

1

Introduction

Fluid mechanics and heat transfer have extensive application. From aeronautical industry to automatic industry, it is applied to several areas. Some of the notable areas are:

- 1) Aeronautical industry – design of airplane to electronic devices
- 2) Automobile industry
- 3) Building and bridge aerodynamics (Selvam 2017)
- 4) Electronic cooling (Silk et al. 2008; Sarkar and Selvam 2009)
- 5) Environmental flow and heat transfer
- 6) Metrological flow and weather prediction
- 7) Hydraulic flow
- 8) Water treatment (Liu and Zhang 2019)
- 9) Wind energy

In all areas, computer modeling has been extensively used in the recent years, and this branch of computation is called computational fluid dynamics (CFD). CFD provides the detail of velocities, pressure, and temperature at every point at each time in the computational domain. This helps to create animation in time and provides the detail of the flow changes in time. To gather this much of information from experiment is very expensive. In certain situation like weather prediction, we cannot do any experiment and computer simulation is the only tool to predict the weather. The major challenge in CFD is to develop a reliable computer model for a particular application. If this is established for a particular application, it will be very useful in the design of the system.

The CFD is applied from single-phase flow to multiphase flow. In the multiphase flow, it can be liquid–vapor flow, solid–liquid flow, and solid–liquid–vapor flow. In these flows, chemical reactions can occur. Some of the challenging flows I encountered in the past 30 years are

Wind–bridge interaction: Here the bridge moves due to wind and hence beyond certain velocities the bridge can flutter as reported in Selvam et al. (2002). Below the critical velocity for flutter, the bridge will not have unlimited oscillations. The concept of moving grid has to be used in addition to regular CFD modeling. The Tacoma Narrow Bridge failed due to flutter for a velocity of 64 km/h (17.8 m/s) in 1940. The critical velocity for flutter for Great Belt East Bridge is 252 km/h (70 m/s) as reported in Selvam et al. (2002). The critical velocity

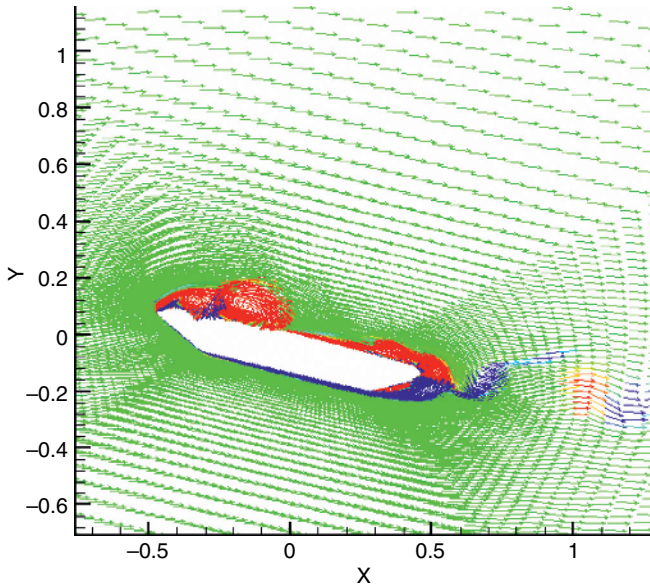


Figure 1.1 Flow around great Belt East Bridge during flutter condition.

depends upon the shape and structural properties of the bridge. The flow features during flutter condition are shown in Figure 1.1.

Heat transfer mechanism in spray cooling: Here, when a liquid droplet impacts a hot plate with a bubble growing in a thin liquid film; heat is removed due to complex interaction of droplet impact and vapor bubble. This high heat removal phenomena are explained in Selvam et al. (2006). For this, multiphase flow modeling of liquid and vapor is considered. In Figure 1.2 the liquid and vapor phases before the droplet impacts a vapor bubble in a liquid film are shown.

Tornado–building interaction: This study is reported in Selvam and Millett (2003, 2005). Here in a tornadic flow how a roof of a building is lifted up is explained using CFD. Figure 1.3 shows the velocity vector over the roof when a tornado-like vortex coincides with the center of a cubical building.

1.1 Brief Review of Steps in CFD Modeling

In the CFD modeling, the steps are very similar to well-established solid mechanics modeling. The major differences being most of the CFD applications are nonlinear and hence several iterations or time steps need to be performed.

Step 1: Grid Generation or Preprocessing: This may be the most time-consuming part if one has a complicated domain. If simple domain where in rectangular grid systems can be used, then the grid generation may be an easier task. Still one has to focus on the grid refinements in the boundary layer and in the regions of steep flow. Also, one has to make sure that grid spacing variation should not be high. The preferred ratio is 1.0–1.5. Very large ratios

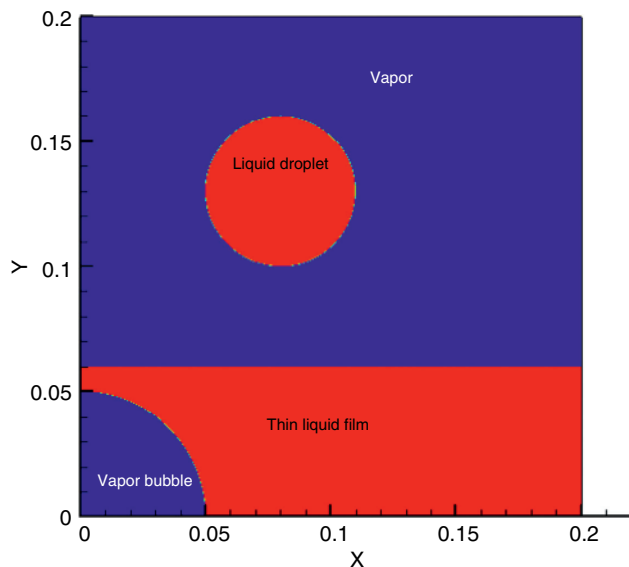


Figure 1.2 Multiphase flow modeling of liquid droplet impacting a vapor bubble in liquid film.

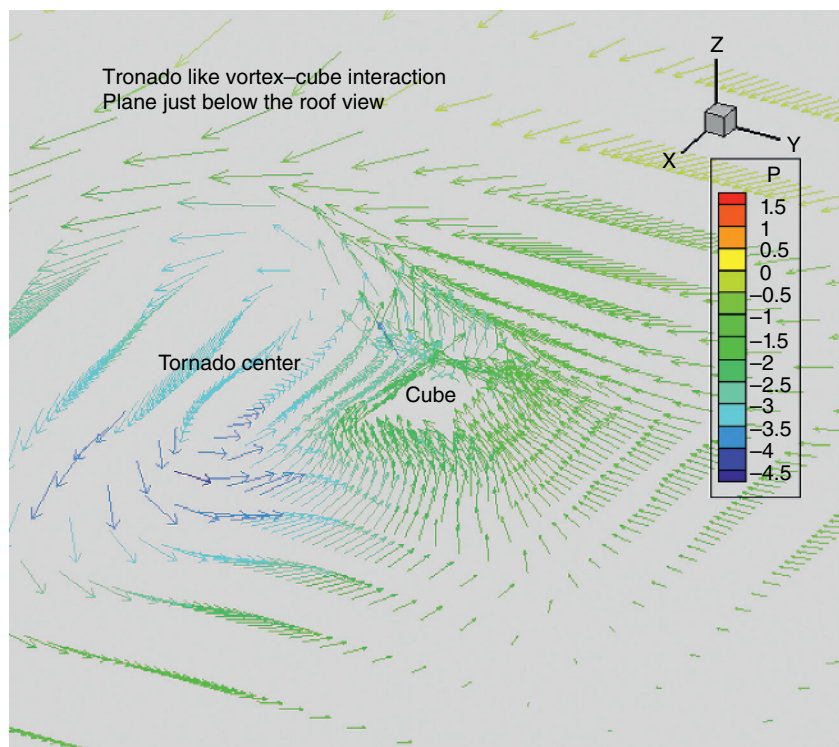


Figure 1.3 Velocity vectors around the roof when a tornado-like vortex coincides with the center of a cubical building.

like more than 5 or 10 are not preferable. For this step, extensive grid generation programs were developed in the recent years.

Step 2: Flow Solver: Once the grid is generated for a particular problem and the proper initial and boundary conditions are given for the problem, one can solve the Navier–Stokes (NS) equations. This is the most computer time-intensive step. For this several methods from direct to iterative procedures are developed to solve the $Ax = b$ equations. To reduce computer time, high performance or parallel computing is also utilized. Sarkar and Selvam (2009) utilized parallel computing to reduce the computer time from 50 to 3 days for spray cooling applications. They also compared the performance of different iterative solvers in the parallel computing environment.

Step 3: Postprocessing: In this step, the output from flow solver is processed to mine valuable information. Here this can be done by regular x–y graphs, contours, vector plots, and the combination of all. If the data is written for several time steps for the whole region, one can make animation using software like TECPLOT, and flow features can be investigated. The flow visualization technique is very sophisticated and some time it is an art than science.

If it is a design, then one can change the parameters of the flow variable or computational domain and further computer runs can be made for further investigations.

Benefits of CFD:

- 1) Data available for all points in space and time.
- 2) Inexpensive comparing to experiment. Especially with the developments in computer speed and memory, CFD programs can run in a personal computer. The major hurdle is validating the CFD with experiment to have reliability.
- 3) Visualization and animation of data to understand the physical problem is easy to implement. This helps anyone to understand complex fluid phenomena.

1.2 CFD for Wind Engineering or Computational Wind Engineering

In wind engineering, the loads on building and bridges are obtained from wind tunnel (WT) measurements or field measurements. The field measurement is very expensive and only very limited field studies are conducted like Texas Tech University building. Currently, WT is the major tool used to investigate forces on buildings and to develop code regulations like ASCE 7-16. In recent years, CFD is emerging as an alternate tool. For more than 30 years, different researchers raised its capabilities and slowly it is becoming a reasonable tool to be used in wind engineering because of the availability of high-performance computers with large storage capacities. The work reported by Selvam (1992) took more than a day for one computer run. With the current computer capabilities, one can solve the same problem in few minutes. Hence, the speed increased may be more than 100 times in a single processor. With multiple processors, we can increase the speed at least 10 times. If the CFD model is well validated with experiments, then it becomes the most economical tool compared to experiments. The way finite element method (FEM) is used in solid mechanics area

in the industry and research nowadays, the hope is someday CFD will be a tool in wind engineering. This book is a stepping stone to achieve the preceding objective.

To apply CFD in wind engineering, one needs to be familiar with the following topics:

- 1) Meteorology or atmospheric flow
- 2) Fluid mechanics
- 3) Turbulence
- 4) Random process or stochastic process
- 5) Numerical techniques like finite difference method (FDM) or FEM for fluid mechanics
- 6) Wind engineering
- 7) Visualization
- 8) Structural dynamics
- 9) Fluid–structure interaction
- 10) Water (wave-storm surge)–wind–structure interaction as in hurricane
- 11) Grid generation
- 12) Parallel computing

In the current work, we may not touch topics beyond point 7 in the preceding list because of lack of time. For the other topics, we will go into detail only what we use in our work. We use simple computational domain to reduce the difficulty of making proper grid. One can see the grid generation complexity in the wind–bridge interaction study, as shown in Figure 1.1. In the industry for complex 3D problem, one or two engineers may be spending two or three months to make a proper grid. Even in wind engineering, we only work on straight wind. We will not discuss much about the other types of winds (tornado and thunderstorm downdraft) due to lack of time. From 1960 onward, field observations and WT testing have been used to find pressures on building. Because CFD takes lots of computer time and memory, only recent years CFD application in wind engineering has emerged with more reliability.

In hurricane-type sever wind, in addition to wind effects on structures, water surge and wave effect produce enormous damage. This leads to multiphase flow (water and air) effect on buildings. Future application may involve water–wind effect on structures.

References

- Liu, X. and Zhang, J. (2019). *Computational Fluid Dynamics: Applications in Water, Wastewater, and Storm Water Treatment*. ASCE Publication.
- Sarkar, S. and Selvam, R.P. (2009). Direct numerical simulation of heat transfer in spray cooling through 3D multiphase flow modeling using parallel computing. *Journal of Heat Transfer* 131: 121007-1–121007-8.
- Selvam, R.P. (1992). Computation of pressures on Texas Tech Building. *Journal of Wind Engineering and Industrial Aerodynamics* 43: 1619–1627.
- Selvam, R.P. (2017). CFD as a tool for assessing wind loading. *The Bridge and Structural Engineer* 47 (4): 1–8. [Review paper-available as opensource].
- Selvam, R.P. and Millett, P.C. (2003). Computer modeling of tornado forces on buildings. *Wind & Structures* 6: 209–220.

- Selvam, R.P. and Millett, P.C. (2005). Large eddy simulation of the tornado-structure interaction to determine structural loadings. *Wind & Structures* 8: 49–60.
- Selvam, R.P., Govindaswamy, S., and Bosch, H. (2002). Aeroelastic analysis of bridges using FEM and moving grids. *Wind & Structures* 5: 257–266.
- Selvam, R.P., Lin, L., and Ponnappan, R. (2006). Direct simulation of spray cooling: effect of vapor bubble growth and liquid droplet impact on heat transfer. *International Journal of Heat and Mass Transfer*. 49: 4265–4278.
- Silk, E.A., Golliher, E.L., and Selvam, R.P. (2008). Spray cooling heat transfer: technology overview and assessment of future challenges for micro-gravity application. *Energy Conversion and Management* 49: 453–468.