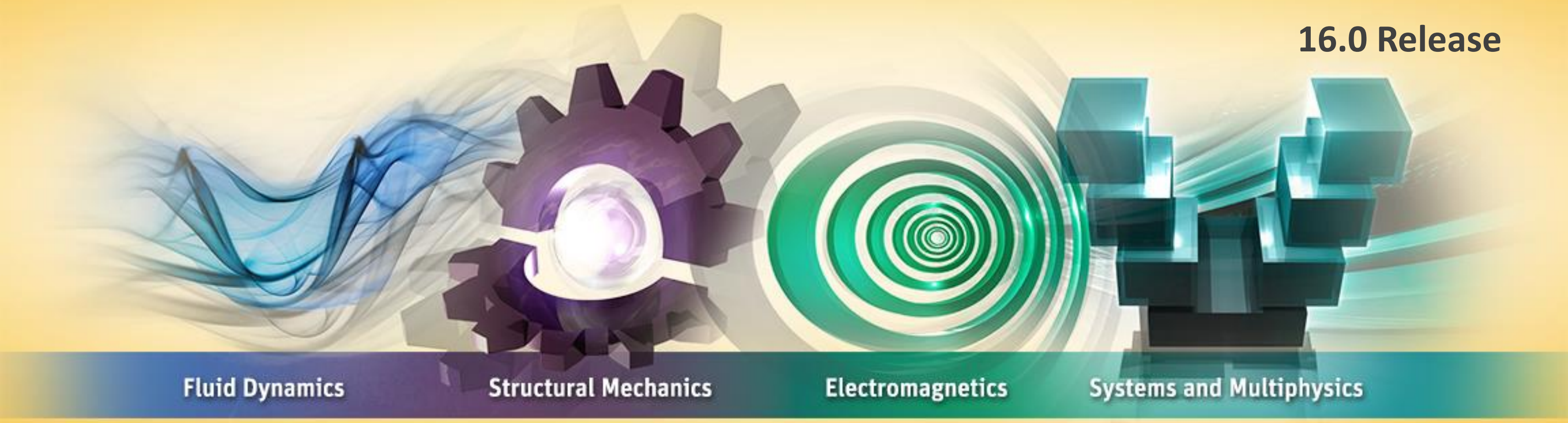
Lecture 7:
Day 1 Review & Tips

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

Introduction

Lecture Theme:

- The focus of Day 1 has been to cover topics (defining material properties, defining boundary conditions and physical models, running the simulation, post-processing the results) which include operations that are common to any CFD analysis in CFX.
- Many analyses will require additional inputs, such as turbulence models or system rotation, which will be the focus of Day 2.

Learning Aims – you have learned:

- Boundary conditions, physical models 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

© 2015 ANSYS, Inc.

ANSYS Confidential

Before you start CFX

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

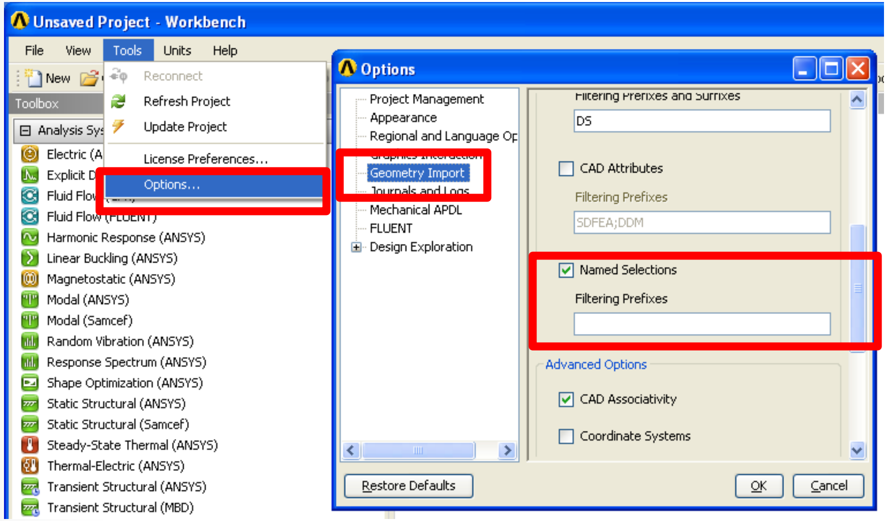
- Consider meshing requirements when creating the geometry
- Do not include unnecessary detail

Create a suitable mesh

- Resolve expected gradients in the solution variables
- Check mesh quality metrics

Working with Workbench

- **Save your Project to set the working directory**
- **Create the workflow by dragging and dropping Analysis and Component Systems on to the Project Schematic**
  - **Systems can share or transfer data by dropping on to an appropriate cell**
- **Configure Tools > Options to suit your needs**



Introduction | **Problem Definition** | Running Simulation | Post-processing | Summary

Domains

Domains define a region of consistent materials and physical models

Use different domains for:

- Different reference frames, e.g. rotating, stationary
- Different domain types – fluid, solid, porous
- Different materials
  - Fluid domains that are connected should use consistent physics – might require multiphase

All regions that have the same physics can be grouped into a single domain

- 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

© 2015 ANSYS, Inc.

ANSYS Confidential

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, Static Pressure Outlet
  - Will give a uniform inlet velocity profile
- Total Pressure Inlet, Mass Flow Outlet
  - Will allow an inlet velocity profile to develop
- Total Pressure Inlet, Static Pressure Outlet
  - Will allow an inlet velocity profile to develop

© 2015 ANSYS, Inc.

ANSYS Confidential

ANSYS

Solver Settings

A good initial guess will assist with Solver stability during the first few iterations

The timestep is an important solver control

- **Smaller Timestep = More Stable, but slower convergence**
- **Larger Timestep = Faster convergence, but too large will cause the solver to fail**

When the solver finishes check:

- **Residuals are converged to less than RMS 1e-4 (or even lower target)**
- **Imbalance are below 1% (or even <0.01%)**
- **Monitor Points for quantities of interest have reached steady values**

© 2015 ANSYS, Inc.

ANSYS Confidential

Post-Processing

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

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