

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

input3d.dat

```
=====
====
=====
====
                MAIN INPUT DATA FILE : 2D CHANNEL FLOW WITH A CONSTRICTION
(A SQUARE BAR)

                                INCOMPRESSIBLE FLOW
                                ISOTHERM

                DIMENSIONLESS LAYOUT :

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

INCOMPRESSIBLE DOWNSTEP FLOW

-----
                | |
                |__|
---> 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= 256
,Cells_Number_J_Direction= 64 ,Cells_Number_K_Direction= 1,
                  Regular_Mesh= .true. /

-----
                GEOMETRY OF THE IMMERSED BODIES
-----

First (and unic) immersed body

&Polyhedral_Immersed_Bodies  Xi_1= 4.5   , Xj_1= 0.5 ,Xk_1= 0.0   , Xi_2=
5.5   , Xj_2= 0.5 ,Xk_2= 0.0 ,
                                Xi_3= 5.5   , Xj_3= 1.0   ,Xk_3= 0.0   , Xi_4=
4.5   , Xj_4= 1.0   ,Xk_4= 0.0 ,
                                Wall_BC_DataSetName= "Set1"/

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

                WALL BOUNDARY CONDITION SETUP
                (DIMENSIONLESS)
=====
=====

-----
DATA SET FOR THE WALL BOUNDARY CONDITIONS
-----

first set of wall boundary condition
(This set corresponds to the default wall boundary conditions for the
```

velocity. It is just shown for example and could be removed)

```
&Velocity_Wall_Boundary_Condition_Setup
```

```
Wall_BC_DataSetName = "Set1",
West_Wall_Velocity_I= 0.0    ,    East_Wall_Velocity_I= 0.0    ,
Back_Wall_Velocity_I= 0.0    ,    Front_Wall_Velocity_I= 0.0    ,
South_Wall_Velocity_I= 0.0   ,    North_Wall_Velocity_I= 0.0   ,
West_Wall_Velocity_J= 0.0    ,    East_Wall_Velocity_J= 0.0    ,
Back_Wall_Velocity_J= 0.0    ,    Front_Wall_Velocity_J= 0.0    ,
South_Wall_Velocity_J= 0.0   ,    North_Wall_Velocity_J= 0.0   ,
West_Wall_Velocity_K= 0.0    ,    East_Wall_Velocity_K= 0.0    ,
Back_Wall_Velocity_K= 0.0    ,    Front_Wall_Velocity_K= 0.0    ,
South_Wall_Velocity_K= 0.0   ,    North_Wall_Velocity_K= 0.0 /
```

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

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 areas of interest.

Inlet : Uniform flowrate profil

```
&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_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)

!--- As "None" is the default setting for this namelist, it can be

removed

```
&Border_Domain_Boundary_Conditions West_BC_Name= "None" , East_BC_Name=
"None" , Back_BC_Name= "None" , Front_BC_Name= "None" , North_BC_Name=
"None" , South_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-Multigrid-
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= 1.7 ,      !---
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+02 ,
                        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 , End_of_Data_Block=
.true. /
&Simulation_Management Probe_Recording_Rate = 10 /

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

!--- Snapshots

&Simulation_Management Fields_Recording_Rate = 5.D+01 /
&Instantaneous_Fields_Listing Name_of_Field = "U      " ,
Recording_Enabled = .true. /      First velocity component
&Instantaneous_Fields_Listing Name_of_Field = "V      " ,
Recording_Enabled = .true. /      Second velocity component

!--- Statistics

&Simulation_Management Start_Time_For_Statistics= 1.D+03
, Time_Range_Statistic_Calculation = 5.D+00 /

&Statistical_Fields_Listing Name_of_Field = "<U>    " ,
Recording_Enabled = .true. /
&Statistical_Fields_Listing Name_of_Field = "<V>    " ,
Recording_Enabled = .true. /

```

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_with_bar_incomp_flow&rev=1512125004

Last update: **2017/12/01 11:43**

