Natural Convection & Transient Natural Convection

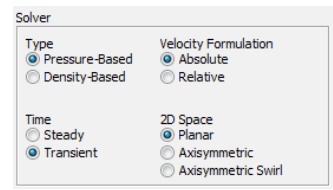
For comparison we first need to simulate a static natural convection scenario. The Boussinesq flow model from lab #3 will satisfy this requirement, as it also has a compelling transient development.

We will simulate the same geometry with the same boundary conditions, but taking the hybrid initialization as the *initial condition* (rather than the *initial guess*).

Setup

- Make a copy of the Boussinesq flow project and start Fluent from that copy.
- In the *General* menu, change the *Time* option to “Transient”.

Solution Methods



- Note that a new option has appeared in the solution methods: *Transient Formulation*. Examine the available options in the pull-down menu.
- For now, we accept the “First-Order Implicit” scheme.

Initialization/Run Calculation

- Initialize the solution (Hybrid) and go to the *Run Calculation* menu.
- Set *Time Step Size* 1, *Number of Time Steps* 20, *Max Iterations/Time Step* 20
- *Calculate*
  The viewport will change to the residual against the iteration number. The sawtooth behavior arises when the simulation starts a time step, reducing the residual by iterations, and then starting a new time step.

Postprocessing

- After the calculation is completed, plot a temperature contour of the domain.
  The contour tells us that convection has not yet begun. The time step size of 1 second happens to be small relative to the evolution of this problem.

Run Calculation

- Change the time step size to 10 seconds and run for another 20 time steps (without re-initializing the solution). *Fluent will warn about changed settings; select “Use setting changes for current calculation only”.*
- Keep an eye on the magnitude of residuals. They are all larger than for the smaller time step.

Postprocessing

- Plot a temperature contour after the calculation. It should show a bit more action this time.
  We have expedited the evolution of solution at the price of reducing accuracy. Such tradeoff holds for any transient simulation.

Convergence Criteria

Extended simulations need to be watched using monitors to verify numerical stability and simulation progress.

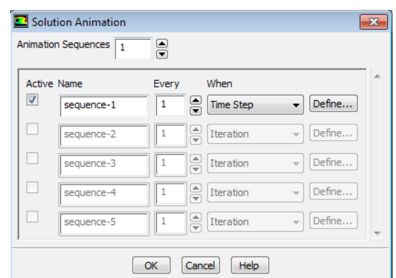
- Go to *Monitors* and click *Residuals—Print, Plot.*
- Change the convergence criterion from “Absolute” to “Relative”.
  If the “absolute” criterion is chosen, the residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a timestep.
  If “relative” is selected, the residual of an equation at an iteration of a timestep is compared with the residual at the start of the timestep. If the ratio of the two residuals is less than a

user-specified value, that equation is deemed to have converged for a timestep.
In many transient flows, the absolute convergence criterion could be too stringent causing a large number of iterations per timestep. For example, the scaling of the continuity equation is based on the value of the continuity residual in the first five iterations. The scaling factor could be low if the initial continuity residual is small and thus the scaled residual could fail to meet the absolute convergence criterion. With the relative convergence criterion, convergence is checked by comparing the residual at an iteration of a timestep with the residual at the beginning of the timestep and hence this problem is alleviated. Thus I recommend relative convergence criterion for unsteady simulations.
In addition to the residual plot, we need to understand the evolution of the solution so that we can identify convergence to the steady state behavior.

- Click *Create* under the *Surface monitor* in the *Monitors* section.
- In the *Surface Monitor* window, check the *Plot* box and output the monitor to window 2. Set *X Axis* to *Time Step*, and *Get Data Every 1 Time Step*.
- Select the *Report Type* to be *Facet Maximum*, and the *Field Variable* to *Velocity Magnitude*.
- Click on the *New Surface* pulldown and make a new *Line/Rake* that is the vertical centerline across the domain.
- Set the monitor to watch this surface.

The monitor will output the maximum velocity along the line at every time step.
Setting Up Animation

- To watch the entire temperature field evolve with time, you need the view the *Calculation Activities* menu and create a *Solution Animation*.
- In the menu that appears, set the *Animation Sequences* to 1, change *When* to "Time Step", and hit the *Define* button.
- In the next window that appears, change *Window* to 3, and *Set*.
- Next, change the *Display Type* to *Contours*, and in the window that appears, change *Contours of* to "Temperature", click *Display* and close the window.
- Click *OK* to finish the definition; "active" should be checked.



Run Calculation/Postprocessing

- Initialize; run the calculation with a time step of 300 seconds for 50 time steps.
- At the top of the *Viewport*, click on the pulldown menu that says "1: scaled residuals" and change it to "2: convergence history".

As the simulation progresses, this plot will show you the value of the velocity along the centerline. Watching this plot will show us when the points on that line have reached a value that no longer changes with time.
This simulation is expected to take some time, so please signal the instructor when you are done to this point and the lecture will proceed.

- After the calculation, go to the *Graphics and Animations* menu, and set up a *Solution Animation Playback*.
- You will see the sequence you just created become available. Hit the *Play* button.

Compare your final results with the time-averaged steady-state results calculated in lab #3. How similar are they? Host your animation on Box and send a link to `me-498admin@illinois.edu`.