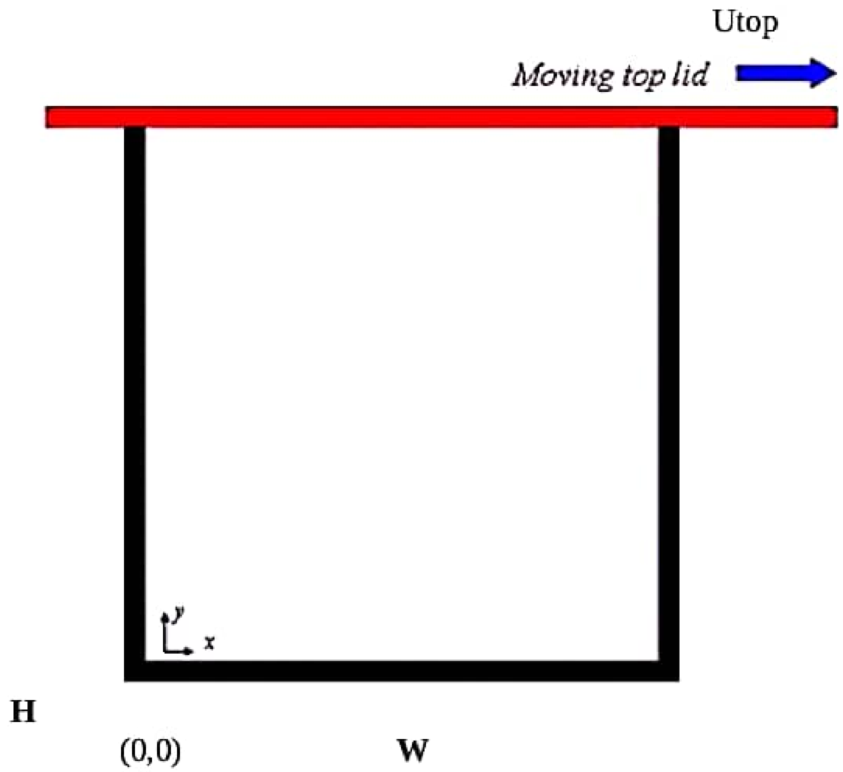

Now this is the time for you to exercise your knowledge and experience in writing an incompressible Navier-Stokes solver.

You will be asked to solve the driven-cavity problem (a rectangular cavity with horizontally moving top), including an exploration of vortex structure in the cavity.

This document describes the validation test cases you should run to test your codes and results. Your report should include, at a minimum, the information requested under **Report:** in each subsection. Also, you should comment on anything else that you come across the strikes you as particularly interesting (or confusing, for that matter). **It is expected the discussion and interpretation of results to be a significant component of your report.**

Scanned with CamScanner

2 Explicit Time Advance: Solver Set up, and Things to watch for:

Now that you have your correct flux function working appropriately, it is time to link it with an explicit time advance. Set up the RK-4 explicit time advance for the computed and verified flux integral, and advance the solution process toward convergence. Play (numerically) with time step to figure out a robust time step to use. (you may choose an appropriate CFL condition)

There are several things that you should look out for (or experiment with) while debugging and optimizing your code for the cases in the following section.

1. **Pressure drift.** With Neumann BC's all the way around the domain, you may (or may not) find that your mean pressure changes gradually. If so, you can fix this by applying a fixed pressure boundary condition for one ghost cell or by forcing the mean pressure to have particular fixed value.
2. **Pressure oscillation.** You will almost certainly find that steady-state pressure distribution have some oscillations. These are present because of de-coupling between pressure in alternate lines of the mesh. To fix this problem, you can add a term to the right-hand side of the pressure equation that looks something like: $A\Delta x^2\nabla^2 P$. You want to choose the constant A so that you smooth out oscillation, but you want A to be as small as possible, since you are after all introducing an error in the steady-state enforcement of continuity. The good news is that this error will be second error.
3. **Choice of β.** While using β=1 is a reasonable choice for these cases, it is probably not optimal. Feel free to experiment with this to see if you can find a value that works significantly better.
4. **Over-relaxation.** Over-relaxing your solution update does in fact improve convergence rate (although including over-relaxation is unlikely to stop your code from blowing up, if it's exhibiting that behavior already). Experiment with this also, if you like.

If you choose to implement or experiment with any of these items, include a description of what you did and the effect it had in your report.

Scanned with CamScanner

3 Flow in a Box with a Moving Top

This problem concerns flow in an enclosed box with a moving top, a shown in Figure 1. The walls all have no-slip boundary conditions. The lid of the box also has a no-slip boundary condition, but in this case the boundary condition is for no slip at the constant velocity U_{top}. We will discuss the pressure boundary condition in more detail shortly. The fluid is isothermal, so buoyancy effects are not present. As the top of the box moves to the right, the fluid in the box circulates clockwise (for a square box). For all cases, use a Reynolds number of 100 and $\beta=1$. Your code should be able to handle a time step of $\Delta t = 0.001$ and possibly higher; higher time steps will of course mean faster convergence. The basic physical results for the problem will be reported using the u-velocity along the line of vertical symmetry, which will of course require averaging, since this line is a control volume boundary.

3.1 Validation case: Stability

Begin with the velocity and pressure distributions of section 1.1. Verify that your code converges toward zero velocity and uniform pressure. Use h=1 (height of the box) and $U_{top}=0$, with the top left corner of the box at (0,0).

Report: Convergence history, plotting the log L_2 norms of u, v, and P versus number for time steps at $\Delta t = 0.001$. Try larger time step if you can.

3.2 Basic solution

For all of these cases, use h=1. You may want to experiment with $\Delta t > 0.05$ for improved convergence. Be sure to indicate the value used so that I can assess your convergence histories in the proper context.

3.2.1 Solution for $U_{top}=1$

Solve the problem for $U_{top}=1$ for a 20×20 mesh.

Report: Convergence history until the largest L_2 norm of the change in solution is smaller than 10-6; Plot of u along the symmetry line. Surface or contour plot of pressure.

3.2.2 Sanity check: Symmetry

Solve the problem for $U_{top}= -1$ for a 20×20 mesh.

Report: Surface or contour plot of $u_{U\ top=1} + u_{U\ top=-1}$, which should be very small everywhere, though probably not quite zero.

3.2.3 Grid convergence

Scanned with CamScanner

Solve the problem for a 10×10 mesh and as many meshes finer than 20×20 as you feel are needed to demonstrate grid convergence.

Report: On the same plot, show u on the symmetry line for each mesh as a function of y. How fine a mesh do you feel is needed for grid convergence for this problem?

3.3 Exploratory case for flow in a box: Effect of increasing *h*

As the height h of the box increases, the single vortex eventually becomes unstable, and a second vortex forms below it (and eventually a third, and so on). The exploratory part of this problem will focus on this formation of additional vortices with increasing h.

For U_{top} = 1, w=1, and h=3, compute the solution for flow in the box. Use meshes (of whatever size you think is appropriate) with Δx=Δy.

Report: Present evidence that you consider convincing that there are two vortices present for this case and that your numerical solution is grid converged. Give the center of the two vortices (and describe how you have made this estimate).

4. Flow in a Duct with Inlet velocity

4.1 Flow in a Duct with Inlet velocity

In this case, you solve a flow inside a duct. The inlet uniform velocity of U=1 is set as the velocity inlet and the outlet pressure is set P=0.0 as the gauge atmospheric pressure. The length and height of the duct are set to be 8 and 1 (everything is non-dimensional). The fixed pressure and fully developed condition are set as the outlet boundary conditions.

4.2 Solve the problem for a mesh 20×10 for Re=50.

Report: Show the convergence history until the largest L2 norm of the change in solution is smaller than 10-6; Plot of u across the duct along the line of X=7. Compare it with the exact solution of the case that you may find in classic fluid mechanics books. Show both velocity profiles on the same graph and verify your numerical solution against the analytical solution. Compute L2 norm of the error of then numerical solution based on the exact solution.

4.3 Grid convergence

Report: Repeat the case for 40×20, 80×40, and 160×80 for the same Re number and make sure the L2 norm of the convergence error is smaller than 10-6 in each case. Estimate the accuracy order of the numerical solution for all four meshes along X=7.

Scanned with CamScanner