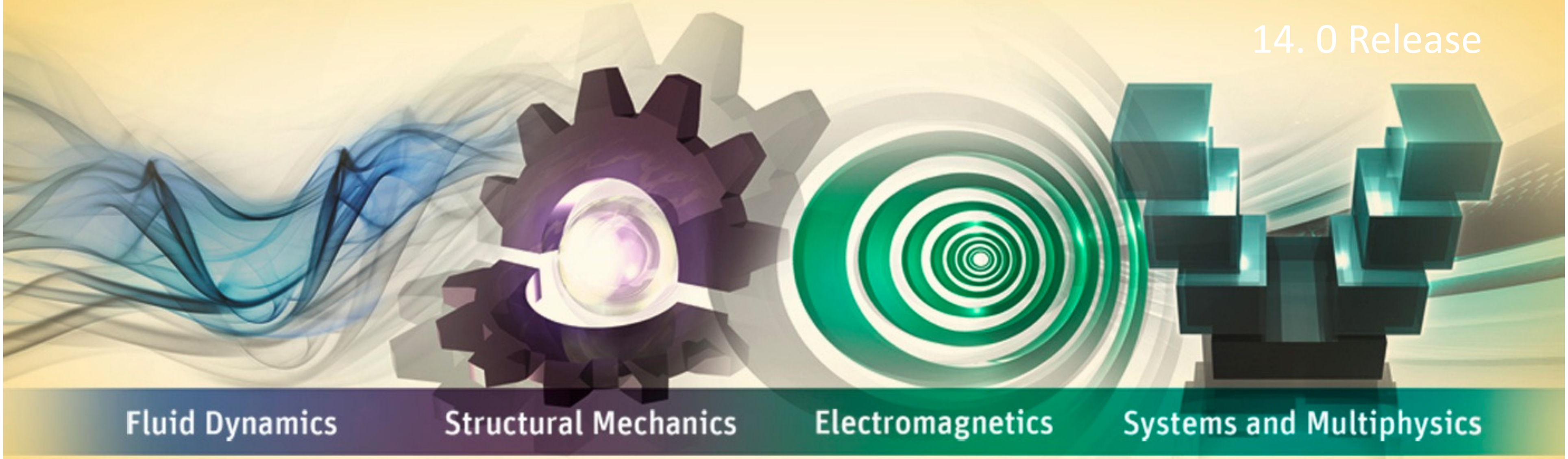
Realize Your Product Promise™

Lecture 5
Day 1 Review and Tips

14. 0 Release



Introduction to ANSYS FLUENT

Introduction

Lecture Theme:

The focus of Day 1 has been to cover topics (defining material properties, defining boundary and cell zone conditions, running the simulation, post-processing the results) which include operations that are common to any CFD analysis in FLUENT.

Many analyses will require additional inputs, such as turbulence models or system rotation, will be the focus of Day 2.

Learning Aims:

You have learned:

Boundary conditions, cell zone conditions, and material properties
Post-processing
Solver settings

Learning Objectives:

After Day 1, you will be able to set up, run and post-process your own CFD simulation.

Before You Start FLUENT

- **Define your modeling goals**
- **Identify the computational domain**
  - Simplify if possible
  - Think about where boundary conditions can be set
  - Avoid placing boundaries in potential recirculation areas when possible
- **Create / Import the Geometry**
  - Consider meshing requirements when creating the geometry
  - Do not include unnecessary detail
- **Create a suitable mesh**
  - Resolve expected gradients in the solution variables
  - Check mesh quality metrics

Cell Zones

- **Cell zones define a region of consistent materials and physical models**

- **Use different cell zones for:**
  - Different reference frames, e.g. rotating, stationary
  - Different domain types – fluid, solid, porous
  - Different materials, e.g. oil, copper and water

- **Fluid cell zones that are connected should use consistent physics**

- **All regions that have the same physics can be grouped into a single cell zone**
  - Regions do not have to be connected
  - Mesh does not have to be continuous

- **The *Reference Pressure* should be set to the operating pressure of the device**

Boundary Conditions

- **It is important to consider the accuracy of the boundary conditions**
  - E.g. a uniform velocity profile is usually not realistic, but can be used if placed a suitable distance upstream

- **Avoid setting boundary conditions in recirculation zones if possible**

- **Use well posed boundary conditions**
  - Mass Flow or Velocity Inlet, Pressure Outlet
    - Will give a uniform inlet velocity profile
      - Use Profiles or UDFs to specify non-uniform or time dependent profiles
  - Pressure Inlet, Pressure Outlet with Target Mass Flow Rate option
    - Total mass flow rate is prescribed but will allow an inlet velocity profile to develop
    - Velocity Inlet with negative velocity or direction vector pointing out of the domain can be used as outlet to achieve the same effect
  - Total Pressure Inlet, Static Pressure Outlet
    - Will allow an inlet velocity profile to develop
    - The mass flow rate through the system will be determined by the pressure difference between inlet and outlet and the losses within the domain

Solver Settings

- **A good initial guess will assist with solver stability during the first few iterations**

- **Underrelaxation factors (or Courant number in coupled solver) are important solver controls**
  - Smaller values = more stable, but slower convergence
  - Larger values = faster convergence, but too large will cause the solver to fail
  - If using the pseudo-transient option, time scale factor plays a similar role

- **When the solver finishes, check that:**
  - Residuals are converged to at least 1e-3
    - Energy should be below 1e-6
  - Energy and mass imbalances in the Flux Reports panel are below 1%
  - Monitor points for quantities of interest have reached steady values
    - This is equally as important as low residuals and flux imbalances

Post-processing

- FLUENT and CFD-Post both contain post-processing tools for flow field visualization and extraction of alphanumeric data such as forces, heat transfer rates, average values, ...
- In CFD-Post
  - Automate post-processing through Session files, State files and Report templates
  - Make use of Expressions and User Variables to extract engineering data
  - Compare solutions using the Multi-file mode and the Case Comparison tools
  - Save images in the 3D CFX Viewer format to provide management or your customers with a better understanding of the flow
- In FLUENT
  - Post-processing can be performed from within the solver, which is often more convenient for most basic post-processing operations

Summary and Conclusions

- **Summary:**
  - Remember to first think about what the aims of the simulation are prior to creating the geometry and mesh
  - All CFD simulations involve some common operations
    - Problem definition
    - Defining boundary conditions and cell zone conditions
    - Defining material properties
    - Post-processing the results
  - Use residual monitors, flux balances and solution monitors to judge convergence
    - Unconverged results can be misleading
- **What Next:**

Day 2 will focus on physical models and best practices