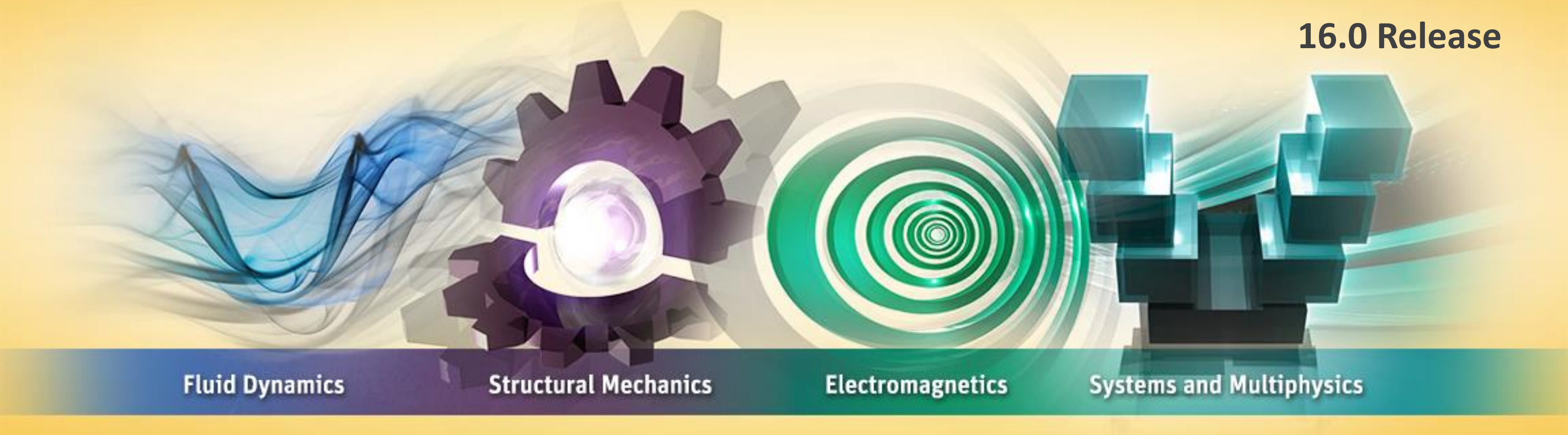
Lecture 14:
Review & Tips

16.0 Release



Introduction to ANSYS CFX

Introduction

Lecture Theme:

The material presented on Day 2 focused on the most widely used physical models in CFX and CFD best practices.

Learning Aims:
You will learn:

- How to select models needed to perform calculations involving turbulent flow, heat transfer, moving objects, and unsteady flow
- How to use domain interfaces
- How to use CEL and additional variables
- How to apply best practices when performing a simulation and creating a mesh to ensure the highest possible accuracy

Learning Objectives:

In addition to the fundamental task of solving the Navier-Stokes equations, most CFD calculations also involve the use of one or more physical models. Understanding the models available in CFX and the background theory helps you make the right choices in your own project.

Turbulence

Estimate the flow Reynolds number ahead of time to determine if the flow is laminar or turbulent

Choose your near-wall modeling strategy ahead of time and check y^+ values to make sure the near-wall mesh is suitable

- $20 < y^+ < 200$ for a Wall Function solution
- $y^+ \approx 1$ with the SST model for a low-*Re* solution

The SST is a good choices for general applications

Be aware of the limitations of the turbulence model chosen

- RANS models resolve the mean flow field, therefore a lot of transient turbulent structures are not captured
  - These may be important when simulating noise and vibration
- The k-ε model can give inaccurate separation predictions

Heat Transfer

Always make sure energy imbalances have reached acceptable levels in CHT cases

If thermal radiation is modeled choose an appropriate model depending on the optical thickness (and memory available on your computer)

Thin wall modeling and thermal contact resistances can be set at domain interfaces

High speed flows (Mach > 0.3) should use the Total Energy Model

Domain Interfaces

Domain Interfaces can be used as part of a meshing strategy as well as for connecting different domains or reference frames together

When the mesh is different on each side of the interface a GGI (General Grid Interface) is used

- This will use more memory in the Solver than a continuous mesh
- Accuracy across a GGI interface is usually not a concern as long as the mesh length scales on each side are similar

Automatic Domain Interfaces are created by CFX-Pre in some cases

- Always check these and don't assume that all the required Domain Interfaces have been created

Moving Zones

Moving boundaries can be simulated in several different ways

- Steady-state Frozen Rotor
- Steady-state Stage
- Transient Rotor-Stator

For rotating walls, a wall velocity can simply be imposed if the motion is purely tangential (e.g. a rotating hub or a solid brake disk)

Mesh motion is usually used to simulate Solid body motion or deforming boundaries

Transient Simulations

In a transient analysis the time step should be small enough to capture the transient behaviour of interest

Boundary conditions can be functions of time

Convergence should be monitored so that each time step is converged

- It is generally better to reduce the overall timestep size to improve convergence rather than increasing the number of coefficient loops

Remember to record Transient Results objects before running

Why Does My Case Fail in the Solver?

- **If the solver has diverged in the first few iterations perform some basic checks:**
  - Examine the mesh for quality problems
  - Are the boundary conditions physical?
  - What's the initial pressure (compressible flow)?
  - Examine the solution fields (Pressure, Velocity, …)
    - Look for the max / min values, they will usually be very high / low

- If there is an error message when the case fails, first **carefully** read the message

Why Does My Case Fail in the Solver?

First *carefully* read the error message

- The error message may recommend setting an Expert Parameter
- This may be an appropriate fix, or it may mask an underlying problem
  - Example:

```
+--------------------------------------------------------------------+
|                 Checking for Isolated Fluid Regions                |
+--------------------------------------------------------------------+

    2 isolated fluid regions were found in domain R1
…
…
turn off this check by setting the expert parameter "check isolated
regions = f".
```

  - This error usually means domain interfaces are missing, so setting the expert parameter would not usually be appropriate

Why Does My Case Not Converge?

- **Sometimes simulations which are run in steady state mode will not converge, even with:**
  - good mesh quality
  - and a well selected timescale

➔ It probably means that the flow is transient

Mesh Refinement Studies

- Errors in a **converged** solution arise from:
  - Numerical Errors
    - E.g. round-off errors, convergence (lack-of) errors
  - Model Errors
    - E.g. accuracy of boundary conditions, physical models
  - Discretization Errors
    - Errors arising from converting the continuous governing equations into a discrete form that can be solved on a computer
- Discretization errors decrease with increased mesh refinement
- Mesh refinement studies are used to estimate the significance of discretization errors on your solution
- Mesh refinement studies are recommended for each new type of simulation you perform

Mesh Refinement Studies

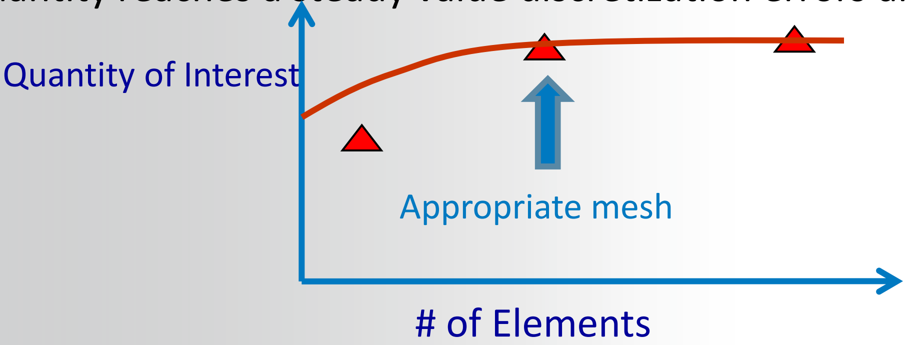
A mesh refinement study consist of solving the same case on progressively finer meshes

- Each mesh should be significantly finer than the previous,

  e.g. 100k nodes, 200k nodes, 400k nodes

The quantities of interest should be evaluated and compared for each mesh

- When the quantity reaches a steady value discretization errors are no longer significant



Advanced Training Courses

And finally:

We hope you have found this week to be useful. Please contact technical support with any queries. In addition, we hope to see you in the next few months in some of our advanced training sessions.

- Examples of advanced courses are:
  - Combustion and radiation
  - Fluid Structure Interaction
  - Multiphase
  - Turbulence Modelling
  - Customisation
- We also offer '**Focus Days**', which are days of 1:1 coaching on your specific problem
- For more information, ask your trainer or see the contact details in Lecture '00' of this course.

What is next?

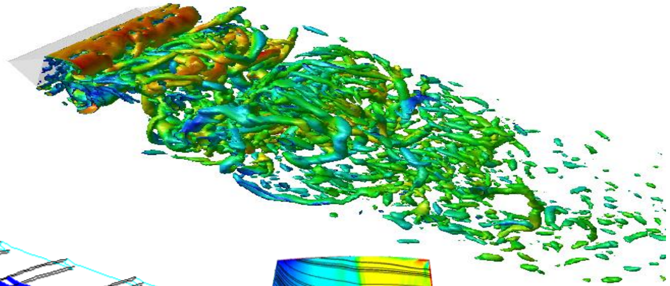
After the introductory course you might still require help & information:

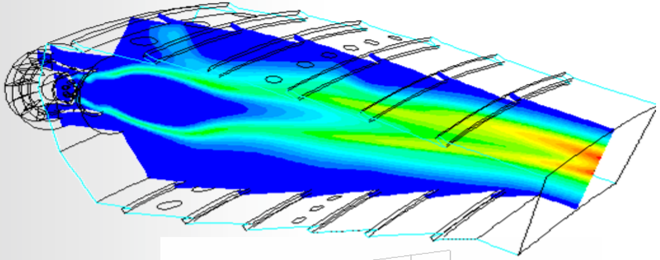
- Visit ANSYS Customer Portal
  - www.ansys.com/customer
- Contact your local Technical Support
  - Network of experts
  - Access to an extensive knowledge base!
- Visit individual training
  - Training is solely based on your particular Geometry/Physics
- Visit advanced training (e.g. combustion, multiphase, User FORTRAN, etc)
- Consulting Project

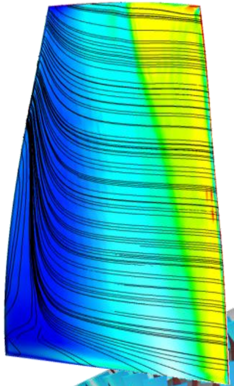
What is next?

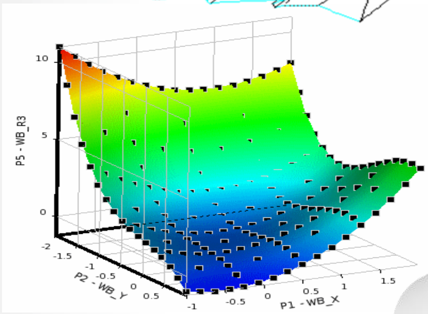
Many models available

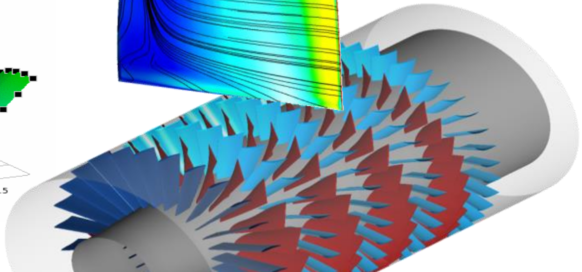
- Turbulence Models
  - E.g. Transition, SAS-SST, Large Eddy, etc
- Radiation
- Combustion
- Multiphase
  - Euler-Euler
  - Euler-Lagrange
- Turbomachinery
- Six Degrees of Freedom/Rigid Body
- Mesh Morphing
- User FORTRAN
- Optimization











Thank you for attending the

Introductory ANSYS CFX Training

Your feedback is important to us.