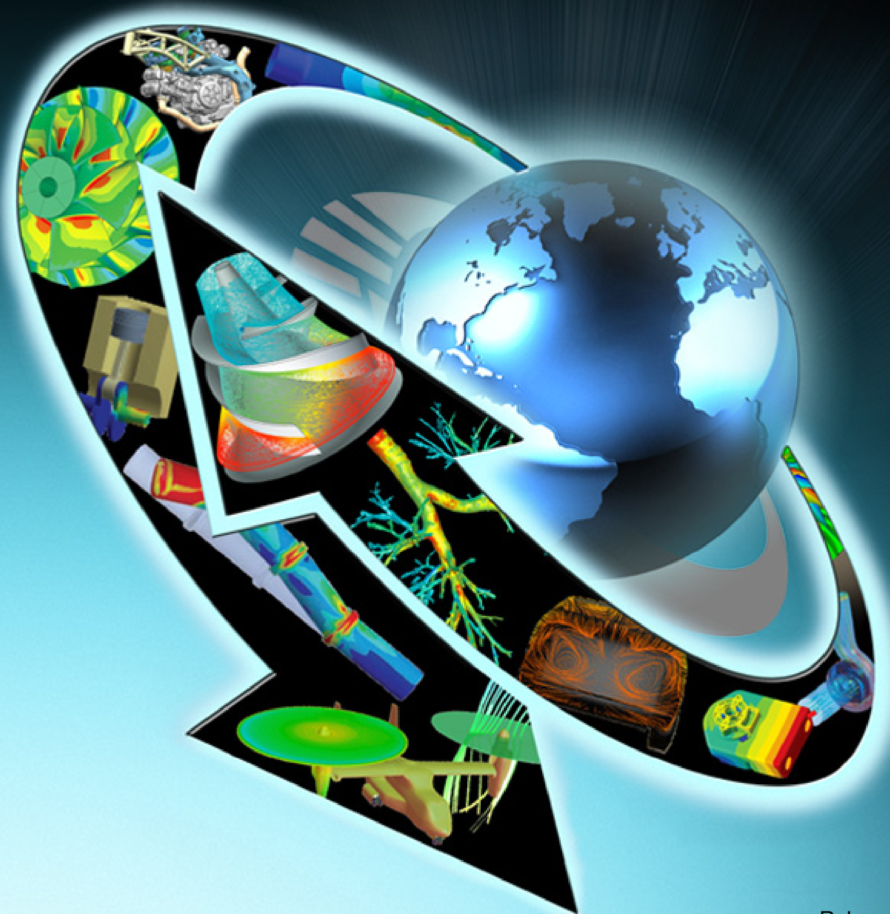
ANSYS

Customer Training Material

Lecture 10

Transient Flow Modeling



Introduction to ANSYS FLUENT

Release 13.0
December 2010

Motivation

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

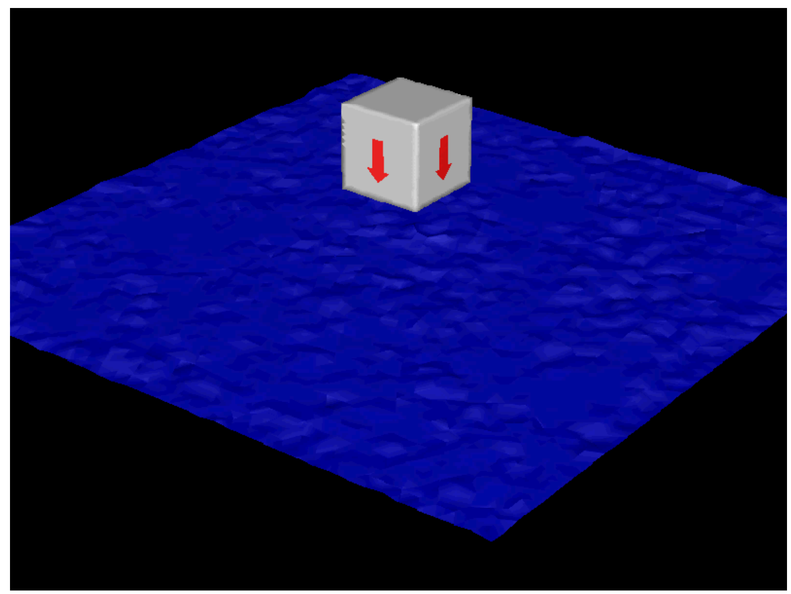
- In CFD, steady-state methods are preferred
  - Lower computational cost
  - Easier to postprocess and analyze

- Many applications require resolution of transient 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
  - Transient Heat Transfer – transient heating and cooling
  - Many more.

Origins of Transient Flow

- Naturally occurring transients
  - Transient 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 transients
  - Time-dependent boundary conditions, source terms drive the transient flow field
  - Examples: pulsing flow in a nozzle, rotor-stator interaction in a turbine stage

Kelvin-Helmholtz Cloud Instability



Buoyant Box Falling Into a Pool of Water

Transient 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
    - Etc.

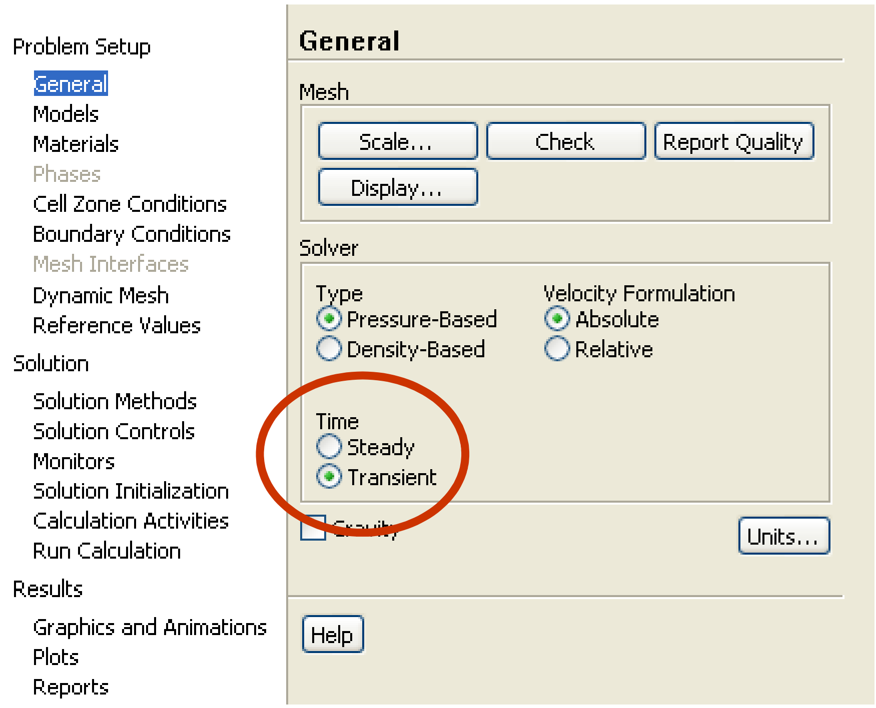
- Extract quantities of interest
  - Natural frequencies (e.g. Strouhal Number)
  - Time-averaged and/or RMS values
  - Time-related parameters (e.g. time required to cool a hot solid, residence time of a pollutant)
  - Spectral data – fast Fourier transform (FFT)

Transient Flow Modeling Workflow

- Enable the transient solver.
- 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.
- Prescribe initial conditions
  - Best to use a physically realistic initial condition, such as a steady solution.
- Assign solver settings and configure solution monitors.
- Configure animations and data output/sampling options
- Select time step and max iterations per time step
- Prescribe the number of time steps.
- Run the calculations (iterate)

Enabling the Transient Solver

- To enable the transient solver, select the Transient button on the General problem setup form:



- Before performing iterations, you will need to set some additional controls.
  - Solver settings
  - Animations
  - Data export / Autosave options

Selecting the Transient Time Step Size

- Time step size, Δt, is set in the Run Calculation form.
  - Δt must be small enough to resolve time-dependent features; make sure convergence is reached within the number of Max Iterations per Time Step
  - The order or magnitude of an appropriate time step size can be estimated as:

$$\Delta t \approx \frac{\text{Typical cell size}}{\text{Characteristic flow velocity}}$$

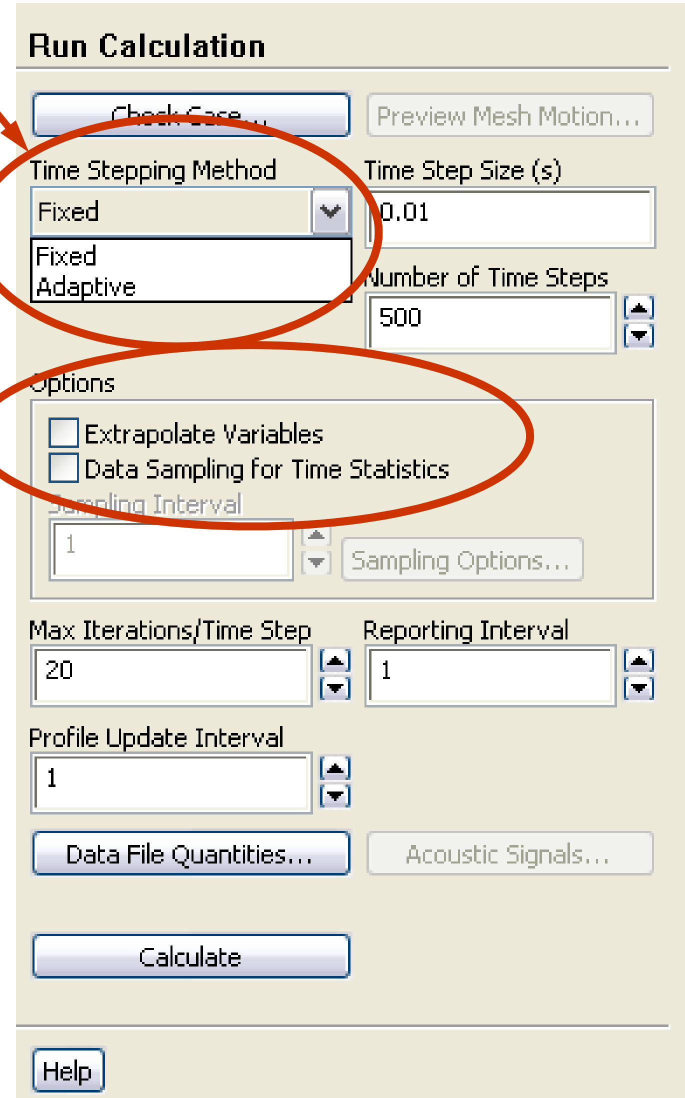
  - Time step size estimate can also be chosen so that the transient characteristics of the flow can be resolved (e.g. flow within a known period of fluctuations)

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

- The PISO scheme may aid in accelerating convergence for many transient flows (set in the Solution Methods form).

Transient Flow Modeling Options

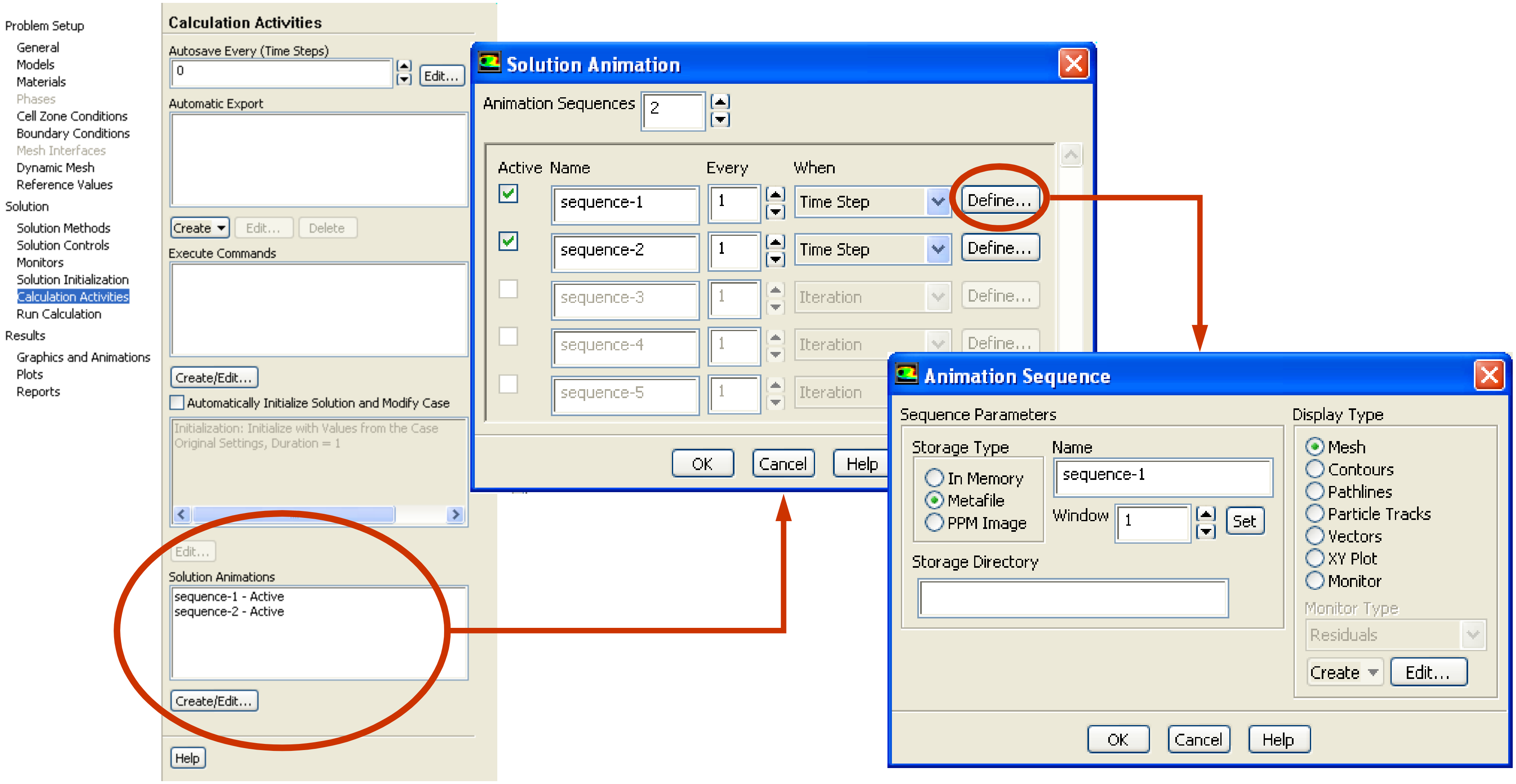
- Adaptive Time Stepping
  - Automatically adjusts time-step size based on local truncation error analysis
  - Customization possible via UDF
- Time-averaged statistics
  - Particularly useful for LES turbulence calculations



- For the density-based solver, the Courant number defines:
  - The global time step size for density-based explicit solver.
  - The pseudo time step size for density-based implicit solver
    - Real time step size must still be defined in the Iterate panel

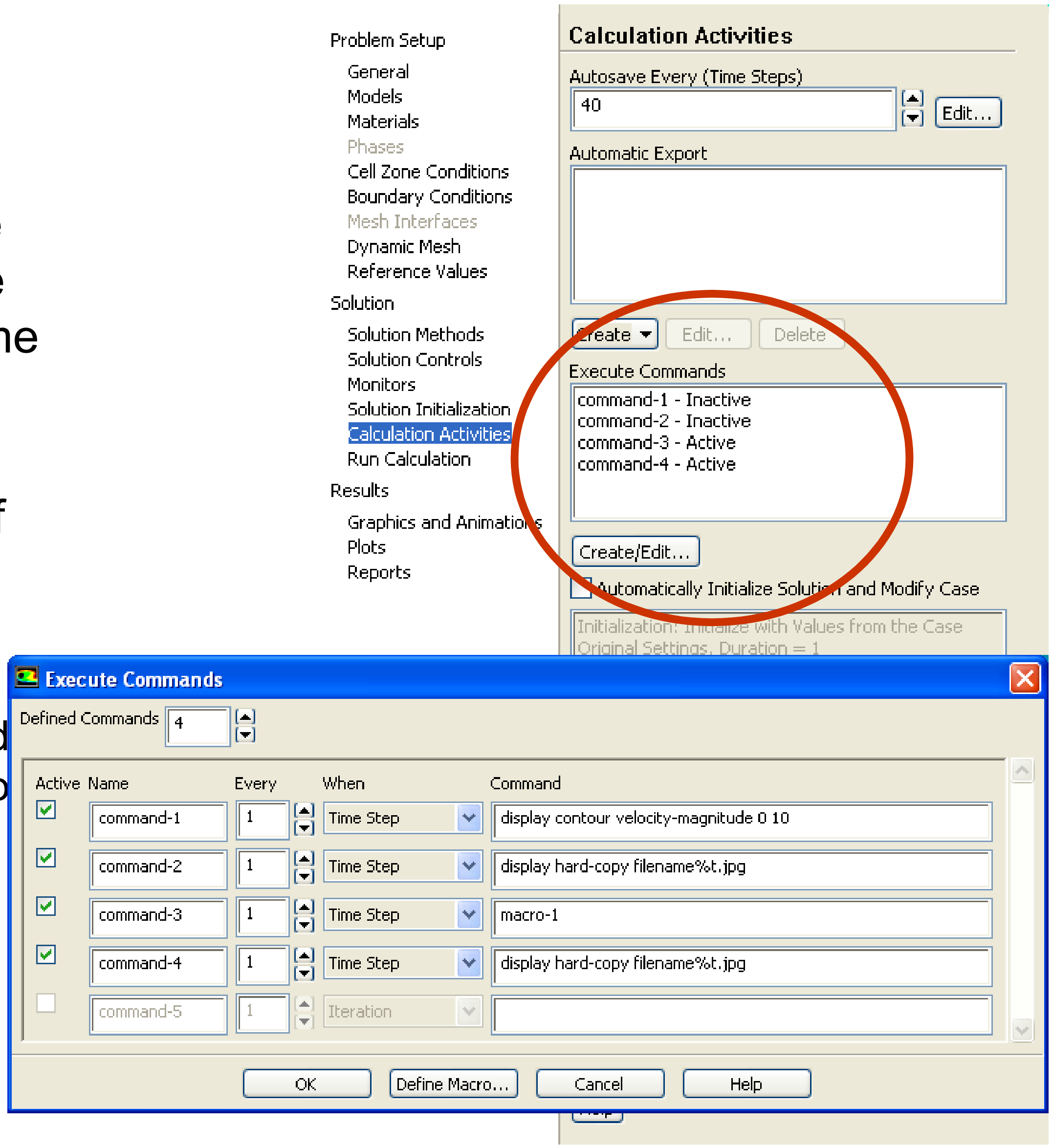
Transient Flow Modeling – Animations

- You must set up any animations BEFORE performing iterations.
  - Animation frames are written/stored on-the-fly during calculations.
  - Settings need defining each time (the setup is not stored in the case file)



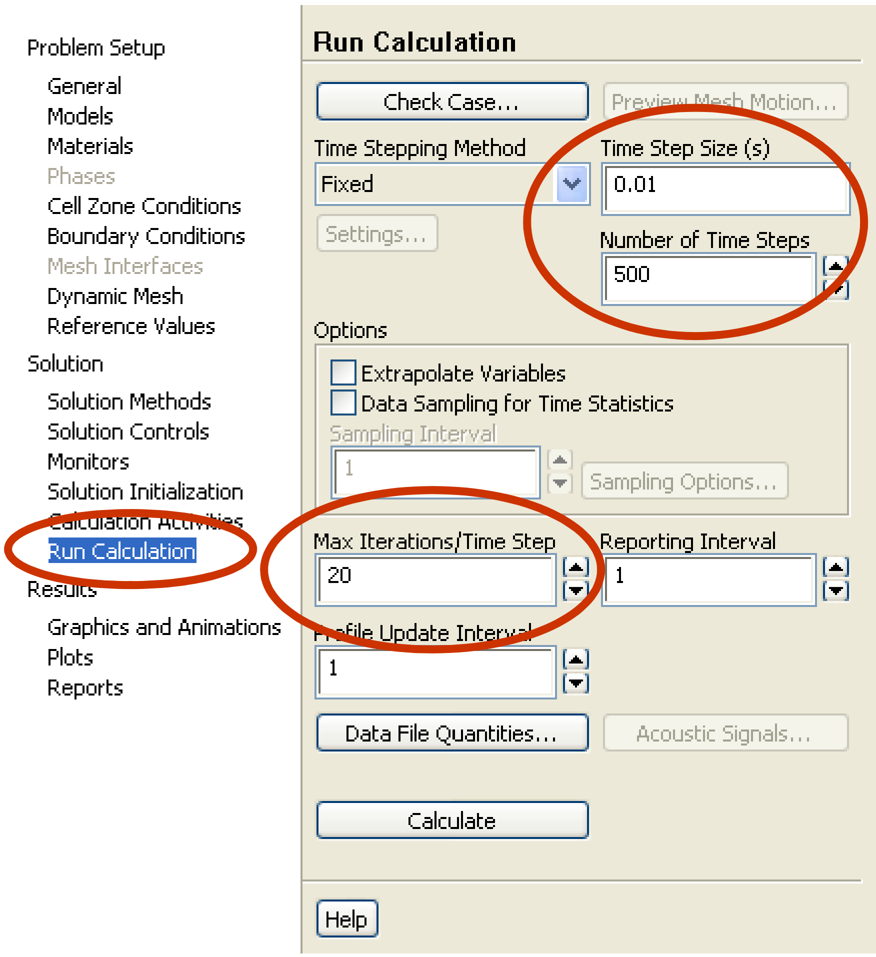
Creating Animations – Alternate Method

- 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.)



Performing Iterations

- The most common time advancement scheme is the iterative scheme.
  - The solver converges the current time step and then advances time.
  - Time is advanced when **Max Iterations/Time Step** is reached or convergence criteria are satisfied.
  - Time steps are converged sequentially until the **Number of Time Steps** is reached.

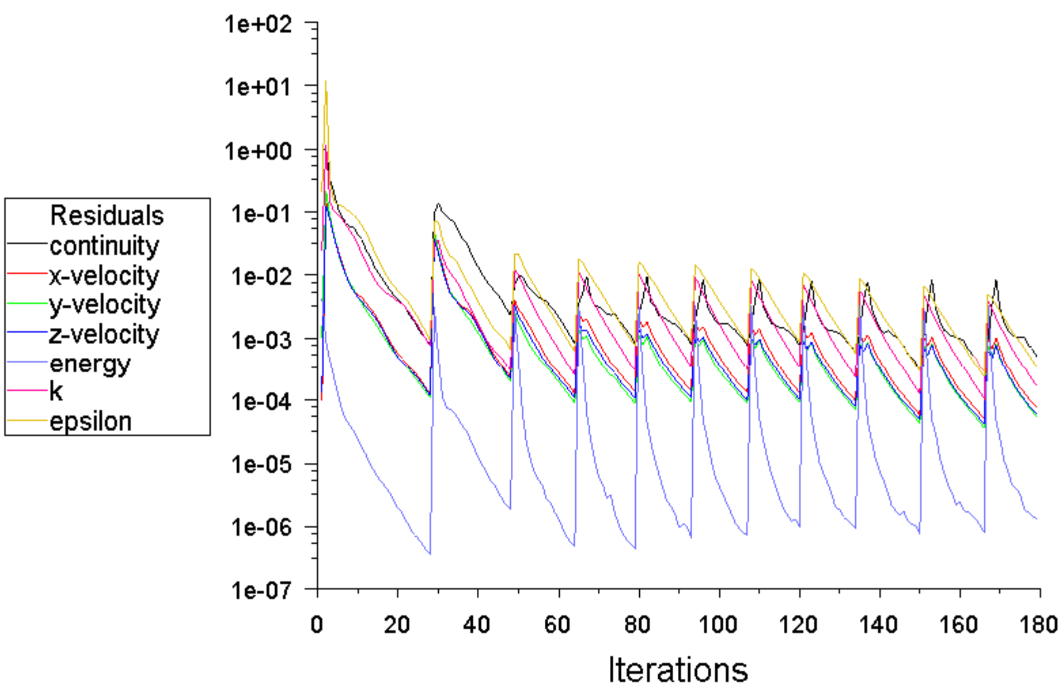


- Solution initialization defines the initial condition and it must be realistic.
  - Sets both the initial mass of fluid in the domain and the initial state of the flow field.

- For some problems, the Non-Iterative Time Advancement scheme will be faster
  - In each timestep, the separate equations (U, V, P) are each converged in isolation, rather than iterating all together.

Convergence Behavior

- Residual plots for transient simulations are not always indicative of a converged solution.
- A residual plot for a simple transient calculation is shown here.
- 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.



Tips for Success in Transient Flow Modeling

- Use PISO scheme for Pressure-Velocity Coupling – this scheme provides faster convergence for transient flows than the standard SIMPLE approach.
- Select the time step size so that the solution converges three orders of magnitude for each time step (of course, convergence behavior is problem-specific).
- 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 just as important as boundary conditions for transient problems – initial condition should always be physically realistic!
- Configure any animations you wish to see before running the calculations.