ME 522
Spring Semester, 2019
Homework Set # 4

Professor J. M. Cimbala

For instructor or TA use only:		
Problem	**Score**	**Points**
1		40
2		20
3		40
Total:		**100**

1. (40 pts)   Prepare a "lecture" on the Kelvin-Helmholtz instability. In other words, pretend you are a professor, and you need to teach this material to your "class." To teach effectively, you must first learn and understand the material yourself! This involves going through the algebra, making sure you understand each step along the way. Use any references you can find. For example, Kundu's book has a section about this as do other books in the library or perhaps on the Internet. Don't be concerned about how long your lecture is; rather, prepare enough so that *you* understand this instability, and how you might explain it to your students. Follow the linear stability analysis procedure outlined in class. The basic idea of this homework problem is that you learn more about this well-known instability, since we will not have time to cover it in detail in class. Also, many of you will end up being faculty members after graduation, and this assignment gives you some experience in preparing a (graduate level) lecture.

2. (20 pts)   In this problem, we fill in some of the gaps in our analysis of the Bénard problem. First, the Boussinesq equations were written as the set of differential equations to consider (step 0). Next, the basic state equations and the basic state were found (step 1). Here, you will fill in some of the algebra, and analyze the basic state temperature and pressure fields in detail.

(**a**) Integrate the energy equation, using appropriate boundary conditions, to verify the expression for $\overline{T}(z)$ given in class. Express your equation in terms of Γ, where $\Gamma = \Delta T/d$.

(**b**) The basic state pressure field is hydrostatic, but not exactly linear since density varies from the top to the bottom of the domain. Find an exact expression for $P(z)$ (exact within the limitations of the Boussinesq approximation). *Note*: Assume boundary condition $P = P_0$ at the lower wall.

(**c**) Plot a comparison of the actual (exact) $P(z)$ with the linear $P(z)$ one would get assuming constant density, and provide a physical explanation for your result. For consistency let the fluid be a liquid with the following properties, constants, and boundary conditions: $\rho_0 = 1000$ kg/m³, $\alpha = 2.70 \times 10^{-3}$ 1/K, $T_0 = 280$°C, $\Delta T = 60$°C, $g = 9.807$ m/s², $d = 0.002$ m, and $P_0 = 0$ Pa (this will make P like a gage pressure with the reference pressure being zero at the bottom wall).

Note: There is another page. →

3. (40 pts)   In this problem you will simulate the Bénard instability problem using computational fluid dynamics (CFD). There are three goals for this exercise:

(1) To get some experience using CFD (for some of you this may be your first exposure to running a CFD code).
(2) To see whether CFD can predict results that are similar to the analytical solution derived in class.
(3) To increase your understanding about flow instabilities and how they form.

You may use any CFD code you want. Three free ones are available for students in our class:

- **STAR CCM+**. A free student license for STAR-CCM+ is available for students in our class to download onto their own computers. Instructions are provided in a separate file, posted on the ME 522 Canvas website, due to security reasons. A STAR-CCM+ video tutorial by Chris Penny is also available for this HW assignment: Benard_Cells.mp4 .
- **ANSYS Workbench** (FLUENT and CFX). This software is available on the MNE PCStudio computers. Also, a *free* student version of ANSYS Workbench is available for students to download onto their own computers. Instructions are at http://www.ansys.com/student. Note that you must use your psu.edu email to prove you are a student. Also, make sure you follow the installation directions carefully. [Warning: Installation is more cumbersome than most software that you download because of the licensing, but it should work if you follow the directions carefully.] An ANSYS Workbench video tutorial by Skylar Schirtzinger (a former TA for this course) is are also available:
  o Part 1: https://youtu.be/oXwNIL0NBDo
  o Part 2: https://youtu.be/IUcPRifgrKc
  Professor Cimbala also has a typed tutorial for older versions of FLUENT. Although some commands have changed, the overall flow of the instructions is still valid. See
  http://www.mne.psu.edu/cimbala/Learning/Fluent/fluent_Benard_problem.pdf .
- **OpenFoam**. This is open source software that is available free to anyone. However, unless you have used it before, the learning curve is probably too steep to learn in time for this homework.

Truncation and round-off errors are typically sufficient as small disturbances that grow as the CFD solution tries to converge, acting very similar to a real experiment.

Details: Consider two parallel walls at distance d apart, with fluid in between. The top wall is colder by some amount ΔT. Following the class notes, all reference fluid properties are evaluated at the bottom wall. In the analytical solution, the walls were infinite in extent. However, in the CFD solution, the computational domain is bounded at either end; *the end conditions influence the geometry of the convection cells*, much like in a real physical experiment. In this problem, the ends are *periodic* (the mesh at the left edge must exactly match that at the right edge). *The computational domain of each student will have a unique aspect ratio so that end effects can be studied*.

- The computational domain for this problem is very simple – just a rectangle, as sketched below.
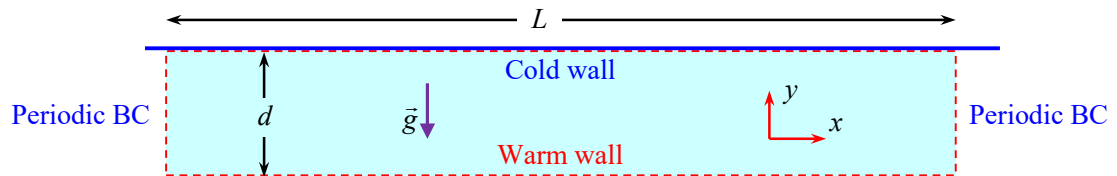
- As sketched, x is the horizontal direction and y is the vertical direction. Gravity acts downward (negative y direction).
- The bottom wall is hotter than the top wall, so that a thermal instability is possible, depending on the Rayleigh number. The bottom wall is held at fixed temperature $T_0 = 60$°C. The top wall temperature is determined by the Rayleigh number.
- For all cases the gap height d between the two walls is set to 2.00 mm (0.00200 m). However, domain length, L varies with each student. ***Each student is assigned a different aspect ratio*** for his/her computational domain, where aspect ratio is defined as AR = L/d, according to the following table [I purposely avoided *integer* values of AR]:

Note: There is another page. →

PSU Login name	Assigned AR = L/d	Calculated λ/d
sja198@psu.edu	4.1	
agc86@psu.edu	4.4	
src347@psu.edu	4.5	
exc5257@psu.edu	4.8	
smc303@psu.edu	4.9	
tmc5478@psu.edu	5.1	
mrc453@psu.edu	5.4	
mje32@psu.edu	5.5	
kmh774@psu.edu	5.8	
iph3@psu.edu	5.9	
auh1002@psu.edu	6.1	
xuh128@psu.edu	6.5	
nuh98@psu.edu	6.9	
nxj190@psu.edu	7.1	
rtj10@psu.edu	7.5	
ajr545@psu.edu	7.9	
acs5213@psu.edu	8.1	
dxs5610@psu.edu	8.5	
rus284@psu.edu	8.9	
cms685@psu.edu	9.1	
yzs220@psu.edu	9.5	
jxs2469@psu.edu	9.9	
emv53@psu.edu	10.1	
auw317@psu.edu	10.5	
ctw39@psu.edu	10.9	
djw68@psu.edu	11.1	
ajw324@psu.edu	11.5	
phw13@psu.edu	11.9	
hzx25@psu.edu	12.1	
jry40@psu.edu	12.3	

(a) Generate an equation for ΔT as a function of thermal conductivity k, viscosity μ, density ρ, specific heat c_p, gravitational constant g, thermal expansion coefficient α, gap height d, and Rayleigh number Ra. For the fluid you choose for your CFD analysis, calculate the ΔT required to achieve Ra = 5,000. *Note*: Make sure the units all cancel correctly!

(b) Using *your* assigned aspect ratio for the computational domain, generate the grid, and run the CFD code at Ra = 5000. Do Bénard cells form? Attach an image showing visualization of your cells.

(c) Measure average cell wavelength λ/d and compare with theory. *Hint*: The easiest way to measure λ is to divide the length of your domain by the number of cell *pairs* (not individual cells since two cells make a wavelength). Do your results agree with theory? Discuss.

(d) In the above table, fill in λ/d for your assigned aspect ratio, and turn this in as part of this assignment. [In a future HW assignment, we will collect all these into Excel and plot them to see if there are any trends.]