Terrestrial Ecosystem Model in R (TEMIR) version 1.0 Manual

§1 – Setup

## Step 1 – Download TEMIR

Download the master version (v1.0) of TEMIR that is the up to date and tested version from GitHub link (<https://github.com/amospktai/TEMIR>) consisting of directories, R scripts and files as follows:

* Parent directory (*TEMIR/*)
  + Code subdirectory (*code\_ v1.0/*)
    - *initialize\_TEMIR\_v1.0.R*
    - *execution\_v1.0.R*
    - *input\_TEMIR\_ecophysiol.R*
    - *PFT\_surf\_data.R*
    - *Farquhar\_Ball\_Berry.R*
    - *FLUXNET\_functions.R*
    - *drydep\_toolbox.R*
    - *Monin\_Obukhov.R*
    - *radiative\_transfer.R*
    - *simulate\_ij.R*
    - *tools.R*
    - *geophys\_const.R*
    - *find\_hist\_stat.R*
  + Input data directory (*TEMIR\_inputs/*)
    - It contains the link to the starter dataset for a basic test run for two simulation days (20090601 and 20090602). Follow the link and download the starter dataset directly into this input data directory.

The default setting of TEMIR is fully functional using the starter data in *TEMIR\_inputs/* so testing is possible without completing the remaining steps §1:2-4 in.

## Step 2 – Download surface data

1. *TEMIR\_inputs/* above contains the default surface data files needed to run TEMIR. A greater collection of surface data can be downloaded from the CESM input data repository (<https://svn-ccsm-inputdata.cgd.ucar.edu/trunk/inputdata/lnd/clm2/>).

## Step 3 – Downloading meteorological data

1. Download meteorological data from the TGABI data repository (<https://gocuhk-my.sharepoint.com/:f:/g/personal/amostai_cuhk_edu_hk/Evqw5zUKBDtGvPIH_-l_o9IBXNImV4YjsfELXMTJNA2qzw?e=jOhEyb>) or directly from the GEOS-Chem input data repository (<http://wiki.seas.harvard.edu/geos-chem/index.php/Downloading_GEOS-Chem_data_directories>). For the password to access the data from the first repository, please email Prof. Amos Tai <[amostai@cuhk.edu.hk](mailto:amostai@cuhk.edu.hk)> directly.

§2 – Initialization

Note: All directory paths must end with a forward slash “/”

## Step 1 – Download required packages and set default directory paths

1. Go to directory *TEMIR/code\_v1.0/*, and open *execution\_v1.0.R*. Examine the beginning of the file, where it lists all R library packages TEMIR needs to run. Install all these packages in R.
2. Under *TEMIR/code\_v1.0/*, open *input\_TEMIR.R*. In the section “Directories”, specify the correct directory paths to the relevant directories; create any of these directories if they do not exist yet.
3. Most importantly, the variable *TEMIR\_dir­* should be set to be the path to the parent directory *TEMIR/*. The paths to the other directories would be correct if you have downloaded the data directory *TEMIR\_inputs/* as a subdirectory under the parent directory *TEMIR/*. Note that this copy of *input\_TEMIR.R* under *TEMIR/code\_v1.0/* is the parent copy that will be copied to the simulation directory for each ensemble of simulations. You may change these paths later in *input\_TEMIR.R* under the simulation directory if you have moved your data around.

## Step 2 – Initialize TEMIR and create simulation directory

1. Go to *TEMIR/code\_v1.0/*, and open *initialize\_TEMIR\_v1.0.R*. Set the variable *TEMIR\_dir­* at the beginning of the script to be the path to the parent directory *TEMIR/*.
2. Set the *simulation\_name* variable to what you would like to call your simulation, e.g., *simulation\_name = my\_simulation*.
3. Set the relevant simulation flags to be TRUE as needed (the default is usually okay).
4. Run the script *initialize\_TEMIR\_v1.0.R*.
5. Doing the above would create a simulation directory *my\_simulation/* under the *TEMIR/* parent directory, which contains copies of the files:
   * execution script *execution\_v1.0.R*
   * input scripts (e.g., *input\_TEMIR.R*)
   * data analysis script *find\_hist\_stat.R*

and subdirectories:

* + *hist\_data/* to contain simulated output files
  + *temp­\_data/* to contain temporary data files

§3 – Execution

## Step 1 – Configure inputs

1. Go to the simulation directory *TEMIR/my\_simulation/*. Set it to be the working directory.
2. Open the script *input\_TEMIR.R*, which has been copied directly from *TEMIR/code\_v1.0/*. Customize and configure it according to your simulation needs. The default file is for running a single-site simulation on a local machine in the debugging mode for Switzerland for two days (20090601 to 20090602) with the default surface data and without ozone damage or dry deposition. You should set options and variables to cater to your own needs.
3. In particular, set variable *debug\_flag* = *TRUE* if you want to run in the debugging mode (recommended for single-site simulations). The output data will not be saved externally, but will remain in the active environment of the R console.

## Step 2 – Run simulation and examine outputs

1. Run the execution script *execution\_v1.0.R* under *TEMIR/my\_simulation/*, either directly in R, or using a command line in a UNIX terminal (e.g., *R --slave -f execution\_v1.0.R > log &*). Command line simulations should never be run in the debugging mode.
2. Done! If you did not run TEMIR in the debugging mode, the output data and possible error messages are all saved in *TEMIR/my\_simulation/hist\_data/*. Output data are by default hourly – to extract daily or monthly statistics, you may make use of the functions in *find\_hist\_stat.R*.

(Last update: Apr 2020)