

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

input3d.dat

```
=====
====
=====
====

      MAIN INPUT DATA FILE : 2D CHANNEL FLOW
                              INCOMPRESSIBLE
                              ISOTHERM

      DIMENSIONLESS LAYOUT :

      Length scale      : h (the channel height)
      Reynolds number    Re_h= rho_0.U_0.h/nu= 100

      dimensionless quantities :
        velocity U*      = U/U_0
        kinetic viscosity= 1/Re_h

      dimensionless domain : Lx/h= 10

      Initialisation = uniform velocity field
      inlet flowrate = uniform profil

-----

---> inflow                outflow --->

-----
J
^
|
|
|
---->I
=====
====
=====
====

&Version File_Version="VERSION2.0"/
+++++
+++++

      GENERAL LAYOUT
      (DIMENSIONLESS)

+++++
+++++

&Fluid_Properties      Reference_Dynamic_Viscosity = 1.00D-02,
Reference_Density= 1.0 /
```

```
&Velocity_Initialization I_Velocity_Reference_Value = 1.0 ,
J_Velocity_Reference_Value = 0.0 , K_Velocity_Reference_Value = 0.0 /
=====
====

                DOMAIN FEATURES
                (DIMENSIONLESS)

=====
====

&Domain_Features Start_Coordinate_I_Direction= 0.00 ,
End_Coordinate_I_Direction= 10.00,
                    Start_Coordinate_J_Direction= 0.00 ,
End_Coordinate_J_Direction= 1.00,
                    Start_Coordinate_K_Direction= 0.00 ,
End_Coordinate_K_Direction= 0.00,
                    Cells_Number_I_Direction= 128
,Cells_Number_J_Direction= 32 ,Cells_Number_K_Direction= 1,
                    Regular_Mesh= .true. /
+++++
+++++

                DEFINITION OF BOUNDARY CONDITIONS
+++++
+++++

=====
=====

                WALL BOUNDARY CONDITION SETUP
                (DIMENSIONLESS)

=====
=====

Not really necessary (Default wall boundary conditions for the velocity
are used, no heat transfer)

=====
=====

                INLET AND OUTLET BOUNDARY CONDITIONS
                (DIMENSIONLESS)

=====
=====

Keep in mind that the domain is enclosed by default.
Here the inlet and outlet conditions are located at the ends of the
domain. They replace the walls by default over the interested areas.

&Inlet_Boundary_Conditions Type_of_BC= "INLET", Direction_Normal_Plan=
1 , Flow_Direction= 1 ,
                        Plan_Location_Coordinate= 0.0      ,
                        Start_Coordinate_of_First_Span = 0.00  ,
End_Coordinate_of_First_Span = 1.00 ,
                        Start_Coordinate_of_Second_Span= 0.0    ,
End_Coordinate_of_Second_Span= 0.0  ,
                        Normal_Velocity_Reference_Value= 1.0 /
```

Outlet : Mass flowrate conservation

```
&Outlet_Boundary_Conditions Type_of_BC= "OUTLET",
Direction_Normal_Plan= 1 , Flow_Direction= 1 ,
                                Plan_Location_Coordinate= 10.0 ,
                                Start_Coordinate_of_First_Span = 0.00 ,
End_Coordinate_of_First_Span = 1.00 ,
                                Start_Coordinate_of_Second_Span= 0.0 ,
End_Coordinate_of_Second_Span= 0.0 /
```

```
=====
=====
```

BORDER BOUNDARY CONDITIONS

```
=====
=====
```

!--- No new boundary conditions are defined at the ends of the domain : walls by default are preserved, the inlet and outlet previously are defined above)

```
&Border_Domain_Boundary_Conditions West_BC_Name= "None" , East_BC_Name=
"None" , Back_BC_Name= "None" , Front_BC_Name= "None" /
```

```
+++++
+++++
```

NUMERICAL METHODS

```
+++++
+++++
```

```
&Numerical_Methods NS_NumericalMethod= "BDF2-Scheme02"
```

```
, !--- BDF2 + 2nd order centered scheme
```

```
                                MomentumConvection_Scheme="Centered-02-
Conservative" , !--- conservative form for solving the velocity
(momentum) equation
```

```
                                Poisson_NumericalMethod="Home-SORMultigrid-
ConstantMatrixCoef" / !--- SOR + multigrid method (homemade release)
for solving the Poisson's equation with constant coefficient matrix
```

```
&HomeData_PoissonSolver SolverName="SOR" , !---
Successive Over-Relaxation (SOR) method based on the red-black
algorithm
```

```
                                Relaxation_Coefficient= 1.8 , !---
Relaxation coefficient of the SOR method ( 1 <= Relaxation_Coefficient
< 2)
```

```
                                Number_max_Grid= 4, !---
Number of grid levels
```

```
                                Number_max_Cycle= 10, !--
- Number of multigrid cycles
```

```
                                Number_Iteration= 0, !---
Maximum number of SOR iterations method applied for any grid level, if
0 (or removed) the 3 next data are considered
```

```
                                Number_Iteration_FineToCoarseGrid= 15, !---
number of SOR iterations applied on any grid level during the
```

```

restriction step (before the coarsest grid computation)
                                Number_Iteration_CoarseToFineGrid= 15, !---
number of SOR iterations applied on any grid level during the
prolongation step (after the Coarsest grid computation)
                                Number_Iteration_CoarsestGrid= 15 ,      !---
number of SOR iterations applied on the coarsest grid
                                Convergence_Criterion= 1.D-08 /           !---
convergence tolerance on the residu of the Poisson's equation

+++++
+++++
                                SIMULATION MANAGEMENT
+++++
+++++
The numerical time step is imposed

&Simulation_Management      Restart_Parameter= 0 ,
                                Steady_Flow_Stopping_Criterion_Enabled =
.true. , Steady_Flow_Stopping_Criterion = 1.D-16,
                                Temporal_Iterations_Number = 100000
, Final_Time = 5.D+01 ,
                                TimeStep_Type = 0 ,
                                Timestep_Max = 1.D-03 ,
                                Simulation_Backup_Rate           = 1000
, Simulation_Checking_Rate = 101 /
=====
=====
                                PROBES MANAGEMENT
=====
=====
                                Probes order      U
, V      , W      , T      , P      , RHO
&Probe_Quantities_Enabled   Temporal_Series_For_Quantity_Enabled(:) =
.true., .true., .false., .false., .true. , .false. /

&Probe_Location   Xi= 3.0 , Xj= 0.5 , Xk= 0.0 /
&Probe_Location   Xi= 6.0 , Xj= 0.5 , Xk= 0.0 /
&Simulation_Management   Probe_TimeIterationRecordingRate= 1 ,
                                Probe_StartTimeIterationRecording= 0 ,
                                Probe_RecordingReset=.false. /

=====
=====
                                FIELDS RECORDING SETUP
=====
=====
&Field_Recording_Setup      Precision_On_Instantaneous_Fields= 2 /

!--- Snapshots

```

```
&Simulation_Management
  InstantaneousFields_RecordingReset=.false.      ,
  InstantaneousFields_TimeRecordingRate= 2.0E+00  ,
  InstantaneousFields_RecordingStartTime= 0.D-00  /

&Instantaneous_Fields_Listing  Name_of_Field = "U      " /      First
velocity component
&Instantaneous_Fields_Listing  Name_of_Field = "V      " /      Second
velocity component

!--- Statistics

END OF FILE
```

[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:2d_channel_flow_incomp_flow

Last update: **2019/11/13 16:33**

