USER'S GUIDELINES FOR THE CODE NEURO-FVIF

Fabien Marpeau

1 Introduction

This code performs the numerical discretization of the Fokker-Planck equation in one space dimension:

$$\partial_t P(t,V) + \partial_V \big(f(V)P(t,V) - D\partial_V P(t,V) \big) = 0, \qquad V \in (V^\infty, V^T)\backslash V^R,$$

coupled with boundary conditions

$$\begin{aligned} f(V^\infty)P(t,V^\infty) - D\partial_V P(t,V^\infty) &= 0, && \text{reflecting b.c. at } V^\infty \\ P(t,V^T) &= 0 && \text{absorbing b.c. at } V^T, \end{aligned}$$

and interior conditions

$$\begin{aligned} \big[P(t,V^R)\big] &= 0, && \text{continuity of } P(t,V), \\ \big[D\partial_V P(t,V^R)\big] &= D\partial_V P(t-\tau_R, V^T), && \text{influx of neurons that crossed } V^T \text{ at } t-\tau_R. \end{aligned}$$

with the finite volume method described in the article "A finite volume method for stochastic integrate–and–fire models" (F. Marpeau, A. Barua, K. Josić), accepted for publication in *Journal of Computational Neuroscience.*
The unknown i the function P, with $P(t,V)$ being the probability that a given neuron is set at potential V at time t. The boundary V^∞ is an arbitrary large negative real number that is used for numerical purposes instead of ∞. All the neurons that reach the threshold potential V^T are reset at the reset potential $V^R < V^T$, see Figure 1.

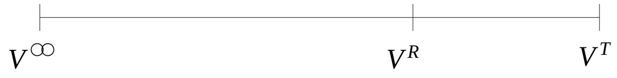

Figure 1: Domain of simulation.

2 Directions

2.1 Run the code

1. In the file called "Makefile", replace pfg95 with the command executing your fortran95 compiler.

2. To compile the code, use the command line

```
./arrow
```

3. Enter your parameters in the file PARAM.DAT (see section 3)
4. To run the code, use the command line

   ```
   ./sabor < RUN.DAT
   ```

5. To view your results, open the file "result.visit" from the visualization freeware called *visit*. Executable files of the freeware *visit* are freely available at

   ```
   https://wci.llnl.gov/codes/visit/executables.html
   ```

2.2 Advanced directions

- To change the name of your data file, open the run file "RUN.DAT" and replace "PARAM.DAT" with another name. To change the name of your result file, open the file "RUN.DAT" and replace "result" with another name, <*file_name*>. To visualize your results, you will have to open the file <*file_name*>.visit.
- Alternatively, you can create another run file instead of "RUN.DAT", <*file_name*>, containing two lines, the first of which is "PARAM.DAT" and the second line is "result". To run the code, replace the command line "./sabor < RUN.DAT" with

  ```
  ./sabor < file_name
  ```

2.3 Examples

All examples provided in the article "A finite volume method for stochastic integrate–and–fire models" are available in the directory "ktest". In all of them, the parameters are $V^{\infty} = -99$, $V^R = 0$, $V^T = 1$, $\tau_R = 0.2$. Tests 1–6 are defined by:
Test 1: $D = 0.01$, $f(V) = -V + 0.5$.
Test 2: $D = 0.1$, $f(V) = -V + 0.5$.
Test 3: $D = 0.01$, $f(V) = -V + 1.5$.
Test 4: $D = 0.1$, $f(V) = -V + 1.5$.
Test 5: $D = 0.1$, $f(V) = (V - 0.1)(V - 0.9) + 0.15$.
Test 6: $D(t) = 0.01 + 0.09 * |\cos(2\pi t)|$, $f(V) = -V + 1 + 0.5 * \sin(2\pi t)$.
To run the example Test i, type

```
cd ktest/testi
./sabor < RUN.DAT
```

3 Enter parameters

Open the file "PARAM.DAT" (or your data file if the name is different) described below and enter your parameter.
Note that the current file "PARAM.DAT" is an example of data set, the file "PARAM.DAT_COMMENTED" defines exactly the same parameters as "PARAM.DAT" but comments describing each entry have been added. A template data file is TEMPLATE.DAT, in which you can enter your own parameters (replace "PARAM.DAT" with "TEMPLATE.DAT" in the run file "RUN.DAT" to use the file "TEMPLATE.DAT" instead of "PARAM.DAT"). The parameters are described in the table below:

Vmin,Vmax	Left and right boundary (ie:threshold) of the domain of simulation
Vr	reset potential
t_0, t_max	Left and right boundary of the time interval of simulation (Note that if the code seeks for steady state, t_max is ignored and the code stops when the steady state is reached)
Vplot_min, Vplot_max	Left and right boundary of the interval on which you want to plot your solution
nbr_stock	Number of times at which you want to plot your solution (this parameter is ignored in case you select the option of creating a movie).
	Right under nbr_stock, write the times at which you want to plot your solution. The times should be written in column; for example, 0.0 0.1 0.2 will plot the numerical solution at times 0, 0.1, 0.2 (Note that in this example, the value of nbr_stock should be 3, as three snapshots are to be plotted)
info	Period (expressed in number of iterations) at which some useful information is plotted on the screed
movie	option T_start T_end number_of_images Option 1 will create a movie starting at T_start, ending at T_end with number_of_images+1 equi-spaced snapshots (In that case, the previous nbr_stock snapshots will not be plotted) Option 0 will not create a movie
steady_state	option convergence_criterion Option 1 will make the code look for a steady state and stop the code when a certain norm of (Pn+1-Pn)/dt is less than convergence_criterion Option 0 will not make the code look for a steady state
go_around	Use 0 to start a new simulation (usual case) Use 1 to start from the end of the previous simulation
schema_transport	Use 1 for 1st order Murman (Godunov) scheme Use 2 for a high-resolution Murman (Godunov) scheme with flux limiters (recommended because a lot more accurate)
cfl	value of the Courant-Friedrich-Levy ratio (select a number between 0 and 1 for stability; recommended value: 0.9)
method_syslin_diff	Iterative method solving linear systems Use 1 for Conjugate gradient Use 2 for Bi-Conjugate gradient Use 3 for GMRES (minimal residual) method Use 4 for Bi-Conjugate gradient with LU preconditioner (recommended)
test_cv_diff	Convergence criterion used by method method_syslin_diff (recommended: 10^{-8})
drift_type	Type of drift Use 1 for constant drift $f(V) = \mu$ Use 3 for LIF $f(V) = -V + \mu$ Use 4 for QIF $f(V) = (V - 0.1) * (V - 0.9) + mu$ Use 5 for time dependent LIF $f(V) = -V + 1 + 0.5 * \sin(2\pi t)$
mu	Value of μ if either LIF or QIF model is used
type_diff	Type of diffusion: Use 1 for constant diffusion $D =$diff_coeff Use 2 for time dependent diffusion $D(t) =$ diff_coeff $+ 0.09 * \lvert\cos(2\pi t)\rvert$
diff_coeff	Value of the diffusion coefficient diff_coeff
tau	value of the refractory period

initial condition	option min max Use option=1; the initial condition is a uniform distribution on the interval (min,max)
nbr_int	Number of sub-domains containing a different mesh
# int 1	
xmin,xmax,nbr_meshes	The first sub-domain is the interval (xmin,xmax) and will be discretized into nbr_meshes meshes
# int 2	
xmin,xmax,nbr_meshes	The second sub-domain is the interval (xmin,xmax) and will be discretized into nbr_meshes meshes
⋮	⋮
# int nbr_int	
xmin,xmax,nbr_meshes	The last sub-domain is the interval (xmin,xmax) and will be discretized into nbr_meshes meshes