**Description of parameter file for "channelflow" code (including rotation in the y-direction, scalar equation for stratification)**

**Hao Lu 11/02/2007, revised on 2/26/2008, 3/5/2008, 4/29/2008, 5/26/2008, 6/1/2008**

1. Filename is "condition.param"
2. **[Example]**

```
1.  nu = 0.25
2.  dtmax = 0.1
3.  CFLmin = 0.1
4.  CFLmax = 0.9
5.  nRac = 20.0
6.  T0 = 0.0
7.  T1 = 10.0
8.  dT = 1.0
9.  Omega = 0.0
10. Usurface = 0.0
11. isurfp = 0
12. Up0 = 0.0
13. Up = 0.0
14. Op0 = 0.0
15. Op = 0.0
16. kpx = 0.0
17. kpz = 0.0
18. ishearstress = 0
19. tau12 = 0.0
20. tau32 = 0.0
21. ihyper = 0
22. nuu = 0.0
23. iscalar = 0
24. g = 0.0
25. kappa = 1.0
26. BC = DbDt
27. bottomvalue = 0.0
28. topvalue = 1.0
29. sbound = 0
30. smax = 1.0
31. smin = 0.0
32. dPdx = 0.0
33. iforce = 0
34. kfmin = 3.5
35. kf = 4.0
36. kfmax = 4.5
37. ef = 0.4
38. forcingtime = 0.0
39. iLES = 0
40. baseflow = PlaneCouette
41. timestepping = CNRK2
42. initstepping = SMRK2
43. nonlinearity = Rotational
44. dealiasing = DealiasXYZ
45. taucorrection = true
46. constraint = PressureGradient
47. Ubulk = 0.0
48. Lx = 2.0
49. Ly = 2.0
50. Lz = 2.0
```

3. There are 50 parameters totally; many of them can be not included in a "condition.param" file if using the default values. The order is arbitrary, but the format "name = value" must be followed for each parameter.
4. **[Explanation] (R: double precision real number; I: integer number)**

nu = 0.25: R; kinetic viscosity; the default value is 0.25

dtmax = 0.1: R; maximum time step for time marching; the default value is 0.1

CFLmin = 0.1 : R; minimum CFL number; the default value is 0.1

CFLmax = 0.9 : R; maximum CFL number; the default value is 0.9

nRac = 20.0: R; maximum local Rayleigh number refer to Ra_c=657.511; the default value is 20.0

T0 = 0.0 : R; initial time; the default value is 0.0
T1 = 10.0 : R; final time; the default value is 10.0
dT = 1.0 : R; time step to output assessments (such as spectrum); the default value is 1.0

Omega = 0.0 : R; rotation rate in the y-direction; the default value is 0.0

Usurface = 0.0 : R; mean velocity on top surface; the default value is 0.0;
isurfp = 0 : I; identification of adding perturbation on top; the default value is 0; isurfp=0: no perturbation;
isurfp=1: u=Usurface+Up0*sin(Op0*t)+Up*sin(Op*t-kpx*x*2π/Lx-kpz*z*2π/Lz); v=w=0
isurfp=2: u=Usurface; v=Up0*sin(Op0*t)+Up*sin(Op*t-kpx*x*2π/Lx-kpz*z*2π/Lz); w=0
isurfp=3: u=Usurface; v=0; w=Up0*sin(Op0*t)+Up*sin(Op*t-kpx*x*2π/Lx-kpz*z*2π/Lz)
Up0 = 0.0 : R; perturbation velocity (zero mode); the default value is 0.0;
Up = 0.0 : R; perturbation velocity; the default value is 0.0;
Op0 = 0.0 : R; frequency for zero mode; the default value is 0.0;
Op = 0.0 : R; frequency; the default value is 0.0;
kpx = 0.0: R; the default value is 0.0
kpz = 0.0: R; the default value is 0.0

ishearstress = 0: I; identification of using shear stress boundary condition; if ishearstress = 0, Dirichlet BC; if ishearstress = 1, shear stress BC, isurfp=1 or 3 cannot be used if using shear BC; the default value is 0
tau12 = 0.0: R; tau12=(∂u/∂y+∂v/∂x); the default value is 0.0
tau32 = 0.0: R; tau32=(∂w/∂y+∂v/∂z); the default value is 0.0

ihyper = 0: I; identification of using hyper-viscosity; if ihyper=0, no hyper-viscosity is included, if ihyper=1, hyper-viscosity will be included in momentum equations; the default value is 0
nuu = 0.0: R; magnitude of hyper-viscosity; the default value is 0.0

iscalar = 0 : I; identification of including scalar equation; the default value is 0; if iscalar=0, then scalar equation is not included; if iscalar=1, then scalar equation is included, but scalar does not effect momentum equation; if iscalar=2, then scalar equation is included, and scalar effect momentum equation (stratification). Note, set iscalar=0 if you don't want to compute a scalar equation, and it will save more than 25% CPU cost.

g = 0.0 : R; value of gravitational acceleration, which describes the density fluctuation effect in momentum equation; the default value is 0.0;

kappa = 1.0 : R; diffusion coefficient for scalar equation; the default value is 1.0

BC = DbDt : ={DbDt, DbNt, NbDt, NbNt}; boundary condition for scalar equation; the default value is DbDt; upper "D", "N" mean Dirichlet and Neumann, lower "b", "t" mean bottom and top

bottomvalue = 0.0: R; value of scalar boundary condition at bottom; the default value is 0.0
topvalue = 1.0: R; value of scalar boundary condition at top; the default value is 1.0

sbound = 0: I; set scalar bound or not; if sbound=1 or 3, s≤smax; if sbound=2 or 3, s≥smin; if sbound=-1 or -3, check s≤smax but not set; if sbound=-2 or -3, check s≥smin but not set; otherwise, no bound; the default value is 0 (no bound)
smax = 0.0: R; maximum value of scalar; the default value is 1.0
smin = 0.0: R; minimum value of scalar; the default value is 0.0

dPdx = 0.0: R; mean pressure gradient; the default value is 0.0

iforce = 0: I; identification of including Gaussian forcing; the default value is 0; if iforce=0, then Gaussian forcing is off; if iforce=1, then Gaussian forcing is on; Gaussian forcing range is kfmin<k<kfmax, the peak wave-number is kf, the energy input rate is ef, and until forcingtime

- kfmin = 3.5: R; the default value is 3.5
- kf = 4.0: R; the default value is 4.0
- kfmax = 4.5: R; the default value is 4.5
- ef = 0.4: R; the default value is 0.4
- forcingtime = 0.0: R; the default value is 0.0 (won't stop forcing if forcingtime ≤0.0)

iLES = 0: I; identification of LES (SGS) model; the default value is 0; if iLES=0, then no LES modeling; if iLES=others, different SGS models will be included according to this id number.

(We suggest using the default values for the following parameters)
baseflow = PlaneCouette: ={Zero, PlaneCouette, Parabolic}; base flow; the default value is PlaneCouette

timestepping = CNRK2: ={CNFE1, CNAB2, CNRK2, SMRK2, SBDF1, SBDF2, SBDF3, SBDF4}; time stepping scheme; the default value is CNRK2; currently, rotation and stratification is included only using CNRK2

initstepping = SMRK2: ={CNFE1, CNAB2, CNRK2, SMRK2, SBDF1, SBDF2, SBDF3, SBDF4}; initial time stepping scheme; the default value is SMRK2; not very serious, can be arbitrary or just use the default value

nonlinearity = Rotational: ={Rotational, Convection, Divergence, SkewSymmetric, Alternating, Alternating_, LinearAboutField, LinearAboutProfile}; method to include nonlinear term; the default value is Rotational; Rotational form is more accurate than Convection form and Divergence form for many cases, and Rotational form is much faster than SkewSymmetric form

dealiasing = DealiasXYZ: ={NoDealiasing, DealiasXZ, DealiasY, DealiasXYZ, DealiasX}; de-aliasing flag; the default value is DealiasXYZ

taucorrection = true: ={true, false}; tau correction or not; the default value is true;

constraint = PressureGradient : = {PressureGradient, BulkVelocity}; flow constraint to be enforced; the default value is PressureGradient

Ubulk = 0.0 : R; bulk velocity value; the default value is 0.0

Lx = 2.0 : R; box dimension in the x-direction; the default value is 2.0
Ly = 2.0 : R; box dimension in the y-direction; the default value is 2.0
Lz = 2.0 : R; box dimension in the z-direction; the default value is 2.0

5. Flags to run this program: there are one optional flag to run the program, and both flags are integer numbers. For example, "run 2". Let's use a general form "run i" to explain the details.

| I | Plot3D (ParaView) | TecPlot X (X>9) | $U^+$ in wall coordinates |
|---|---|---|---|
| ±1 | Yes | No | No |
| ±2 | No | Yes | No |
| ±3 | Yes | Yes | No |
| ±4 | Yes | No | Yes |
| ±5 | No | Yes | Yes |
| ±6 | Yes | Yes | Yes |
| ±7 | No | No | Yes |
| Otherwise (or not set) | No | No | No |

If i<0, then program won't save velocity/pressure/scalar data. For example, set i=−8 (or no input of i) in order to turn off everything.