Lecture 2

Introduction to CFD Methodology

Introduction to ANSYS FLUENT
What is CFD?

• **Computational Fluid Dynamics (CFD)** is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations
  – Conservation of mass
  – Conservation of momentum
  – Conservation of energy
  – Conservation of species
  – Effects of body forces

• **The results of CFD analyses are relevant in:**
  – Conceptual studies of new designs
  – Detailed product development
  – Troubleshooting
  – Redesign

• **CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.**
How Does CFD Work?

- **ANSYS CFD solvers are based on the finite volume method**
  - Domain is discretised into a finite set of control volumes
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes

\[
\frac{\partial}{\partial t} \int \rho \phi \, dV + \int \rho \phi \mathbf{V} \cdot d\mathbf{A} = \int \Gamma \phi \nabla \phi \cdot d\mathbf{A} + \int S_{\phi} \, dV
\]

  - Partial differential equations are discretised into a system of algebraic equations
  - All algebraic equations are then solved numerically to render the solution field

* FLUENT control volumes are cell-centered (i.e. they correspond directly with the mesh) while CFX control volumes are node-centered
Introduction to the CFD Methodology

CFD Modeling Overview

Problem Identification
1. Define goals
2. Identify domain

Pre-Processing
3. Geometry
4. Mesh
5. Physics
6. Solver Settings

Solve
7. Compute solution

Post Processing
8. Examine results

9. Update Model
Introduction to the CFD Methodology

1. Define Your Modeling Goals

   Problem Identification
   1. Define goals
   2. Identify domain

- What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?
  - What are your modeling options?
    - What physical models will need to be included in your analysis
    - What simplifying assumptions do you have to make?
    - What simplifying assumptions can you make (i.e. symmetry, periodicity)?

- What degree of accuracy is required?

- How quickly do you need the results?

- Is CFD an appropriate tool?
2. Identify the Domain You Will Model

Problem Identification
1. Define goals
2. Identify domain

• How will you isolate a piece of the complete physical system?

• Where will the computational domain begin and end?
  – Do you have boundary condition information at these boundaries?
  – Can the boundary condition types accommodate that information?
  – Can you extend the domain to a point where reasonable data exists?

• Can it be simplified or approximated as a 2D or axisymmetric problem?

Domain of Interest as Part of a Larger System (not modeled)

Domain of interest isolated and meshed for CFD simulation.
3. Create a Solid Model of the Domain

- How will you obtain a model of the fluid region?
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?

- Can you simplify the geometry?
  - Remove unnecessary features that would complicate meshing (fillets, bolts...)?
  - Make use of symmetry or periodicity?
    - Are both the solution and boundary conditions symmetric / periodic?

- Do you need to split the model so that boundary conditions or domains can be created?
A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes.

- **What degree of mesh resolution is required in each region of the domain?**
  - The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients.
  - Can you predict regions of high gradients?
  - Will you use adaption to add resolution?

- **Do you have sufficient computer resources?**
  - How many cells/nodes are required?
  - How many physical models will be used?
For flow-aligned geometries:

- Quad/hex meshes can provide higher-quality solutions with fewer cells/nodes than a comparable tri/tet mesh.

- Quad/hex meshes show reduced numerical diffusion when the mesh is aligned with the flow.

- It does require more effort to generate a quad/hex mesh.
For complex geometries:

- It would be impractical to generate a structured (flow-aligned) hex mesh.
- You can save meshing effort by using a tri/tet mesh or hybrid mesh
- Quick to generate

• Hybrid meshes typically combine tri/tet elements with other elements in selected regions
  - For example, use wedge/prism elements to resolve boundary layers.
  - More efficient and accurate than tri/tet alone.
Non-conformal meshes:

- Usually when meshing, where two volumes meet, the mesh should exactly match (conformal mesh).
- This will be the case if, in ANSYS DesignModeler all the bodies are combined to form 1 part.

- If you have multiple parts the mesh will not match, and in FLUENT you MUST set up a non-conformal interface to pair the surfaces.

- Typical scenarios for using non-conformals are when meshing very complex geometries and for sliding mesh applications.
• For a given problem, you will need to:
  – Define material properties
    • Fluid
    • Solid
    • Mixture
  – Select appropriate physical models
    • Turbulence, combustion, multiphase, etc.
  – Prescribe operating conditions
  – Prescribe boundary conditions at all boundary zones
  – Provide initial values or a previous solution
  – Set up solver controls
  – Set up convergence monitors

For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time.
• The discretised conservation equations are solved iteratively until convergence.

• Convergence is reached when:
  – Changes in solution variables from one iteration to the next are negligible.
    • Residuals provide a mechanism to help monitor this trend.
  – Overall property conservation is achieved
    • Imbalances measure global conservation
  – Quantities of interest (e.g. drag, pressure drop) have reached steady values.
    • Monitor points track quantities of interest.

• The accuracy of a **converged** solution is dependent upon:
  – Appropriateness and accuracy of physical models.
  – Mesh resolution and independence
  – Numerical errors
Examine the Results

• Examine the results to review solution and extract useful data
  – Visualization Tools can be used to answer such questions as:
    • What is the overall flow pattern?
    • Is there separation?
    • Where do shocks, shear layers, etc. form?
    • Are key flow features being resolved?
  – Numerical Reporting Tools can be used to calculate quantitative results:
    • Forces and Moments
    • Average heat transfer coefficients
    • Surface and Volume integrated quantities
    • Flux Balances

Examine results to ensure property conservation and correct physical behavior. High residuals may be caused by just a few poor quality cells.
Consider Revisions to the Model

- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?

- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?

- **Is the mesh adequate?**
  - Can the mesh be refined to improve results?
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?
Introduction to the CFD Methodology

Demonstration of FLUENT Software

- Start FLUENT (assume the mesh has already been generated).
  - Set up a simple problem.
  - Solve the flow field.
  - Postprocess the results.
Using our Training Machines

• Log in to your workstation
  – Login name: training
  – Password: training

• Directories
  – Workshop mesh/case/data files can be found in
    c:\Users\Training_Materials\originals\course_name
  – We recommend that you save your work into a central working folder:
    c:\Users\Training\your_name

• To start FLUENT and/or Workbench, use the desktop icons.

• It is recommended that you restart FLUENT and/or Workbench for each tutorial to avoid mixing solver settings from different workshops.