


```

- Pr = 0.707
- T0 = 300 K
- P0 = 101325 Pa
- Uniform molar fraction of H2O = 0.02

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

FLUID PROPERTIES
=====
====
INCOMPRESSIBLE FLUID FLOW --> Constant Density
HEAT DRIVEN FLOW          --> Activation of Heat Transfer
BOUSSINESQ ASSUMPTION     --> Thermal Expansion Coefficient = 1/T0 (
here beta = 0 ==> beta = 1/T0 )

&Fluid_Properties   Variable_Density   = .false.   ,
Constant_Mass_Flow   = .true.   , Heat_Transfer_Flow = .true.   ,
                    Heat_Capacity_Ratio = 1.4   , Reference_Density= 1.225,
Reference_Dynamic_Viscosity= 1.852D-05,
                    Reference_Temperature= 300.0   , Prandtl = 0.707,
Reference_Heat_Capacity = 1004.D0 , Thermal_Expansion_coefficient =
0.0/

=====
====
INITIALIZATION OF THE VELOCITY COMPONENTS, THE TEMPERATURE AND
SPECIES
=====
====
START FROM FLOW AT REST
AND UNIFORM TEMPERATURE at T0 = 300 K

&Velocity_Initialization   I_Velocity_Reference_Value = 0.0   ,
J_Velocity_Reference_Value = 0.0   , K_Velocity_Reference_Value = 0.0   ,
                    Initial_Field_Option_For_Velocity_I = 0,
Initial_Field_Option_For_Velocity_J = 0 ,
Initial_Field_Option_For_Velocity_K = 0/

&Temperature_Initialization   Temperature_Reference_Value = 300.0,
Initial_Field_Option_For_Temperature = 0 /

=====
====

GRAVITY

```

```

=====
====
FORCE GRAVITY ALONG THE VERTICAL AXIS POINTING DOWNWARD ( i.e. gravity
= -g.\vec{z} )
CONSIDERING DIMENSIONAL PARAMETER g = 9.81 m/s^2

&Gravity Gravity_Enabled= .true. , Gravity_Angle_IJ= 90.0 ,
Gravity_Angle_IK= 0.0 , Reference_Gravity_Constant= 9.81 /

=====
====

                        RADIATION

=====
====
AS RADIATION IS CONSIDERED :
- ACTIVATE THE RADIATIVE SOLVER [default = .false.] ( ONLY FOR 3D
CARTESIAN PROBLEMS !! )
- SOLVE THE RADIATIVE PROBLEM EVERY 5 CONVECTIVE TIMESTEP ( LIMIT TIME
CONSUMPTION , KEEP THIS PARAMETER LOWER THAN 5~8 FOR STABILITY ... )
[default = 1]
- IF STARTED FROM SCRATCH, FORCE THE SOLVER TO ITERATE OVER
FirstIterations=200 LOCAL ITERATIONS FOR INCIDENT FLUXES CONVERGENCES
AT WALLS AND VOLUMIC RADIATIVE SOURCE TERM [default = 20]
- FOR EACH RADIATIVE PROBLEM SOLVING STEPS, ITERATE OVER
RadiativeLocalIterations=20 SUB-ITERATIONS OR UNTIL
RadiativeConvergenceTolerance=5.E-05 RESIDUAL
ERROR IS REACHED [default = 1.E-15]
- WallRadCoeff AND VolRadCoeff ARE FOR DEVELOPEMENT ONLY ... [default
= 1]
- CONSIDER THE "LATHROP" SCHEME TO INTERPOLATE THE CELL-FACES
RADIATIVE INTENSITY [default = STEP]
- CONSIDER THE ANGULAR DISCRETISATION WITH S10 LEVEL SYMMETRIC
QUADRATURES Squad = 10 ( 120 DIRECTIONS IN VOLUMES, 60 DIRECTIONS ON
WALLS) [default = 8]
- CONSIDER BLACK WALLS ON DIRICHLET WALLS AND REFLECTIVE WALLS ON THE
OTHERS [default = 0.1]
- CONSIDER THE MEDIUM AS A REAL GAS MIXTURE :
+ ACTIVATE THE SLW MODEL ActivateGas=.true. [default = .false.]
+ SPLIT THE ABSORPTION COEFFICIENT DOMAIN IN 8 WEIGHTED SUM OF GRAY-
GASES NbGas = 8 [default = 1]
+ ka_min AND ka_max REPRESENTS THE MINIMUM AND MAXIMUM RANGE OF THE
ABSORPTION COEFFICIENT DOMAIN in m^{-1} [default = 0]
+ CONSIDERS THE MEDIUM AS AN AIR-H2O GAS MIXTURE WITH UNIFORM MOLAR
FRACTION x = 0.02 [default = 0.07]

&Radiative_Heat_Transfer_DOM activateRadiation=.true. ,
RadiativePeriod = 5, FirstIterations=200,
RadiativeLocalIterations=20,
RadiativeConvergenceTolerance = 5.E-05,
WallRadcoeff = 1.0 , VolRadCoeff = 1.0, RadiativeScheme
= "LATHROP",

```

```

        ActivateGas=.true., NbGas = 8, ka_max=570.,
ka_min=6.3e-07,
        Pref=101325.0, Href = 1.0, spec='H2O',xaref=0.02,
xaUniform=0.02,
        Squad = 10, WallEmissivity = 1.0 1.0 0.0 0.0 0.0 0.0 /

=====
=====
                        DOMAIN FEATURES
=====
=====
- CONSIDER HERE A CUBICAL CAVITY WITH WALL REFINED CELLS GIVEN IN
SEPARATE MESH FILES
- WE CONSIDERS AN MPI DOMAIN DECOMPOSITION PROBLEM ON 2x2x3 MPI
PROCESSES

&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= 1.00,
                Cells_Number_I_Direction= 40
,Cells_Number_J_Direction= 40 ,Cells_Number_K_Direction= 30,
                Number_OMP_Threads= 1,
                MPI_Cartesian_Topology= .true. ,
                Total_Number_MPI_Processes= 12,
                Max_Number_MPI_Proc_I_Direction= 2 ,
Max_Number_MPI_Proc_J_Direction= 2, Max_Number_MPI_Proc_K_Direction= 3,
                Regular_Mesh= .false. /

+++++
+++++
                        DEFINITION OF BOUNDARY CONDITIONS
+++++
+++++
=====
=====
                        WALL BOUNDARY CONDITION SETUP
=====
=====
- WE CONSIDER DIRICHLET TEMPERATURE CONDITION ON HOT AND COLD WALLS
(Heat_BC_Option = 0)
- AND WALL CONVECTION-RADIATION COUPLING AT THE OTHER WALLS
(Heat_BC_Option = 5)

&Heat_Wall_Boundary_Condition_Setup
        Wall_BC_DataSetName ="Set1",
        West_Heat_BC_Option = 0 , East_Heat_BC_Option = 0 ,
Back_Heat_BC_Option = 5 , Front_Heat_BC_Option = 5 ,

```

```

South_Heat_BC_Option = 5 , North_Heat_BC_Option = 5,
    West_Wall_BC_Value= 300.005 , East_Wall_BC_Value= 299.995 ,
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
=====
=====
!--- 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
+++++
+++++
PARTIAL DIAGONALISATION TECHNIQUE IS EMPLOYED FOR THE POISSON PROBLEM
==> Numerical_Method_Poisson_Equation    = 3

&Numerical_Methods  NS_NumericalMethod= "BDF2-Scheme02"
,      !--- BDF2 + 2nd order centered scheme
                                MomentumConvection_Scheme="Centered-02-
Conservative" ,      !--- conservative form for solving the velocity
(momentum) equation
                                TemperatureAdvection_Scheme="Centered-02-
Conservative",      !--- conservative form for solving the temperature
(enthalpy) equation
                                Poisson_NumericalMethod="Home-
PartialDiagonalization" /      !--- Partial Diagonalization for
Poisson's equation
+++++
+++++
                                SIMULATION MANAGEMENT
+++++
+++++
- START FROM SCRATCH IF Restart_Parameter= 0 OR FROM EXISTING FILES IF
Restart_Parameter= 3
- WE CONSIDERS THAT THE PROBLEM WILL REACH A STEADY STATE AND WILL
EVOLVE IN TIME WITH FIXED CFL PARAMETER

&Simulation_Management    Restart_Parameter= 3 ,
                                Steady_Flow_Stopping_Criterion_Enabled =
.true. , Steady_Flow_Stopping_Criterion = 1.D-14,

```

```

Temporal_Iterations_Number = 100 , Final_Time
= 3.D+04 ,
TimeStep_Type = 1 ,
CFL_Min = 0.3 , CFL_Max = 0.3 ,
Timestep_Min = 1.D-03 , Timestep_Max =
1.D+01 ,
Iterations_For_Timestep_Linear_Progress= 1,
Probe_Recording_Rate = 1000
,
Simulation_Backup_Rate = 5000
, Simulation_Checking_Rate = 20 /

=====
=====
PROBES MANAGEMENT
=====
=====
=====
=====
FIELDS RECORDING DECLARATION
=====
=====
&Field_Recording_Setup Check_Special_Features=
"NOHeat_Driven_Cavity_Flow" /
&Simulation_Management Fields_Recording_Rate = 5.D+02 /
&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
&Instantaneous_Fields_Listing Name_of_Field = "W" ,
Recording_Enabled = .true. / Third velocity component
&Instantaneous_Fields_Listing Name_of_Field = "T" ,
Recording_Enabled = .true. / Temperature
&Instantaneous_Fields_Listing Name_of_Field = "P" ,
Recording_Enabled = .true. / Pressure
&Instantaneous_Fields_Listing Name_of_Field = "divU" ,
Recording_Enabled = .true. / Momentum divergence

```

From:
<https://sunfluidh.lisn.upsaclay.fr/> - Documentation du code de simulation numérique SUNFLUIDH

Permanent link:
https://sunfluidh.lisn.upsaclay.fr/doku.php?id=sunfluidh:3d_heat_driven_cavity_incomp_flow_gasrad&rev=1511968952

Last update: 2017/11/29 16:22

