

# 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 omitted 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 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,...)

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

Fluid Properties	
<a href="#">Numerical_Methods (new version)</a>	Data setup used to select the numerical methods used to solve the discretized equations.
<a href="#">Level_Set_Method</a>	Data setup for the level set method (immiscible fluids).
<a href="#">Simulation_Management</a>	Data setup used to set the time parameters of the simulation.
Data acquisition	
<a href="#">Probe_Quantities_Enabled</a>	Define physical quantities that will be recorded by probes during the simulation to build time series.
<a href="#">Probe_Location</a>	Coordinates of probes used to record time series of physical quantities.
<a href="#">Instantaneous_Fields_Listing</a>	List of physical quantities whose instantaneous fields will be recorded.
<a href="#">Statistical_Fields_Listing</a>	List of statistical fields to record.
<a href="#">Field_Recording_Setup</a>	Precision (simple or double) on the result fields.

[Click here to go up](#)

[Click here to come back to the previous page](#)

From:  
<https://sunfluidh.lisn.upsaclay.fr/> - **Documentation du code de simulation numérique SUNFLUIDH**

Permanent link:  
[https://sunfluidh.lisn.upsaclay.fr/doku.php?id=sunfluidh:sunfluidh\\_data\\_setup&rev=1506093145](https://sunfluidh.lisn.upsaclay.fr/doku.php?id=sunfluidh:sunfluidh_data_setup&rev=1506093145)

Last update: **2017/09/22 17:12**

