

Fluent

PENG

Created: 20260917, Version: 20260917

Contents

1	Introduction	1
2	Basic Steps for CFD Analysis	1
3	Executing Ansys Fluent	1

1 Introduction

Fluent User's Guide

Definition 1.1 (Ansys Fluent). *Ansys Fluent a computer program written in C for modelling fluid flow, heat transfer, and chemical reactions in complex geometries.*

2 Basic Steps for CFD Analysis

1. define the modeling goals: simplifying assumptions, boundary conditions
2. create the model geometry and mesh:
 - unstructured meshes, default units: meters
 - mesh resolution sufficient in each region of the domain
 - mesh check: cell skewness < 0.98
3. set up the solver and physical models
 - boundary conditions at all boundary zones
 - initial solution
 - converged solution is not necessarily a correct one
 - use **second-order upwind discretization** for better accuracy
4. compute and monitor the solution
 - number of iterations to reach a converged solution
5. examine and save the results
 - flow features: flow pattern, separation, .etc
 - quantitative results: forces and moments
6. revisions to the numerical or physical model parameters

3 Executing Ansys Fluent