## ASSIGNMENTS

## Introduction to Finite Volume method

1. *Consider the computational domain* $\Omega = [0,1]^2$. *The source term is* $\mathbf{f} = \mathbf{0}$. *The exact solution of such a problem is:* $\mathbf{u}_{ex}(x,y) = (y,-x)$ *and* $p_{ex}(x,y) = 0.5(x^2+y^2) - 1/3$. *Perform a mesh convergence analysis based on Cartesian uniform orthogonal grids for* $\nu = 10$.

2. *Consider the lid driven cavity benchmark. Employ the following initial condition and source term for the temperature transport equation:*

$$T_0(x,y) = 1 \quad \text{for} \quad \sqrt{(x-0.05)^2 + (y-0.05)^2} \leq 0.01 \quad \text{otherwise} \quad T_0(x,y) = 0,$$
$$f_T(x,y) = \alpha T \quad \text{with} \quad \alpha \in R.$$

   *Note that the source term needs to be treated in an implicit way.*

## Navier Stokes equations in stream function vorticity formulation

1. *Perform the vortex merger simulation by using* icoFoam *and carry out a comparison with the results obtained by the stream-function vorticity formulation in terms of all the variables involved (velocity, pressure, stream function and vorticity). Concerning the computation of the pressure for the stream-function vorticity formulation, you can refer, e.g., to* *https://www.iist.ac.in/sites/default/files/people/psi-omega.pdf*

2. *Perform the lid driven cavity simulation by using the stream-function vorticity formulation and carry out a comparison with the results obtained by* icoFoam *in terms of all the variables involved (velocity, pressure, stream function and vorticity). Concerning the boundary conditions to be enforced for the vorticity, you can refer, e.g., to* *https://www.iist.ac.in/sites/default/files/people/psi-omega.pdf*

## Quasi Geostrophic equations for ocean flows

1. *Perform the double-gyre wind forcing simulation for* $Re = 200$ *and* $Ro = 0.0016$ *on meshes* $4 \times 8$, $8 \times 16$ *and* $16 \times 32$ *and carry out a comparison (efficiency and accuracy) between QGE and BV-*$\alpha$ *models.*

2. *Employ the BV-Bardina model (https://www.lume.ufrgs.br/bitstream/handle/10183/128048/000974186.pdf?sequence=1) and carry out a comparison (efficiency and accuracy) between it and the BV-*$\alpha$ *model for the double-gyre wind forcing simulation for* $Re = 450$ *and* $Ro = 0.0036$ *on meshes* $4 \times 8$ *and* $8 \times 16$.

## Compressible Euler equations for nearly hydrostatic flows

1. *Following what done for the Straka benchmark, build the rising thermal bubble simulation by http://camp.cos.gmu.edu/papers/ijnmf-fwave.pdf. Plot the contour of* $\theta'$ *for* $t =$ *1020 s and* $\theta'_{max}$ *v.s.* $t$ *for the meshes* $h = 250$, $h = 125$ *and* $h = 62.5$.

2. *Modify the pressure splitting in the source code as* $p = p' + p_{ref}$ *where* $p_{ref} = 1000\left(1 - \dfrac{gz}{C_p\theta_0}\right)^{\frac{C_p}{R}}$ *and make a comparison between the new code and the old one for the Straka benchmark for* $h = 100\ m$.

## Cardiovascular flows

1. *Extend to a three-dimensional configuration the idealized blood vessel benchmark. Then, consider a more complex Windkessel model based on a* $RCR$ *circuit providing the outlet boundary condition for the pressure.*