LECTURE 5: COMPUTATIONAL FLUID DYNAMICS (CFD) AND CFD SOFTWARES

5.0 Introduction

In engineering there are several occurrences that obey and can be explained using the fundamental laws of mechanics. Using these laws in the case of fluids, governing equations for the fluid's behavior are obtained.

Some of the equations are:-

The conservation of mass equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = 0$$

The conservation of momentum

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho\left(\vec{V} \cdot \nabla\right)\vec{V} = -\nabla p + \rho \vec{g} + \nabla \cdot \tau_{ij}$$

These equations with the conservation of energy equation form a set of non-linear partial differential equations which cannot be solved analytically for most real life engineering problems.

Approximate solutions can be obtained through the use numerical methods solutions of the governing equation of engineering problems. These solutions are computer-based solutions and this is the basis of Computational Fluid Dynamics (CFD).

CFD is widely adopted by researchers in the industry because it is less expensive when compared to physical testing. The downside being the possibility of errors in solutions obtained are high with complex flow simulations hence, engineering expertise is need to obtain acceptable solutions.

5.1 CFD Strategy

CFD strategy is the replacement of continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain.

5.2 Introduction to CAE and CFD Open-Source Softwares

Introduction

Linux distribution has a software package which is open sources and free for Computer Aided Engineering (CAE) activities.

While several software packages for Computer Aided Engineering (CAE) activities such as Amira, ANSYS, ANSYS FLUENT, SolidWorks, MSC Nastran, etc. the common thing amongst them is that they are expensive which makes them almost impossible to have and use by lowly income earners because of affordability. There also exist some free and good opensource software packages that can compete to a large extent with the earlier listed commercial software. These include:- OpenFOAM for Computational Fluid Dynamics analysis, SOLOME is used for 3D modeling, meshing and pre/post-processing, Elmer is used for multiphysical simulation, etc.

Code_Saturne is one of the free**,** open-source software for computational fluid dynamics (CFD) applications and developed and released by EDF to solve. Code_Saturne for Computational Fluid Dynamics analysis and it uses finite-volume approach for analysis.

This software is relevant in solving laminar or turbulent, Navier-Stokes equations for 2D, 2D-axisymmetric and 3D flows, steady or unsteady, incompressible or weakly dilatable, isothermal or not, with scalars transport if required.

Several turbulence models are available, from Reynolds-Averaged models to Large-Eddy Simulation models. In addition, a number of specific physical models are also available as modules: gas, coal and heavy-fuel oil combustion, semi-transparent radiative transfer, particle-tracking with Lagrangian modeling, Joule effect, electrics arcs, weakly compressible flows, atmospheric flows, rotor/stator interaction for hydraulic machines.

OpenFOAM is divided into a set of precompiled libraries that are dynamically linked during compilation of the solvers and utilities. Libraries such as those for physical models are supplied as source code so that users may conveniently add their own models to the libraries.