Benchmarking Ansys CFX and Fluent: A Study of Solver Parameters and Turbulence Models

General Information:

Laboratory: Laboratory for Applied Mechanical Design (LAMD)
Supervisor: Anupam Jena, Issam Soukhmane, Selahittin Atilay Zervent, Prof. Jürg Schiffmann
Location: Microcity, Neuchâtel (EPFL will provide Travel Cost)
Contacts: anupam.jena@epfl.ch, issam.soukhmane@epfl.ch, atilay.zervent@epfl.ch, jurg.schiffmann@epfl.ch

Description:

This project involves a comparative study between two widely used CFD solvers, Ansys CFX and Fluent. While both solvers are capable of solving complex fluid flow problems, they have distinct methodologies and solver settings. The goal of this project is to understand these differences and achieve comparable results in terms of convergence and accuracy.

Tasks:

- Compare the convergence behavior of Ansys CFX and Fluent for given CFD problems
- Identify key solver parameters in Fluent that replicate the convergence quality achieved in CFX
- Evaluate and benchmark the performance of different turbulence models in both solvers
- Gain hands-on experience in understanding the nuances of CFD solvers

Project Phases:

Understanding the Problem Setup in CFX

- Gain familiarity with the CFX setup and its default solver parameters
- Review the provided CFX model and CFD simulation results
- Understand boundary conditions, turbulence models, and solver settings used
- Analyze the convergence criteria achieved in the CFX simulation

Setting Up the Fluent Model

- Replicate the CFX simulation in Fluent
- Import geometry and mesh from the CFX model into Fluent or recreate the setup in Fluent
- Configure boundary conditions and physical models to match those in the CFX simulations
- Use Fluent's default solver settings initially to establish a baseline

Parameter Tuning for Fluent

- Identify and adjust Fluent's solver parameters to achieve convergence comparable to CFX
- Investigate Fluent's solver parameters such as Pressure-velocity coupling schemes (e.g., SIMPLE, PISO, Coupled); Discretization schemes for momentum, turbulence, and other equations; Under-relaxation factors; Time step or pseudo time step control (if transient)
- Perform parametric studies to observe the effect of these settings on convergence behavior

Turbulence Model Benchmarking

- Compare the performance of various turbulence models in CFX and Fluent
- Run simulations using different turbulence models (e.g., k-ε, k-ω, SST, LES) in both solvers
- Evaluate the results in terms of convergence, accuracy, and computational cost
- Identify any solver-specific sensitivities to turbulence models

Deliverables:

- **Simulation Setup Files:** CFX and Fluent simulation files for all cases studied
- **Parameter Study Results:** A detailed summary of Fluent solver parameter tuning efforts
- **Turbulence Model Comparison:** Tabulated and visual results showing performance metrics
- **Final Report:** A comprehensive report including methodology, results, analysis, and recommendations
- **Presentation:** Two concise presentations to the lab summarizing the project outcomes

Tools:

- **Software**: Ansys CFX, Ansys Fluent, and Ansys CFD-Post
- **Hardware**: SCITAS High-performance computing resources and/or LAMD workstations