

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

```
=====
=====
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
+++++
++
                For Navier-Stokes Equations : Numerical scheme --> BDF2 - Spatial
discretization --> 2nd order centered scheme
                                convection flux ---> conservative
form

```

For Poisson's equation : Iterative method --> SOR coupled with a multigrid method.

Parameters of the multigrid method :

```
&Numerical_Methods Numerical_Scheme= 1 ,
                    Convective_Flux_Discretization_Type = 1
                    Numerical_Method_Poisson_Equation= 1,
                    Number_max_Grid= 4 , Number_max_Cycle= 10 ,
Number_Iteration= 15 ,
                    Relaxation_Coefficient = 1.80 , Convergence_Criterion =
1.D-08 /
```

+++++

#### 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.  /
```

[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=1506954967](https://sunfluidh.lisn.upsaclay.fr/doku.php?id=sunfluidh:2d_channel_flow_with_bar_incomp_flow&rev=1506954967)

Last update: **2017/10/02 16:36**

