-free background).



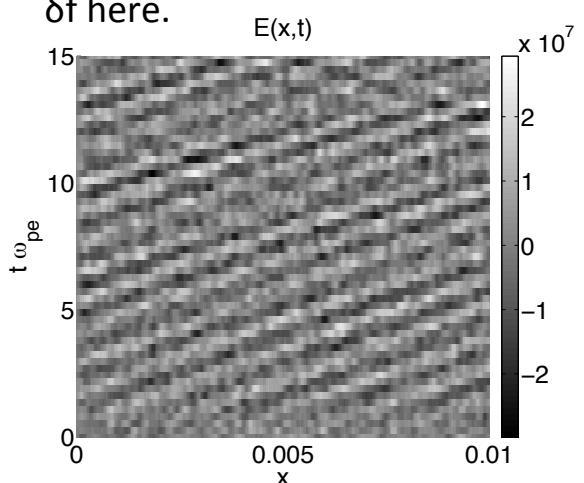
Why does this help?

- Noise in fields/densities lower by factor ($\delta f/f$).
- Moments of distribution have much lower error: diagnostics easier.
- Unphysical numerical collisions much weaker.
- Allows linear simulations:
 - These are pretty difficult/annoying in standard PIC.
 - Much easier to compare with analytic theory.

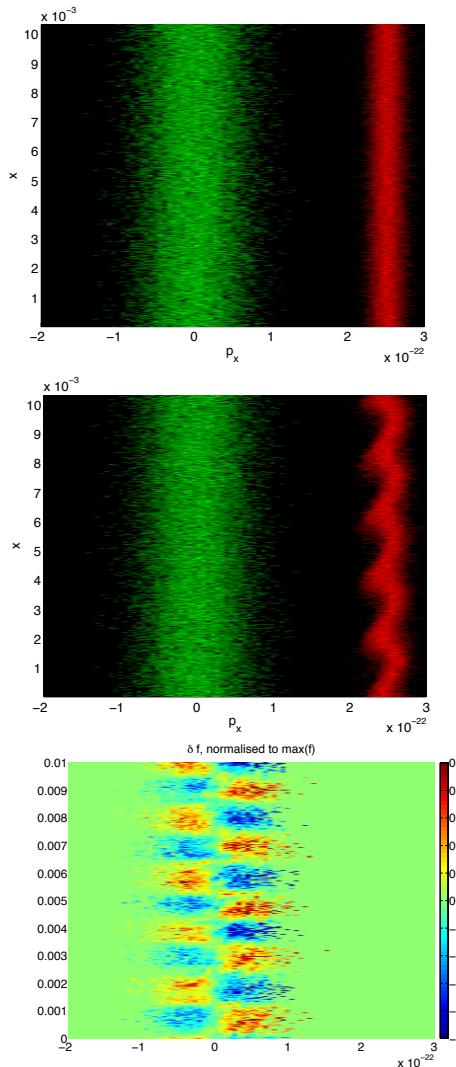
An example: weak beam two stream



- Background plasma 1000 times the density of beam (red)
- Normal PIC handles this quite well: noise in background doesn't affect the instability.
- Electric field quite noisy (even smoothed).
- Using a δf simulation: the beam is far from Maxwellian so δf not helpful. The background perturbation is small so could use δf here.



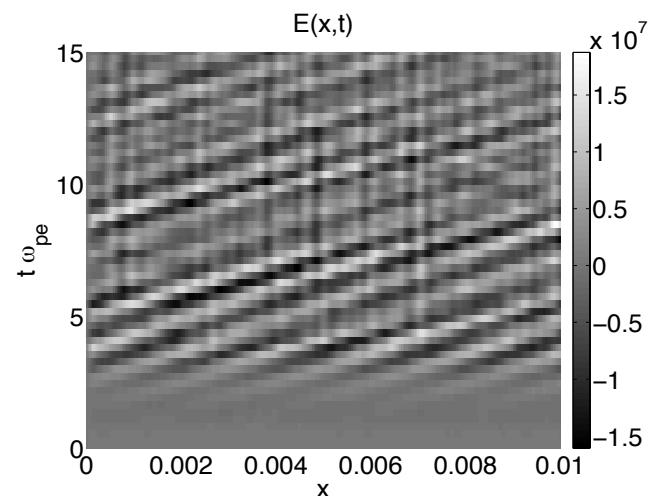
Weak beam, δf



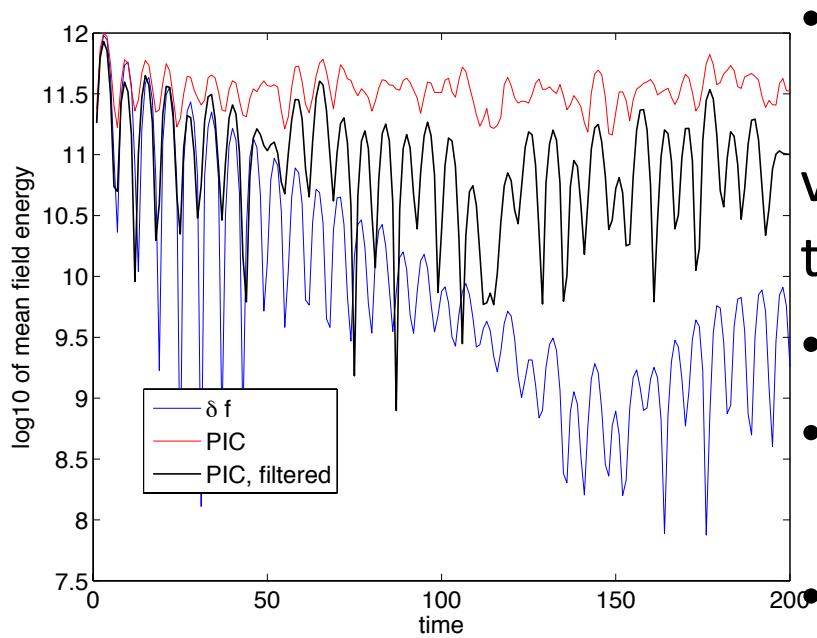
Perturbed distribution function.

Where is sampling important?

- Instability grows later due to lower initial fluctuations->find growthrate.
- Otherwise similar results for beam distribution.
- Using a δf simulation: much lower noise in electric field. Can see fine structure but care needed in tail of background.
- Can also determine how the background distribution has evolved.



Landau damping.



- Initial condition:
1% sinusoidal variation to velocity of thermal plasma.
- 300 particles per cell
- δf gives clean decay trace.
- Standard PIC needs filtering to see anything.

Conclusions.

- δf PIC used widely in tokamak research.
- Complementary tool to standard PIC simulation.
- Implemented in an EPOCH (Arber, 2015, PPCF) code extension.
- Often very easy to run a δf simulation: change three lines in an EPOCH input file.
- Particularly useful for nearly-linear physics or problems where most particles respond weakly.
Waves+beams...
- Needs some real examples.

D References

- [1] T. D. Arber, K. Bennett, C. S. Brady, A. Lawrence-Douglas, M. G. Ramsay, N. J. Sircombe, P. Gillies, R. G. Evans, H. Schmitz, A. R. Bell, and C. P. Ridgers, “Contemporary particle-in-cell approach to laser-plasma modelling,” *Plasma Physics and Controlled Fusion*, vol. 57, no. 11, pp. 1–26, Nov. 2015.
- [2] A. Blinne, D. Schinkel, S. Kuschel, N. Elkina, S. G. Rykovanov, and M. Zepf, “A systematic approach to numerical dispersion in Maxwell solvers,” *Computer Physics Communications*, 2017. [Online]. Available: <https://arxiv.org/pdf/1710.06829.pdf>
- [3] O. Buneman, “TRISTAN: The 3-D Electromagnetic Particle Code.” in *Computer Space Plasma Physics: Simulations Techniques and Software*, 1993.
- [4] B. M. Cowan, D. L. Bruhwiler, J. R. Cary, E. Cormier-Michel, and C. G. R. Geddes, “Generalized algorithm for control of numerical dispersion in explicit time-domain electromagnetic simulations,” *Phys. Rev. Accel. Beams*, vol. 16, no. 4, p. 041303, apr 2013.
- [5] R. Duclous, J. G. Kirk, and A. R. Bell, “Monte carlo calculations of pair production in high-intensity laserplasma interactions,” *Plasma Phys. Contr. F.*, vol. 53, no. 1, p. 015009, 2011.
- [6] A. V. Higuera and J. R. Cary, “Structure-preserving second-order integration of relativistic charged particle trajectories in electromagnetic fields,” *Physics of Plasmas*, vol. 24, no. 5, p. 052104, 2017. [Online]. Available: <https://doi.org/10.1063/1.4979989>
- [7] R. Lehe, A. F. Lifschitz, C. Thaury, V. Malka, and X. Davoine, “Numerical growth of emittance in simulations of laser-wakefield acceleration,” *Phys. Rev. Accel. Beams*, vol. 16, no. 2, p. 021301, 2013.
- [8] K. Nanbu and S. Yonemura, “Weighted particles in coulomb collision simulations based on the theory of a cumulative scattering angle,” *Journal of Computational Physics*, vol. 145, pp. 639–654, 09 1998.
- [9] F. Pérez, L. Gremillet, A. Decoster, M. Drouin, and E. Lefebvre, “Improved modeling of relativistic collisions and collisional ionization in particle-in-cell codes,” *Physics of Plasmas*, vol. 19, no. 8, p. 083104, Aug. 2012.
- [10] A. Pukhov, “Three-dimensional electromagnetic relativistic particle-in-cell code VLPL (Virtual Laser Plasma Lab),” *J. Plasma Phys.*, vol. 61, no. 3, pp. 425–433, 1999.
- [11] C. Ridgers, J. Kirk, R. Duclous, T. Blackburn, C. Brady, K. Bennett, T. Arber, and A. Bell, “Modelling gamma-ray photon emission and pair production in high-intensity lasermatter interactions,” *J. Comput. Phys.*, vol. 260, pp. 273 – 285, 2014.
- [12] J. Roden and S. Gedney, “Convolution pml (cpml): An efficient fdtd implementation of the cfs-pml for arbitrary media,” *Microw. Opt. Technol. Lett.*, 2000.
- [13] Y. Sentoku and A. J. Kemp, “Numerical methods for particle simulations at extreme densities and temperatures: Weighted particles, relativistic collisions and reduced currents,” *J. Comput. Phys.*, 2008.
- [14] A. Tafove and S. C. Hagness, *Computational Electrodynamics: The Finite-Difference Time-Domain Method*. Artech House, 2000.
- [15] J.-L. Vay and B. B. Godfrey, “Modeling of relativistic plasmas with the particle-in-cell method,” *Comptes Rendus Mcanique*, vol. 342, no. 10, pp. 610 – 618, 2014, theoretical and numerical approaches for Vlasov-maxwell equations. [Online]. Available: <http://www.sciencedirect.com/science/article/pii/S163107211400148X>

- [16] J. Vyskočil, O. Klímo, and S. Weber, “Simulations of bremsstrahlung emission in ultra-intense laser interactions with foil targets,” *Plasma Physics and Controlled Fusion*, vol. 60, no. 5, p. 054013, 2018.
- [17] D. Wu, X. He, W. Yu, and S. Fritzsche, “Particle-in-cell simulations of laser–plasma interactions at solid densities and relativistic intensities: the role of atomic processes,” *High Power Laser Science and Engineering*, vol. 6, 2018.