

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

input3d.dat

MAIN INPUT DATA FILE : 2D HEAT-DRIVEN CAVITY FLOW

DIMENSIONLESS FORM :

```

Ra= 1.D+04
Density scale    : rho_0 (fluid
density)
cavity
(k/H).Ra**0.5 (k thermal diffusivity)
hot ; Tc= T cold)
U/U0
(T-Tc)/(Th-Tc)
viscosity= Pr/Ra**0.5
diffusivity= 1/Ra**0.5
= Pr.T*
Ly/H= 1
Density scale    : rho_0 (fluid
Length scale  H : height of
Velocity scale ---> U0=
Temperature scale Th - Tc (Th= T
Dimensionless Velocity    U*=
Dimensionless Temperature T*=
dimensionless kinematic
dimensionless thermal
dimensionless buoyancy term
dimensionless domain Lx/H= 1 ,

```



```

=====
====

                GRAVITY
            (DIMENSIONLESS FORM)

=====
====
&Gravity Gravity_Enabled= .true. , Gravity_Angle_IJ= 90.0 ,
Gravity_Angle_IK= 90.0 , Reference_Gravity_Constant= 0.71/

=====
====

                DOMAIN FEATURES

=====
====
&Domain_Features Start_Coordinate_I_Direction= 0.00 ,
End_Coordinate_I_Direction= 1.00,
                    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= 64
,Cells_Number_J_Direction=64 ,Cells_Number_K_Direction= 1,
                    Regular_Mesh= .true. /

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

=====
=====

WALL BOUNDARY CONDITION SETUP

=====
=====

Keep in mind that the domain is enclosed by default.
No new boundary conditions are defined at the ends of the domain : the
walls by default are preserved

&Heat_Wall_Boundary_Condition_Setup
    Wall_BC_DataSetName ="Set1",
    West_Heat_BC_Option = 0 , East_Heat_BC_Option = 0 ,
    Back_Heat_BC_Option = 1 , Front_Heat_BC_Option = 1 ,
    South_Heat_BC_Option = 0 , North_Heat_BC_Option = 0,
    West_Wall_BC_Value= 1.0 , East_Wall_BC_Value= 0.0 ,
    Back_Wall_BC_Value= 0.0 , Front_Wall_BC_Value= 0.0 ,
    South_Wall_BC_Value= 0.0 , North_Wall_BC_Value= 0.0 /

=====
=====

BORDER BOUNDARY CONDITIONS : The walls located by default at the ends

```

of the domain remain unchanged

```
=====
!--- 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-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= 1.7 , !---
Relaxation coefficient of the SOR method ( 1 <= Relaxation_Coefficient
< 2)
```

```
Number_max_Grid= 5, !---
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= 3, !---
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 estimated by means of the CFL coefficient

```

&Simulation_Management      Restart_Parameter= 0 ,
                             Steady_Flow_Stopping_Criterion_Enabled =
.true. , Steady_Flow_Stopping_Criterion = 1.D-16,
                             Temporal_Iterations_Number = 1000000
, Final_Time = 5.D+02 ,
                             TimeStep_Type = 1 ,
                             Timestep_Min = 1.D-02
, Timestep_Max = 1.D-02 ,
                             CFL_Min      = 0.5
, CFL_Max      = 0.5 ,
                             Iterations_For_Timestep_Linear_Progress= 1,
                             Simulation_Backup_Rate      =
1000 , Simulation_Checking_Rate = 101 /

```

```

=====
=====
PROBES MANAGEMENT
=====
=====

```

NO PROBE

```

=====
=====
FIELDS RECORDING SETUP
=====
=====

```

```

&Simulation_Management
    InstantaneousFields_RecordingReset=.false.      ,
    InstantaneousFields_TimeRecordingRate= 5.0E+00 ,
    InstantaneousFields_RecordingStartTime= 0.D-00 /
&Field_Recording_Setup      Check_Special_Features=
"Heat_Driven_Cavity_Flow", Precision_On_Instantaneous_Fields= 2 /
Here, a special variable devoted to results of heat driven cavity flows
is active

```

```

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

```

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_heat_driven_cavity_incomp_flow&rev=1573659132

Last update: **2019/11/13 16:32**

