# Problem C1.6. Vortex transport by uniform flow

## Overview

This problem is aimed at testing a high-order method's capability to preserve vorticity in an unsteady inviscid flow. Accurate transport of vortexes at all speeds (including Mach << 1) is very important for Large-Eddy and Detached-Eddy simulations, possibly the workhorse of future industrial CFD simulations, as well as for aeronautics/rotorcraft applications.

## Governing Equations

The governing equations are the unsteady 2D Euler equations, with a constant ratio of specific heats of $\gamma = 1.4$ and gas constant $R_{gas}=287.15$ J/kg K.

## Flow Conditions

The domain is first initialized with a uniform flow of pressure $P_\infty$, temperature $T_\infty$ and given Mach number (see **Testing Conditions** bellow), and a vortical movement of characteristic radius $R$ and strength, $\beta$, is superposed around the point of coordinates ( $X_c$ , $Y_c$ ):

$$\delta u=-(U_\infty*\beta)*(y-Y_c)/R*\exp(-r^2/2)$$
$$\delta v=(U_\infty*\beta)*(x-X_c)/R*\exp(-r^2/2)$$
$$\delta T=0.5*(U_\infty*\beta)^2*\exp(-r^2)/C_p$$

$$u_0=U_\infty+\delta u, v_0=\delta v$$

where

$$C_p=\frac{\gamma}{(\gamma-1)}*R_{gas}$$

$$r=\sqrt{((x-X_c)^2+(y-Y_c)^2)}/R$$

and $U_\infty=M_\infty*\sqrt{(\gamma*R_{gas}*T_\infty)}$ is the speed of the unperturbed flow.

Fluid's pressure, temperature and density are prescribed such that the over imposed vortex is a steady solution of the stagnant (e.g. without uniform transport) flow situation :

$T_0=T_\infty-\delta T$ , $\rho_0=\rho_\infty*(T_0/T_\infty)^{\left(\frac{1}{(\gamma-1)}\right)}$ and $\rho_\infty=P_\infty/(R_{gas}*T_\infty)$ . Pressure is computed as $P_0=\rho_0*R_{gas}*T_0$ .

The superposed vortex should be transported without distortion by the flow. Thus, the initial flow solution can be used to assess the accuracy of the computational method (see **Requirements** bellow).

## Geometry

The computational domain is rectangular, with $(x,y)=[0..L_x]x[0..L_y]$ .

## Boundary Conditions

Translational periodic boundary conditions are imposed for the left/right and top/bottom boundaries respectively.

## Testing Conditions

Assume that the computational domain dimensions (in meters) are $L_x=0.1, L_y=0.1$ and set $X_c=0.05$ [m], $Y_c=0.05$ [m] (marking the center of the computational domain), and $P_\infty=1.e5$ N/m2 and $T_\infty=300\,K$ .

Consider the following two flow configurations:

1. "Slow vortex" : $M_\infty=0.05$ , $\beta=1/50$ , R = 0.005.
2. "Fast vortex" : $M_\infty=0.5$ , $\beta=1/5$ , R = 0.005.

Define the time-period **T** as $T=L_x/U_\infty$ and perform a "long" simulation, where solution is advanced in time for *50 time-periods (dT = 50 T).*

The above flow configurations and simulation time define two different testing conditions.

## Requirements

1. For both testing conditions, perform two sets of simulations, on both *regular* meshes (uniform Cartesian nodes distribution) and the corresponding *randomly perturbed* meshes (meshes provided, see **Additional Notes** bellow).

2. Compute solutions on a series of minimum three successively refined grids, with grid-sizes $h$ : $L_x/32$ (grid 1), $L_x/64$ (grid 2), $L_x/128$ (grid 3), etc.

3. Advance the solution in time as required (50 time periods, **T**) and compute the **L2**-norm of the error at the end of the simulation, as advised in the guidelines, using the two velocity-vector components (u, v), $L_2(err)$ .

4. Compare the numerical solution at the end of simulation, with the exact solution (i.e. the solution after initialization, $u_0$ and $v_0$ ).

5. For each test condition considered, perform a sensitivity study to determine the appropriate time-step size, ***dt***. The final results obtained *on the finest mesh and for the duration considered,* while using the time-step ***dt***, should be time-step size insensitive: the difference in the measured $L_2(err)$ should not change with more than 0.1%, if the time-step size is reduced from ***dt*** to 0.5***dt***.

6. Study the numerical order of accuracy, e.g. $L_2(err)$ v.s. a characteristic grid-size $\bar{h}$ , defined as $\breve{h}=1/(nDOFs)^{\frac{1}{ND}}$ (where ND = {2, 3} for 2D and 3D respectively) *(*see **Guidelines***)*, and discretization order *p*.

7. Submit the following sets of data (for each testing condition considered):

   - *L2(err)* v.s. $\bar{h}$ , for different characteristic grid-sizes $\bar{h}$ and discretization orders *p*. Note: at least 3 data points are required for each regression line.

– The computational cost (in work units) to perform the entire simulation (on both the regular and perturbed meshes), for different discretization orders *p*.

## Additional Notes

- Only the "slow vortex" simulations (on regular and perturbed meshes) are mandatory.

- Successively refined regular 2D meshes (both tri- and quad-meshes) and respective 3D meshes (tet- and hex-meshes) in GMESH format for four different mesh sizes, are provided for convenience.

File names: ***2d_[tri/quad]_grid-[1/2/3/4].msh*** and **3*d_[tet/hex]_grid-[1/2/3/4].msh*** .

- Randomly perturbed meshes, of corresponding average mesh-sizes *(h)*, where the mesh's nodes are randomly displaced with a maximum distance $\delta_{max}$ :

$$\delta_{max}=0.15*h$$

in both X- and Y-coordinate directions, are also provided.

File names: **rp_*2d_[tri/quad]_grid-[1/2/3/4].msh*** and **rp_3*d_[tet/hex]_grid-[1/2/3/4].msh*** .