TUTORIAL to the Guincho model and software

By Vasco Manuel Nobre de Carvalho da Silva Vieira

MARETEC, Instituto Superior Técnico, Universidade Técnica de Lisboa, Av. Rovisco Pais, 1049-001, Lisboa, Portugal.

vasco.vieira@tecnico.ulisboa.pt

**Model overview and evolution**

This Guincho 2.0, published in Vieira et al. (2015a,b) is an upgrade to the published in Ocean Science by Vieira et al (2013). Developed in Matlab R2011a and later adapted to Matlab 2015, it focuses on speed of calculus. The first upgrade was calculus vectorization, replacing the for-loops by matrix algebra, thus improving speed about 12× faster. This enables running the gas flux model in parallel processing using the Single Program Multiple Data (spmd) solution. However, spmd duplicates ram memory usage, turning it massive and thus demanding partitioning the spmd into blocks so that the data no longer required may be cleared in-between. Loading massive amounts of data into the ram memory is a slow process in older laptops, revoking the advantages of parallel processing.

The Matlab Parallel Processing Toolbox has a strict set of rules for the use of variables. The 'transparency' rule disabled the hierarchical script Child.m run inside Parent.m management strategy. However, this problem was solved by estimating solubilities, transfer velocities, sea-surface agitation, atmospheric stability, etc from their respective @myfunction.m. These are all named after the respective property being modelled as 'estimate<property>.m'. Neither the calculus part of the GasFlux files nor the @myfunction files are to be edited by the user.

The air-water gas flux is estimated in single processing running the GasFluxExe_field.m. This file was used for the model testing with the Baltic field data in Vieira et al (2015a,b). The parallel processing alternative simulating the gas fluxes in the Mediterranean and North Atlantic was done by the GasFluxExe_mp.m. The variables, values for the constants, calculus settings and numerical options must all be defined in advance by the user in the beginning of the script.

**The constants**

These are divided in constants for the water, air and gas properties, automatically loaded into Matlab workspace by defining them in their respective blocks in the beginning of the script. The gas constants are specific for each gas. The script already has the values for the greenhouse gas $CO_2$, $CH_4$ and $N_2O$. These can be chosen un-commenting the values for the desired gas while commenting the values for the remaining gases. The values for other gases can be found in the works by Sander (1999) and Sarmiento and Gruber (2013).

**The variables**

These are automatically loaded into Matlab workspace by defining the file name, the directory and the respective data arrays. If it is simulated an area over a period, then 'location' should be along the lines and 'time' along the columns. This way, the 2D ocean surface (longitude-latitude) must be squeezed into a column vector.

**The numerical options**

The model simulates several geophysical processes occurring at the air-water interface. Each of these can be simulated by alternative formulations, all taken from the literature. The user must define them in the respective block in the beginning of the script. There, it is provided the tag for the respective formulation and bibliographic reference.

**Coordination between variables and numerical options**

Certain $k_w$ formulations require $u_*$, the wind at the sea-surface. If this is not given, it will be estimated the wind log-linear profile, given the wind at height z, the sea-surface roughness and atmospheric stability of the surface boundary layer. However, upon the lack of data, the model uses a chosen drag coefficient (CD) formulation.

   Other $k_w$ formulations require $u_{10}$, the wind at 10m heights, which sometimes is not available but instead the wind ($u_z$) at other heights. In these cases it must be run a preliminary simulation with a $k_w$ formulation that uses $u_*$ instead. The model will use the given $u_z$ to estimate $u_{10}$ from the wind log-linear profile and stored it in 'Results.U10'. Then, the user will have to edit the variables in the 'GasFluxExe' to U= Results.U10; and zu=10*ones(size(U)); for single processing or U=codistributed(U10); and zu=codistributed(10*ones(size(U))); for multiple processing. Finally, run the model with the $k_w$ parameterization requiring $u_{10}$.

**Estimating transfer velocities from observed gas fluxes**

The observed Eddy-Covariance gas fluxes and associated transfer velocities ($k_w$) are estimated by the 'getKfromEC.m' using the WPL correction for fluctuations in humidity and heat (Webb et al 1980). The file has two 'Settings' sections to be edited by the user and two 'Calculus' sections not to be edited by the user.

Good luck and have fun,
Vasco Vieira