Computational Fluid Dynamics (ME4095)

Final Exam - take home project

08.01.2024

Prepare a case to be solved using FLUENT software or MATLAB CFD Tools among the following three cases. Search through the web or library to find an article (or thesis) and simulate the case in the article and compare the results and discuss it in your report.

- First create a mesh and then give the proper boundary conditions using preprocessor of the program. Diluted mesh will be printed in your report.
- Run the case to get the solution flow field with appropriate residuals, under-relaxation factors etc. Use first order scheme for convective terms and SIMPLE algorithm with segregated solvers.
- Afterwards, plot the results using the post-processor which include vector plots of the flow field and pressure contours, stream lines etc.

Cases can be chosen from one of the following:

1. Backward-facing Step Flow – Plane (UG or MSc students)
2. Backward-facing Step Flow – Axisymmetric (UG or MSc students)
3. Lid-Driven cavity flow (UG students only)

Notes:

1. Fluid is steady-incompressible (water or air)
2. If flow is turbulent use standard **k-ε** model.
3. Flow velocities are within low subsonic region (Mach number < 0.3)
4. All cases are going to be assumed 2-dimensional.
5. Prepare a report in MS-Word describing your case, plot the grid mesh created (do not use triangular cells), Show basic steps of the solution. Present the results and compare with your reference. Also cite your references.
6. Due date is 26.01.2024 - 23:59.