SHIELD Global Heliosphere Model

1 Installation and configuration

To simulate with the outer heliosphere (OH) model of the Space Weather Modeling Framework (SWMF), one first needs to install SWMF using the instructions in SWMF README. Since the outer heliosphere model is using a lot of computer memory and CPU time, it is advisable to run this model on a supercomputer platform.

To configure the outer heliosphere, go into the SWMF/GM/BATSRUS/ directory and use the following configuration:

```
./Config.pl -u=OuterHelio -e=OuterHelio -g=8,8,8
```

The next step is to compile BATSRUS:

```
make BATSRUS -j
```

and to compile the PostPDL for post processing IDL output files

```
make PIDL
```

To run the OH model, we still need to make a rundir in the SWMF/GM/BATSRUS directory:

```
make rundir component=OH
```

This will generate a run directory within SWMF/GM/BATSRUS and one can find the BATSRUS executable and post processing scripts within this subdirectory. An example input to run the outer heliosphere model is PARAM.in. Copy this file into your run directory. The executable to run the OH model is BATSRUS.exe which can be found in the run directory.

2 Visualizing the Output

The OH component has the capability to produce solutions in IDL and Tecplot formats. Although the model can produce outputs in various forms (i.e. 1D, 2D, 3D, x=0, y=0, z=0, cut, 2D slice, etc.), here we will focus on the simplest cases of meridional slices (y=0) and the full 3D solution. For more details on other forms of saving output, please refer to the BATS-R-US user manual and search for details on the #SAVEPLOT command in Chapter 3.

2.1 Saving the Output

In the following, we show the command to be included in PARAM.in if you wish to output to IDL:

```
#SAVEPLOT
4                nPlotFile
y=0 MHD idl      StringPlot ! y=0 plane plot with MHD variables
1000             DnSavePlot
-1.0             DtSavePlot
0.25             DxSavePlot ! resolution (for IDL plots)
3d MHD idl       StringPlot ! 3D plot with MHD variables
-1               DnSavePlot
100 y            DtSavePlot
-1               DxSavePlot ! resolution (for IDL plots)
```

```
y=0 MHD tcp     StringPlot ! y=0 plane plot with MHD variables
1000            DnSavePlot
-1.0            DtSavePlot
3d MHD tcp      StringPlot ! 3D plot with MHD variables
-1              DnSavePlot
100 y           DtSavePlot
```

In this #SAVEPLOT command, 'nPlotFile' corresponds to the number of different outputs that you are requesting. In this case, four different outputs are requested: meridional slices (y=0) and full 3D outputs that cover the entire spatial domain. If this number is less than the number of specified outputs below it, the command will produce the outputs in the order provided. For example, if 'nPlotFile' is set to two in the above command, only two outputs would be produced for the 'y=0 MHD idl' and '3d MHD idl' cases.

'StringPlot' corresponds to the type of output you wish to produce. The first part specifies the shape of the output (i.e., y=0, 3d, etc.). The second part specifies the variables that will be included in the output. Here, 'MHD' will produce the spatial variables (x,y,z), the plasma density, the velocity components of the plasma, the magnetic field components, the level set variable (tracks the heliopause location - see Onubogu et al. 2025), the plasma pressure, and the current density components. Additionally, if the simulation is run with multi-fluid neutrals (Opher et al. 2009), the density, velocity components, and pressure of each neutral fluid are outputted as well. The third part of 'StringPlot' specifies the format of the output, 'idl' for IDL format and 'tcp' for Tecplot format.

'DnSavePlot' and 'DtSavePlot' correspond to the frequency of the solution output. Only one should be chosen, as the two correspond to different output criteria. 'DnSavePlot' reflects how many simulation steps should pass before each output. This is best when running with #TIMEACCURATE set to false (different time steps in different grid cells within the domain) considering in this case there is no set time within the simulation. 'DtSavePlot' reflects how much simulation time should pass before each output. This is best when running with #TIMEACCURATE set to true (one uniform time step for the entire domain). For 'DtSavePlot', the unit of time can also be set, with '1 h' corresponding to one hour and '1 y' corresponding to one year. If 'DnSavePlot' is used to set the output frequency, 'DtSavePlot' should be set to -1.0, and if 'DtSavePlot' is used to set the output frequency, 'DnSavePlot' should be set to -1. Please note that 'DtSavePlot' should be a float, while 'DnSavePlot' should be an integer.

For IDL only, it is also possible to set the output resolution when saving with 'DxSavePlot'. If 'DxSavePlot' is set to -1, the resolution from the simulation is used for the output. However, a uniform grid can also be produced as output. In this case, 'DxSavePlot' having a value of 0.25 means that instead of using the resolution from the simulation, the output is produced on a grid of uniform spacing with 0.25 au resolution. Please note that depending on the resolution of the simulation, this will likely produce an output with interpolated values. This command is not used with the Tecplot format.

2.2 Processing the output files

As the simulation progresses, it will produce the output files in OH/IO2 within the run directory. These output files will not yet be usable as they need to be processed first. In order to do so, within your run directory, simply execute 'PostProc.pl'. This perl script will process the output files within OH/IO2. When completed, the output will be either .out files for IDL or .plt files for Tecplot.