Introduction to CFD Analysis

Introductory FLUENT Training

What is CFD?

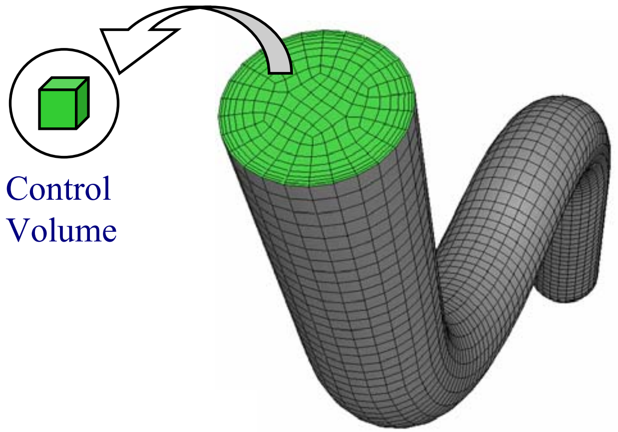
- Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations.
  - Conservation of mass, momentum, energy, species, etc.
- The results of CFD analyses are relevant in:
  - conceptual studies of new designs
  - detailed product development
  - troubleshooting
  - redesign
- CFD analysis complements testing and experimentation.
  - Reduces the total effort required in the experiment design and data acquisition

How Does CFD Work?

- FLUENT solvers are based on the finite volume method.
  - Domain is discretized onto a finite set of control volumes (or cells).
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes.

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\,\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

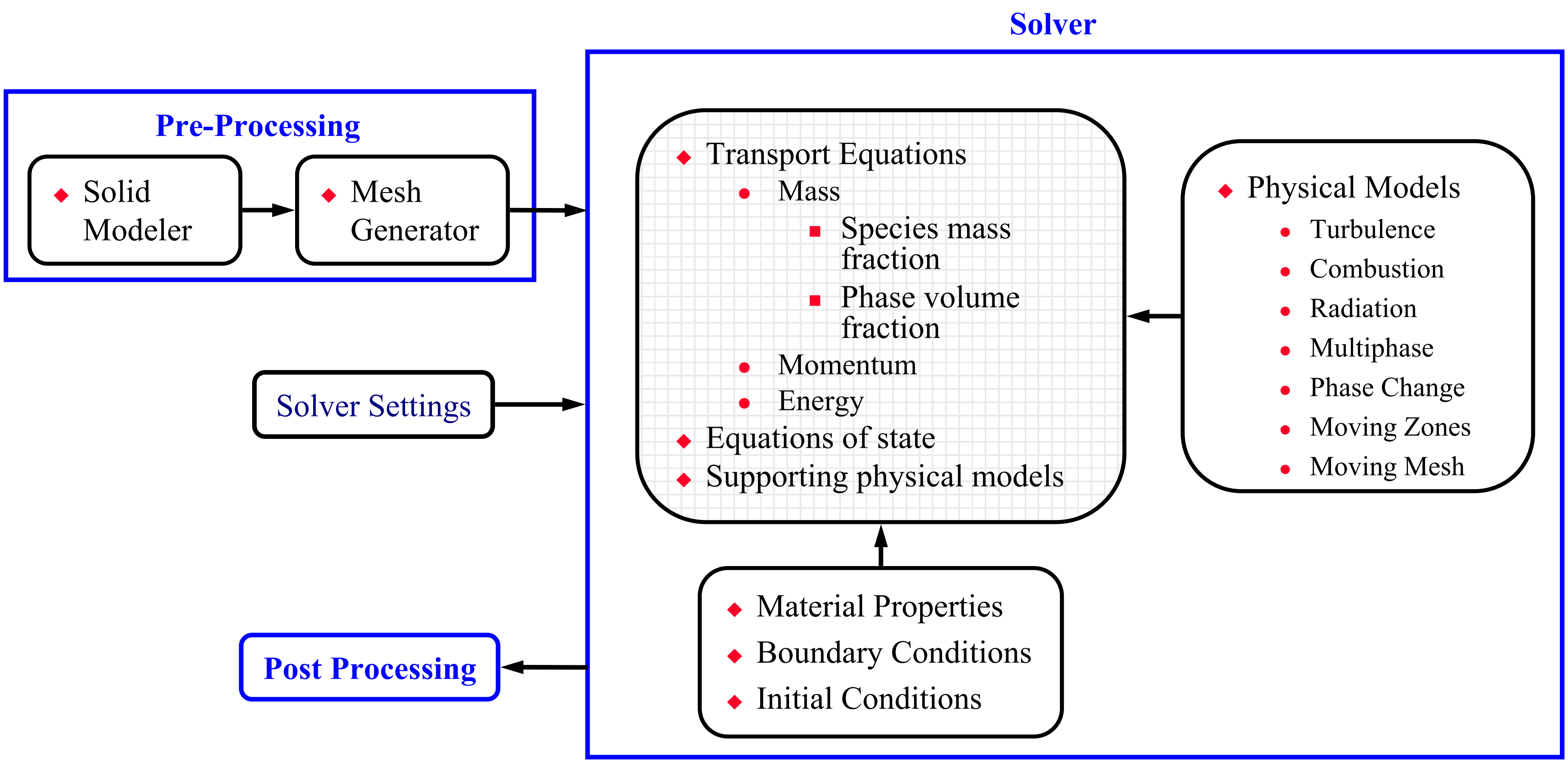
  - Partial differential equations are discretized into a system of algebraic equations.
  - All algebraic equations are then solved numerically to render the solution field.



Fluid region of pipe flow is discretized into a finite set of control volumes (mesh).

Equation	Variable
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

CFD Modeling Overview



CFD Analysis – The Basic Steps

- Problem Identification and Preprocessing
  1. Define your modeling goals.
  2. Identify the domain you will model.
  3. Design and create the grid.

- Solver Execution
  4. Set up the numerical model.
  5. Compute and monitor the solution.

- Post-Processing
  6. Examine the results.
  7. Consider revisions to the model.

Define Your Modeling Goals

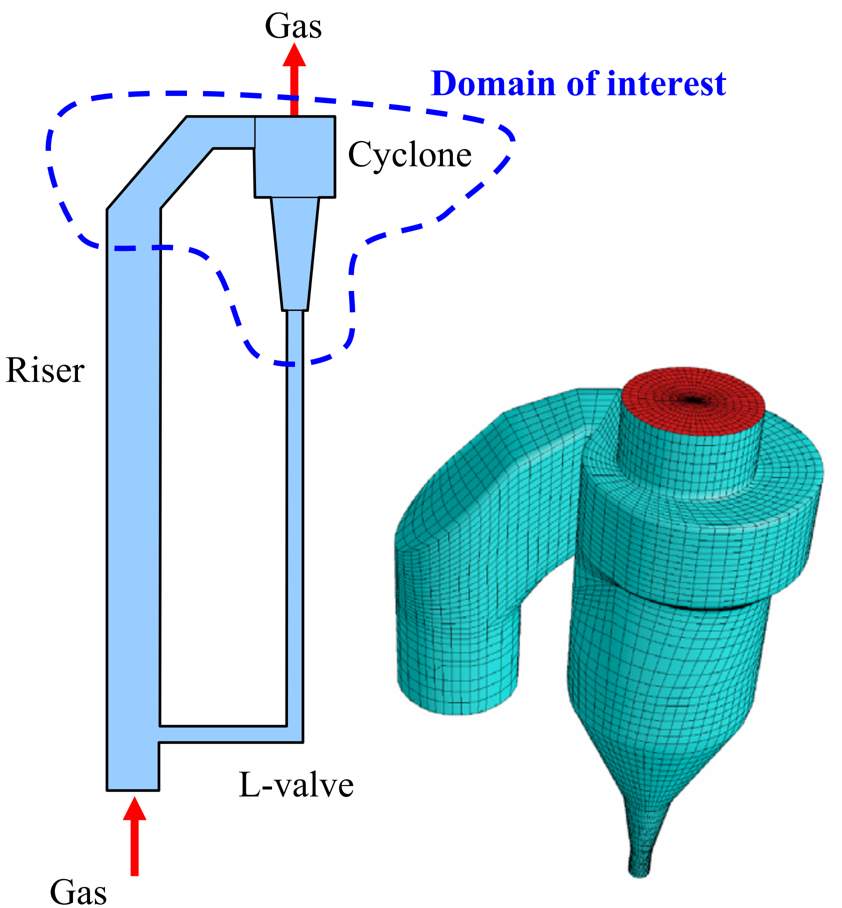
- Problem Identification and Pre-Processing
  1. Define your modeling goals
  2. Identify the domain you will model
  3. Design and create the grid

- What results are you looking for (i.e. pressure drop, mass flowrate), and how will they be used?
  - What are your modeling options?
    - What physical models will need to be included in your analysis (i.e. turbulence, compressibility, radiation)?
    - What simplifying assumptions do you have to make?
    - What simplifying assumptions can you make (i.e. symmetry, periodicity)?
    - Do you require a unique modeling capability?
      - User-defined functions (written in C) in FLUENT 6
- What degree of accuracy is required?
- How quickly do you need the results?

Identify the Domain You Will Model

- Problem Identification and Pre-Processing
  1. Define your modeling goals
  2. Identify the domain you will model
  3. Design and create the grid

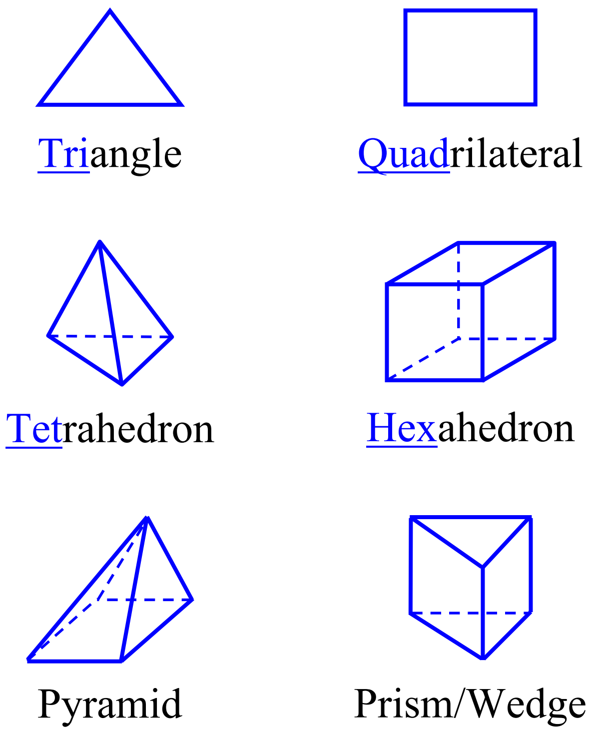
- How will you isolate a piece of the complete physical system?
- Where will the computational domain begin and end?
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?
- Can it be simplified or approximated as a 2D or axisymmetric problem?



Example: Cyclone Separator

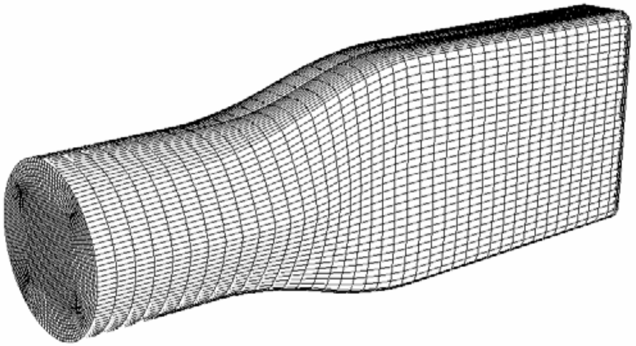
Design and Create the Grid

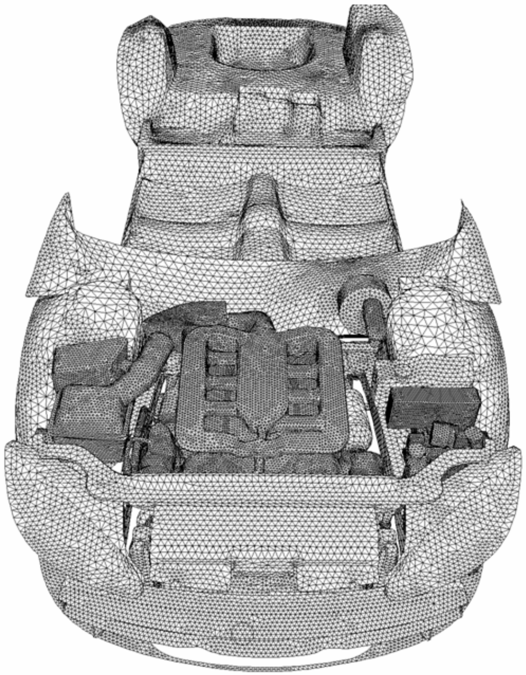
- Problem Identification and Pre-Processing
  1. Define your modeling goals
  2. Identify the domain you will model
  3. Design and create the grid



- Can you benefit from another code, such as MixSim, IcePak, or AirPak?
- Can you use a quad/hex grid or should you use a tri/tet grid or hybrid grid?
  - How complex is the geometry and flow?
  - Will you need a non-conformal interface?
- What degree of grid resolution is required in each region of the domain?
  - Is the resolution sufficient for the geometry?
  - Can you predict regions with high gradients?
  - Will you use adaption to add resolution?
- Do you have sufficient computer memory?
  - How many cells are required?
  - How many models will be used?

Tri/Tet vs. Quad/Hex Meshes

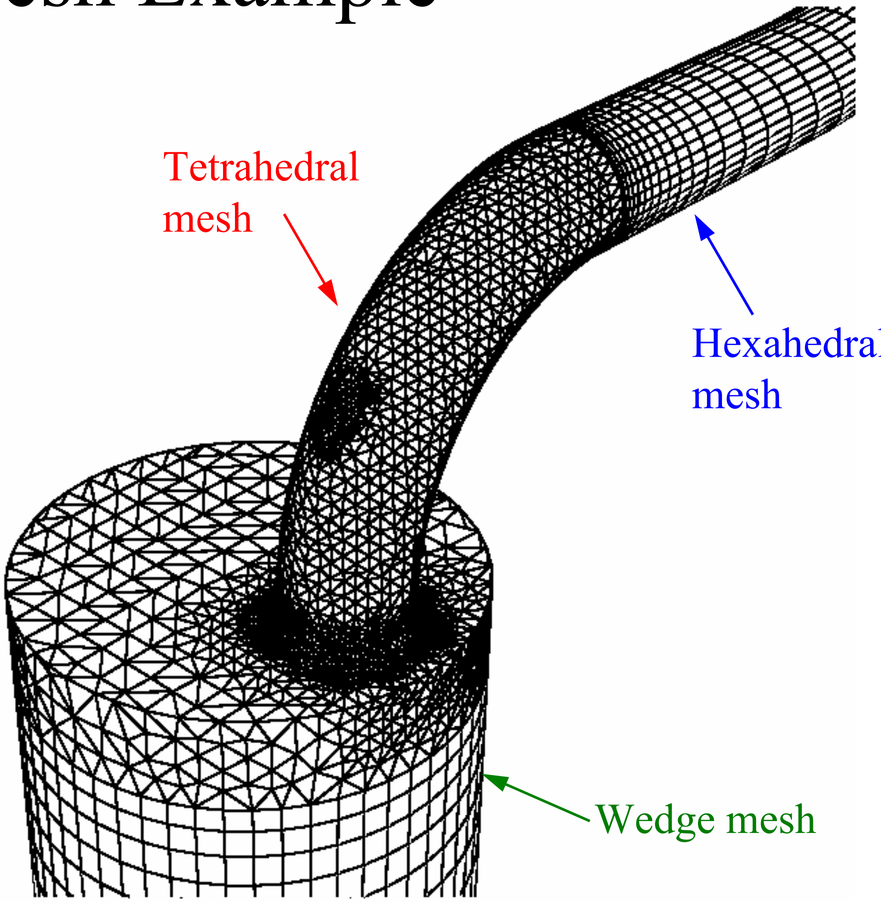




- For **simple** geometries, quad/hex meshes can provide higher-quality solutions <u>with fewer cells</u> than a comparable tri/tet mesh.
  - Shows reduced false diffusion when the grid is aligned with the flow.
- For **complex** geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh.
  - Flow is generally not aligned with the grid.

Hybrid Mesh Example

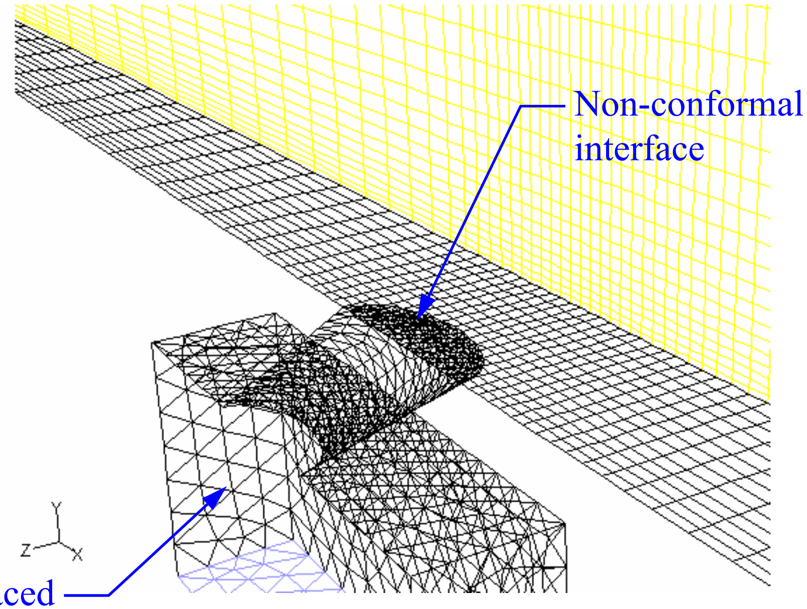
- Valve port grid
  - Specific regions can be meshed with different cell types.
  - Both efficiency and accuracy are enhanced relative to a hexahedral or tetrahedral mesh alone.
  - Tools for hybrid mesh generation are available in GAMBIT and TGrid.



Hybrid mesh for an IC engine valve port

Non-Conformal Mesh Example

- A non-conformal mesh is a mesh in which the grid nodes to not match across an interface.
  - Can be helpful for meshing complex geometries.
  - Required for sliding mesh applications.
- Example:
  - 3D film cooling
    - Coolant is injected into a duct from a plenum.
    - Plenum is meshed with tetrahedral cells.
    - Duct is meshed with hexahedral cells.



Set Up the Numerical Model

- Solver Execution
  4. Set up the numerical model.
  5. Compute and monitor the solution.

Solving initially in 2D will provide valuable experience with the models and solver settings for your problem in a short amount of time.

- For a given problem, you will need to:
  - Select appropriate physical models.
    - Turbulence, combustion, multiphase, etc.
  - Define material properties.
    - Fluid
    - Solid
    - Mixture
  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Provide an initial solution.
  - Set up solver controls.
  - Set up convergence monitors.

Compute the Solution

- Solver Execution
  4. Set up the numerical model.
  5. Compute and monitor the solution.

- The discretized conservation equations are solved iteratively.
  - A number of iterations are usually required to reach a converged solution.
- 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.
- The accuracy of a converged solution is dependent upon:
  - Appropriateness and accuracy of physical models.
  - Grid resolution and independence
  - Problem setup

A converged and grid-independent solution on a well-posed problem will provide useful engineering results!

Examine the Results

- Postprocessing
  6. Examine the results
  7. Consider revisions to the model

Examine results to ensure property conservation and correct physical behavior. High residuals may be attributable to only a few cells of poor quality.

- 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

Consider Revisions to the Model

- Postprocessing
  6. Examine the results
  7. Consider revisions to the model

- Are physical models appropriate?
  - Is flow turbulent?
  - Is flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- Are boundary conditions correct?
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- Is grid adequate?
  - Can grid be adapted to improve results?
  - Does solution change significantly with adaption, or is the solution grid independent?
  - Does boundary resolution need to be improved?

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.

- Online help and documentation is available on each panel by pressing the help button
  - Requires that you have the documentation installed and properly connected to your web browser.

Navigating the PC at Fluent

- Log in to your workstation
  - Login name: **`fluent`**
  - Password: **`fluent`**
- Directories
  - Your FLUENT session will start in **`c:\users`**
  - Tutorial mesh/case/data files can be found in:

    `c:\Student Files\fluent\tut\`
  - We recommend that you save your work into a central working folder:

    `c:\users`
  - Working folder shown on the desktop is a shortcut to **`c:\users`**
- To start FLUENT and/or GAMBIT
  - From the Start menu or desktop, launch the appropriate icon.
  - From a system prompt, enter either **`fluent`** or **`gambit`**
- Your support engineer will save your work at the end of the week.
- **It is recommended that you restart FLUENT for each tutorial to avoid mixing solver settings from different tutorials.**