VI1 - Vortex transport by uniform flow

how5@cenaero.be

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. This verification case is particularly suitable for verifying temporal order of accuracy.

Governing Equations

The governing equations are the unsteady 2D Euler equations, with a constant ratio of specific heats of γ = 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_∞ , temperature T_∞ and given Mach number M_∞ (see Testing Conditions below), and a vortical movement of characteristic radius R and strength ß, is superposed around the point of coordinates (X_c,Y_c), leading to the following expressions for the local velocity components u and v, as well as the temperature T

$$u = U_\infty\left(1 - \beta \cdot \frac{y - Y_c}{R} \cdot e^{-r^2/2}\right)$$

$$v = U_\infty \beta \cdot \frac{x - X_c}{R} \cdot e^{-r^2/2}$$

$$T = T_\infty - \frac{U_\infty^2 \beta^2}{2C_p} e^{-r^2}$$

here we define the heat capacity at constant pressure C_p

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

and the non-dimensional distance r to the (initial) vortex core position

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

whereas the free stream velocity U_∞ is given by

$$U_\infty = M_\infty \sqrt{\gamma R_{gas} T_\infty}$$

The fluid pressure p, temperature T and density ρ are prescribed such that the over imposed vortex is a steady solution of the stagnant (e.g. without uniform transport) flow situation :

$$\rho_\infty = \frac{p_\infty}{R_{gas} T_\infty}$$

$$\rho = \rho_\infty \left(\frac{T}{T_\infty}\right)^{\frac{1}{\gamma-1}}$$

$$p = \rho R_{gas} T$$

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 below).

Geometry

The computational domain is rectangular, with (x , y)=[0 ..Lx] x [0.. Ly] .

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 Lx=0.1 ,Ly=0.1 and set Xc=0.05 [m], Yc=0.05 [m] (marking the center of the computational domain), and $P_\infty=10^5$ N/m2 and T_∞=300K . Consider the following two flow configurations:

1. "Slow vortex" : M_∞=0.05 , ß=1/50 , R = 0.005.
2. "Fast vortex" : M_∞=0.5 , ß=1/5 , R = 0.005.

Define the time-period T as T=Lx /U_∞ 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 below).
2. Compute solutions on a series of minimum three successively refined grids, with grid-sizes h : Lx /32 (grid 1), Lx /64 (grid 2), Lx /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), L2 (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 L2 (err) should not change with more than 0.1%, if the time-step size is reduced from dt to 0.5dt.
6. Study the numerical order of accuracy, e.g. L2 (err) v.s. a characteristic grid-size h˜, defined as h≅1 /(nDOFs) $^{1/ND}$ (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. h˜, for different characteristic grid-sizes 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 GMSH format for four different mesh sizes, are provided for convenience. File names: 2d_[tri/quad]_grid-[1/2/3/4].msh and 3d_[tet/hex]_grid-[1/2/3/4].msh .
- Randomly perturbed meshes of corresponding average mesh sizes h, where the mesh nodes are randomly displaced with a maximum distance δ_{max}=0.15h in both x and y

directions, are also provided. File names: rp_2d_[tri/quad]_grid-[1/2/3/4].msh and rp_3d_[tet/hex]_grid-[1/2/3/4].msh