Homework

Turbulence and CFD models: Theory and applications
Hands-on session 3

Task 1

Re-run tutorial 3, "Flow around a circular cylinder - 2D LES simulation", at a different Reynolds number from the one used during the hands-on session, and complete the following tasks,

- Compute lift and drag coefficients.
- Plot the signals of the lift and drag coefficients.
- Compute the descriptive statistics of the lift and drag coefficient signals. You must compute the following quantities; mean, variance, standard deviation, and minimum and maximum values.
- Using FFT, compute the shedding frequency (dominant frequency) of the lift signal and drag signal. You can use Fluent's built-in FFT calculator or an external application.
- Compute the Strouhal number.
- Plot contours of velocity, pressure, and vorticity (vorticity magnitude and ω_z).
- Plot velocity vectors.

You must run an unsteady simulation, in laminar or turbulent regime. If you run in turbulent regime, feel free to use a different turbulence model and compare the outcome with that of the LES simulation.

In the video tutorial, we covered geometry and mesh generation, so feel free to use a different domain and mesh. You can use any near-the-wall treatment (wall resolving or wall modeling).

This case has plenty of experimental and numerical data, so compare your solution with the values found in the literature and quantity the error.

You must write a small overview of your case setup, that is, turbulence model used, mesh used, boundary conditions, discretization schemes, solution method, average and maximum y^+, short discussion of the results, and so on.

Compulsory questions

1. If you plot the signals of the lift and drag coefficients, you will notice that the shedding frequencies are different. Why the frequency of the drag signal is twice the value of the frequency of the lift coefficient?

2. What criterion can you use to determine when to stop an unsteady simulation? Illustrate your arguments using the data gathered during this simulation.

Task 2 (optional)

If you want to go for the extra mile, try to complete the following optional tasks,

- Compare the outcome of the Fractional Step method against the outcome of the PISO method (do not run the PISO method using the non-iterative marching option). Is the computing time the same? Are the solutions the same?
- Run the simulation using a very large time-step ($CFL > 10$), and quantify the error between this solution and the solution obtained using a moderate-to-low CFL number ($CFL < 4$).
- Measure the width of the wake at two different locations.
- Plot the turbulent energy spectrum.
- Generate a movie showing the vortex shedding. You can plot contours of velocity, pressure, and vorticity. Do not send me the movie, simply share a link.

General guidelines

- You must run the case at a Reynolds number different from the one used during the hands-on session.
- Be technical and concise when written the overview of your case setup. This is very important when writing technical reports.
- If you want to share large files, please share a link (Dropbox, Onedrive, Google Drive, etc). Do not send large files to my email.
- For movies, share a YouTube link. The movie must be a least 20 seconds long.
- Please, do not overdo. You can write this report in no more than 10 pages (figures and tables included).
- You can write your report in English or Italian.
- Do not hesitate to contact me if you have any question.

Deadline

The deadline to submit your homework is 8 May 2020. You can send it to my email: joel.guerrero@unige.it