Lecture

Fluid flow calculation using ANSYS Fluent
Goal

Lecture Theme:

Fluent requires inputs (solver settings) which determine how the solution is initialized and calculated. While default settings can be used for many cases, understanding the role of the most important settings will help to ensure optimal solution convergence. Emphasis will be placed on convergence, which is critical for the CFD simulation.

Learning Aims:

You will learn:

• How to specify the solver and set the discretization schemes
• How to initialize the solution
• How to monitor and judge solution convergence

Learning Objectives:

You will be able to choose appropriate solver settings for your CFD simulation and be able to monitor and judge solution convergence
Navigation Pane Guides Basic Workflow

- Read and check mesh (scale if needed in standalone mode)
- Select Physical Models
  - Energy
  - Turbulence
  - Multiphase
  - ............
- Create/Assign Materials
- Assign Cell & Boundary Conditions
  - Choose Solver Settings
  - Create Solution Monitors
  - Initialize Solution
  - Run Calculation
- Post-Process Results
W3: Solver

Materials

- Fluent materials database
  - Provides access to a number of pre-defined fluid, solid and mixture materials
  - Properties listed depend on the models used
  - Materials can be copied to the case file and edited if required

- User-Defined material database
  - Custom databases can be created, accessed and modified from the standard materials panel in Fluent
Fluid density

- **For incompressible flow with \( \rho = \text{constant} \)**
  - Select constant for density

- **Ideal gas properties**
  - **Incompressible flow, \( \rho = f(T) \)**
    - Polynomial or piecewise-polynomial
    - Incompressible ideal gas law \( \rho = \frac{p_{\text{operating}}}{RT} \)
      - Set \( p_{\text{operating}} \) close to the mean pressure in the problem ➔ **see next slide**

  - **Compressible flow, \( \rho = f(p,T) \)**
    - Use ideal-gas for density \( \rho = \frac{p_{\text{absolute}}}{RT} \)
      - For low-Mach-number flows, set \( p_{\text{operating}} \) close to mean pressure of the problem to avoid round-off errors
      - Use Floating Operating Pressure for unsteady flows with large, gradual changes in absolute pressure (segregated solver only)
Operating Pressure

- Represents the absolute pressure datum from which all relative pressures are measured

\[ P_{\text{absolute}} = P_{\text{operating}} + P_{\text{relative}} \]

- Pressures specified at boundary conditions and initial conditions are relative to the Operating Pressure

- Used to avoid problems with round-off errors which occur when the dynamic pressure differences in a fluid are small compared to the absolute pressure level

\[ P_{\text{rel,max}} = 100,001 \text{ Pa} \]
\[ P_{\text{rel,min}} = 99,999 \text{ Pa} \]

Ex. 1: \( P_{\text{operating}} = 0 \text{ Pa} \)

\[ P_{\text{rel,max}} = 1 \text{ Pa} \]
\[ P_{\text{rel,min}} = -1 \text{ Pa} \]

Ex. 2: \( P_{\text{operating}} = 100,000 \text{ Pa} \)
W3: Solver

Cell zones / boundary zones

- The mesh consists of a large number of finite volumes, or cells
- The cells are grouped into one or more cell zones
  - For instance in a conjugate heat transfer calculation there may be one cell zone for the fluid region and a second cell zone for the solid material
- Each cell is bounded by a number of faces
- These faces are grouped into a number of face zones
- Some of these faces are located on the boundaries of the model
- The zones to which such faces belong are called boundary zones

Cell zone conditions are applied to all cell zones

Boundary conditions are applied to all boundary zones
W3: Solver

Cell Zone - panel

• Expand the Cell Zone Conditions branch of the Tree

• A list of all cell zones will appear

• Right click on the zone of interest and select Edit (or double click on the name of the zone)
  • The cell zone conditions panel can be used to define a porous zone, prescribe energy sources in solid and fluid zones, specify inputs for rotating machinery, fix the values of one or more solution variables and many other operations
Boundary conditions

- To define a problem that results in a unique solution, you must specify information on the dependent (flow) variables at the domain boundaries
  - Specify fluxes of mass, momentum, energy, etc. into the domain

- Poorly defined boundary conditions can have a significant impact on your solution

- Defining boundary conditions involves:
  - Identifying types (e.g. inlets, walls, symmetry,...)
  - Identifying location
  - Supplying required data depending on boundary type, location and physical models

- Choice depends on:
  - Geometry
  - Availability of data at the boundary location
  - Numerical considerations
Boundary conditions - Type

Zones and zone types are initially defined in the preprocessing phase.

To change the boundary condition type for a zone:
- Right click on the zone name in the Tree
- Select "Type" to open a list of available boundary types and select a new type from the list.
Boundary conditions - Types

External Boundaries
- General
  - Pressure Inlet
  - Pressure Outlet
- Incompressible
  - Velocity Inlet
  - Outflow (not recommended)
- Compressible
  - Mass Flow Inlet
  - Pressure Far Field
- Other
  - Wall
  - Symmetry
  - Axis
  - Periodic
- Special
  - Inlet / Outlet Vent
  - Intake / Exhaust Fan

Internal Boundaries
- Fan
- Interior
- Porous Jump
- Radiator
- Wall
Symmetry boundary conditions can be applied at symmetry planes:

- No inputs are required
- Both the geometry and the flow field must be symmetric:
  - Zero normal velocity at symmetry plane
  - Zero normal gradients of all variables at symmetry plane
  - Must take care to correctly define symmetry boundary locations
Specifying Well Posed Boundary Conditions

Boundaries near recirculation zones

- **Ideal Location:** Apply an outlet downstream of the recirculation zone to allow the flow to develop. This will make it easier to specify accurate boundary conditions.

- **Bad Location:** Difficult to apply the correct backflow conditions for turbulence, temperature, species, etc. if the pressure outlet is located here.
Specifying Well Posed Boundary Conditions

External Flow

- In general, if the object (building, wind turbine, automobile,...) has height $H$ and width $W$, you would want your domain to be at least more than 5H high, 10W wide, with at least 2H upstream of the building and 10H downstream of the building.
- You would want to verify that there are no significant pressure gradients normal to any of the boundaries of the computational domain. If there are, then it would be wise to enlarge the size of your domain.

Concentrate mesh in regions of high gradients.
Locations and types of boundary condition are extremely important for good convergence and accurate results.
Solution Procedure Overview

The sketch to the right shows the basic workflow for any simulation.

This lecture will cover most items in the chart:

- **Solution parameters**
  - Choosing the solver
  - Discretization schemes
- **Initialization**
- **Calculate the solution and monitor convergence**
  - Monitoring convergence
  - Stability
    - Setting Under-relaxation
    - Setting Courant number
    - Setting Pseudo-timestep
  - Accelerating convergence
- **Accuracy**
  - (Discussed in Lecture 09, "Best Practices for CFD")
• Fluent has two types of solver, pressure-based and density-based
• Pressure-Based is the default and should be used for most problems
  • Handles the range of Mach numbers from 0 to \( \sim 2-3 \)
• Density-Based is normally only used for higher Mach numbers, or for specialized cases such as capturing interacting shock waves
Available Solvers

Pressure-Based

- Segregated
  - Solve U-Momentum
  - Solve V-Momentum
  - Solve W-Momentum
  - Solve Continuity; Update Velocity

- Coupled
  - Solve Mass & Momentum
  - Solve Energy
  - Solve Species
  - Solve Turbulence Equation(s)
  - Solve Other Transport Equations as required

Density-Based

- Coupled Implicit
  - Solve Mass, Momentum, Energy, Species

- Coupled-Explicit
  - Solve Mass, Momentum, Energy, Species
Pressure-based Solver (PBS)

• The pressure-based solvers
  
  – Velocity field is obtained from the momentum equation
  
  – Mass conservation (continuity) is achieved by solving a pressure correction equation
    • Pressure-velocity coupling algorithms are derived by reformatting the continuity equation
    • The pressure equation is derived in such a way that the velocity field, corrected by the pressure, satisfies continuity
  
  – Energy equation (where appropriate) is solved sequentially
  
  – Additional scalar equations are also solved in a segregated (sequential) fashion
Density-based Solver (DBS)

- The governing equations of continuity, momentum, and (where appropriate) energy and species transport are solved simultaneously (i.e., coupled together)
- Additional scalar equations are solved in a segregated fashion
- The density-based solver can be run implicit or explicit
Pressure-Based Solver Inputs

Pressure-Velocity Coupling needed by Pressure-Based Solver

Default is **SIMPLE**
- Good for majority of routine incompressible flow applications

For compressible flows choose **Coupled**
- Often referred to as pressure-based coupled solver, or PBCS
- Also preferred for incompressible flow cases involving buoyancy or rotation
- Use in place of SIMPLE for any case that has convergence problems

The other selections are only used in specific situations
- PISO is normally only used for transient calculations (Lecture 10)
- SIMPLEC is primarily of academic interest
Implicit under-relaxation factors are used for SIMPLE, SIMPLEC, PISO

- The under-relaxation factor, $\alpha$, is included to stabilize the iterative process for the pressure-based solver
- The final, converged solution is independent of the under-relaxation factor
  - Only the number of iterations required for convergence is dependent

The default settings are suitable for a wide range of problems

- You can reduce the values when necessary
- Appropriate settings are best learned from experience!

\[ \phi_P = \phi_{P,\text{old}} + \alpha \Delta \phi_P \]
Pressure-based solver control

Two methods are available to control the solution when using the pressure-based coupled solver

- Courant number: default = 200
  - Can be reduced to 10-50 for problems that are difficult to converge or for complex physics such as multiphase and combustion
  - In general, lower Courant number values make the solution more stable, while higher values allow the solution to converge faster
    - If the value used is too high, the solution will probably diverge
  - As with under-relaxation factors, optimal values can be somewhat problem dependent and are best learned from experience

- Pseudo-transient (next slide)
Using the Pseudo Transient option with the pressure-based coupled solver can lead to better convergence for meshes with high aspect ratio cells. This option requires inputs for the calculation of the pseudo time step (see below):

- For internal flows, the default settings of Automatic and Length Scale Method = Conservative work well in the majority of cases.
- For external flows, use Automatic with User-Specified length scale equal to a characteristic length of the geometry, e.g. airfoil chord length.
- More details can be found in the Appendix.
W3:Solver

PBCS-Coupled

![Solution Methods]

- **Scheme**: Coupled
- **Gradient**: Least Squares Cell Based
- **Pressure**: Second Order
- **Momentum**: Second Order Upwind
- **Turbulent Kinetic Energy**: Second Order Upwind
- **Specific Dissipation Rate**: Second Order Upwind
- **Frozen Flux Formulation**: Pseudo Transient

![Run Calculation]

- **Length Scale Method**: Conservative
- **Number of Iterations**: 250
- **Reporting Interval**: 1

ANSYS Warsztaty 2016 AGH
• Using the Pseudo Transient option with the pressure-based coupled solver can lead to better convergence for meshes with high aspect ratio cells
• This option requires inputs for the calculation of the pseudo time step (see below)
  • For internal flows, the default settings of Automatic and Length Scale Method = Conservative work well in the majority of cases
  • For external flows, use Automatic with User-Specified length scale equal to a characteristic length of the geometry, e.g. airfoil chord length

  – Approximately 2250 iterations of SIMPLE (default) in 3.5 hours
  – Approximately 120 iterations of coupled 13 minutes
Choosing a Solver – Density Based

The **density-based** solver is applicable when there is a strong coupling, or interdependence, between density, energy, momentum, and/or species.

**Density-based Coupled Implicit**
- The implicit option is generally preferred over explicit since explicit has a very strict limit on time scale size (CFL constraint) as implicit does not have.
- Examples: *High speed compressible flow with combustion, hypersonic flows, shock interactions*

**Density-based Coupled Explicit**
- The explicit approach is used for cases where the characteristic time scale of the flow is on the same order as the acoustic time scale.
- Example: *propagation of high-Mach shock waves, shock tube problem*
DBS Iterative Procedure – Courant Number

A pseudo-transient term is included in the density-based solver even for steady state problems.

- The Courant number (CFL) defines the time scale size.
- The pseudo-transient option is available for DBS as well as PBS.

For density-based explicit solver:
- Stability constraints impose a maximum limit on the Courant number (<2).

For density-based implicit solver:
- The Courant number is theoretically not limited by stability constraints.
  - Default value is 5
    - (can be reduced for start up to 0.1-2)
    - Values of 100 – 1000 are common in external aero.
Use of the default settings for spatial discretization is recommended for most cases.

- For natural convection problems, where gravity has been activated, the pressure discretization must be changed to PRESTO! or Body-Force Weighted.
Fluent requires that all solution variables be initialized before starting iterations

- Basically this means that in every individual cell in the mesh a value must be assigned for every solution variable to serve as an initial guess for the solution
- A realistic initial guess improves solution stability and accelerates convergence
- In some cases a poor initial guess may cause the solver to fail during the first few iterations

5 initialization methods are available **

- Hybrid initialization (default)
  - Use this for most cases

- FMG initialization
  - Provides a more realistic initial guess, but the initialization process takes much longer than other methods
  - Can be especially beneficial for compressible flows and rotating machinery

- Standard initialization

- Patch values

- Starting from a previous solution
Initialization

Standard Initialization:
All cells have the same value

Hybrid Initialization:
Slightly more realistic non-uniform initial guess

FMG Initialization:
Much more realistic non-uniform initial guess, however takes longer to generate

Final converged solution

In general, the closer the initial guess is to the final solution, the fewer iterations will be needed to reach convergence.
Run Calculation

- Number of iterations solution will run
- Continues from current solution
- Solution will stop sooner if convergence monitor checks are met

• Time step size and number of time steps solution will run
• Continues from current solution
Convergence

The solver must perform enough iterations to achieve a converged solution.

At convergence, the following should be satisfied:

- All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance (Residual).
  - The residual measures the imbalance of the current numerical solution and is related to but NOT EQUAL to the numerical error.
- Overall mass, momentum, energy, and scalar balances are achieved.
- Target quantities reach constant values:
  - Integral: e.g. Pressure drop
  - Local: e.g. Velocity at specified position
Monitoring convergence using residual history

- Generally, a decrease in residuals by **three orders of magnitude** can be a sign of convergence
- Scaled energy residual should decrease to $10^{-6}$ (for the pressure-based solver)
- Scaled species residual may need to decrease to $10^{-5}$ to achieve species balance

Best practice is to also monitor quantitative variables to decide convergence

- Ensure that overall mass/heat/species conservation is satisfied
- Monitor other relevant key variables/physical quantities for confirmation
  - The use of one or more monitors like this is strongly recommended for all simulations
Convergence Monitors – Residuals

- Residual plots show when the residual values have reached the specified tolerance
  - It is possible to modify or disable the default checking criterion for convergence
- Prevents calculation being stopped prematurely

![Residual plots showing convergence](image)

All equations converged

- Residuals: continuity, x-velocity, y-velocity, z-velocity, energy, k, epsilon

- Example of Residual Monitors settings:
  - Residual error criteria:
    - Continuity: 1e-03
    - x-velocity: 1e-06
    - y-velocity: 1e-06
    - z-velocity: 1e-06
  - Iterations to Plot: 1000
  - Iterations to Store: 1000

- Residual monitors check convergence based on absolute criteria.
Checking Overall Flux Conservation

- The net flux imbalance (shown in the GUI as Net Results) should be less than 1% of the smallest flux through the domain boundary.
In addition to residuals, you can also monitor:

- Lift, drag and moment coefficients
- Relevant variables or functions (e.g. surface integrals) at a boundary or any defined surface

These additional monitored quantities are important convergence indicators

- The use of one or more of this type of solution monitor is strongly recommended for all calculations

These monitors can also be used to determine when iterations stop (details in Appendix)
Convergence - problems

Numerical instabilities can arise with an ill-posed problem, poor-quality mesh and/or inappropriate solver settings

- Exhibited as increasing (diverging) or “stuck” residuals
- Diverging residuals imply increasing imbalance in conservation equations
- Unconverged results are very misleading!

Troubleshooting

- Ensure that the problem is well-posed
- Compute an initial solution using a first-order discretization scheme
- For the pressure-based solver, decrease underrelaxation factors for equations having convergence problems
- For the density-based solver, reduce the Courant number
- Remesh or refine cells which have large aspect ratio or large skewness.
  - Remember that you cannot improve cell skewness by using mesh adaption!
Convergence speed-up

Convergence can be accelerated by:

- Supplying better initial conditions
  - Starting from a previous solution (using file/interpolation when necessary)
- Gradually increasing under-relaxation factors or Courant number
  - Excessively high values can lead to solution instability and convergence problems
  - You should always save case and data files before continuing iterations
- Starting with a good quality mesh with appropriate mesh resolution
  - The orthogonal quality reported in Mesh > Info > Quality should have a minimum value of 0.1 and an average value that is much higher
A converged solution is not necessarily an accurate solution

Accuracy depends on:
- Order of the discretization schemes ($2^{nd}$ order schemes are recommended)
- Mesh resolution
- Boundary Conditions
- Model limitations
- Geometry simplifications
- Precision of the solver ($2d/3d$ or $2ddp/3ddp$)
- ...
Questions ?