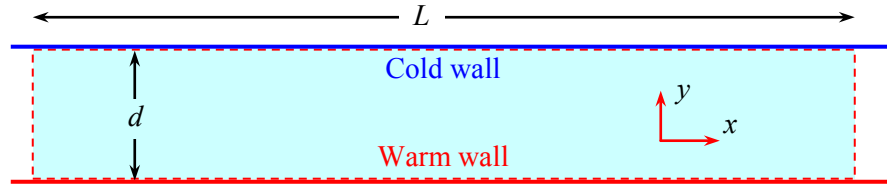


# ANSYS-FLUENT – Instability: The Bénard problem

Author: John M. Cimbala, Penn State University  
Latest revision: 25 February 2016

## Introduction and Instructions:

This learning module contains abbreviated instructions for solving the free convection problem between two parallel horizontal plates, as sketched, with the bottom wall hotter than the top wall. The CFD program, FLUENT, is used as the solver. The solution illustrates the Bénard instability, and Bénard cells should form. It is assumed that a simple rectangular mesh has been generated via ANSYS Workbench or some other mesh generation software.



**Note:** If you are using ANSYS Workbench to generate your mesh, there are some issues with specifying periodic boundary conditions that can be recognized properly by FLUENT. Here is a work-around, thank to Erik Brown:

1. Create your 2D geometry then go into the Mesher.
2. Create named selections for each of the four sides (top, bottom, left, right).
3. Mesh the left and right sides with the same mesh control (you can select both by using the control key and clicking), and mesh the top, bottom, and face as you normally would.
4. Exit the mesher and in the boundary condition menu of setup, set both your left and right sides to the “interface” boundary condition.
5. Go to the “Define” drop down menu at the top and then select Mesh Interfaces.
6. In the “Create/Edit Mesh Interfaces” GUI, select the right and left named sides (one in each of the two boxes on the right of the menu), name the new interface by typing a name (call it whatever you like) in the top left box and then click on “Periodic Boundary Condition” in the box to the bottom left and be sure “Translational” is selected below that.
7. Click “Create” and the two interfaces should reduce to one new “Periodic” boundary condition, with one being a 'shadow' of the other.

## Log on and launch FLUENT:

1. Log on to a computer that has access to the FLUENT software.
2. Start FLUENT, and select 2d, the two-dimensional, double precision option. Run. [Note: If you are using a Unix or Linux machine, go to the desired directory, and enter the command fluent 2d & (the & symbol lets FLUENT run in background mode so that the Unix shell is still usable).]
3. After a few seconds, the main *FLUENT* window should appear on your screen.

## Read the grid points and geometry of the flow domain:

1. Select File-Read-Case. Browse as necessary to locate the mesh file that you previously created. Highlight this file (click on it), and OK. FLUENT will read in the grid geometry and mesh, and the word “Done” should appear. Some warnings may appear about the axis boundary condition.
2. Check the validity of the grid: Grid-Check. If the grid is valid, no errors should appear. If there are errors, you may have done something wrong in the grid generation, and will have to go back and regenerate the grid.
3. Look at the grid. Display-Grid-Display. A new graphical display window opens up showing the grid. If this window is too big or too small, re-scale it by dragging the edges of the window. It is best if the graphical display window is small enough that both it and the *FLUENT* window are visible simultaneously. The *FLUENT* window and/or the graphical display window may need to be moved to accomplish this.
4. The graphical display can be zoomed-in or zoomed-out with the *middle* mouse button. If you start on the *lower left* and draw a rectangle with the middle mouse button towards the upper right, the display zooms in on what is included in the rectangle. Meanwhile, the *left* mouse button can be used to drag the image to a new location. If you draw a rectangle *backwards* with the middle mouse button, from right to the left, it zooms out. Zoom in if necessary until the grid is shown nicely in the window. Close the *Grid Display* window; the display itself will remain.

## Set up the appropriate models and operating conditions:

1. Turn on heat transfer: Define-Models-Energy, check Energy Equation. OK.

2. Turn on gravity in the negative  $y$ -direction: Define-Operating Conditions. Turn on Gravity. Enter “-9.807” ( $\text{m/s}^2$ ) in the  $Y$  text box.
3. Notice the default operating temperature under *Boussinesq Parameters*. Set this temperature to  $60^\circ\text{C}$  ( $333.15\text{ K}$ ), which will be the temperature of the lower wall. It is tempting to specify an operating density here too, but the help panels say that it is not necessary to specify operating density when using the Boussinesq approximation. OK.

**Define the fluid as liquid water with the Boussinesq approximation:**

1. The default fluid is air, which must be changed to water. In the main *FLUENT* window, Define-Materials-FLUENT Database. Select water-liquid from the *Fluid Materials* drop down list. Copy. Close.
2. Now turn on the Boussinesq model: In the *Materials* window, make sure water-liquid is selected from the *FLUENT Fluid Materials* drop down list. In the section of this window called *Properties*, open the drop down list for *Density*. Change from constant (the default) to boussinesq.
3. For some reason (I think this is a bug in FLUENT), the default reference density becomes zero. Change this to the appropriate reference density for water at  $60^\circ\text{C}$ , which is  $983.3\text{ kg/m}^3$ .
4. Similarly, scroll down through the other fluid properties, and enter the appropriate values at our reference temperature ( $c_p = 4185\text{ J/kg}\cdot\text{K}$ ,  $k = 0.654\text{ W/m}\cdot\text{K}$ ,  $\mu = 4.67\text{e-}4\text{ kg/m}\cdot\text{s}$ , and  $\alpha = 5.17\text{e-}4\text{ 1/K}$ ). *Note*: FLUENT does not automatically update these properties at our specified reference temperature, so they must be changed by hand. Change/Create. [These properties are used to define the Rayleigh number.]
5. Finally, Close the *Materials* window. *Caution*: This has added liquid water (with Boussinesq) into the list of fluids available as boundary conditions, but has not actually *changed* the fluid from air to water. This must be done when specifying the boundary conditions.

**Define the boundary conditions:**

1. Define-Boundary Conditions. Select fluid. Set. Choose water-liquid from the list of *Material Names*. OK.
2. Choose the lower wall. Set. Change *Thermal Conditions* to Temperature, and enter the desired temperature of the lower wall, which is the reference temperature in this problem,  $60^\circ\text{C}$  ( $333.15\text{ K}$ ). OK.
3. Similarly, define the (cooler) temperature of the upper wall, according to your desired Rayleigh number.
4. Close the *Boundary Conditions* window.
5. Make sure that the left and right edges of the domain are *periodic*.

**Initialize and iterate:**

1. To initialize the solution, Solve-Initialize-Initialize. The default initial value of pressure is okay, but I recommend a small value for the initial velocity field (something like  $0.001\text{ m/s}$  in the  $y$  direction), just to give the flow some initial “kick” to help set up the instability. Also, set the temperature everywhere to some kind of average between the lower and upper limits. Init and Close.
2. Solve-Monitors-Residual. Turn on the Plot option and set the convergence criteria for the residuals to something very small, like  $1\text{.E-}08$ , as described in previous learning modules.
3. Solve-Controls-Solution. Reduce the under-relaxation values for pressure, momentum, and energy to about half of their default values to ensure better convergence.
4. In addition, the discretization method recommended for natural convection flows is PRESTO!. OK.
5. It is recommended that you save your case and data files at this point, in case anything goes wrong.
6. Iterate towards a solution. If having difficulty converging, try this: Turn off the flow equations first, and just solve for the heat transfer. (Solve-Controls-Solution, and turn *off* the Flow equations.) This should converge very quickly to the basic state (steady state pure conduction through the fluid). Verify by plotting temperature contours. Turn the Flow equations back on and iterate some more.
7. If cells still do not form, you may need to adapt the grid. Sometimes the CFD code needs a little non-uniformity in the grid in order to initialize formation of the cells. For this reason I suggest adapting by region to refine the grid in a region of the domain. First display the grid, then Adapt-Region-Select Points with Mouse, and select an arbitrary rectangular region in the domain by clicking with the right mouse button. Adapt-Yes.
8. Iterate some more. After some trial and error, adapting more as necessary, re-initializing, etc., the residuals should start to go down slowly but nicely. You may not be able to converge completely to the convergence criteria. Use your judgment as to when you have a converged solution. It may take several thousand iterations.
9. Bénard cells, if they appear, should be readily visible by displaying velocity vectors, streamlines, or a contour plot of temperature.

10. After the solution has converged, make a hardcopy plot of the velocity vectors, colored by velocity magnitude, if there are clear Bénard cells. It is desirable here to zoom in to a couple cells so that the flow pattern is clear. Follow the directions given in previous learning modules if you don't remember how to generate hardcopies. ***Be sure to add your name and the Rayleigh number to the plot title.***
11. Generate a plot of the streamlines ***over the entire computational domain*** if there are Bénard cells: Display-Contours. Choose contours of Velocity and choose Stream Function. Display. Generate a hardcopy of the streamline pattern (make sure the entire domain is visible). This plot will be necessary for measuring the wavelength of the Bénard cells.

**Save your calculations and exit FLUENT:**

1. In the main *FLUENT* window, File-Write-Case & Data. OK. It is okay to overwrite the files, so OK again.
2. Repeat the calculations for the other Rayleigh number (all you need to change is the temperature of the upper wall). I would suggest re-initializing the flow field. Make hardcopy plots as described above, but be sure to change the Rayleigh number on the plot title. Save these calculations as different file names if you have enough disk space available.
3. Exit FLUENT by File-Exit.