CFD Logic Use

PAFSE Project: Scenario 1: Droplets & the Physics of Viruses Transmission

ADDITIONAL DOCUMENT

Software Usage CFD Logic

Sustainable Development Goals	
Domains of Citizenship Education	Sustainable Development Health
Competencies when finishing Compulsory Schooling	Critical and creative thinking, well-being, health and environment Scientific, technical and technological knowledge Languages and texts Information and Communication

Framework

Briefly, *Computational Fluid Dynamics (CFD),* consists of analyzing and solving problems involving phenomena of fluid dynamics and energy transfer, such as chemical reactions through computational power. Among the main areas of application of CFDs are the following: [1]

- Aerodynamics in the aviation and automotive sectors;
- Ship hydrodynamics;
- Internal combustion of engines and steam turbines;
- Turbomachinery;
- Ventilation of electrical systems;
- Chemical engineering processes such as separation and mixing and polymer modeling;
- HVAC system (heating, ventilation and air conditioning) in buildings;
- Environmental engineering: assessment of the emission of pollutants into the atmosphere;
- Weather;
- Biomedical Engineering: evaluation of blood flow in arteries and veins.

One of the main challenges of using CFD technology is related to the fact that the analysis and processing of this type of problems is complex, since it requires a complete and detailed description of the phenomenon to be studied, and therefore, the computational costs are relatively high, as well as the costs of maintaining machines. However, the results produced from the application of CFD techniques for problem solving have their advantages that counterbalance the identified challenges, such as: [1]

- Substantial reduction of economic costs of new production lines;
- Possibility of studying systems in which obtaining experimental results is difficult or even impossible to obtain (e.g., large systems);
- Possibility of studying systems on harsh conditions at and above their operating limit (e.g., evaluation of the probability of accidents);
- Virtually unlimited level of highly detailed results.

CFD Application Logic

There are several codes available for the use of CFD, which have already incorporated numerical algorithms that allow solving fluid flow problems and energy transfer phenomena. These codes, through their interfaces, allow the user to enter the parameters of the problem and to examine the results of the simulations. Essentially, all CFD codes have three fundamental elements, which are used in the following order:

1. Pre-processing;
2. Solver;
3. Post-Processing.

Pre-processing

It consists of the introduction and characterization of a fluid flow or energy transfer problem in the CFD program, usually through an interface between program and user. In this step, the tool user performs the following activities: [1]

- Definition and modeling of the geometry to be analyzed: the computational domain;
- Mesh generation – The subdivision of the domain into a set of smaller subdomains (cells);
- Selection of the chemical or physical phenomenon that needs to be modeled (e.g. indication of the numerical sub-models to be applied for each problem);
- Definition of fluid properties;
- Appropriate specification of boundary conditions in cells that coincide with domain boundaries;

About 50% of the time spent on a CFD job is dedicated to the definition and modeling of geometry, and the generation of mesh.

In the images illustrated below, the steps of generating a CFD work mesh are exemplified.

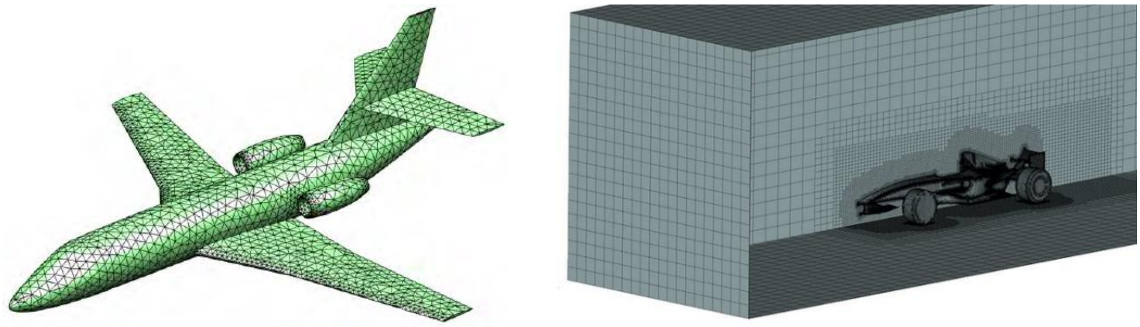

Figure 1. Mesh Generations CFD Jobs

Solver

Essentially, there are three methods of numerical problem solving: finite difference method, finite element method and finite volume method (FVM). The latter is usually the most widely used method in CFD. The numerical resolution goes sequentially through the following steps:

1. Integration of the general conservation equations governing the flow of a fluid in the volume controls (cells) of the domain;
2. Discretization – conversion of the result of the integration of the equations into an algebraic system of equations (matrix);
3. Solving algebraic equations through the iterative method;

CFD codes already have several integration, discretization, and equation resolution techniques, adaptable to the most diverse problems, depending on the user's choice. Among the main CFD codes on the market, ANSYS/FLUENT, STAR-CCM+ and OpenFOAM stand out.

Post-processing

It consists of the analysis phase of the results obtained in the simulations. Depending on the graphical and computational capabilities, result visualization tools take different forms, including: [1]

- Visualization of the modeled geometry, as well as the generated mesh;
- Plots of fluid properties (speed, temperature, pressure, among others);
- Shaded line and contour plots;
- 2D and 3D surface plots;

- Measurement of the particle trajectory;
- Manipulation of graphical display orientations (translation, rotation, zoom in zoom out);
- Animation production (videos).

The images below illustrate several types of results that are used to analyze CFD works.

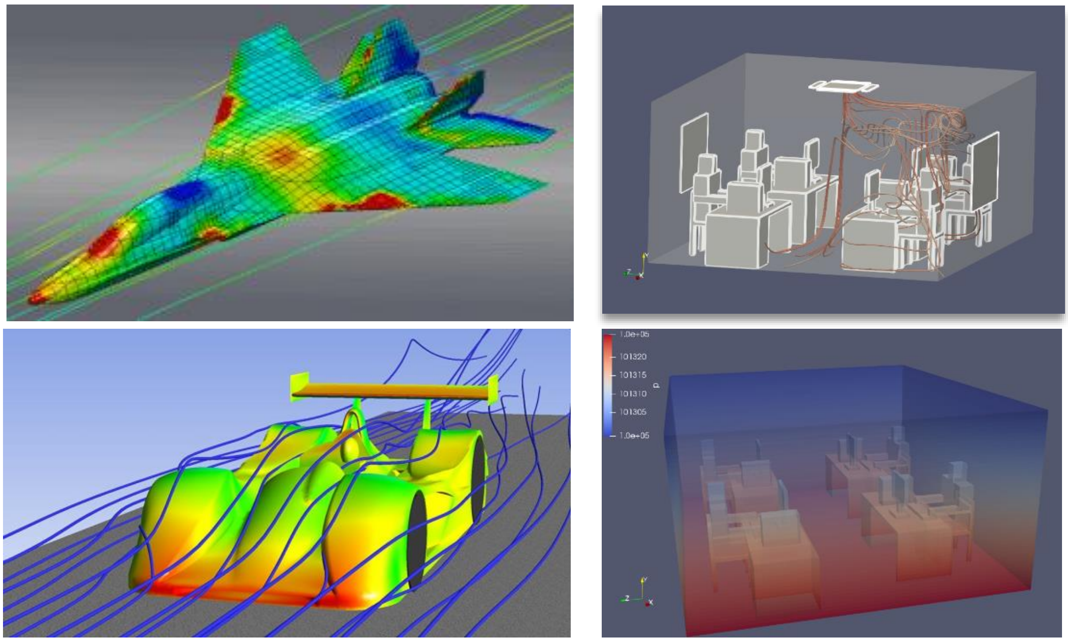

Figure 2. Types of Results Obtained in CFD

Videos such as those that students will analyze in classes 3 and 4 are also elements of analysis of CFD simulation results.

Bibliography

[1] Versteeg, H.K., Malalasekera, W. (1995). An Introduction to Computation Fluid Dynamics, The Finite Volume Method. 2nd Edition, Pearson Education Limited. Edinburgh Gate, Harlow, England. Available on Dropbox.