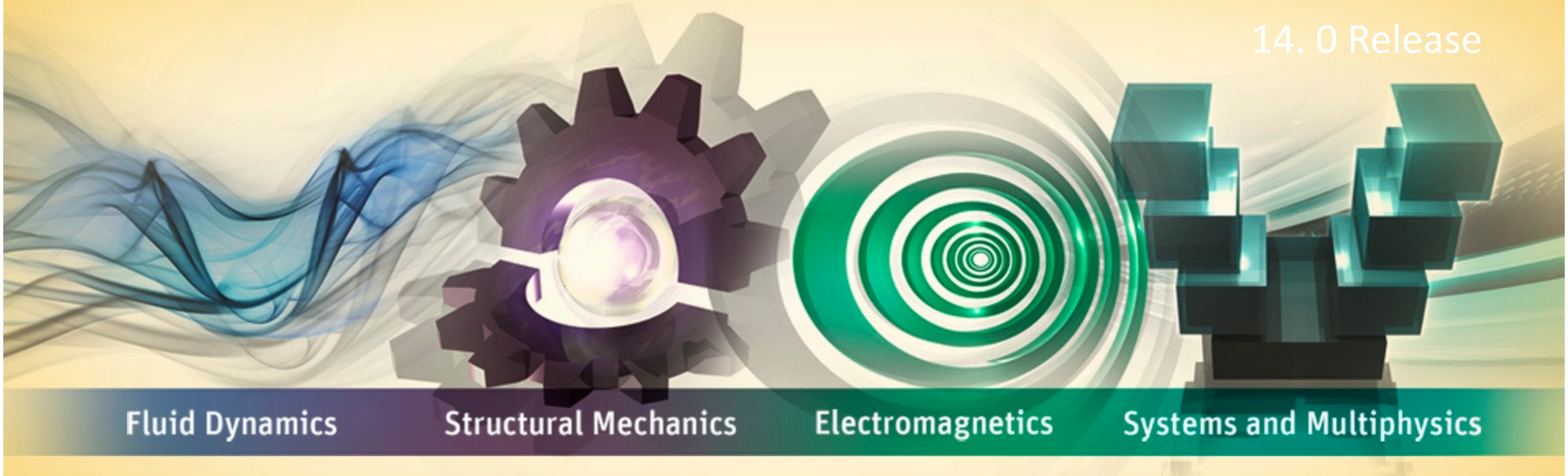
ANSYS

Realize Your Product Promise™

# Lecture 12
# Review and Tips

14. 0 Release




# Introduction to ANSYS FLUENT

# Introduction

**Lecture Theme:**

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

**Learning Aims:**
To recap on:

- How to select models needed to perform calculations involving turbulent flow, heat transfer, moving objects, and unsteady flow
- How to use non-conformal interfaces and periodic boundaries
- How to write and use a simple UDF
- 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 FLUENT 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
  - $30 < y^+ < 300$ for a Wall Function solution
  - $y^+ <= 2$ with the SST model for a near-wall resolved solution with enhanced wall treatment
- The Realizable k-$\varepsilon$ or SST k-$\omega$ models are 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-$\varepsilon$ model can give inaccurate separation predictions

# Heat Transfer

- The double precision setting for the Solver is recommended for *Conjugate Heat Transfer* (CHT) simulations (i.e. when a solid cell zone is included)
- 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 shell conduction can be used at external and internal walls
- Thin wall modeling and thermal contact resistances can be set at coupled walls (wall\wall-shadow)

# Moving Zones

- Moving boundaries can be simulated in several different ways
  - Steady-state Moving Reference Frame (SRF & MRF)
  - Steady-state Mixing Plane
  - Unsteady Sliding Mesh
  - Unsteady Dynamic Mesh

- 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)

- Dynamic mesh is usually used to simulate solid body motion or deforming boundaries
  - Easy set-up
  - 3 methods :
    - Smoothing
    - Dynamic Layering
    - Remeshing

# 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 time step size to improve convergence rather than increasing the number of iterations per time step
- Remember to record data (either images and / or result data files) during the solution process

# 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
    - Also check that it has been properly scaled
  - 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 Not Converge?

- **Sometimes simulations which are run in steady state mode will not converge, even with**
  - good mesh quality
  - and a well selected timescale (coupled solver) or under-relaxation factors (segregated solver)

**➔ It probably means that the flow is transient**

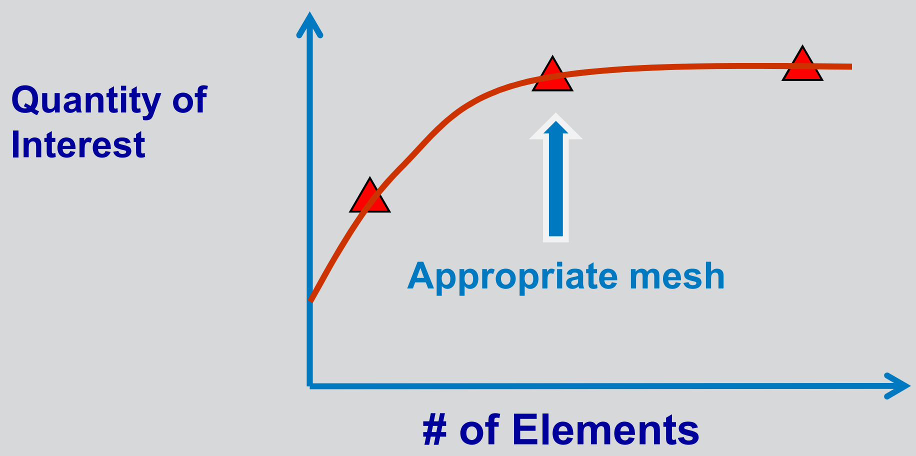
# Mesh Refinement Studies

- 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 mesh spacing
- 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.

- For FLUENT users, we have advanced courses in:
  - Combustion
  - Dynamic Mesh
  - Heat Transfer
  - Multiphase
  - Turbulence Modelling
  - UDFs

  *+ many more….*

- We also offer '**Application Days**', which are days of 1:1 coaching on your specific problem with one of our engineers

- For more information, ask your trainer, or see the contact details in Lecture '00' of this course.

Introduction | Day 2 Models | CFD Tips | Conclusion