

Introduction to Computational Fluid Dynamics (CFD) – Part II

Fluent simulation Demonstration

Prepared by:

Dr Milinkumar Shah and Dr Ranjeet Utikar

Senior Lecturer, Chemical Engineering Discipline, Curtin University,
Western Australia

ELECTRONIC WARNING NOTICE FOR COPYRIGHT STATUTORY LICENCES

WARNING

This material has been reproduced and communicated to you by or on behalf of **Curtin University** in accordance with section 113P of the *Copyright Act 1968* (**the Act**)

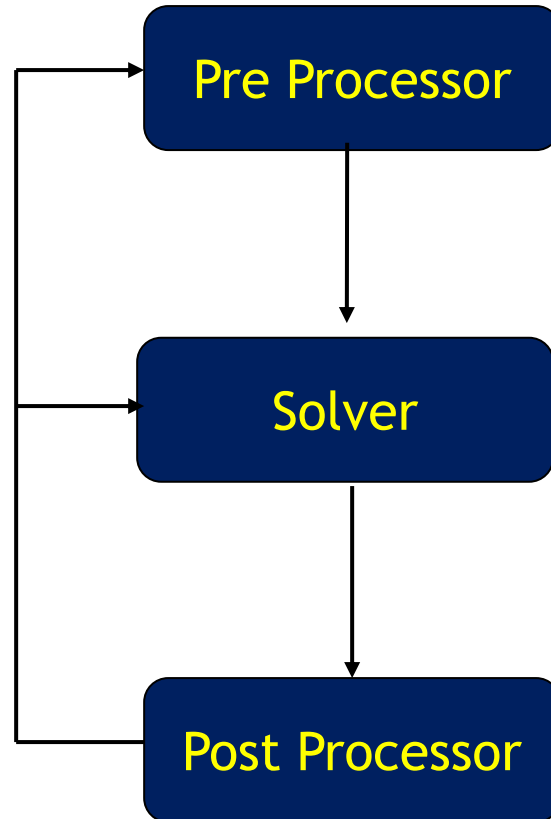
The material in this communication may be subject to copyright under the Act. Any further reproduction or communication of this material by you may be the subject of copyright protection under the Act.

Do not remove this notice.

Outline

- Fluent simulations basics
- Fluent solver basics
- Simulations settings and demonstration

Fluent Simulation Basics



Pre-processor

Geometry generation

Geometry cleanup

Meshing

Solver

Problem specification

Additional models

Numerical computation

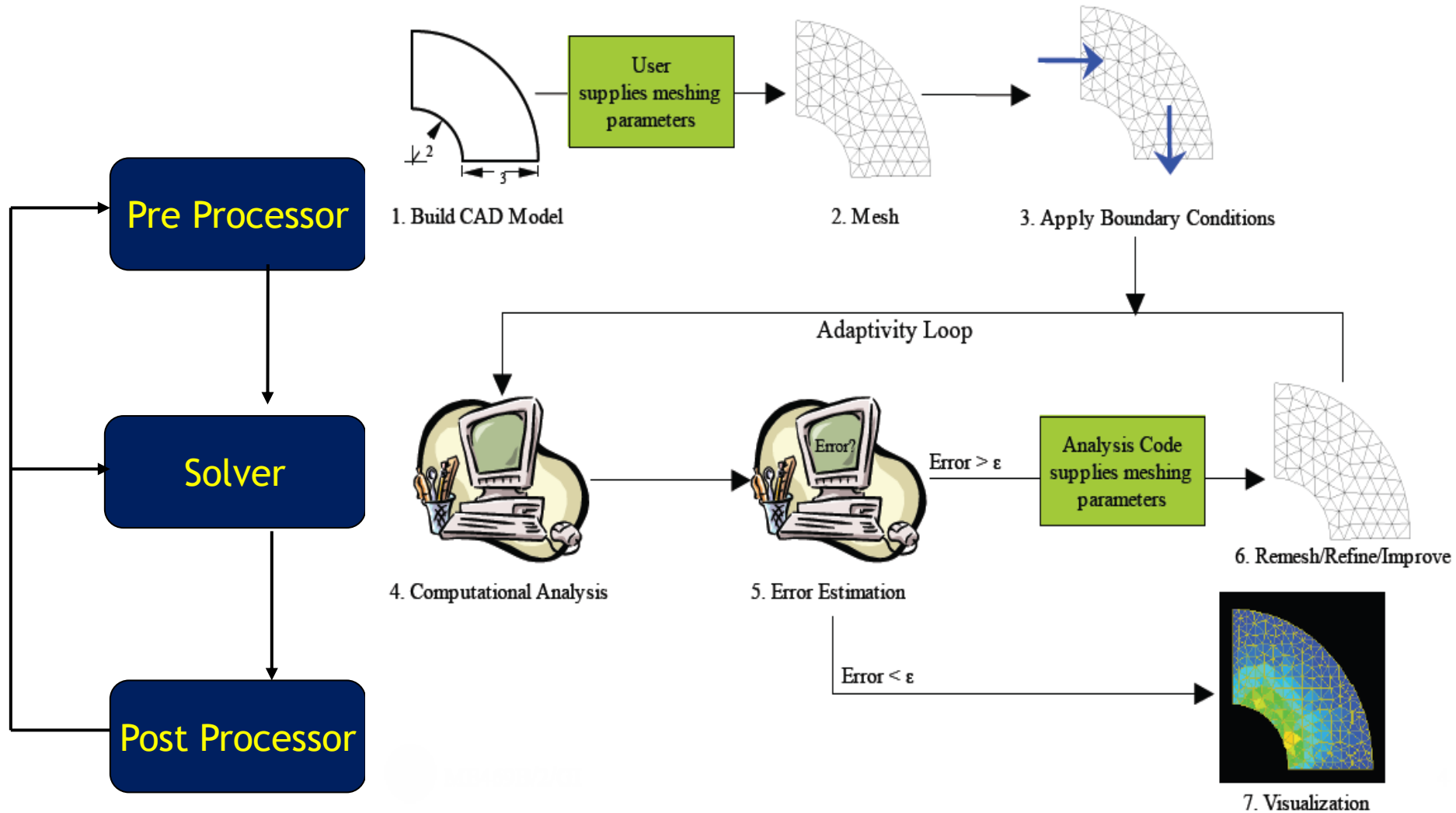
Post-processor

Line and Contour data

Average Values

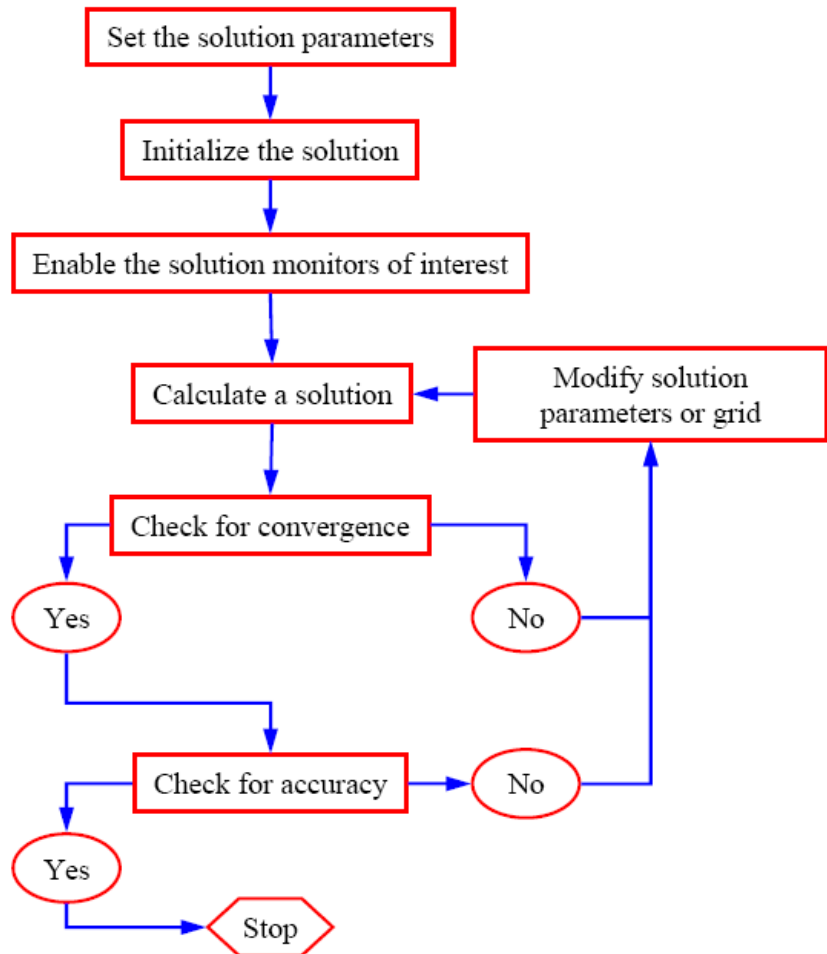
Report Generation

Fluent Simulation Basics



Fluent Solver Basics

- Solution parameters
 - Choosing the solver
 - Discretization schemes
- Initialization
- Convergence
 - Monitoring convergence
 - Stability
 - Setting Under-relaxation
 - Setting Courant number
 - Accelerating convergence
- Accuracy
 - Grid Independence
 - Adaption

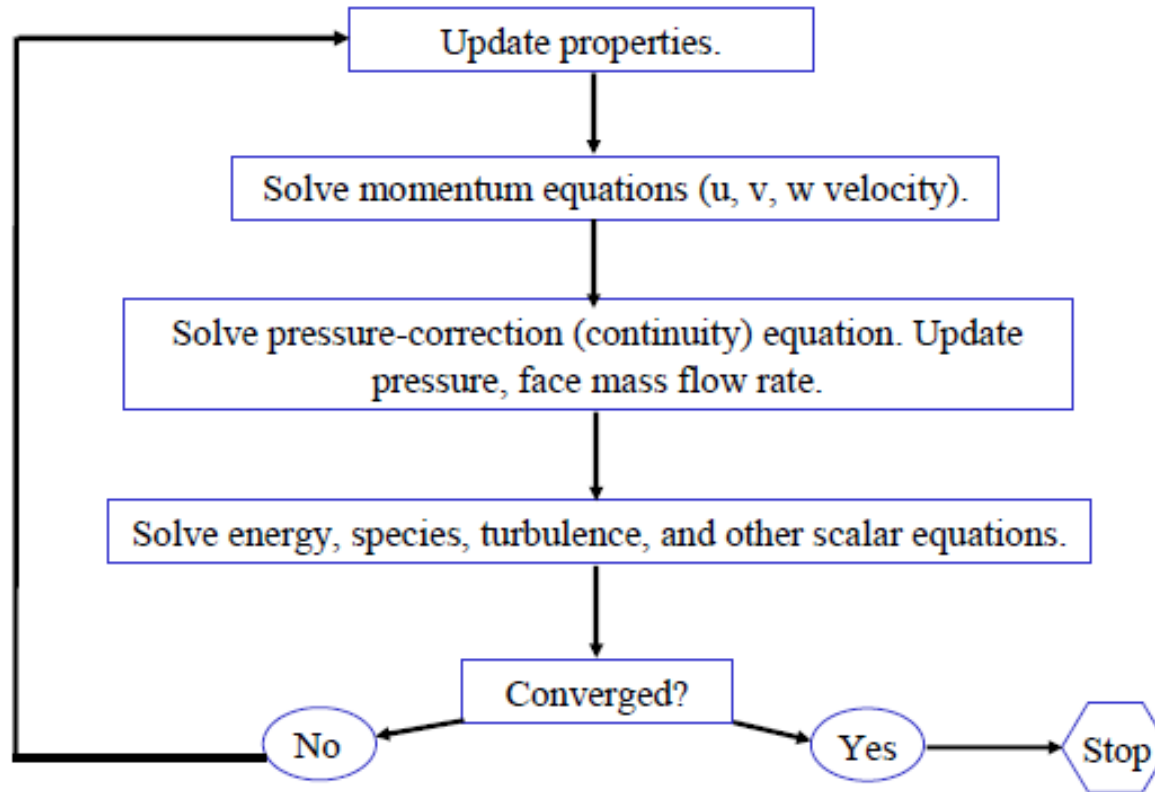


Fluent Solver Basics

- The finite volume solution method can either use a “**segregated**” or a “**coupled**” solution procedure.
 - With segregated methods an equation for a certain variable is solved for all cells, then the equation for the next variable is solved for all cells, etc.
 - With coupled methods, for a given cell equations for all variables are solved, and that process is then repeated for all cells.
- **The segregated solution method is the default method** in most commercial finite volume codes. It is best suited for incompressible flows or compressible flows at low Mach number. (**Pressure-based solver**)
- Coupled solver is suitable for compressible flows at high Mach number. (**Density-based solver**)

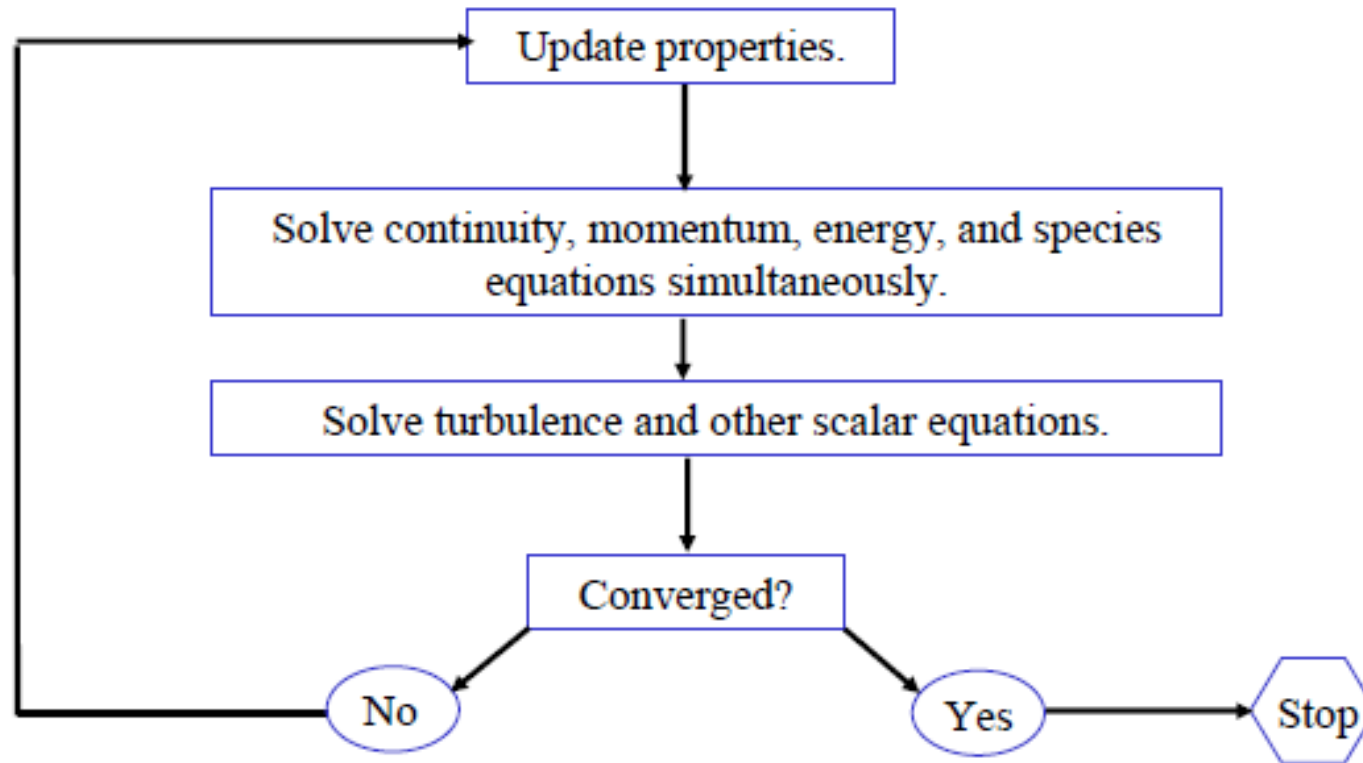
Fluent Solver Basics

Segregated solution procedure



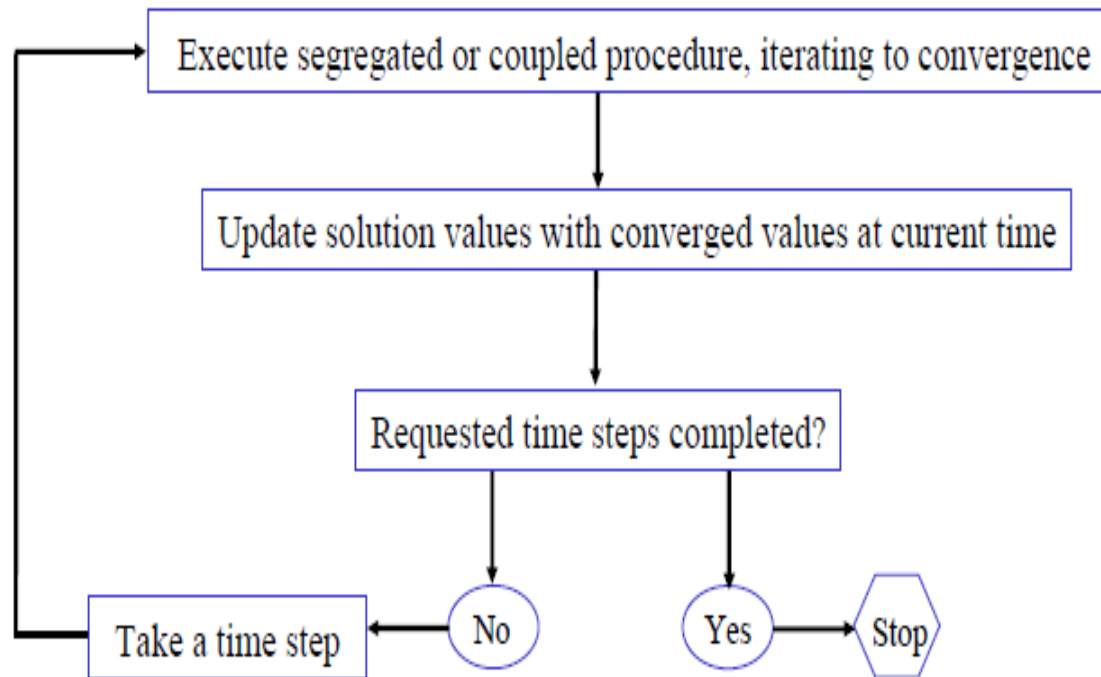
Fluent Solver Basics

Coupled solution procedure



Fluent Solver Basics

Unsteady solution procedure



Fluent Simulation - Demonstration

- Lab -1: Make geometry of a simple pipe using ANSYS-DesignModeller
- Lab-2: Using ANSYS meshing
- Lab 3: Simple flow in pipe
- Lab 4: Flow in a mixing elbow with heat transfer

Lab files are available in a separate folder