

Computational Fluid Dynamics (CFD) with ANSYS/Fluent

Computational Fluid Dynamics (CFD) with ANSYS/Fluent is a simulation technique used to analyze fluid flow, heat transfer, and related phenomena in engineering applications. It provides a virtual environment to test and optimize designs before physical prototyping.

Key Features:

- **Fluid Flow Simulation:** Models laminar, turbulent, and multiphase flows.
- **Heat Transfer Analysis:** Studies conduction, convection, and radiation.
- **Coupled Physics:** Integration with structural and thermal analysis.
- **Visualization:** Detailed graphical outputs of velocity, pressure, and temperature.

Benefits:

- Reduced costs by minimizing physical prototypes.
- Improved accuracy in predicting product performance.
- Faster design cycles with virtual testing.
- Enhanced innovation with design optimization.

Applications:

- Aerospace for aerodynamics and engine performance.
- Automotive for drag reduction and cooling systems.
- HVAC for airflow and thermal comfort studies.
- Energy for turbine and combustion analysis.

CFD with ANSYS/Fluent is a powerful engineering tool that enhances product development through accurate simulation and analysis.