Boundary conditions

The problem gets different type of boundary conditions :

- For the velocity :
    - Common wall boundary conditions : No slip and impermeable boundary conditions
    - Inflow boundary condition : uniform velocity profile imposed.
    - Outflow boundary condition : mass flowrate conservation
- For the heat transfer :
    - Wall boundary conditions : Temperature imposed
    - Inflow boundary condition : Temperature imposed
    - Outflow boundary condition : zero temperature gradient

Wall boundary conditions

The walls are associated with two different immersed bodies:

- the domain ends which gets the top and bottom walls
- the step

The wall boundary conditions for the velocity are identical for every walls of the domain and they correspond to the wall boundary conditions stated by default in the code SUNFLUIH. As a consequence, they need not to be explicitly declared.
The wall temperature are however different according to the immersed bodies cited above ($T_c*=1$ for the bottom and the top walls, $T_h*=2$ for the step walls). We must define two sets of boundary conditions for the temperature.

Set 1 (for the walls of domain ends) : Only the data associated to the top (front) and bottom (back) walls are present, the other ones are useless as the corresponding walls are absent.

```
 &Heat_Wall_Boundary_Condition_Setup
  Wall_BC_DataSetName     ="Set1",
  Back_Heat_BC_Option     = 0    , Front_Heat_BC_Option    = 0 ,    !---
option value for  temperature imposed
  Back_Heat_Function_Type = 0    , Front_Heat_Function_Type= 0 ,    !---
option value for defining a uniform value of T over the wall
  Back_Wall_BC_Value      = 1.0  , Front_Wall_BC_Value     = 1.0 /  !---
temperature value (here Tc)
```

Set 2 (for the walls of the step)_ : Only the data associated to the top (noted back for immersed bodies) and right (noted west for immersed bodies) walls are present, the other ones are useless as the corresponding walls have no role in the current problem.

```
 &Heat_Wall_Boundary_Condition_Setup
  Wall_BC_DataSetName     ="Set2",
  Back_Heat_BC_Option     = 0    , East_Heat_BC_Option    = 0 ,    !---
option value for  temperature imposed
  Back_Heat_Function_Type = 0    , East_Heat_Function_Type= 0 ,    !---
option value for defining a uniform value of T over the wall
```

```
  Back_Wall_BC_Value      = 2.0  , East_Wall_BC_Value     = 2.0 /  !---
temperature value (here Tc)
```

We must now build the immersed bodies.

On the walls of the domain ends :

This is already made. Keep in mind that the computational domain is enclosed by default. Walls are automatically placed at the domain ends by the code except when other boundary conditions are specified by the user (which are going to replace the walls).

For building the step :

```
 &Polyhedral_Immersed_Bodies
                         Xi_1= 0.0  , Xj_1= 0.0  ,Xk_1= 0.0 , !---
Coordinates of the 1st corner
                         Xi_2= 2.0  , Xj_2= 0.0  ,Xk_2= 0.0 , !---
Coordinates of the 2nd corner
                         Xi_3= 2.0  , Xj_3= 1.0  ,Xk_3= 0.0 , !---
Coordinates of the 3rd corner
                         Xi_4= 0.0  , Xj_4= 1.0  ,Xk_4= 0.0 , !---
Coordinates of the 4th corner
                         Wall_BC_DataSetName     ="Set2"/     !--- ID name
associated to the suitable set of wall boundary conditions
```

Keep in mind :

- By default, the set of wall boundary conditions identified as "Wall_BC_DataSetName=Set1" is automatically associated to walls placed at the domain ends.
- For other immersed solid bodies, the variable "Wall_BC_DataSetName" links the body walls with the set of wall boundary conditions that have got the same name.
- For more information on the rules of construction Click here
- For more details on the data setup of the wall boundary conditions
  - Click here for the wall boundary conditions on the heat flux
  - Click here for wall boundary conditions on the velocity
  - Click here for wall boundary conditions on the species mass fraction"
- For more details on the data setup for building immersed solid bodies
  - Click here for the Polyhedral Immersed Bodies
  - Click here for the Cylindrical Immersed Bodies"

Return to the page Tutorial : How to build the input data file ?

Inlet boundary conditions

The inflow conditions are uniform profiles of velocity ($U_b=1$) and temperature ($T_c= 1$).

```
 &Inlet_Boundary_Conditions
              Type_of_BC                    = "INLET",  !--- Specify inflow
conditions (mass flowrate and other physical quantities imposed)
              Direction_Normal_Plan         = 1    ,    !--- Normal vector
of the inlet plan oriented along the I-direction
              Plan_Location_Coordinate      = 0.0  ,    !--- Position of
the inlet along the normal direction
              Start_Coordinate_of_First_Span = 1.0  ,    !--- Start
coordinate of the inlet along the direction related to the 1st span (here J-
direction, just above the step)
              End_Coordinate_of_First_Span   = 2.0  ,    !--- End coordinate
of the inlet along the direction related to the 1st span (here J-direction,
at the top wall)
              Start_Coordinate_of_Second_Span= 0.0  ,    !--- Start
coordinate of the inlet along the direction related to the 2nd span (here K-
direction)
              End_Coordinate_of_Second_Span  = 0.0  ,    !--- End coordinate
of the inlet along the direction related to the 1st span (here K-direction)
              Flow_Direction                = 1    ,    !--- The flow is
oriented along the increasing I-index
              Define_Velocity_profile       = 0    ,    !--- Option value
for a uniform velocity profile
              Normal_Velocity_Reference_Value= 1.0  ,    !--- Value of the
normal velocity-component (here Ub= 1) , the other ones are null.
              Temperature_Reference_Value    = 1.0 /     !--- Value of the
temperature (uniform distribution, here Tc= 1)
```

- The geometry of inlets is rectangular except when the domain is defined in cylindrical geometry (inlets fit the grid topology).
- More details on the inlet data setup can be found here
- Other examples can be found here

Return to the page Tutorial : How to build the input data file ?

Outlet boundary conditions

We here select usual outflow boundary conditions based on the flowrate conservation for treating the normal velocity component. The outflow boundary conditions other physical quantities are zero gradient condition.

```
 &Inlet_Boundary_Conditions
              Type_of_BC                    = "OUTLET", !--- Specify the
outflow conditions cited above
              Direction_Normal_Plan         = 1    ,    !--- Normal vector
of the inlet plan oriented along the I-direction
              Plan_Location_Coordinate      = 0.0  ,    !--- Position of
```

```
the inlet along the normal direction
              Start_Coordinate_of_First_Span = 0.0  ,    !--- Start
coordinate of the inlet along the direction related to the 1st span (here J-
direction, at the floor wall)
              End_Coordinate_of_First_Span   = 2.0  ,    !--- End coordinate
of the inlet along the direction related to the 1st span (here J-direction,
at the top wall)
              Start_Coordinate_of_Second_Span= 0.0  ,    !--- Start
coordinate of the inlet along the direction related to the 2nd span (here K-
direction)
              End_Coordinate_of_Second_Span  = 0.0  ,    !--- End coordinate
of the inlet along the direction related to the 1st span (here K-direction)
              Flow_Direction                 = 1    /    !--- The outflow is
mainly oriented along the increasing I-index
```

- The geometry of outlets is rectangular except when the domain is defined in cylindrical geometry (outlets fit the grid topology).
- More details on the outlet data setup can be found here
- Other examples can be found here

Return to the page Tutorial : How to build the input data file ?

Border boundary condition

The boundary conditions related to our example are already specified (walls, inflow, outflow conditions). No more boundary conditions must be defined. The namelist “Border_Domain_Boundary_Conditions” is therefore not required :

```
  &Border_Domain_Boundary_Conditions
        West_BC_Name = "None"   ,              !--- Boundary conditions
already defined for the left end of the domain (corresponding to the lower
I-index)
        East_BC_Name = "None"   ,              !--- Boundary conditions
already defined for the right end of the domain (corresponding to the upper
I-index)
        Back_BC_Name = "None"   ,              !--- Boundary conditions
already defined for the bottom end of the domain (corresponding to the lower
J-index)
        Front_BC_Name= "None" /                !--- Boundary conditions
already defined for the top end of the domain (corresponding to the upper J-
index)
```

- In this case, this namelist could be removed
- This dataset is used for specifying periodical or symmetrical boundary conditions

at the domain ends. Click here for more details and here for other examples)

Return to the page Tutorial : How to build the input data file ?