1 Introduction

E4D is a powerful electrical resistivity tomography program written by Tim Johnson at PNNL. It can be freely downloaded and installed from `https://e4d.pnnl.gov/`. The program's strength are that it is parallelized and can therefore solve large problems (many inversion cells) very quickly. The disadvantage of E4D is that it is harder to learn than for example BERT.

E4D uses several types of configuration files which we will discuss in the following. The Matlab/Octave scripts that come with this tutorial will be helpful to generate the content for many of these input files.

In this tutorial we will go step by step, starting with generating a simple mesh and moving through calculating inversions. Preparing the input files for complex meshes is the hardest part of using E4D, so we will go step by step from simple (but usually useless) meshes to complicated meshes.

Generally the work flow for using E4D will be to first construct a mesh, and then run the inversion on a prepared mesh. E4D always requires an input file named `e4d.inp`. In this input file, the user specifies what needs to be done (e.g. mesh generation, inversion), and the names of the configuration and data files.

2 Mesh generation

On your desktop, or in any folder you would like to work in, create a file named `e4d.inp`. You can do that for example with emacs by running in your folder

```
emacs e4d.inp
```

To make e4d create a mesh, the first line of `e4d.inp` needs to be the number 1. The second line needs to be the name of our mesh configuration file. Let's call it `myfirstmesh.cfg`. So make your `e4d.inp` text file look like

```
1
myfirstmesh.cfg
```

Of course, we now need to create the mesh configuration file `myfirstmesh.cfg`. To do that, create it again using emacs

```
emacs myfirstmesh.cfg
```

This file will now be a bit more complicated than the input file. A mesh file will always consist of the following components:

- The mesh quality (how deformed can the tetrahedra be), and maximum tetrahedra volume
- the bottom elevation of the mesh
- some information on how to build the mesh (this will always be the same so don't worry about this now)
- The points around which the mesh will be generated including electrode positions, topography, boundary points for the internal and external zones, electrode depth points for mesh refinement around the electrodes, etc.
- Internal planes to delineate between mesh zones
- holes (I have not yet worked with holes in the mesh so we will skip this in this tutorial)
- zone descriptions
- information about writing a paraview/exodus file

Let's start with the simplest possible configuration. We first need to learn how we write points. A point always has the form

pointnumber xcoord ycoord zcoord type

type can be subsurface point (0), or surface point (1) or external boundary point (2).

The external boundary points define the outside boundary of our region so we will always need them.

Let's write our `myfirstmesh.cfg` file. Let's make the mesh quality 1.3 and the maximum volume 10^{12}. The surface will be defined by the external boundary points (which by definition are on the surface), and the bottom will be at -10. We want the external boundary to be at at four points (-1/-3/0), (5/-0.5/1), (4/7/-1.2), and (-2/6/0).

So the lines in our file `myfirstmesh.cfg` will be:

```
1.3 1e12   (mesh quality, max volume)

-10   (bottom elevation of mesh)
1
"tetgen"
"triangle"

4   (number of points)
1 -1 -3 0 2   (first boundary point)
2 5 -0.5 1 2  (second boundary point)
3 4 7 -1.2 2  (etc)
4 -2 6 0 2

0   (no internal planes)
0   (no holes)

1   (one zone)
1 0 0 -2 5 0.01   (point in zone, max mesh size this zone, conductivity)

1   (save mesh as exodus file)
"bx"  (use the program "bx")

1   (mesh translation option)
```

If you want you can omit the comments in parenthesis. These will not be used by the program but it may make it easier for you to edit the mesh configuration file.

In this file we needed to define parameters such as maximum tetrahedra volume for each zone (at this point we only have one zone), and the conductivity. To do this we needed to pick a random point within our zone, we picked (0/0/-2), and set the maximum tetrahedra volume, we selected 5 m, and the conductivity, we selected 0.01 S/m. These values are for zone 1, therefore the line read

*(*1 0 0 -2 5 0.01)

To create the mesh, run in your command line

```
e4d
```

The program will automatically look in the input file `e4d.inp` and use the mesh configuration file you specified there and will create a lot of auxiliary files. Open the resulting file `myfirstmesh.exo` using paraview.

Exercise: Play around with the maximum mesh size for the zone and see if you can create a finer mesh.

NEXT: PUT IN ELECTRODES THEN: TO REFINE MESH AROUND ELECTRODES, PUT IN POINTS BELOW ELECTRODES.