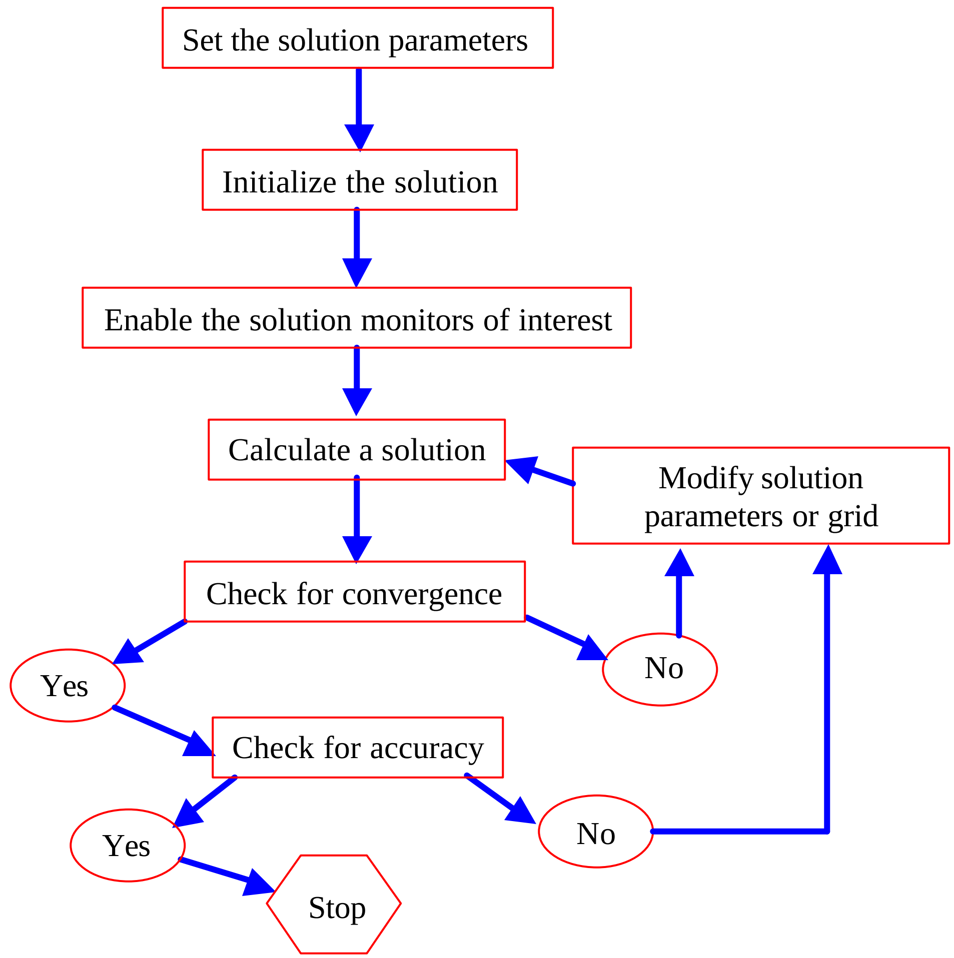
Solver Settings

Outline

- Using the Solver
  - Setting Solver Parameters
  - Convergence
    - Definition
    - Monitoring
    - Stability
    - Accelerating Convergence
  - Accuracy
    - Grid Independence
    - Adaption
- Appendix: Background
  - Finite Volume Method
  - Explicit vs. Implicit
  - Segregated vs. Coupled
  - Transient Solutions

Solution Procedure Overview

- Solution Parameters
  - Choosing the Solver
  - Discretization Schemes
- Initialization
- Convergence
  - Monitoring Convergence
  - Stability
    - Setting Under-relaxation
    - Setting Courant number
  - Accelerating Convergence
- Accuracy
  - Grid Independence
  - Adaption



Choosing a Solver

- Choices are Coupled-Implicit, Coupled-Explicit, or Segregated (Implicit)
- The Coupled solvers are recommended if a strong inter-dependence exists between density, energy, momentum, and/or species.
  - e.g., high speed compressible flow or finite-rate reaction modeled flows.
  - In general, the Coupled-Implicit solver is recommended over the coupled-explicit solver.
    - Time required: Implicit solver runs roughly twice as fast.
    - Memory required: Implicit solver requires roughly twice as much memory as coupled-explicit *or* segregated-implicit solvers!
  - The Coupled-Explicit solver should only be used for unsteady flows when the characteristic time scale of problem is on same order as that of the acoustics.
    - e.g., tracking transient shock wave
- The Segregated (implicit) solver is preferred in all other cases.
  - Lower memory requirements than coupled-implicit solver.
  - Segregated approach provides flexibility in solution procedure.

Initialization

- Iterative procedure requires that all solution variables be initialized before calculating a solution.

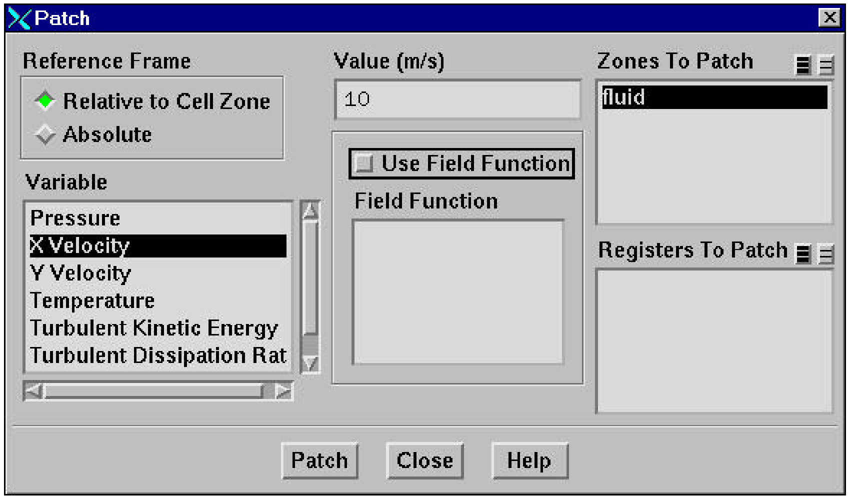
  Solve → Initialize → Initialize...

  - Realistic ‘guesses’ improves solution stability and accelerates convergence.
  - In some cases, **correct** initial guess is required:
    - Example: high temperature region to initiate chemical reaction.

- “Patch” values for individual variables in certain regions.

  Solve → Initialize → Patch...

  - Free jet flows
    (patch high velocity for jet)
  - Combustion problems
    (patch high temperature for ignition)



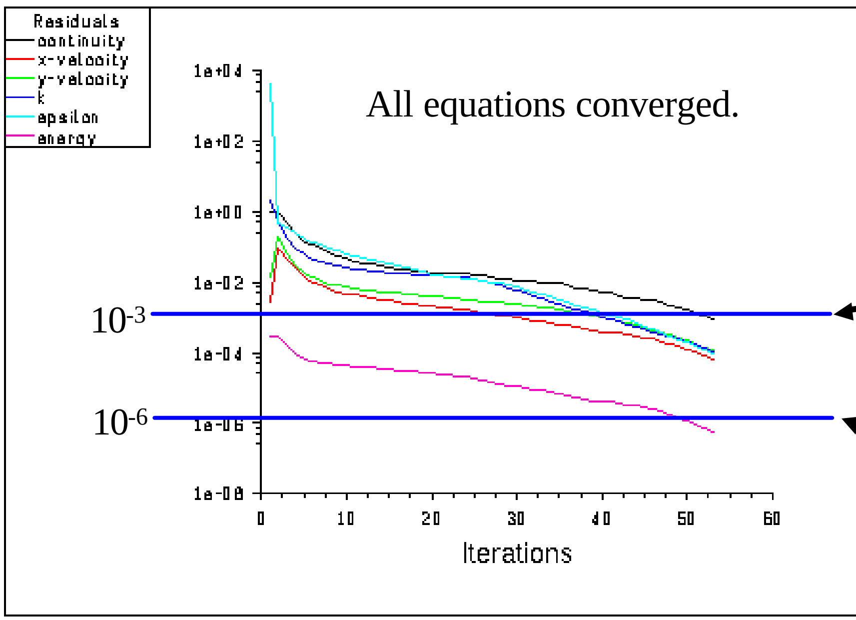
Convergence

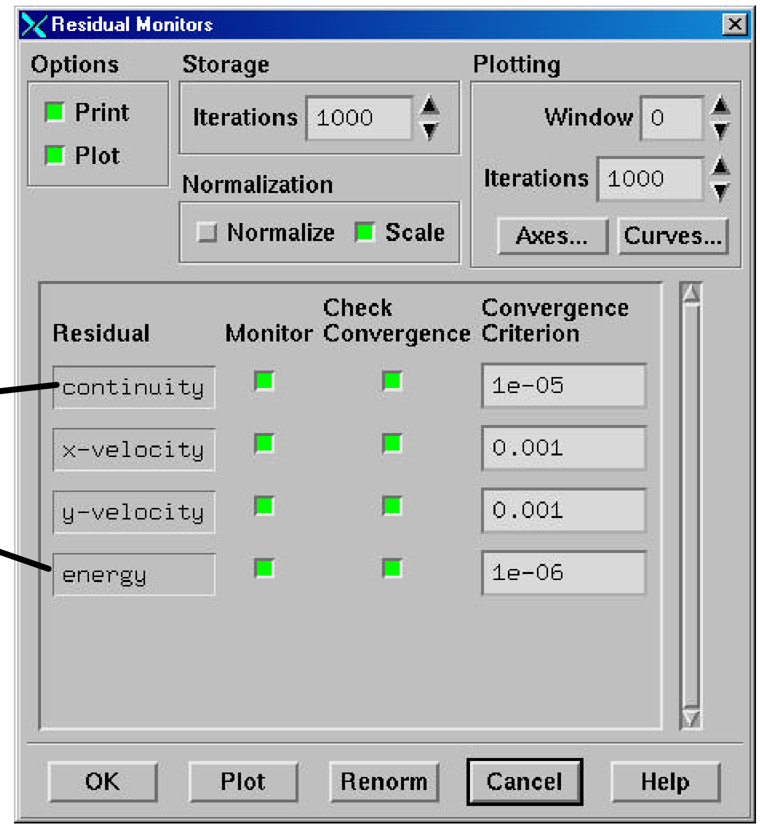
- At convergence:
  - All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells *to a specified tolerance*.
  - Solution no longer changes with more iterations.
  - Overall mass, momentum, energy, and scalar balances are obtained.
- Monitoring convergence with residuals:
  - Generally, a decrease in residuals by 3 orders of magnitude indicates at least qualitative convergence.
    - Major flow features established.
  - Scaled energy residual must decrease to 10^{-6} for segregated solver.
  - Scaled species residual may need to decrease to 10^{-5} to achieve species balance.
- Monitoring quantitative convergence:
  - Monitor other variables for changes.
  - Ensure that property conservation is satisfied.

Convergence Monitors: Residuals

- Residual plots show when the residual values have reached the specified tolerance.

Solve ⟶ Monitors ⟶ Residual...



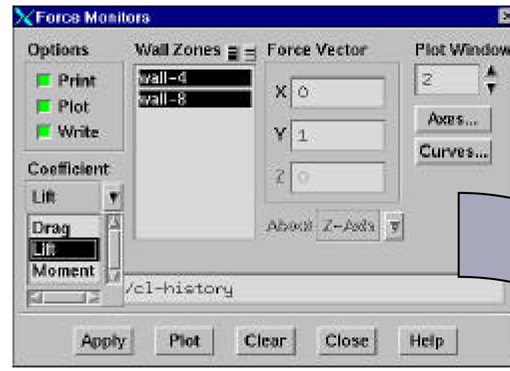


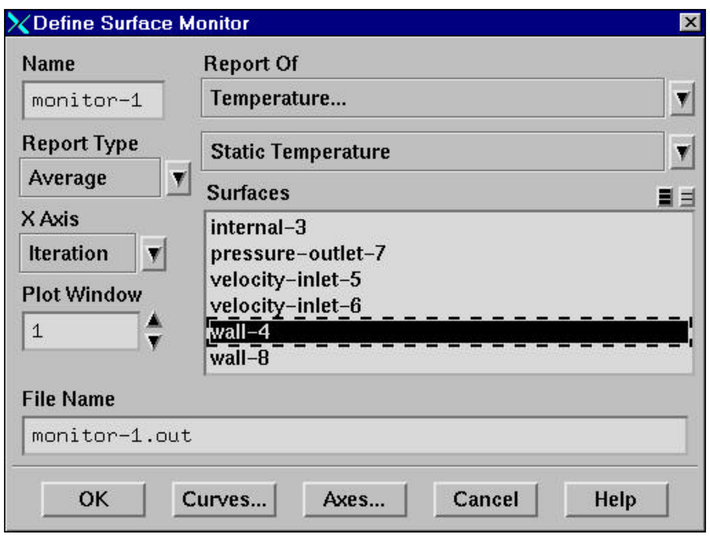
Convergence Monitors: Forces/Surfaces

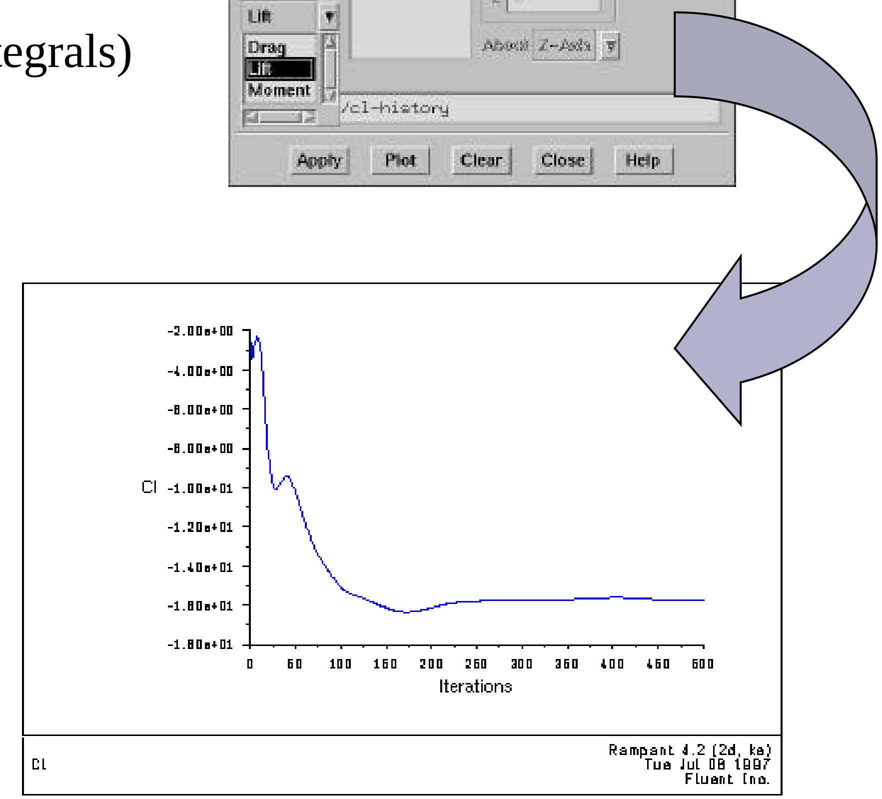
- In addition to residuals, you can also monitor:
  - Lift, drag, or moment

    Solve → Monitors → Force...
  - Variables or functions (e.g., surface integrals) at a boundary or any defined surface:

    Solve → Monitors → Surface...



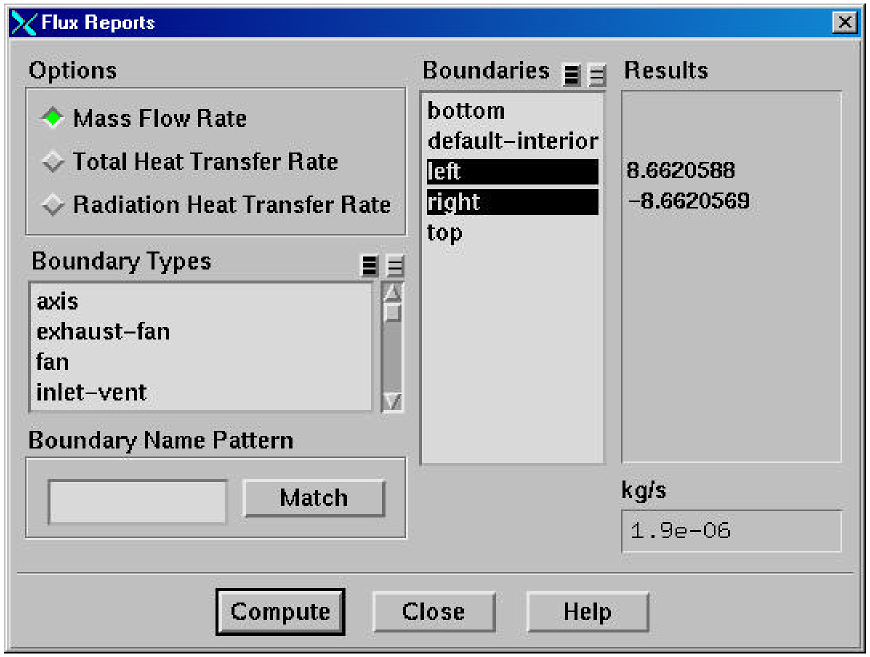




Checking for Property Conservation

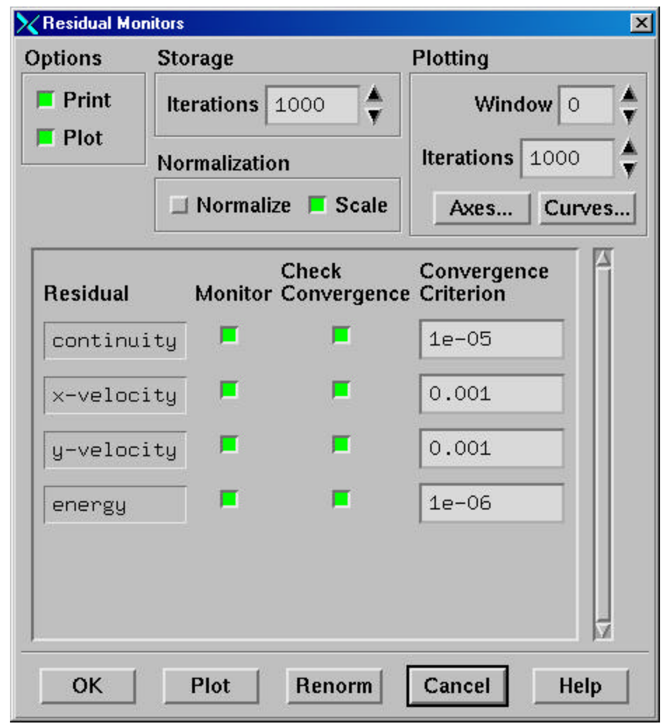
- In addition to monitoring residual and variable histories, you should also check for overall heat and mass balances.
  - At a minimum, the net imbalance should be less than 1% of smallest flux through domain boundary.

Report → Fluxes...



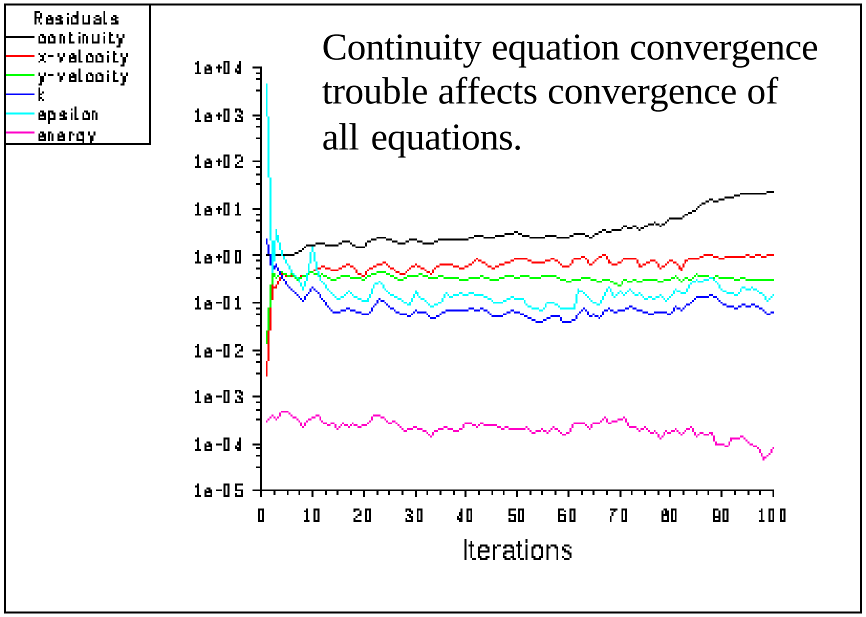
Decreasing the Convergence Tolerance

- If your monitors indicate that the solution is converged, but the solution is still changing or has a large mass/heat imbalance:
  - Reduce Convergence Criterion or disable Check Convergence.
  - Then calculate until solution converges to the new tolerance.



Convergence Difficulties

- Numerical instabilities can arise with an ill-posed problem, poor quality mesh, and/or inappropriate solver settings.
  - Exhibited as increasing (diverging) or “stuck” residuals.
  - Diverging residuals imply increasing imbalance in conservation equations.
  - Unconverged results can be misleading!
- Troubleshooting:
  - Ensure problem is well posed.
  - Compute an initial solution with a first-order discretization scheme.
  - Decrease under-relaxation for equations having convergence trouble (segregated).
  - Reduce Courant number (coupled).
  - Re-mesh or refine grid with high aspect ratio or highly skewed cells.

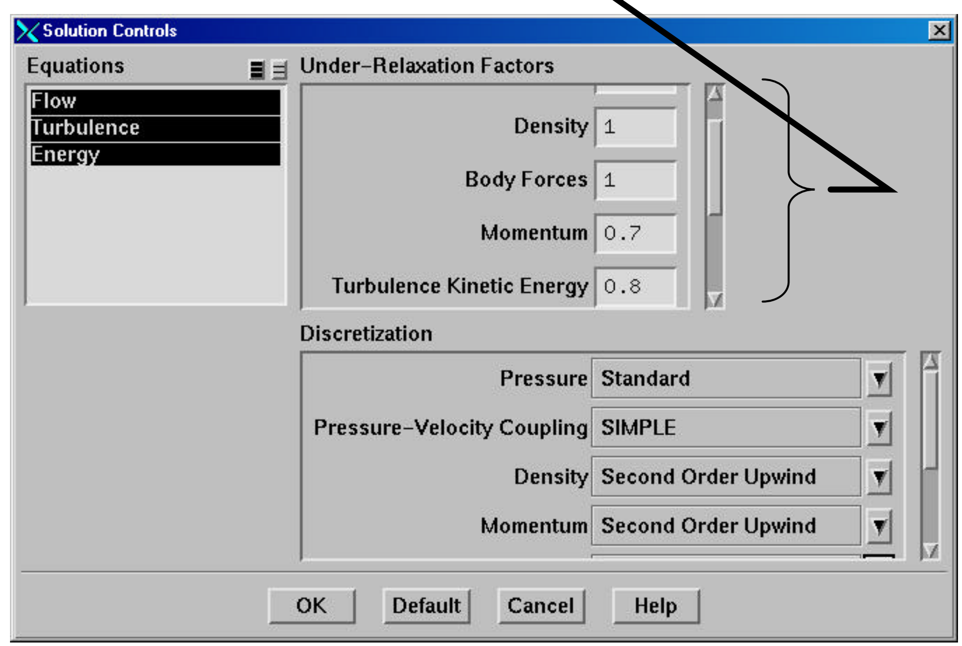


Modifying Under-relaxation Factors

- Under-relaxation factor, *a*, is included to stabilize the iterative process for the segregated solver.
- Use default under-relaxation factors to start a calculation.

  Solve → Controls → Solution...
- Decreasing under-relaxation for *momentum* often aids convergence.
  - Default settings are aggressive but suitable for wide range of problems.
  - ‘Appropriate’ settings best learned from experience.

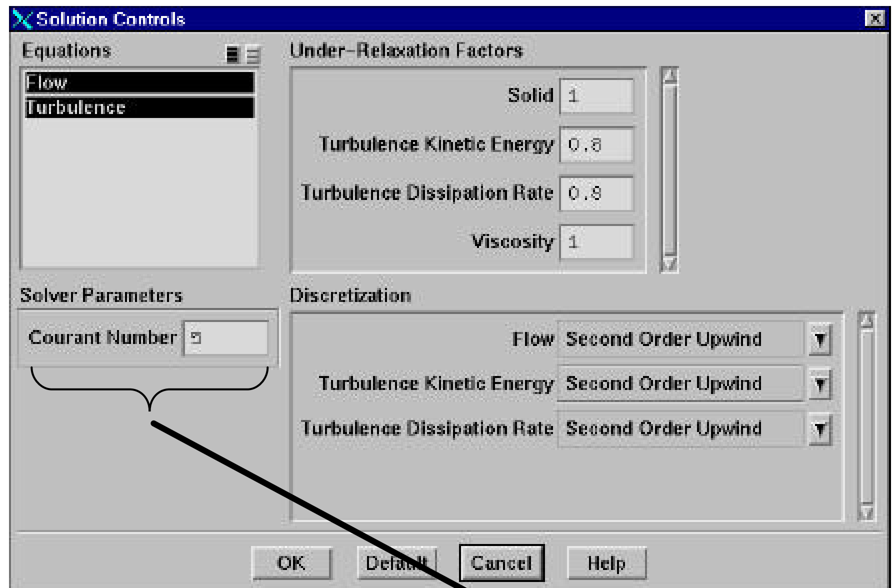
$$f_p = f_{p,old} + a \Delta f_p$$



- For coupled solvers, under-relaxation factors for equations *outside* coupled set are modified as in segregated solver.

Modifying the Courant Number

- Courant number defines a 'time step' size for steady-state problems.
  - A transient term is included in the coupled solver even for steady state problems.
- For coupled-explicit solver:
  - Stability constraints impose a maximum limit on Courant number.
    - Cannot be greater than 2.
      - Default value is 1.
    - Reduce Courant number when having difficulty converging.
- For coupled-implicit solver:
  - Courant number is not limited by stability constraints.
    - Default is set to 5.



$$\Delta t = \frac{(\mathrm{CFL})\Delta x}{u}$$

Accelerating Convergence

- Convergence can be accelerated by:
  - Supplying good initial conditions
    - Starting from a previous solution.
  - Increasing under-relaxation factors or Courant number
    - Excessively high values can lead to instabilities.
    - Recommend saving case and data files before continuing iterations.
  - Controlling multigrid solver settings.
    - Default settings define robust Multigrid solver and typically do not need to be changed.

Accuracy

- A converged solution is not necessarily an accurate one.
  - Solve using 2nd order discretization.
  - Ensure that solution is grid-independent.
    - Use adaption to modify grid.
- If flow features do not seem reasonable:
  - Reconsider physical models and boundary conditions.
  - Examine grid and re-mesh.

Mesh Quality and Solution Accuracy

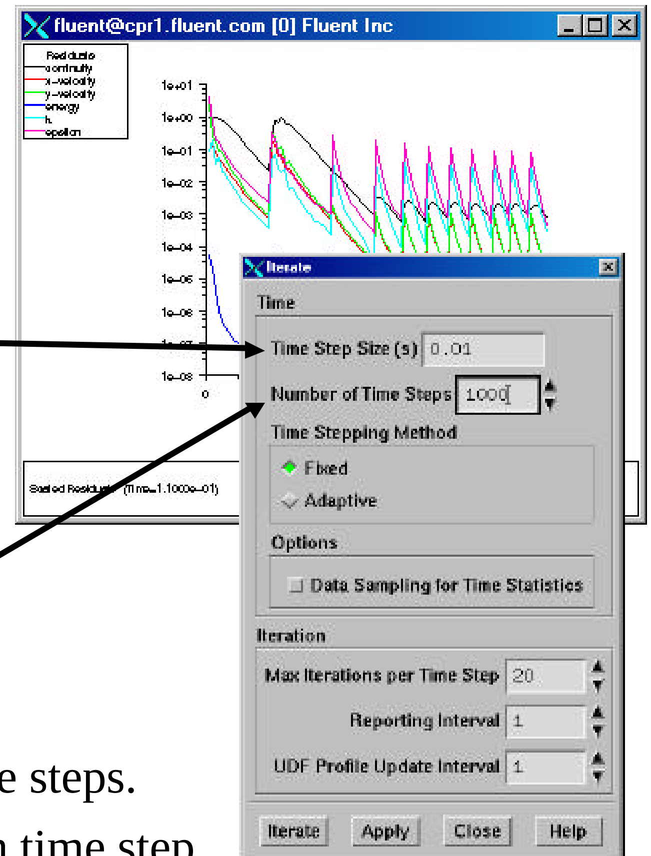
- Numerical errors are associated with calculation of cell gradients and cell face interpolations.
- These errors can be contained:
  - Use higher order discretization schemes.
  - Attempt to align grid with flow.
  - Refine the mesh.
    - Sufficient mesh density is necessary to resolve salient features of flow.
      - Interpolation errors decrease with decreasing cell size.
    - Minimize variations in cell size.
      - Truncation error is minimized in a uniform mesh.
      - Fluent provides capability to adapt mesh based on cell size variation.
    - Minimize cell skewness and aspect ratio.
      - In general, avoid aspect ratios higher than 5:1 (higher ratios allowed in b.l.).
      - Optimal quad/hex cells have bounded angles of 90 degrees
      - Optimal tri/tet cells are equilateral.

Determining Grid Independence

- When solution no longer changes with further grid refinement, you have a "grid-independent" solution.
- Procedure:
  - Obtain new grid:
    - Adapt
      - Save original mesh before adapting.
        - If you know where large gradients are expected, concentrate the original grid in that region, e.g., boundary layer.
      - Adapt grid.
        - Data from original grid is automatically interpolated to finer grid.
    - file → write-bc and file → read-bc facilitates set up of new problem
    - file → reread-grid and File → Interpolate...
  - Continue calculation to convergence.
  - Compare results obtained w/different grids.
  - Repeat procedure if necessary.

Unsteady Flow Problems

- Transient solutions are possible with both segregated and coupled solvers.
  - Solver iterates to convergence at each time level, then advances automatically.
  - Solution Initialization defines initial condition and must be realistic.
- For segregated solver:
  - Time step size, Δt, is input in Iterate panel.
    - Δt must be small enough to resolve time dependent features and to ensure convergence within 20 iterations.
    - May need to start solution with small Δt.
  - Number of time steps, N, is also required.
    - N*Δt = total simulated time.
  - To iterate without advancing time step, use ‘0’ time steps.
  - PISO may aid in accelerating convergence for each time step.



Summary

- Solution procedure for the segregated and coupled solvers is the same:
  - Calculate until you get a converged solution.
  - Obtain second-order solution (recommended).
  - Refine grid and recalculate until grid-independent solution is obtained.
- All solvers provide tools for judging and improving convergence and ensuring stability.
- All solvers provide tools for checking and improving accuracy.
- Solution accuracy will depend on the appropriateness of the physical models that you choose and the boundary conditions that you specify.