Evaluating the accuracy of the computational fluid dynamics solver, FLUENT

Jumpei Nomura[1,2], Andrew Meade[3] and Javier Villarreal[3]

[1] Department of Social and Environmental Engineering, Vehicle and Environmental Systems, Hiroshima University, Higashi-Hiroshima, Japan

[2] Nakatani RIES: Research & International Experiences for Students, Rice University, Houston, Texas, U.S.A.

[3] Department of Mechanical Engineering, Rice University, Houston, Texas, U.S.A.

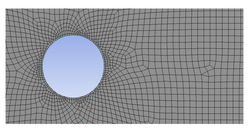
What is Computational Fluid Dynamics (CFD)?

- Branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flow.
- Experimental development can be too costly and time consuming.
- The equations of fluid mechanics are solvable for only a limited number of flows.

Research Project Objectives

- Simulation of viscous and vortex flow of a circular cylinder at various values of the Reynolds number
- Simulation of viscous flow of an NACA 0012 airfoil at various angles of attack

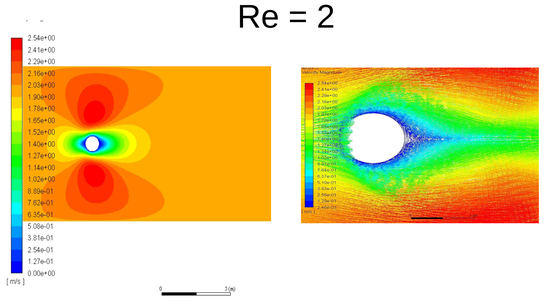
Software Analysis Using FLUENT



- Geometry and Mesh
  - Design geometry to calculate a flow
  - Create mesh on that geometry
  - The finer the mesh, the more accurate the data
- Physical Model
  - Set the model to solve the flow problem
  - Steady, unsteady, incompressible, viscous flow used for a cylinder
  - Incompressible, viscous flow used for an airfoil
- Boundary Condition
  - Set for velocity, pressure and viscosity

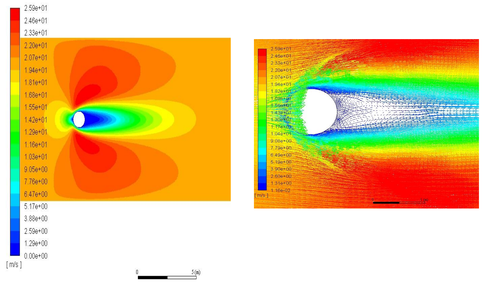
Result 1: – Steady Incompressible Flow Over a Cylinder

Re = 2



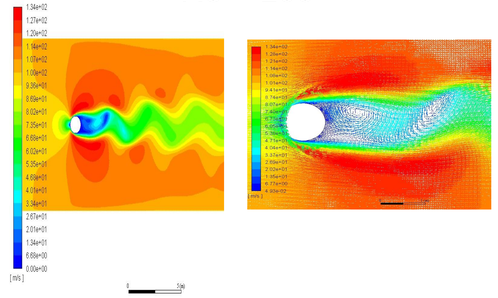
- No vortex

Re = 20



- Vortices form in the wake of the cylinder

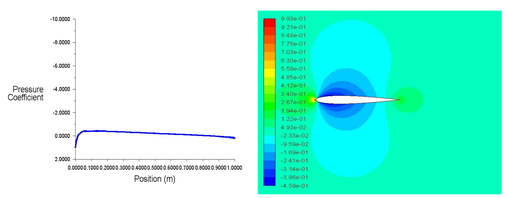
Re = 100



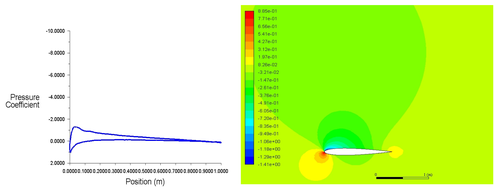
- Oscillation of the solution due to vortex shedding

Result 2: – Incompressible Flow Over an NACA 0012 Airfoil

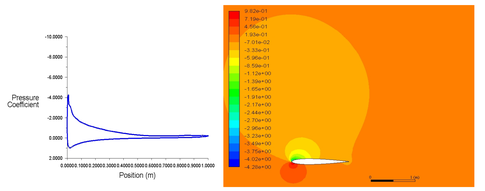
- Simulation of Pressure Coefficient at various angles
- Re = 3E+6, M = 0.3



Angle of attack = 0 deg.



Angle of attack = 3.86 deg.



Angle of attack = 9.86 deg.

Conclusion

- The wake characteristics behind a cylinder depend on Re number.
- Cp distribution around an airfoil is dependent on angle of attack and Mach number.

Future Work

- Run simulations of transonic and supersonic flow
- Observe the formation of shock waves across airfoils
- Model wings and even full aircraft models

Reference

[1] Charles D. Harris, “Two-Dimensional Aerodynamic Characteristics of the NACA0012 Airfoil in the Langley 8-foot Transonic Pressure Tunnel, *NASA Technical Memorandum*, (1981), p. 57-61

[2] Joel H. Ferziger and Milovan Peric, “Computational Methods for Fluid Dynamics”, *Springer*, (1999), p. 1-37

Acknowledgements

This research internship was conducted as part of 2017 Nakatani RIES Fellowship supported by the Nakatani Foundation.

Special thanks to Prof. Andrew Meade, Javier Villarreal, Prof. Junichiro Kono, Sarah Phillips, Kenji Ogawa, and Aki Shimada.