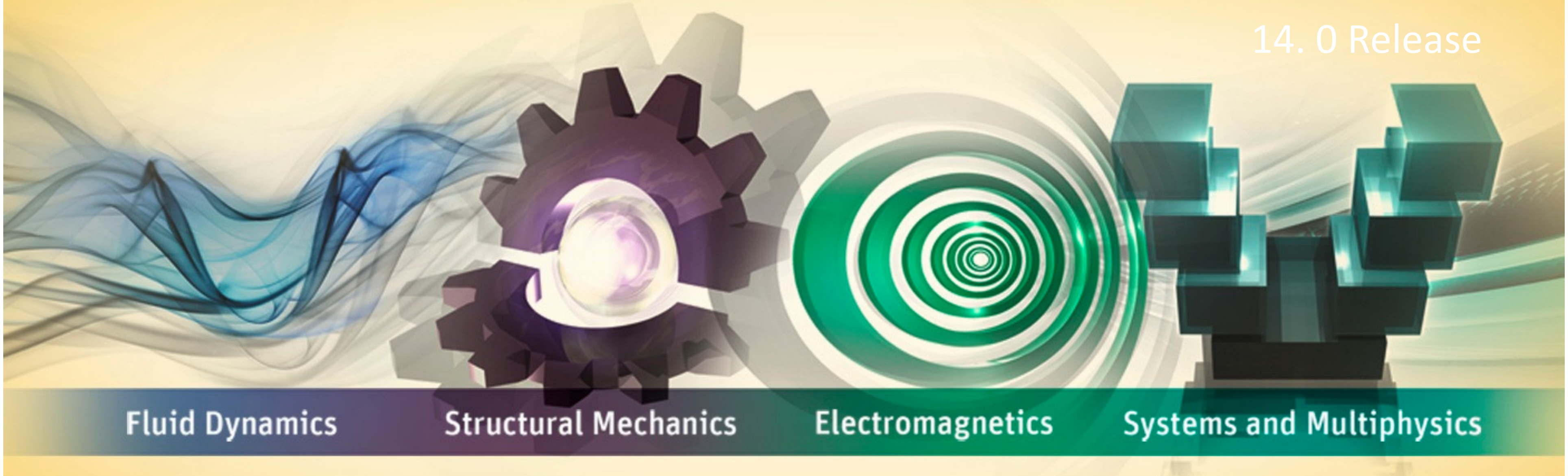
ANSYS

Realize Your Product Promise™

Lecture 10
Transient Flow Modeling

14. 0 Release



Introduction to ANSYS FLUENT

Introduction

Lecture Theme:

Performing a transient calculation is in some ways similar to performing a steady state calculation, but there are additional considerations. More data is generated and extra inputs are required. This lecture will explain these inputs and describe transient data post-processing.

Learning Aims:

You will learn:

- How to set up and run transient calculations in FLUENT
- How to choose the appropriate time step size for your calculation
- How to post-process transient data and make animations

Learning Objectives:

Transient flow calculations are becoming increasingly common due to advances in high performance computing (HPC) and reductions in hardware costs. You will understand what transient calculations involve and be able to perform them with confidence.

Motivation

- **Nearly all flows in nature are unsteady!**
  - Steady-state assumption is possible if we:
    - Ignore unsteady fluctuations
    - Employ ensemble/time-averaging to remove unsteadiness
      - this is what is done in modeling RANS turbulence

- **In CFD, steady-state methods are preferred**
  - Lower computational cost
  - Easier to post-process and analyze

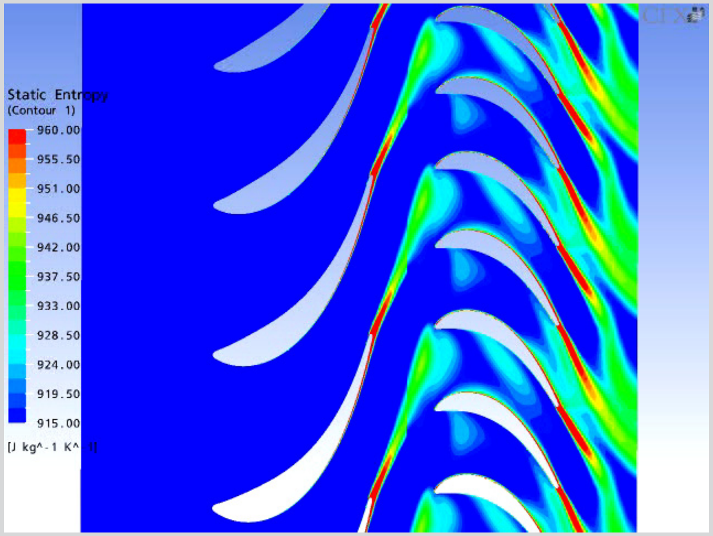
- **Many applications require resolution of unsteady flow:**
  - Aerodynamics (aircraft, land vehicles, etc.) – vortex shedding
  - Rotating Machinery – rotor/stator interaction, stall, surge
  - Multiphase Flows – free surfaces, bubble dynamics
  - Deforming Domains – in-cylinder combustion, store separation
  - Unsteady Heat Transfer – transient heating and cooling
  - Many more …

ANSYS

Origins of Unsteady Flow

- **Natural unsteadiness**
  - Unsteady flow due to growth of instabilities within the fluid or a non-equilibrium initial fluid state
  - Examples: natural convection flows, turbulent eddies of all scales, fluid waves (gravity waves, shock waves)
- **Forced unsteadiness**
  - Time-dependent boundary conditions, source terms drive the unsteady flow field
  - Examples: pulsing flow in a nozzle, rotor-stator interaction in a turbine stage

Kelvin-Helmholtz Cloud Instability



Rotor-Stator Interaction in an Axial Compressor

Unsteady CFD Analysis

- **Simulate a transient flow field over a specified time period**
  - Solution may approach:
    - Steady-state solution – Flow variables stop changing with time
    - Time-periodic solution – Flow variables fluctuate with repeating pattern
  - Your goal may also be simply to analyze the flow over a prescribed time interval
    - Free surface flows,
    - Moving shock waves,
    - …
- **Extract quantities of interest**
  - Natural frequencies (e.g. Strouhal Number)
  - Time-averaged and/or RMS values
  - Time-related parameters (e.g. time required to cool a hot solid, residence time of a pollutant)
  - Spectral data – Fourier Transform (FT)

Unsteady CFD Analysis

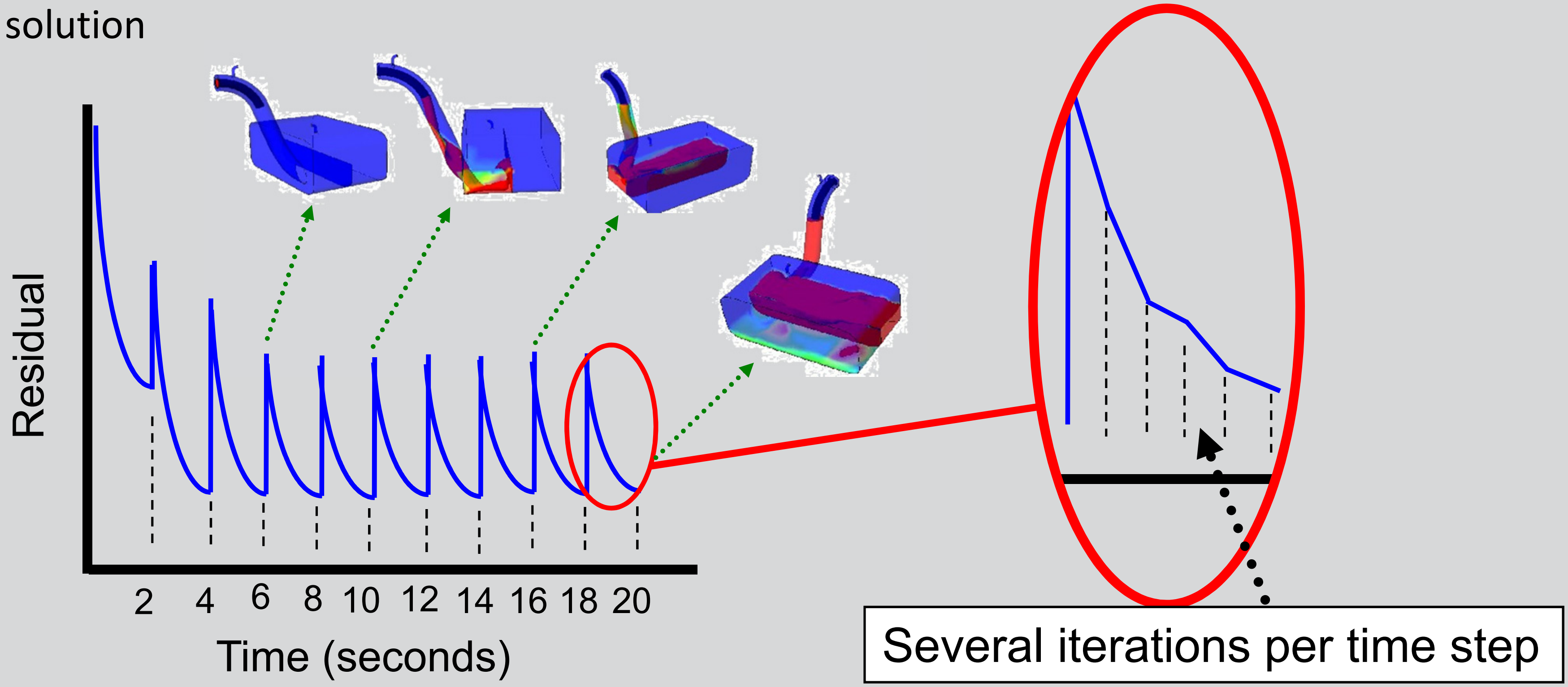
- Transient simulations are solved by computing a solution for many discrete points in time

Time steps = 2 s

Initial Time = 0 s

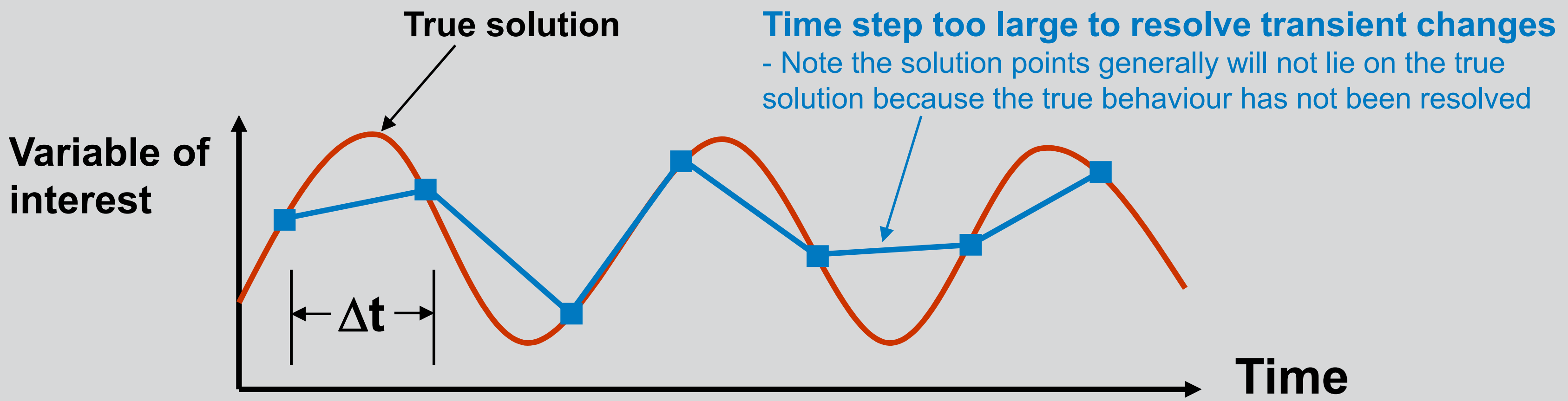
Total Time = 20 s

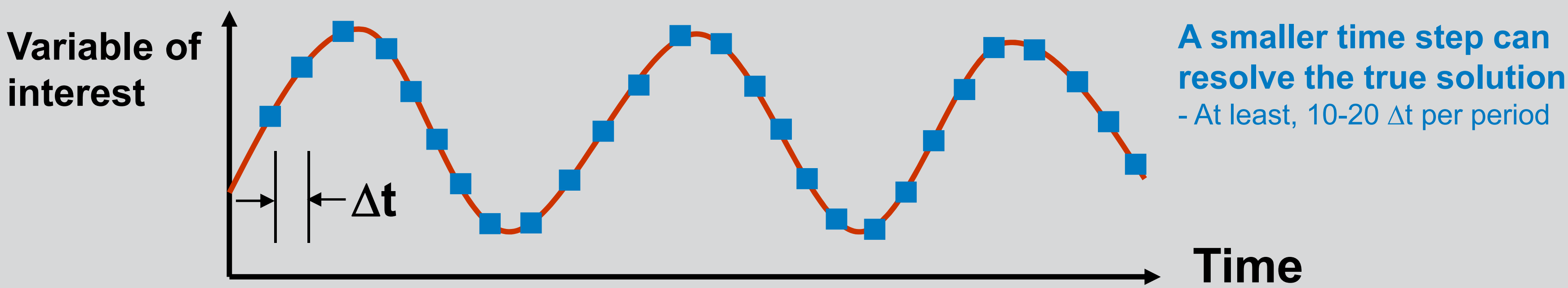
- At each time point we must iterate to the solution



Selecting the Transient Time Step Size

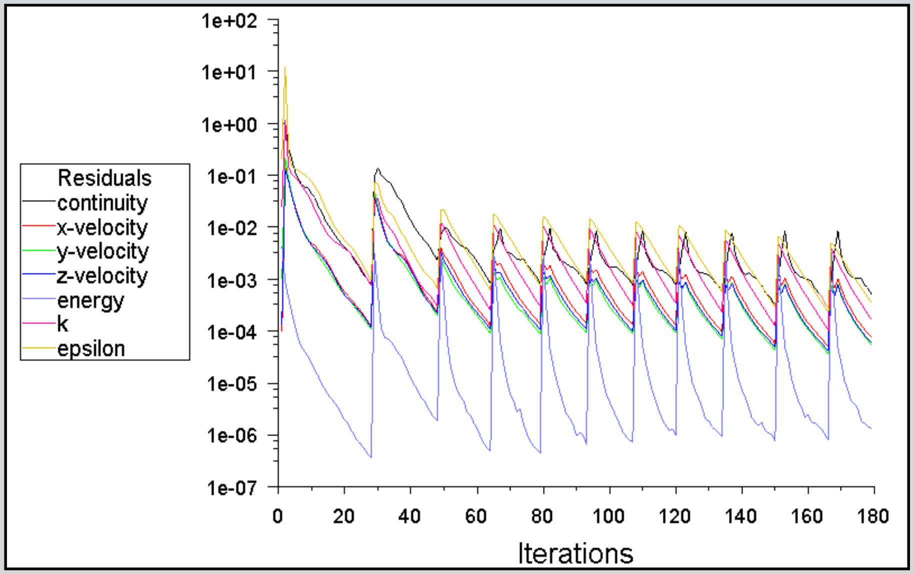
- **The time step size is an important parameter in transient simulations**
  - **Δt must be small enough to resolve time-dependent features …**





Convergence Behavior

- Residual plots for transient simulations are not always indicative of a converged solution
- You should select the time step size such that the residuals reduce by around **three orders of magnitude** within one time step
  - This will ensure accurate resolution of transient behavior
  - For smaller time steps, residuals may only drop by 1-2 orders of magnitude - look for a monotonic decrease throughout the time step
- A residual plot for a simple transient calculation is shown here



Introduction | Unsteady Flow | Time Step | Setup | Post-Processing | Summary

Selecting the Transient Time Step Size

- **… and it must be small enough to maintain solver stability**
  - The quantity of interest may be changing very slowly (e.g. temperature in a solid), but you may not be able to use a large time step if other quantities (e.g. velocity) have smaller timescales

- **The Courant Number is often used to estimate a time step:**

$$\text{Courant Number} = \frac{\text{Characteristic flow velocity} \times \Delta t}{\text{Typical Cell Size}}$$

  - This gives the number of mesh elements the fluid passes through in one time step
  - Typical values are 1 – 10, but in some cases higher values are acceptable

Selecting the Transient Time Step Size

- **Tips & Tricks for the estimation of the time step:**

- Usual Case : $$\Delta t = \frac{1}{3}.\frac{L}{\mathrm{V}}$$
  - restrictive but safe for convergence with L=cell characteristic size

- Turbomachinery : $$\Delta t = \frac{1}{10}.\frac{\text{Number of Blades}}{\text{Rotational Velocity}}$$

- Natural Convection : $$\Delta t = \frac{L}{(\mathrm{g}.\beta.\,\Delta\mathrm{T.L})^{1/2}}$$

- Conduction in solids : $$\Delta t = \frac{L^2}{\left(\dfrac{\lambda}{\rho.Cp}\right)}$$

L = Characteristic length
V = Characteristic velocity

- **A smaller time step will typically improve convergence**

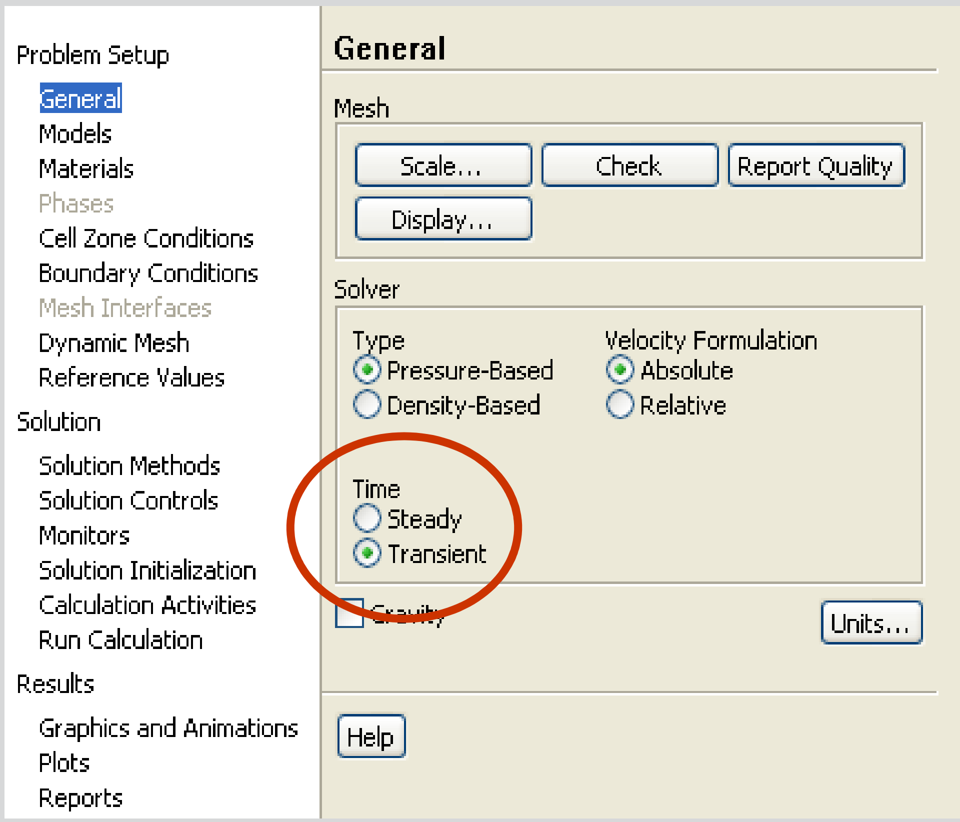
Transient Flow Modeling Workflow

- **Similar set-up as steady-state simulation, then:**

1. **Enable the unsteady solver**
2. **Set up physical models and boundary conditions as usual**
   - Transient boundary conditions are possible – you can use either a UDF or profile to accomplish this
3. **Prescribe initial conditions**
   - Best to use a physically realistic initial condition, such as a steady solution
4. **Assign solver settings and configure solution monitors**
5. **Configure animations and data output/sampling options**
6. **Select time step and max iterations per time step**
7. **Prescribe the number of time steps**
8. **Run the calculations (Iterate)**

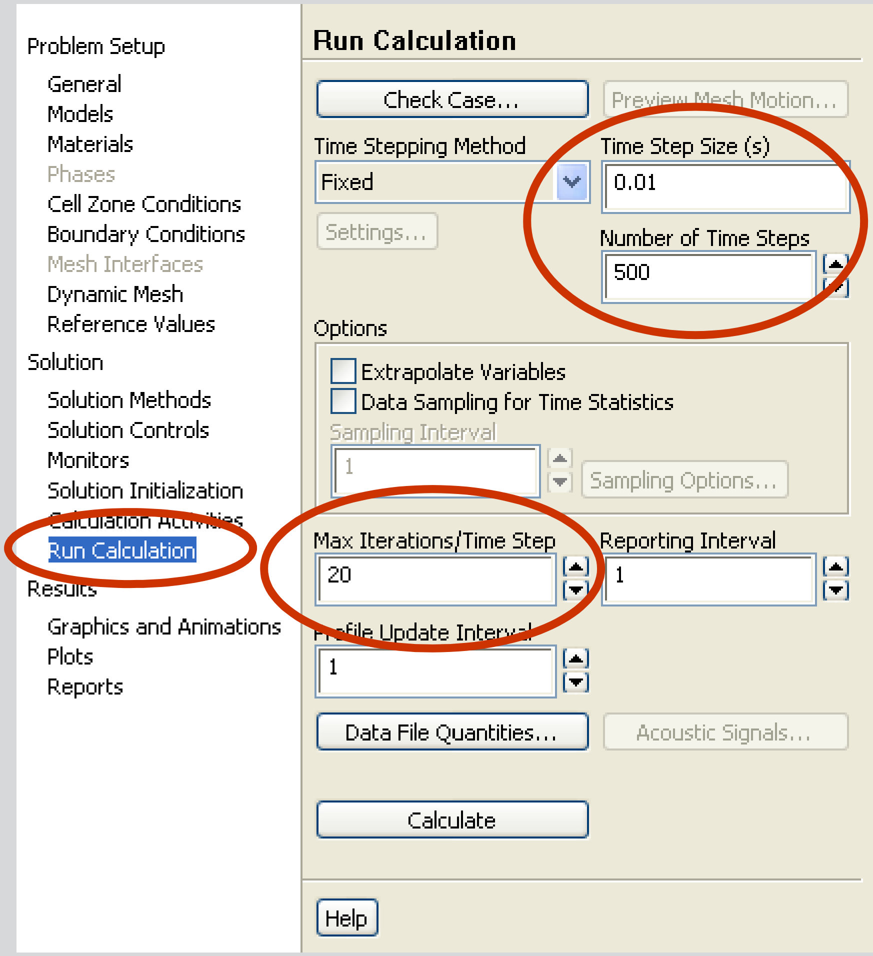
Enabling the Transient Solver

- To enable the unsteady solver, select the **Transient** button on the General problem setup form



Set Up Time Step Size

- **Set the time step size**
  - This controls the spacing in time between the solution points
- **Options are:**
  - Number of time steps
  - Maximum number of iterations per time step

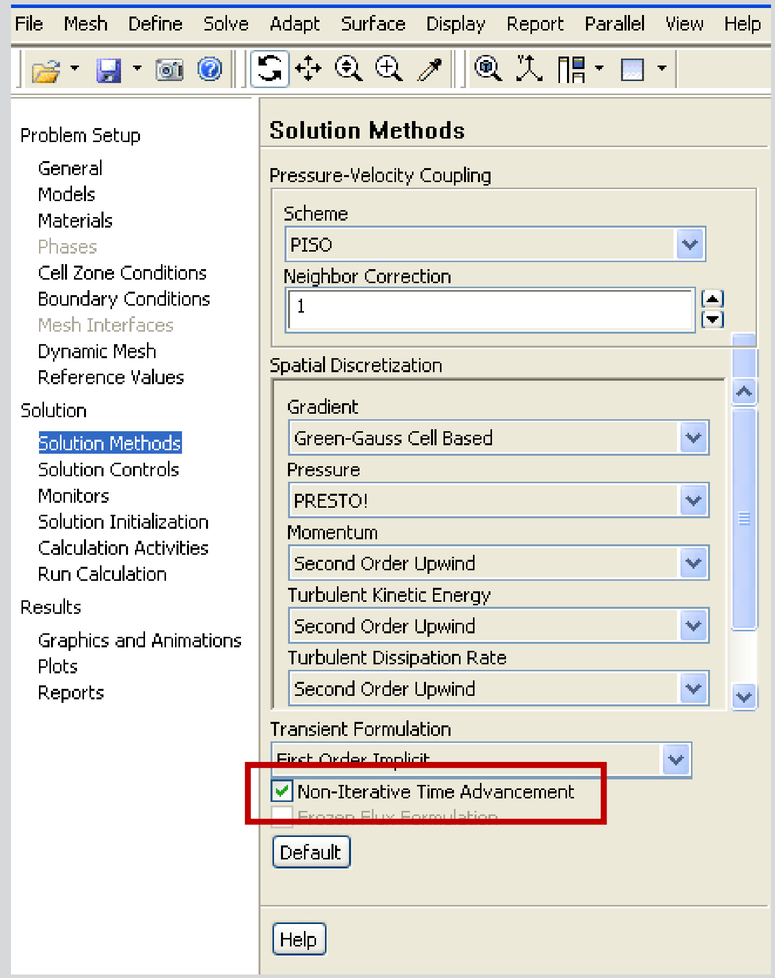


Introduction | Unsteady Flow | Time Step | Setup | Post-Processing | Summary

ANSYS

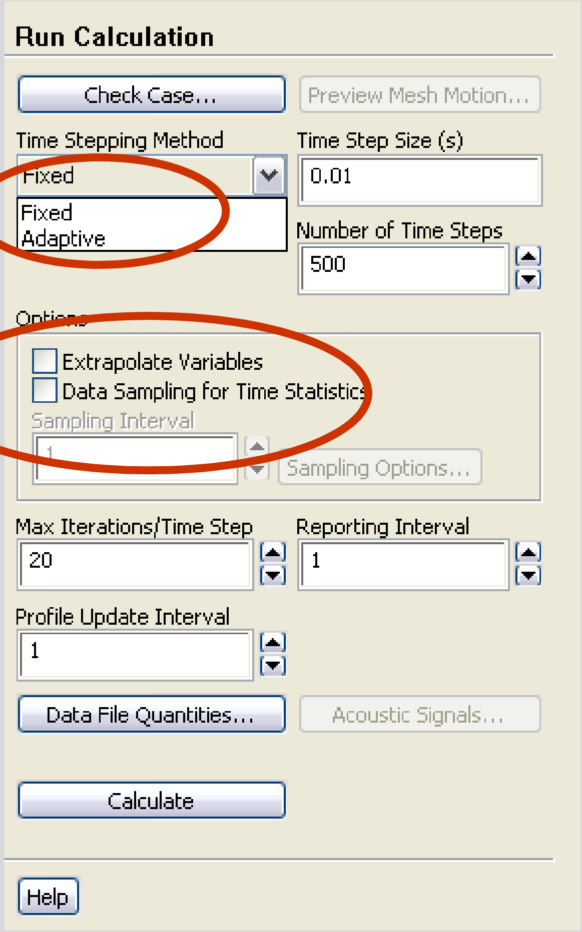
Non-iterative Time Advancement

- Non-iterative Time Advancement (**NITA**) is available for faster computation time
  - NITA runs about 2x to 10x as fast as ITA scheme
- **Limitations:** Available with pressure-based solvers only
  - NITA schemes are not available for multiphase (except VOF), reacting flows, radiation models, porous media, fan models, etc.
- Consult the Appendix and FLUENT Documentation for additional details



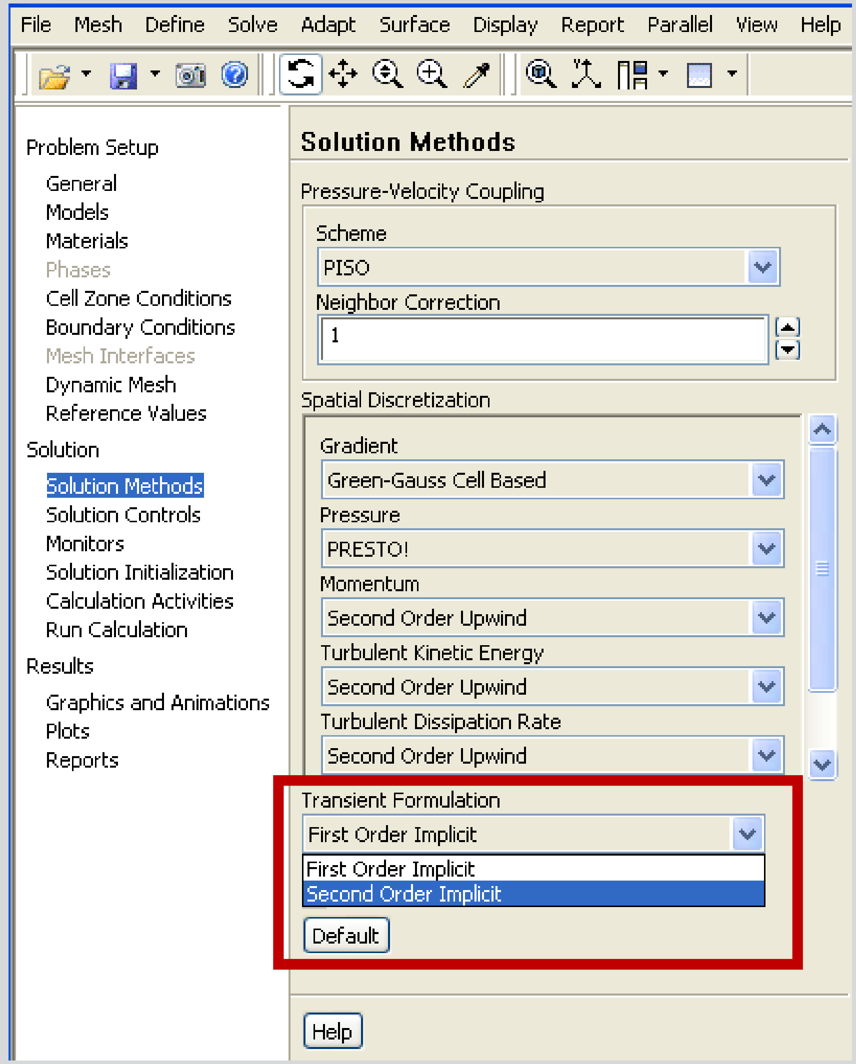
Unsteady Flow Modeling Options

- **Adaptive Time Stepping**
  - Automatically adjusts time-step size based on local truncation error analysis
  - Customization possible via UDF
- **Extrapolate Variables**
  - Speed up the transient solution by reducing required sub-iteration
    - Using Taylor series expansion solution will be extrapolated to the next time level to improve the predicted initial value
- **Data Sampling for Time Statistics**
  - Particularly useful for LES turbulence calculations



Solver Control

- **The transient scheme defines the numerical algorithm for the transient term**
- **Two Implicit time-stepping schemes are available:**
  - First Order Backward Euler
    - (more stable)
  - Second Order Backward Euler
    - (more accurate)

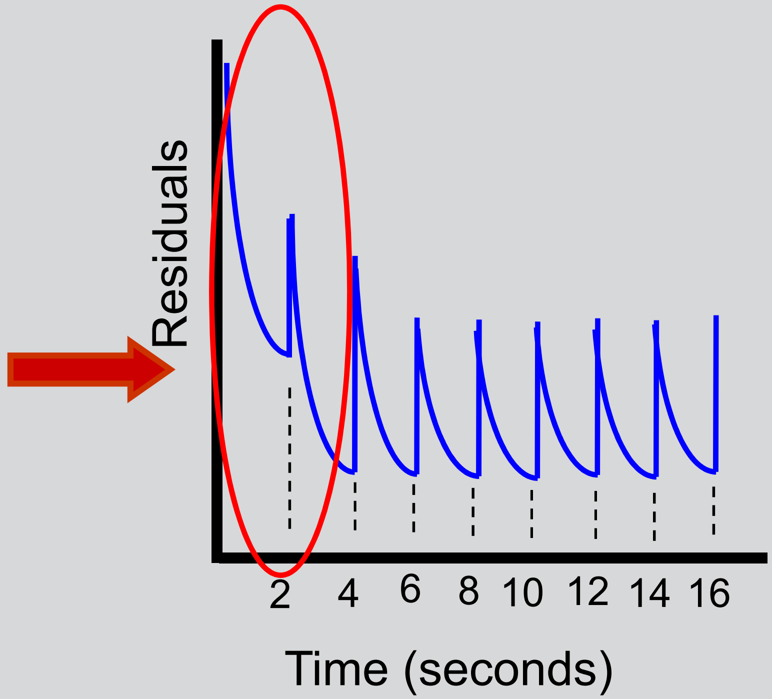


ANSYS

Initialization

- **Physically realistic initial conditions should be used**
  - A converged steady state solution is often used as the starting point

- **If a transient simulation is started from an approximate initial guess, the initial transient will not be accurate**
  - The first few time steps may not converge
  - A smaller time step may be needed initially to maintain solver stability
  - For cyclic behavior the first few cycles can be ignored until a repeatable pattern is obtained



Tips for Success in Transient Flow Modeling

- With Pressure-based Solvers, use **PISO** scheme for Pressure-Velocity Coupling – this scheme provides faster convergence for unsteady flows than the standard SIMPLE approach

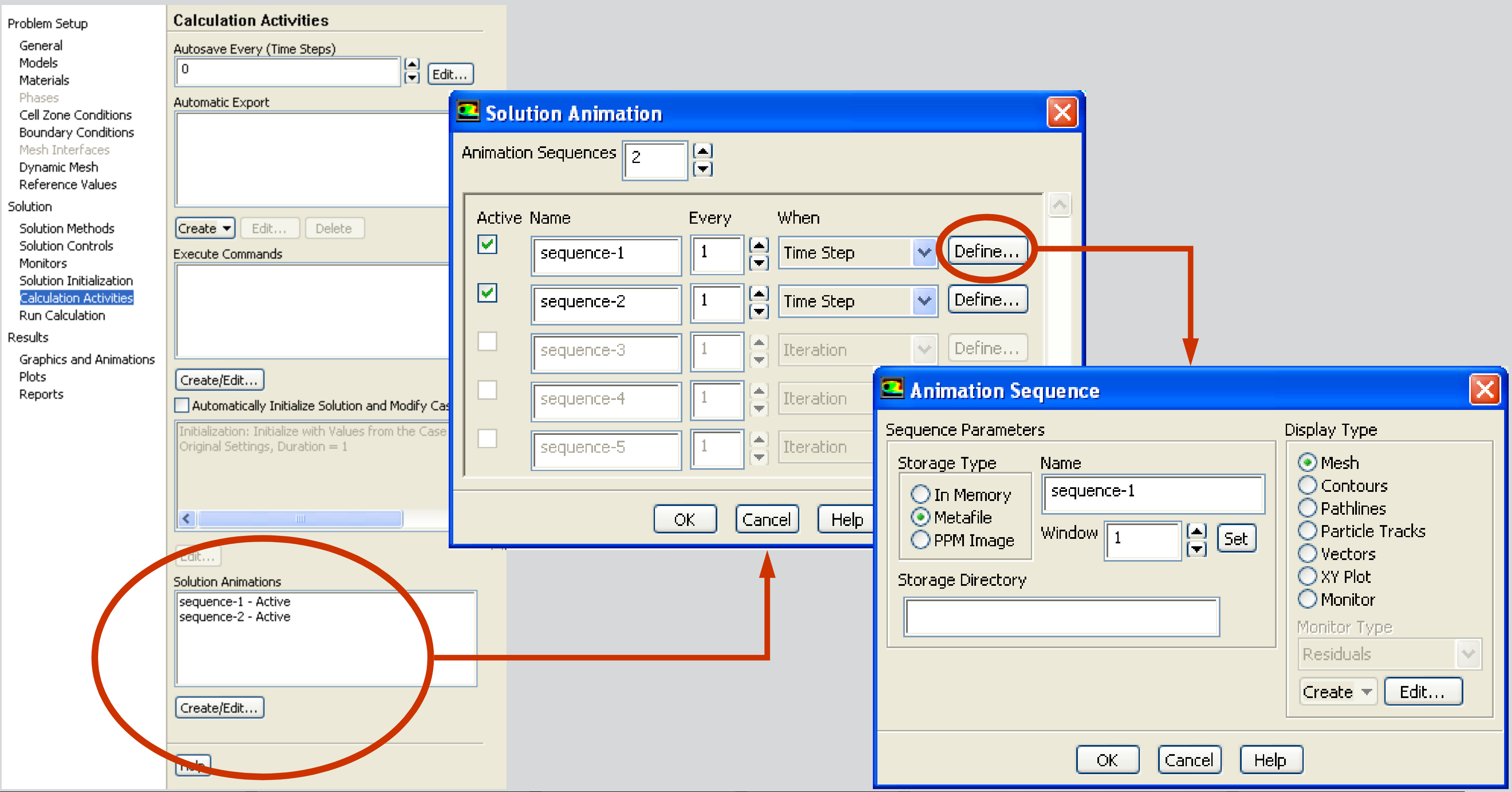
- Select the number of iterations per time step to be **around 20**
  - it is better to reduce the time step size than to do too many iterations per time step

- Remember that accurate initial conditions are as important as boundary conditions for unsteady problems
  - **initial condition should always be physically realistic!**

- To iterate without advancing in time, specify zero time steps
  - This will instruct the solver to converge the current time step only

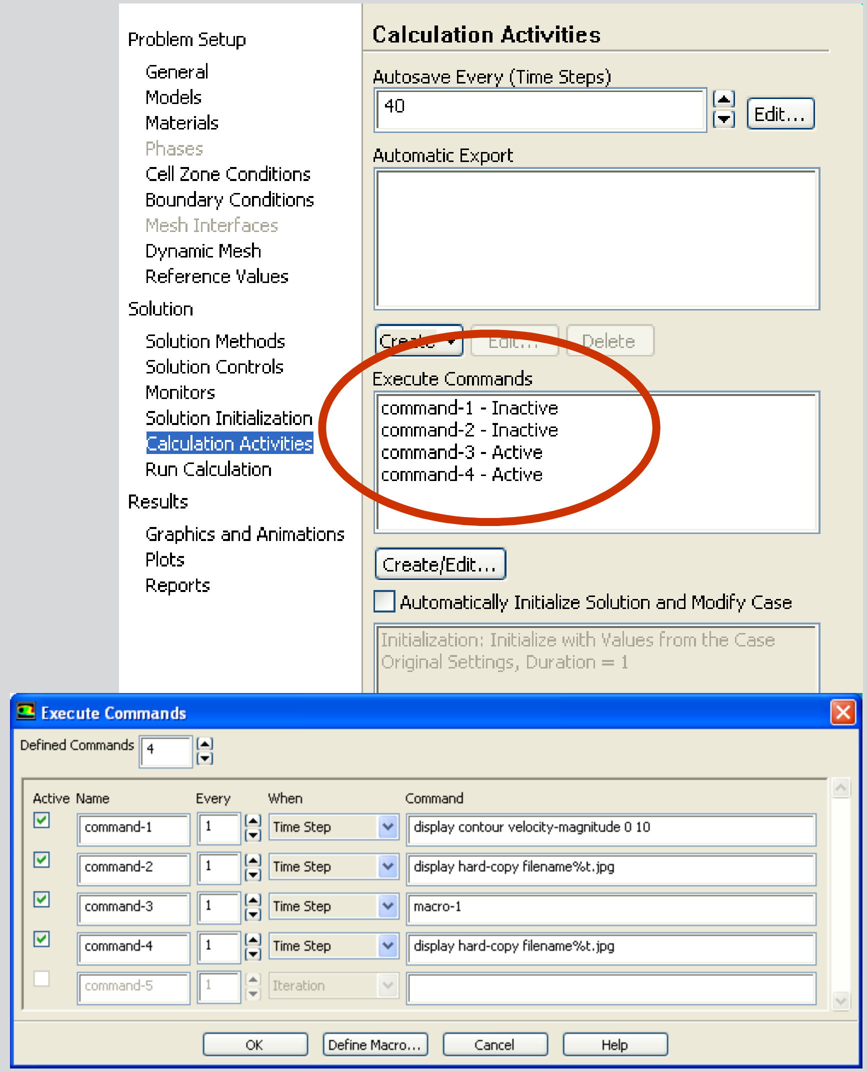
Unsteady Flow Modeling – Animations

- **You must set up any animations BEFORE performing iterations**
  - Animation frames are written/stored on-the-fly during calculations



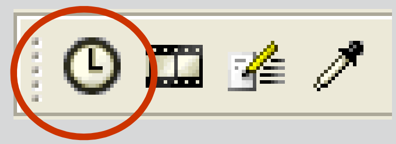
Unsteady Flow Modeling – Animations

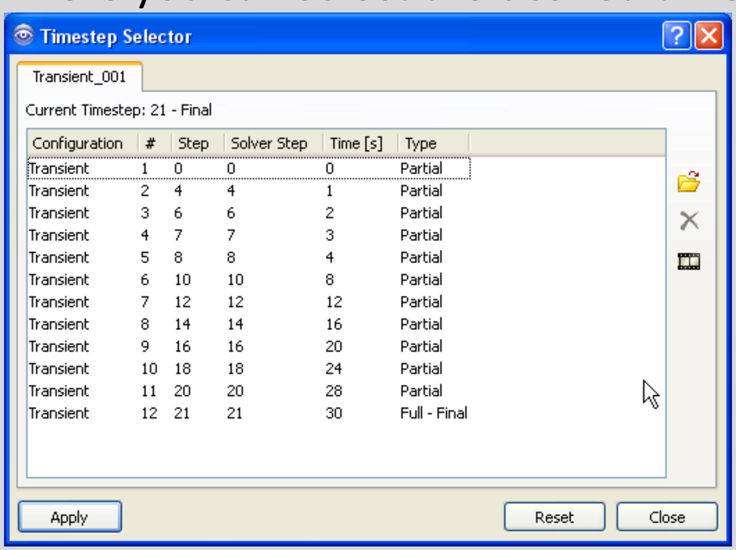
- **Another method in FLUENT is available which makes use of the Execute Commands feature**
- **Text commands or macros can be defined which are executed by the solver at prescribed iteration or time step intervals**
- **This approach is very useful in creating high-quality animations of CFD results**
  - A command is defined which generates an animation frame (contour plot, vector plot, etc.) and then writes that frame to a hard copy file.
  - Third-party software can then be used to link the hard copy files into an animation file (AVI, MPG, GIF, etc.)



CFD-Post: Timestep Selector

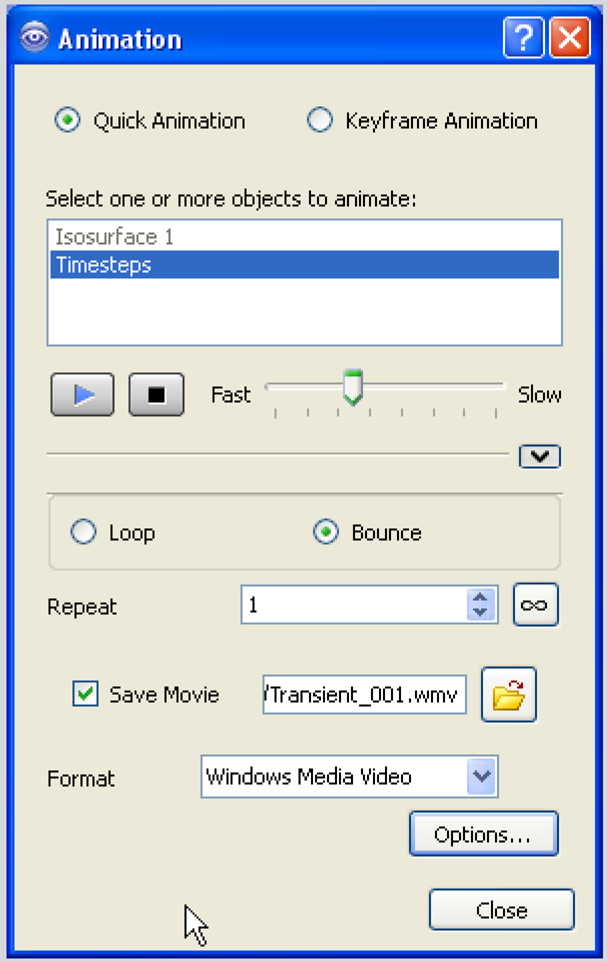
- **Transient results can also be post-processed in CFD-Post.**
  - You need to instruct FLUENT **when** to export data (every *n* time steps)
  - *Otherwise, by default, FLUENT does not write any data to disk during the solution, and interim results will be forgotten.*
  - So **in FLUENT**, under ‘Calculation Activities’, select ‘Autosave’
  - **In CFD Post**, select the first results file, then tick “Load Complete History as a Single Case”.
  - The clock icon will open up a window from where you can select the desired time step





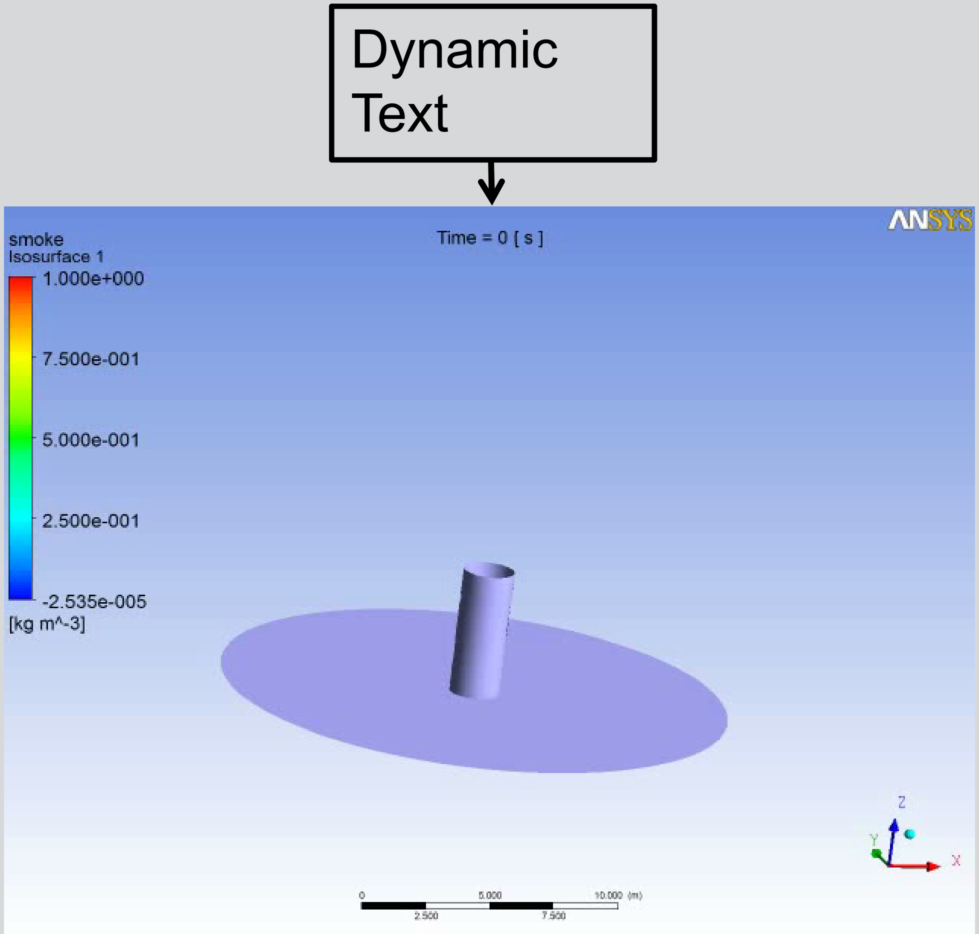
CFD-Post: Animations

- **Two modes:**
  – Quick Animation
  – Keyframe



CFD-Post: Animations

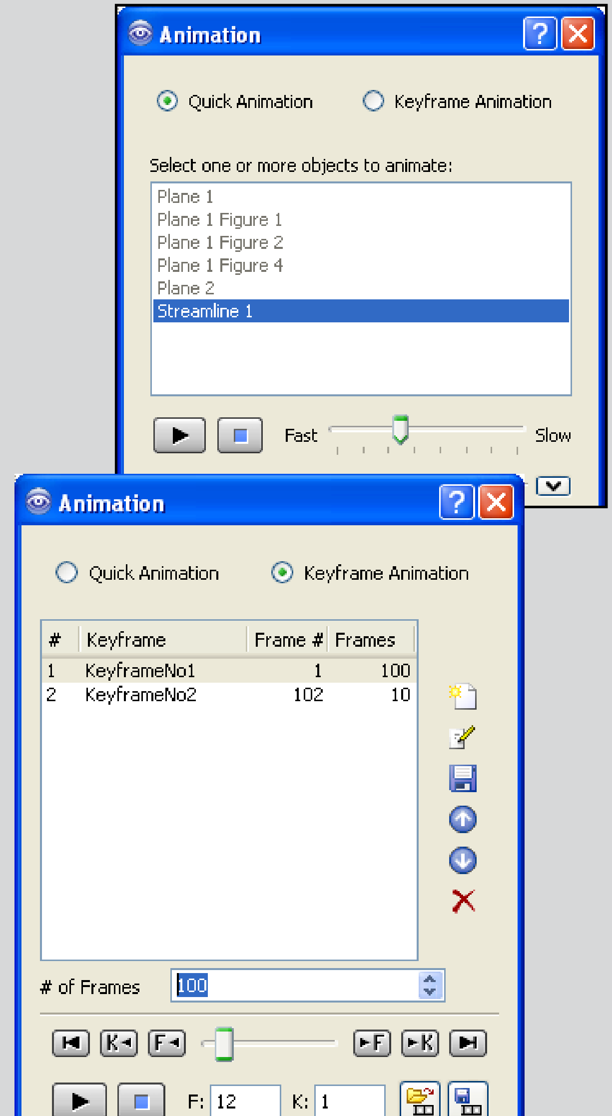
- **Quick Animation:**
  - Selection of animation objects
  - Animation of "primary" object variable:
    - Plane: Coordinate
    - Streamline: Dynamic, velocity dependent symbols on streamlines
    - Time steps
  - Limited control



Introduction | Unsteady Flow | Time Step | Setup | Post-Processing | Summary

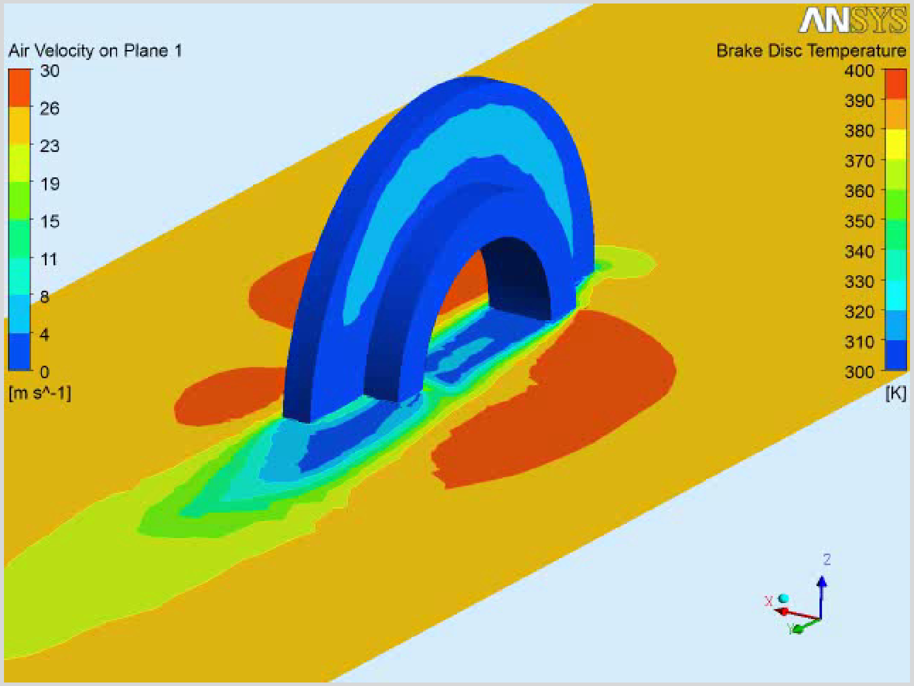
CFD-Post: Keyframe Animations

- **Keyframe Animation:**
  - More control
  - When you create a Keyframe, a snapshot of the current state is stored with that Keyframe
  - A series of Keyframes represents a series of different states
    - Camera position, object visibility, selected time step, or anything else can be different between Keyframes
  - An animation needs at least two Keyframes (one for the start and one for the end)
  - Enter the *# of Frames* between each Keyframe
    - Everything that is different between the Keyframes gets interpolated over the # of Frames



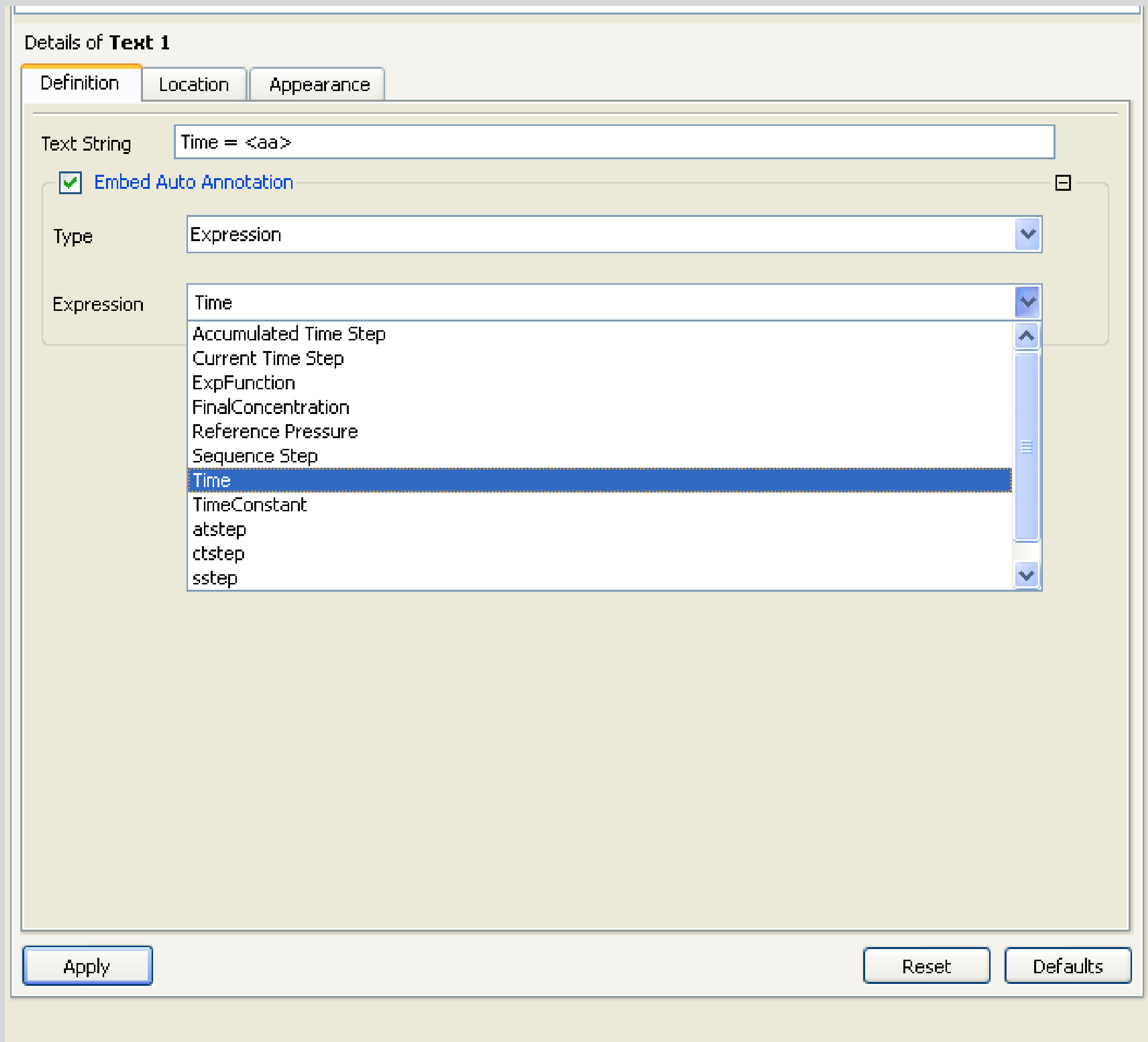
CFD-Post: Typical Keyframe Animation Procedure

1. ***Timestep Selector* load the first time step**
2. **Create necessary plots and position the view**
3. **Create the first *Keyframe***
4. **Load the last time step**
5. **If necessary change the plots and the view**
6. **Create the second *Keyframe***
7. **Select the first *Keyframe* and set the *# of Frames***
   - This is the # of Frames in between the first and second Keyframes
   - If we have a total of 100 time steps, then setting *# of Frames* to 98 will produce a total of 100 frames (98 plus first, plus last) and therefore 1 frame per time step
8. **Set the Movie options**
9. **Rewind to the first *Keyframe* and click *Play***



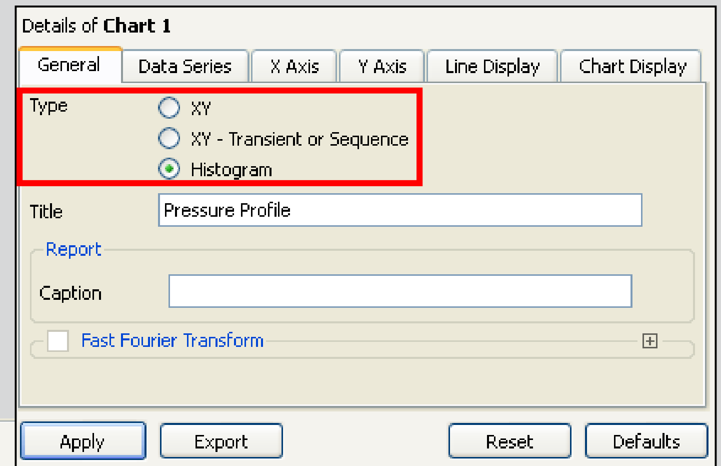
In this example the first and second Keyframes used a different view position and the Transparency of the Plane was changed from 0 to 1. The changes between Keyframes are interpolated gradually over each animation frame

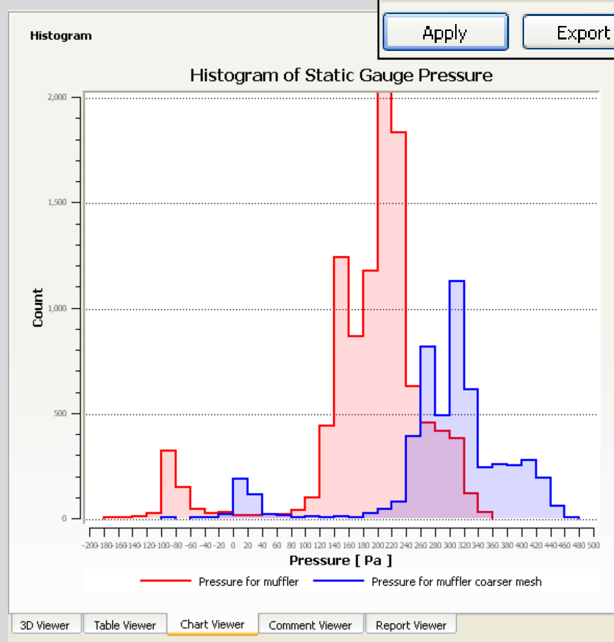
CFD-Post: Dynamic Text Objects



CFD-Post: Transient Charts

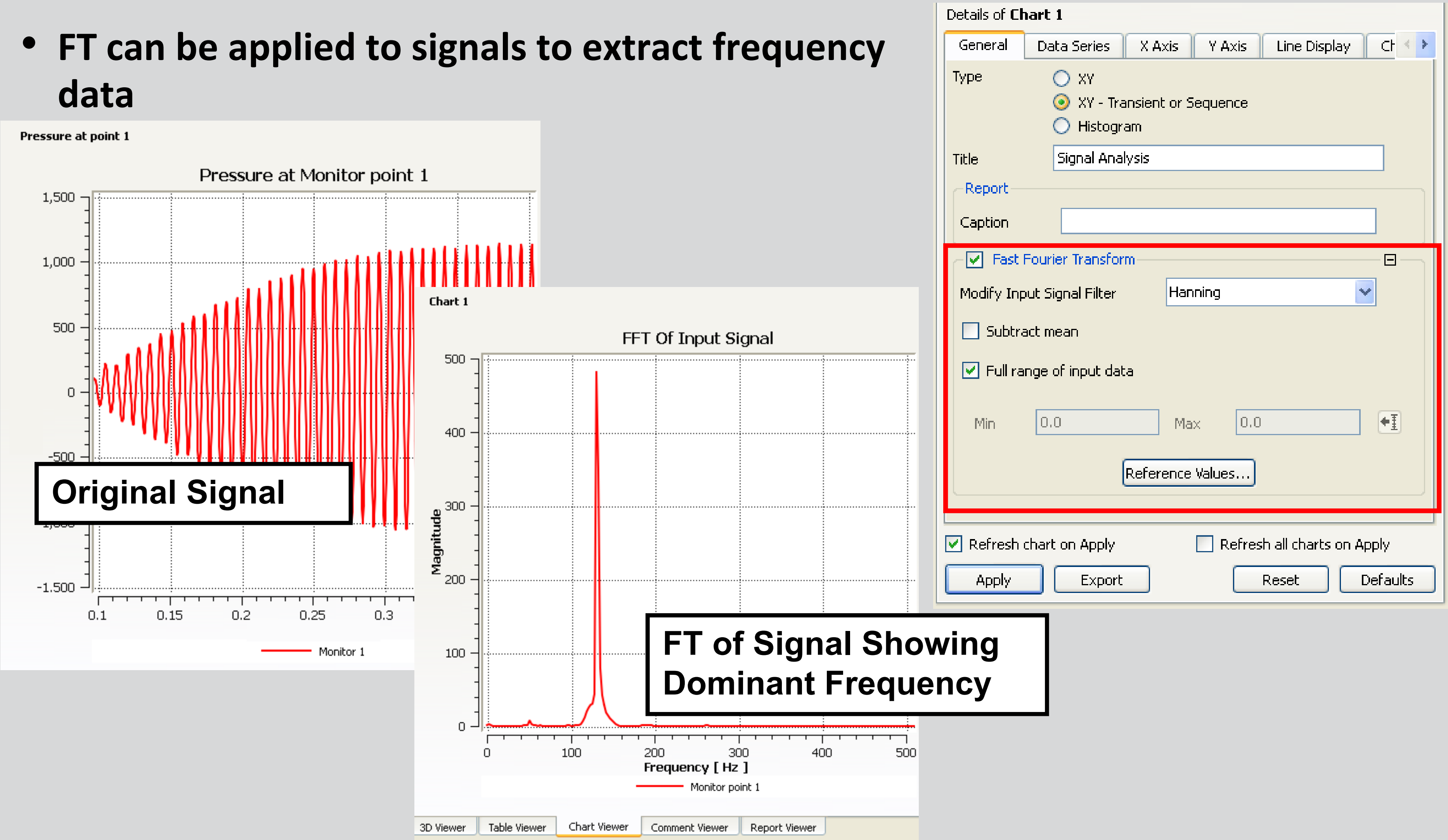
- **XY – Transient or Sequence**
  - Plots an expression (usually Time) versus a variable at a point locator
  - Typically used to show the transient variation of a variable at a point
    - Unless the data from every time step is saved and loaded into CFD-Post, create a point surface in FLUENT and write the data at every time step
      - If desired, the data in the FLUENT monitor output file can be plotted in a CFD-Post chart





CFD-Post: Fourier Transform

- **FT can be applied to signals to extract frequency data**



Introduction | Unsteady Flow | Time Step | Setup | **Post-Processing** | Summary

Summary

- **No matter what solver is being used**
  - **The time step size will be determined by the minimum of**
    - **The value at which the solution will converge**
    - **The value needed to resolve mean flow physical time scales (e.g. vortex shedding frequency given by Strouhal number) and/or turbulent eddies (Courant number <= 1)**
  - **The solution must converge at every time step**
    - **Non-convergence within the very first steps may be acceptable when there is a non-physical initial condition**
    - **If the solution is not converging, it is almost always more efficient to reduce the time step size**
  - **Solution monitors are an important tool for ensuring the solution is correct**
    - **Watch out for physically unrealistic behavior of monitored variables**
  - **Second order temporal discretization is almost always preferred**

Appendix

Non-iterative Time Advancement

ITA versus NITA

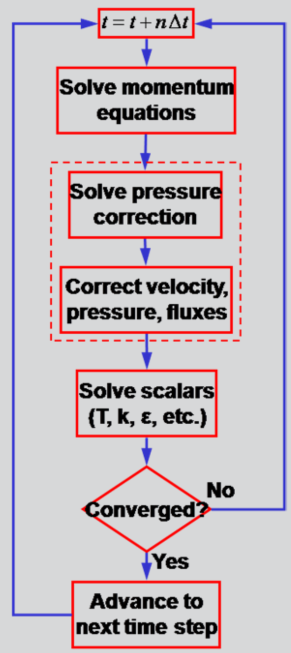
NITA runs about <u>two to ten times </u>as fast as ITA scheme

- NITA scheme reduce the splitting error to $O(\Delta t^2)$ by using sub-iterations per time step

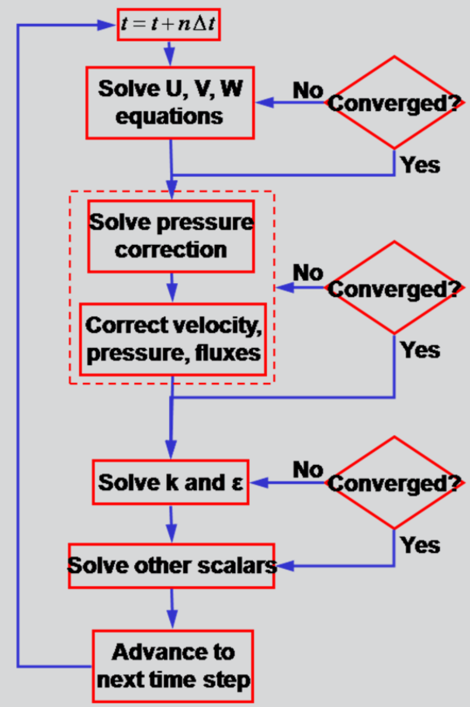
Overall time-discretization error for 2nd-order scheme: $O(\Delta t^2)$	=	Truncation error: $O(\Delta t^2)$	+	Splitting error (due to eqn segregation): $O(\Delta t^n)$

- Two flavors of NITA schemes available

- PISO (NITA/PISO)
- Fractional-step method (NITA/FSM)
  - About 20% cheaper than NITA/PISO on a per time-step basis



ITA



NITA

– **<u>Limitations:</u>**
- Available with Pressure-based Solvers only
- NITA schemes are not available for multiphase (except VOF), reacting flows, radiation models, porous media, fan models, etc.

Consult Fluent User's Guide for additional details