Fluid2D: Parameter Description

general

Name: modelname
Default value: advection
Available value: advection,euler,boussinesq,quasigeostrophic
Description: Type of model

Name: expname
Default value: myexp
Description: Name of the experiment. Gives the prefix to the netcdf output files: "advection_his.nc" contains the snapshots and "advection_diag.nc" contains integrals timeseries

domain and resolution

Name: nx
Default value: 128
Description: Number of gridpoints in x direction in the global grid.

Name: ny
Default value: 128
Description: Number of gridpoints in y direction in the global grid.

Name: Lx
Default value: 1.
Description: Domain length along x

Name: Ly
Default value: 1.

Description: Domain length along y

Name: geometry
Default value: disc
Available value: disc, perio, square, ychannel, xchannel
Description: Domain shape. It offers a bunch of predefined masks. You may change it in your script.

Name: isisland
Default value: False
Description: Activate to impose non-zero streamfunction along certain boundaries

Name: mpi
Default value: 0
Available value: 0,1
Description: whether mpi is activated or not

Name: myrank
Default value: 0
Description: rank of the process, working on the rank-th subdomain

Name: npx
Default value: 1
Description: number of subdomains (cores) in x

Name: npy
Default value: 1
Description: number of subdomains (cores) in y

time

Name: adaptable_dt
Default value: True
Description: Set whether the time step is recomputed in realtime according to the cfl

Name: dt
Default value: 0.1
Description: model time step if adaptable_dt is False

Name: cfl
Default value: 0.5
Description: desired cfl that sets the time step in the case of 'adaptable_dt'. dt is computed in core/fluid2d.py

Name: dtmax
Default value: 5.
Description: Maximum time step of the model. It is import to control it for accelerated flows starting from the rest state and/or for flows where the stability is dominated by waves propagation

Name: rescaledtime
Default value: none
Available value: none,enstrophy
Description: Measure time with either the model time (none) or a rescaled time (enstrophy)

Name: ninterrestart
Default value: 10
Description: Number of restart outputs during the integration.

numerics

Name: timestepping
Default value: RK3_SSP
Available value: EF,LF, Heun, RK3_SSP,AB2,AB3,RK4_LS
Description: Time scheme.

Name: order
Default value: 5
Description: Order of the advective flux discretization. Should be in [1,2,3,4,5] 1,3,5 are upwind fluxes, 2,4 are centered fluxes

Name: nh
Default value: 3
Description: Number of ghost points in the halo

physics

Name: diffusion
Default value: False
Description: Add or not a diffusion term on tracer

Name: customized
Default value: False
Description: Customize the time step. If activated you should provide the name of the custom module

Name: Kdiff
Default value: 0.
Description: Diffusion coefficient. Has to be set manually. It should depends on the resolution.

Name: noslip
Default value: False
Description: Add a noslip condition along the boundaries

Name: enforce_momentum
Default value: False
Description: Enforce the global momentum conservation in closed domain in unforced case

plotting options

Name: plot_interactive
Default value: True
Description: Set on the interactive plotting. Turn it off to speed up the computation

Name: freq_plot
Default value: 1
Description: number of iterations between two plotting refresh

Name: colorscheme
Default value: minmax
Available value: minmax,symmetric,imposed
Description: Method to adjust the colorscale

Name: plotting_module
Default value: plotting
Available value: anything
Description: give the name of your plotting module (python script). It should look like core/plotting.py