SUNFLUIDH DATA SETUP

Click here to come back to the previous page

The file **input3d.dat** is the main data file to prescribe input values to sunfluidh. It contains the major part of data : geometric layout, flow features, boundary conditions, simulation management ... Each part of the file is structured by means of the NAMELIST concept. (Example of a minimal input3d.dat that contains all the mandatories NAMELIST needed by Sunfluidh)

Namelist statement

The user finds here some things to know about the namelist statement.
The NAMELIST statement defines a group of variables or arrays. It specifies a group name, and lists the variables and arrays of that group. The namelist starts with a & directly followed by the name of the namelist and ends with the / character. For instance :

```
  &Fluid_Properties    Reference_Dynamic_Viscosity= 1.84D-05, ! a comment
can be written here
                   Reference_Density= 1.19, Reference_Temperature= 293.0,
Prandtl= 0.72 /
```

Here the variable Reference_Density in the namelist Fluid_Properties is set to 1.19. In input3d.dat, a namelist or some variables in a namelist can be ommitted according to the context. For instance, for a numerical simulation without heat transfer, the namelist Fluid_Properties can be reduced to :

```
  &Fluid_Properties    Reference_Dynamic_Viscosity= 1.84D-05,
Reference_Density= 1.19 /
```

This allows us to define the simulation setup only from the useful data. If they are not set in the file input3d.dat, the data are initialized with default values.
The entire set of input data (default and user's values) are written in the file check_namelist_data.dat by sunfluidh at the beginning of each run.

BEWARE : A same namelist can be used several times in the data file in order to initialize different physical quantities belonging to a same type. For instance, if you want to initialize the physical properties of different species, you will use the same namelist (named Species_Properties_Namelist, see below) for each species. When a namelist could be used several times, the last must contains the variable End_of_Data_Block = .true. in order to specify the end of the namelist series.

Example :

```
&Species_Properties      Species_Name = "H2" , .... /
&Species_Properties      Species_Name = "O2" , .... /
```

```
 &Species_Properties      Species_Name = "N2" , ...., End_of_Data_Block =
.true. /
```

The first namelist contains the group of data describing the properties of the di-hydrogen, the second for the di-oxygen and the third for the Nitrogen. Only the last namelist contains the variable End_of_Data_Block = .true. If only one species is initialized, the namelist must however contain the variable End_of_Data_Block = .true.

Click here to come back to the previous page

Content of the data file

Herein the user finds the content of the data file before addressing in details each namelist of the data set:

- The fluid properties
  - Physical properties of the fluid (incompressible fluids, low Mach number hypothesis, multi-components gas, two-phase flows, reactive flow)
  - The reference values of the main physical quantities.
- Field initialization
  - Velocity
  - Temperature
  - Species mass fraction
  - Two-fluids interface (density distribution with discontinuity)
- Forces applied to the fluid
  - Gravity and buoyancy force
  - Sustaining force, etc ...
- Domain configuration
  - geometry of the domain
  - Immersed bodies characteristics (in order to build complex geometries)
  - domain decomposition methods (for parallel computing)
  - mesh characteristics
  - boundary conditions
- Simulation management
  - Numerical methods for solving equations
  - Numerical time step
  - time range of simulation
  - Stopping criteria
  - Backup files, checking files
  - recording rates of data acquisitions
- Data acquisition
  - Time-series
  - Instantaneous fields
  - Statistics

Namelist lookup

Click on the highlighted namelist to get more details about it (type of data, definition of data,...)

Fluid Properties	
Fluid_Properties	General properties of the fluid.
Species_Properties	Specific properties of components of the fluid (multi-component flows).
Chemical_Reactions_Features	Group of data on the chemical reaction features (reactive flows only).
Initialization of the velocity, temperature and density over the domain	
Velocity_Initialization	Data for the velocity field initialization.
Temperature_Initialization	Data for the temperature field initialization.
Species_Initialization	Data for the species mass fraction initialization.
Two_Fluids_Initialization	Data for the density field initialization in the special case of immiscible fluids.
Forces applied to the fluid	
Gravity	Data to define the direction and the magnitude of the gravity.
External_Force	To define a bulk force on the flow.
Domain configuration (geometries, domain decomposition (if MPI parallelization) & boundary conditions	
Domain_Features	Data on the geometry, the size of the domain and the mesh size as well as the domain decomposition layout (for parallel simulations only).
Heat_Wall_Boundary_Condition_Setup	Data setup on the wall boundary conditions for the heat flux.
Velocity_Wall_Boundary_Condition_Setup	Data setup on the wall boundary conditions for the velocity.
Species_Wall_Boundary_Condition_Setup	Data setup on the wall boundary conditions for the mass flux of species.
Polyhedral_Immersed_Bodies	Data setup for building every polyhedral solid objects.
Cylindrical_Immersed_Bodies	Data setup for building every cylindrical solid objects.
Inlet_Boundary_Conditions	Data setup used to define inflow boundary conditions.
Outlet_Boundary_Conditions	Data setup used to define outflow boundary conditions.
Border_Domain_Boundary_Conditions	Data setup used to define boundary conditions at the ends of the domain.
Large Eddy Simulation models	
SGS_Model	Data related to Large Eddy Simulation models that are available.
Wall and Gas Radiation modeling	
Radiative_Heat_Transfer_DOM	Data related to the Discrete Ordinates Method and SLW gas model.
Simulation management (choice of numerical methods, time parameters of simulation, recording rates of data, etc)	
Numerical_Methods (old version)	Data setup used to select the numerical methods used for solving the discretized equations.

Fluid Properties	
Numerical_Methods (new version)	Data setup used to select the numerical methods used for solving the discretized equations.
HomeData_PoissonSolver	Data used for solving Poisson's equation with the “homemade” iterative methods.
HypreData_PoissonSolver	Data used for solving Poisson's equation with the HYPRE library tools.
Level_Set_Method	Data setup for the level set method (immiscible fluids).
Simulation_Management	Data setup used to set the time parameters of the simulation .
Data acquisition	
Probe_Quantities_Enabled	Define physical quantities that will be recorded by probes during the simulation to build time series.
Probe_Location	Coordinates of probes used to record time series of physical quantities.
Instantaneous_Fields_Listing	List of physical quantities whose instantaneous fields will be recorded.
Statistical_Fields_Listing	List of statistical fields to record.
Field_Recording_Setup	Precision (simple or double) on the result fields.

Click here to go up

Click here to come back to the previous page