22.315 - APPLIED COMPUTATIONAL FLUID DYNAMICS AND HEAT TRANSFER

You are required to submit answers prepared only by yourself

TAKE HOME MIDTERM QUIZ

Due 64 hrs. after receipt, 11:59 p.m. on CANVAS

GENERAL RECOMMENDATIONS *(useful)*

1. **Do not panic!** It is long but structured and I want to see how much you have learned so far.
2. **Start from problems 2 and 3,** which should be quick. Dedicate 10 minutes to problem 2 and 30 minutes to problem 3**. Don't let these take more than an hour to 90 minutes.**
3. **Focus on building the problem correctly** and check that the solution makes sense using a medium mesh. Only after all the postprocessing is set start comparing methods and meshes.
4. **If you run out of time,** focus on answering questions appropriately and either reduce the number of meshes or the number methods (for example only use MUSCL). You need a minimum of 3 runs to come up with some conclusions.

Problem 1 – ROTATING DISK DESCRIPTION (70 Points)

We have discussed in class code verification. This test case is designed to evaluate the accuracy of implemented numerical approximations in your CFD software, with particular attention to the code applicability to transient flow simulations and eddy resolving methods (e.g LES).

The test case consists of a **2D simulation of a vortex advected by a uniform inviscid flow field.** After a convective time $\tau_{adv}=20s$, supposing that there are no interactions between the various vortices on the infinite domain, the vortex profile **should be the same as the initial one**, but translated by a distance $=U_{adv} * \tau_{adv}$. The test is illustrated in Fig. 1.

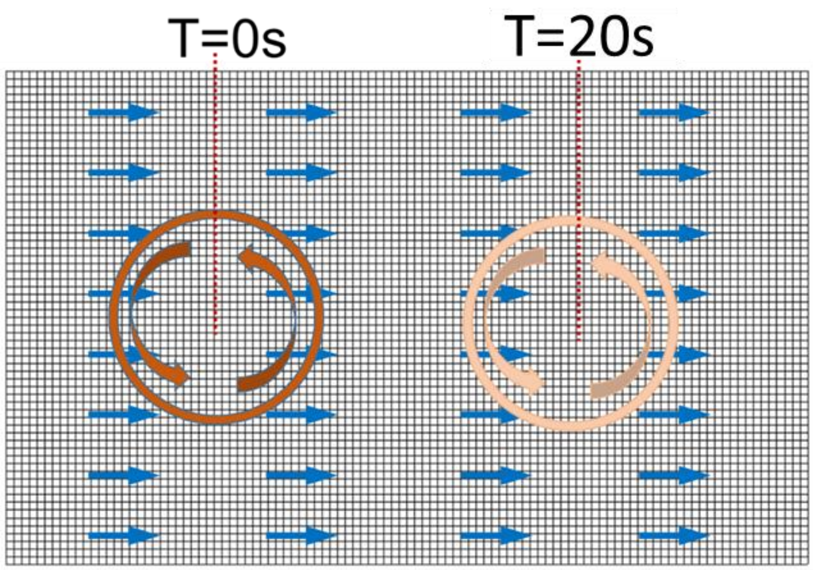


Figure 1. Principle of vortex advection case

The vortex flow field is defined as:

$$u(x,y) = -\frac{y}{r}\frac{f_t}{f_{max}} \tag{1}$$

$$v(x,y) = \frac{x}{r}\frac{f_t}{f_{max}} \tag{2}$$

where *r, x and y* are the distances from any given point to the origin of the vortex system, f_{max} = 0.3 is the maximum tangential velocity and f_t is given by:

$$f_t = \frac{\tanh(r)}{\cosh^2(r)} \tag{3}$$

The uniform flow has a constant velocity of 0.25 m/s. The parameters of the fluid and vortex are presented in Table 1. The fluid is considered to be inviscid (and no turbulence model is used). ***Thus, the only source of dissipation comes from the numerical schemes.***

Figure 2 shows the geometrical configuration to be adopted, and the location of the coordinate system where the vortex should be located at T=0s.:

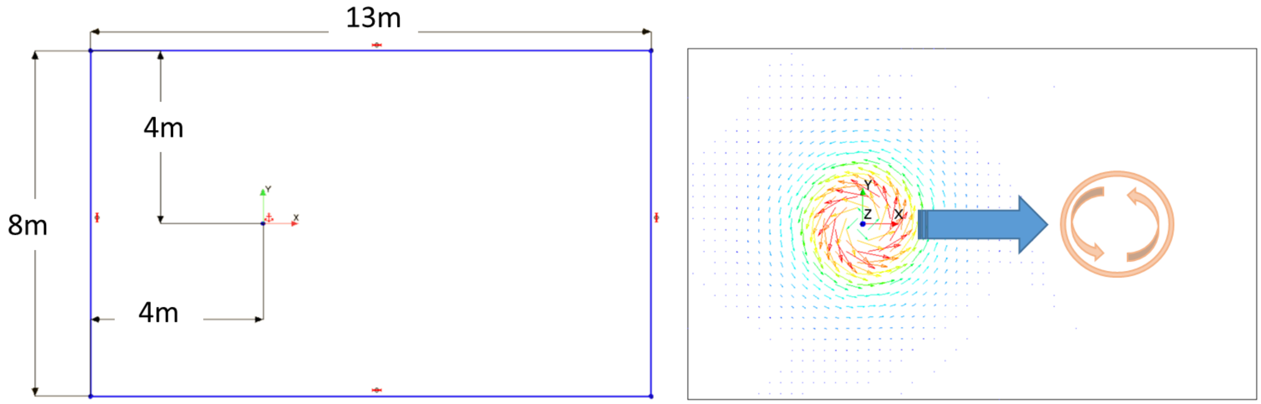


Figure 2. Geometrical configuration (left panel), Initial vortex field (right panel).

Table 1 Fluid Properties and Convective Velocities

P_0=1.013E5 Pa
ρ = 1
U_{adv}= 0.25 m/s
V_{adv}= 0 m/s

ORDER OF SPATIAL CONVERGENCE (70 Points)

Generate 2 separate Plots presenting (35 Points)

- the L2 error variation vs the grid spacing $\Delta x = \Delta y$
- the Order of Accuracy *p* vs the grid spacing $\Delta x = \Delta y$
  using 4 regular quadrangle grids for a combination of interpolation schemes shown in table. The errors should be computed at t = 20 seconds of simulation time. UD=Upwind Based with gradient/slope limiting and MUSCL/CD is a MUSL based higher order scheme.

Recommended cell sizes for the exercise:

- (1) $\Delta x = \Delta y = 0.5$m - (2) $\Delta x = \Delta y = 0.25$m - (3) $\Delta x = \Delta y = 0.125$m - (4) $\Delta x = \Delta y = 0.0625$m

CASE 1	**CASE 2**	**CASE 3**
SIMPLE algorithm	SIMPLE algorithm	SIMPLE algorithm
Momentum	***Momentum***	***Momentum***
1st Order UD	2nd Order UD	MUSCL/CD 0.1 blending
Time	***Time***	***Time***
2ndOrder	2ndOrder	2ndOrder

- Discuss the details of how you implement appropriate boundary conditions. (6 Points).
- Discuss the influence of iteration error and how you have estimated its impact on the evaluation of the order of spatial convergence (6 Points).
- Discuss the influence of temporal discretization and how you have estimated its impact on the evaluation of the order of spatial convergence (6 Points).
- Discuss the errors of the different methods tested, are they of the same type, and how do you judge them (6 Points).
- Is the order of spatial convergence consistent with the expected accuracy of the methods? is it expected to vary with the grid resolution and why? (6 Points).
- Discuss any expected/unexpected findings. Are the results consistent with your expectations and if not, how do you justify the discrepancy (5 Points).

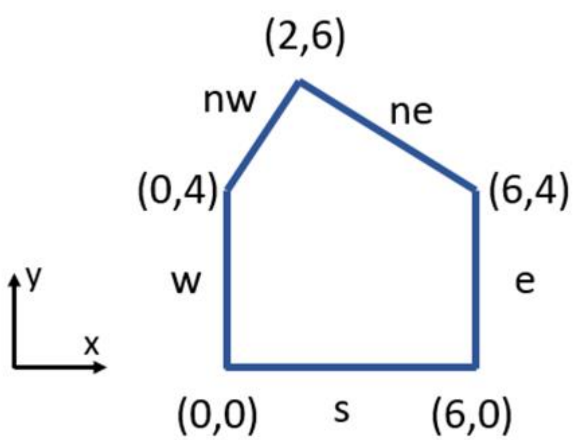
Note 1 – ***Make sure to postprocess!*** In order to confirm the correctness of the setup and in order to be able to answer all the questions. Conclusions should be based on your results. The last question can be very useful in case something in your solution is incorrect, so make sure to discuss unexpected findings.

Problem 2 (18 Points) – Short Questions, short answers.
(NOTE: the more you write the higher the risk of writing something wrong, so be brief)

1. Under the assumption of incompressible behavior, what difference exists in the fully developed profile of velocity between water and liquid mercury at equal Reynolds number conditions? Explain why. **(3 Points)**
2. For an incompressible Navier-Stokes solution, is it necessary to input the value of the absolute pressure? If so, why? **(3 Points)**
3. Define the term "bounded" when applied to an advection scheme and state, without proof, whether a 2nd order scheme can be bounded? **(3 Points)**
4. What is meant by the statement that an advection scheme is Total-Variation-Diminishing (TVD)? (*note: math is better than words*) **(3 Points)**
5. If co-located pressure and velocity are used in a finite volume solver and advective velocities are calculated by linear interpolation *decoupling* or *checkerboarding* of the pressure field appears. Suggest two common remedies that have been used to remove *checkerboarding*. **(3 Points)**
6. Evidence suggests that PISO is more efficient in time-dependent calculations, but SIMPLE and its variants are better in direct iteration to steady state. Explain, briefly, why you think this occurs. **(3 Points)**

Problem 3 (12 Points) – Short Numerical Question.

a. The figure below depicts a 2-d polyhedral cell in a finite-volume calculation. Vertices are given in the figure along with velocity in the adjacent table. At this instant $\rho = 1.0$ everywhere.
   1. Calculate the volume flux out of each face (assume unit depth). **(6 Points)**
   2. Show that the flow is not incompressible and find the time derivative of density. **(6 Points)**



Face	*u*	*v*
w	6	3
s	4	4
e	3	7
ne	2	3
nw	1	3.5

Useful Definitions

We can define an L2 norm for problem 1 as below:

$$L2\ error = \sqrt{\frac{1}{ncells}\sum_{cells}\left(\left(U - U_{analytical}\right)^2 + \left(V - V_{analytical}\right)^2\right)}$$

Remember from class that we can define

- **Order of Accuracy** {p} based on the L2 norm of the solutions as:

$$p = \frac{\ln\left(\frac{L2error_{2h}}{L2error_h}\right)}{ln(r)}$$

Where r is the mesh size reduction factor.

Example of L2 Norm Plots and order of Accuracy plots (NOTE: figure below are only representative and not specific this exam test case)