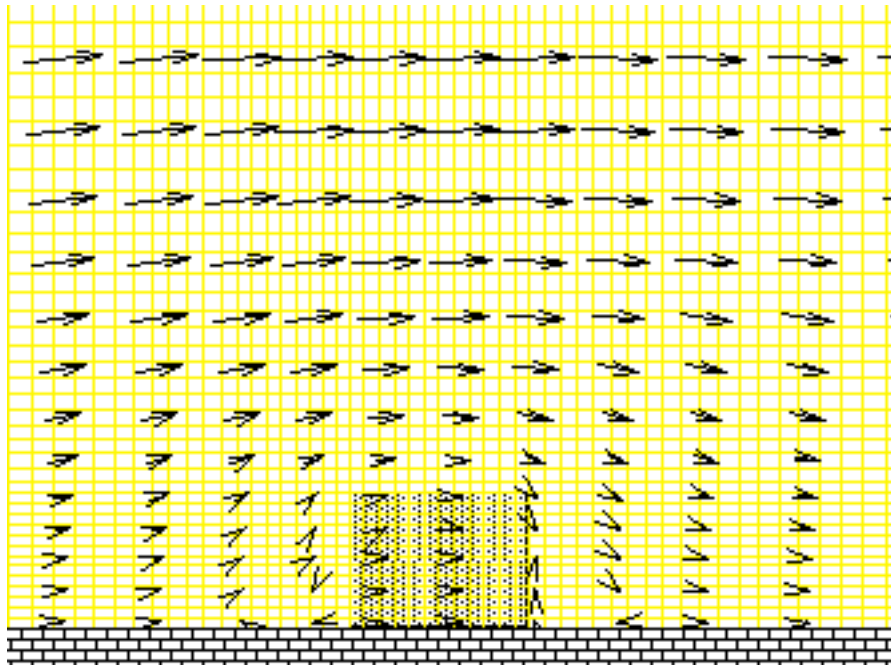


Module II:
Two-dimensional fluid flow

- application notes and website tutorial

- Problems



QnS Computational Fluid Dynamics program (2 Feb 96)

<http://www.me.pdx.edu/~gerry/QnS/>

by

Scott Forbes <MacIsBack@aol.com> University of Akron, Akron, OH
Gerald Recktenwald <gerry@me.pdx.edu>Portland State University, Portland, OR

QUICK 'n SIMPLE, or QnS, allows users to set up and solve two-dimensional, laminar, incompressible flow problems using the Apple Macintosh(TM). QnS is intended primarily as a tool for teaching fluid mechanics and computational fluid dynamics (CFD). Without at least some training in these areas you will not fully understand how to accurately specify a problem and interpret the results.

On the other hand, QnS, as its name also implies, can be used to quickly and simply set up and solve fluid dynamics problems that are impossible to solve analytically. We have designed QnS to work in the natural way that fluid dynamicists describe the physical aspects of the problems they are attempting to solve.

To solve a flow problem you

- #Specify the size of the computational domain.
- #Assign boundary conditions: inflow, outflow.
- #Generate a grid that is consistent with the boundary conditions
- #Tell QnS to solve the flow field

Unlike many other CFD programs, QnS generates the grid after the physical problem is specified. We believe this is more intuitive, since the grid, after all, is merely an artifact of the solution method.

The name of the program is derived from the QUICK algorithm of B.P.Leonard [1] and the SIMPLE algorithm of S.V. Patankar and D.B.Spalding [2]. QnS incorporates both the SIMPLE and SIMPLER algorithms for the pressure-velocity coupling. QnS allows the user to select one of QUICK, pure upwinding, or the Power-Law difference method for modeling the convective transport.

QUICK 'n SIMPLE is an interactive educational tool for studying the behavior of typical fluid flows. It is not intended to compete with complex and expensive industrial strength CFD codes. In keeping with this purpose, we have only incorporated features which maintain the "quick" and "simple" nature of the user interface.

Features

- QnS is capable of solving the Navier-Stokes equations for laminar, incompressible flow in two-dimensional Cartesian or axisymmetric coordinate systems
- The user can easily chose from the following convection modeling schemes: QUICK, pure upwind, or Power-Law differencing
- Grid generation is easy and flexible. Nonuniform grids can be prescribed in a number of ways.
- Inflow, outflow, moving, and symmetry boundary conditions are selected by clicking and dragging with the mouse.
- Inflow velocity profiles can be uniform, linear, parabolic, or custom
- Internal obstacles in the flow field can be added by drawing them with the mouse
- Transport of a passive scalar, e.g., smoke or pollutant can also be modelled with a single source anywhere in the domain.
- Convergence is displayed graphically while calculations are under way.

- Solution to the flow field may be displayed graphically as velocity vectors, streamlines along with contours of pressure, vorticity, velocity magnitude, and pollutant concentration.
- The user can choose fluid properties of air or water at any temperature, or custom fluid properties can be used.

Limitations

The graphical user interface of QnS has allowed us to simplify the problem specification and hide the complexity of the solution algorithm. However, you must still understand enough about fluid mechanics to specify a meaningful problem and to interpret the results. Though QnS may solve your problem, it is no substitute for common sense and engineering experience. QnS merely permits you to spend less time on the tedious details of the problem solution and concentrate on the more important aspects of the fluid mechanics. We hope that by working with QnS you will develop your intuition of fluid behavior.

Availability

QnS version 1.1 is available as freeware. You can download the application and (skimpy) documentation from the QnS directory of the anonymous ftp server at the PSU school of Engineering and Applied Science. You are free to copy and distribute it in an unmodified form. If you find it useful, (or if you find bugs) please send us an email message.

References

- Leonard, B.P., (1979), "A stable and accurate convective modelling procedure based on quadratic upstream interpolation" , Comp. Meth. in Appl. Mech. and Engrng., vol. 19, pp. 59-98.
- Patankar, S.V. and D.B. Spalding, (1972), "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", Int. J. Heat Mass Transfer, vol. 15, p. 1787.

website & first page of application

Location:  <http://www.me.pdx.edu/~gerry/QnS/>

For text-only, choose ----> [\[Overview\]](#) [\[Examples\]](#) [\[Tutorial\]](#) [\[Details\]](#) [\[Download\]](#)


[Overview](#) [Examples](#) [Tutorial](#) [Details](#) [Download Program](#)



A Computational Fluid Dynamics program by [Scott Forbes](#) and [Gerald Recktenwald](#)

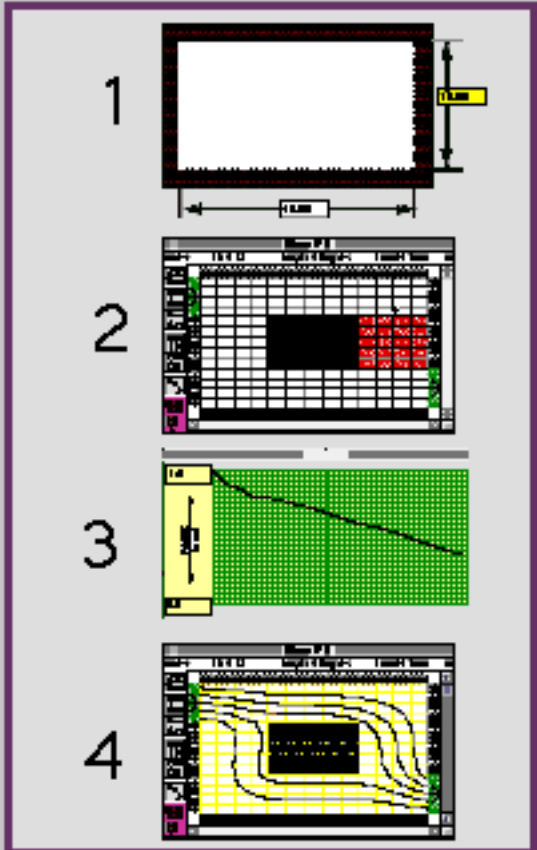
Tutorial Overview

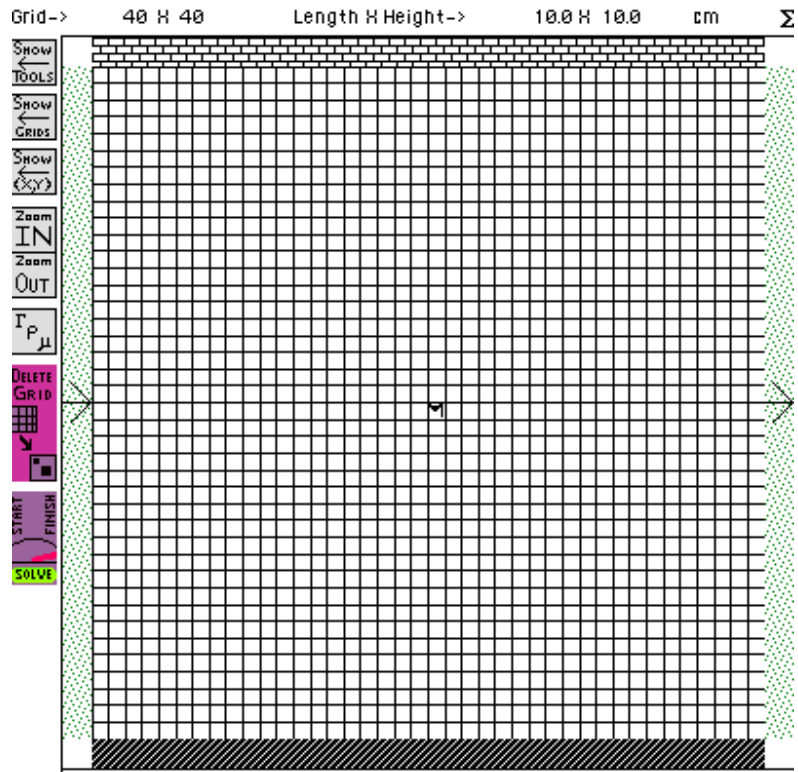
To familiarize you with use of the QnS interface, we will walk through the following sequence of steps in Problem 1A:

 Return to Main

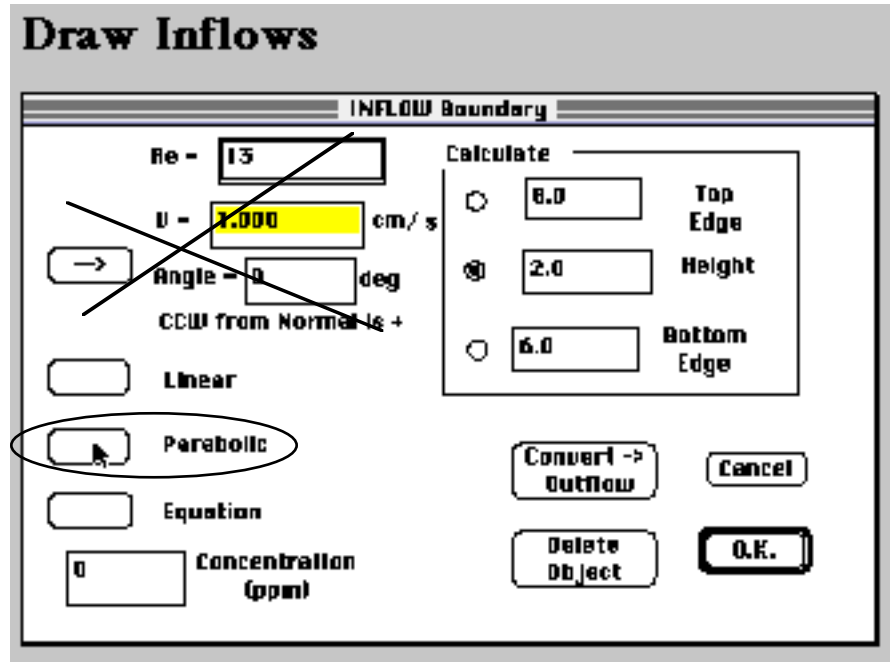
Steps

1. [Specify Problem](#)
2. [Generate Grid](#)
3. [Solve It](#)
4. [Visualize Results](#)

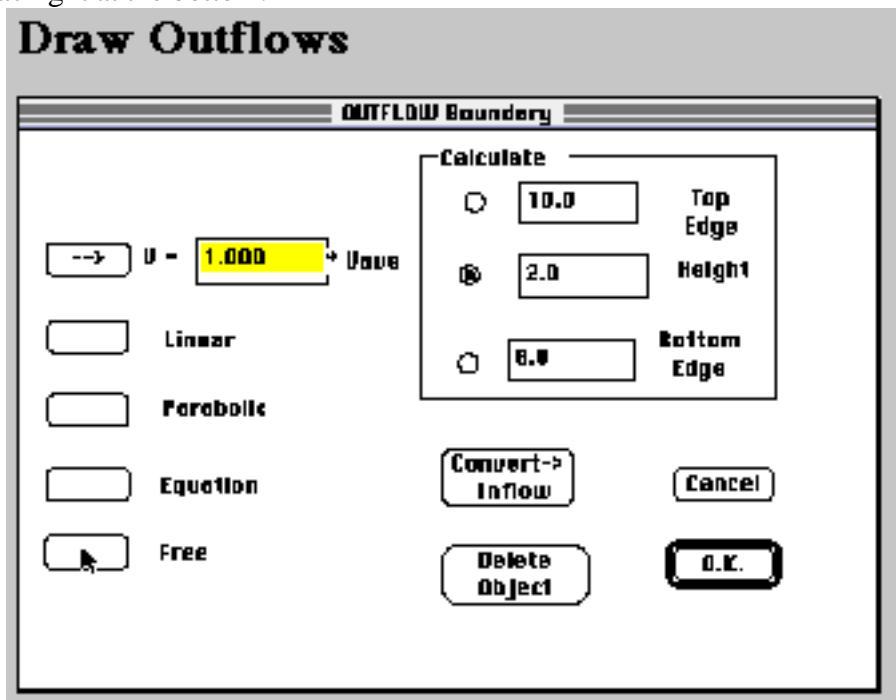




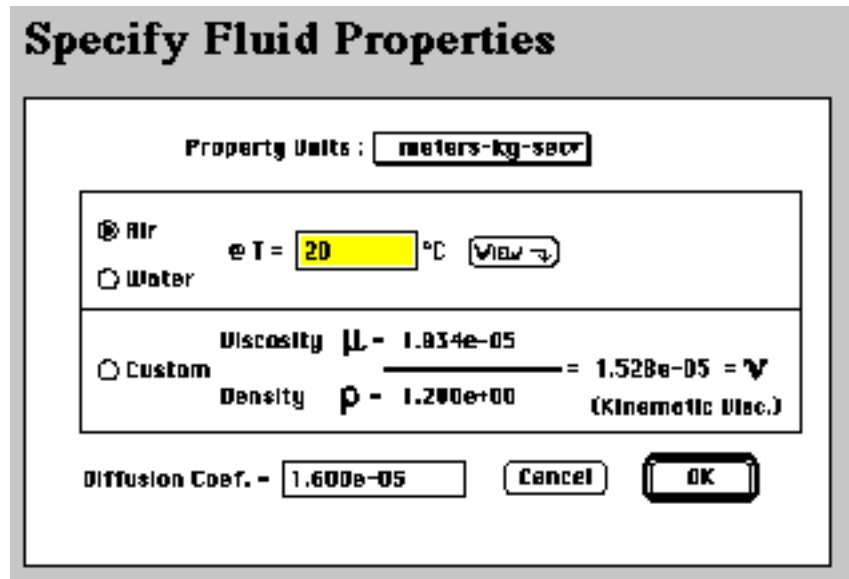
To specify the desired parabolic inflow, click on **IN** so that the window below opens. Once you click on **Parabolic**, a second box will pop up; enter $V = 1$ cm/sec and make this the maximum at the lower corner.



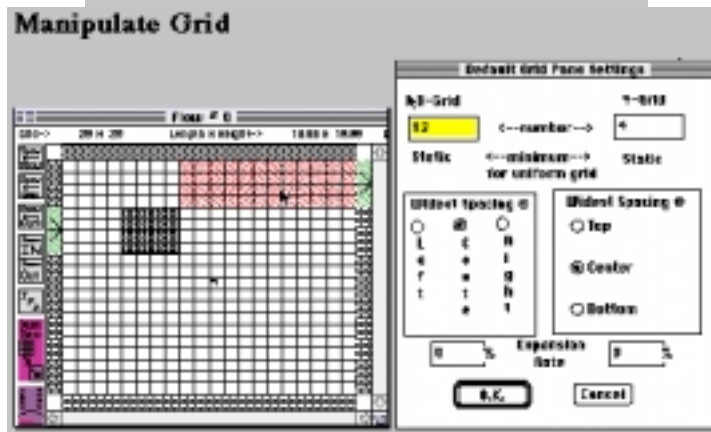
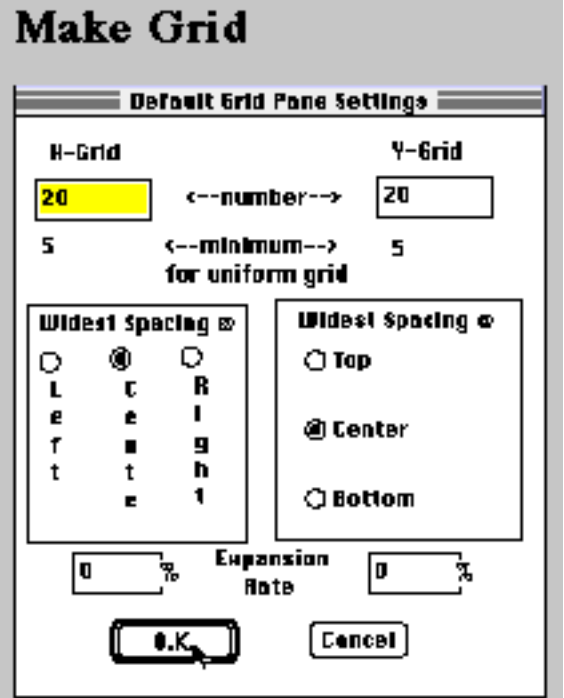
To specify the desired free boundary outflow, click on **OUT** for the window below to open, and then click on **Free**. Finally, also make the bottom of the domain symmetric by clicking on the **S** - button and placing it at the bottom.



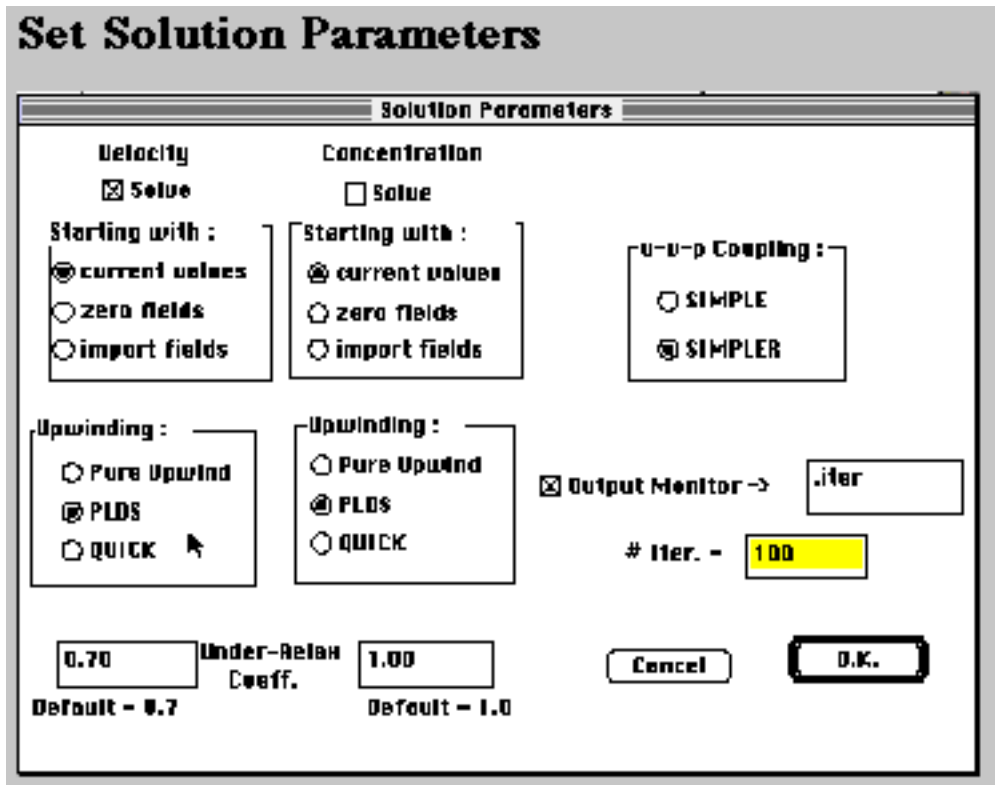
To specify the fluid to be water, click on the $\Gamma\rho\mu$ - button for the window below to open, and then click on **Water**.



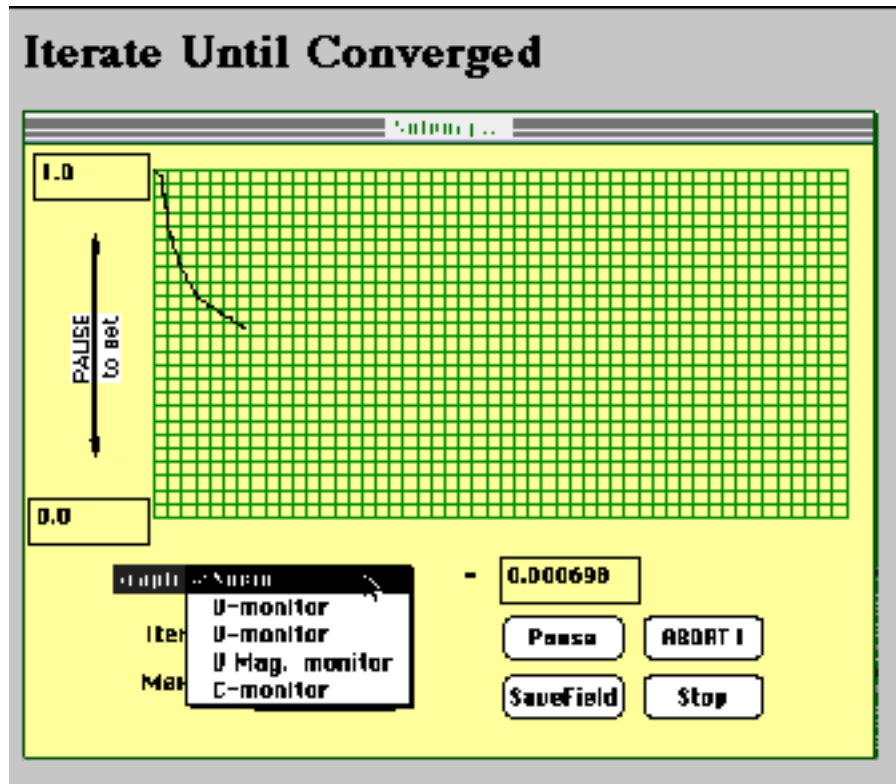
To impose a mesh on the domain, click on the **Make Grid** - button. The window below will open, and you should make a **30** by **30** grid. By default, a uniformly square mesh will appear. Later, when we insert block objects in the flow, a non-uniform mesh will be desirable. You will be able to obtain this interactively by clicking on subdomains of the mesh; the window reappears and allows you to refine a mesh locally around an object.



To solve for the flow in the domain, click on the **SOLVE START FINISH** - button for the window below to open, and then set the iterations to **300**. The other default parameters should prove adequate.

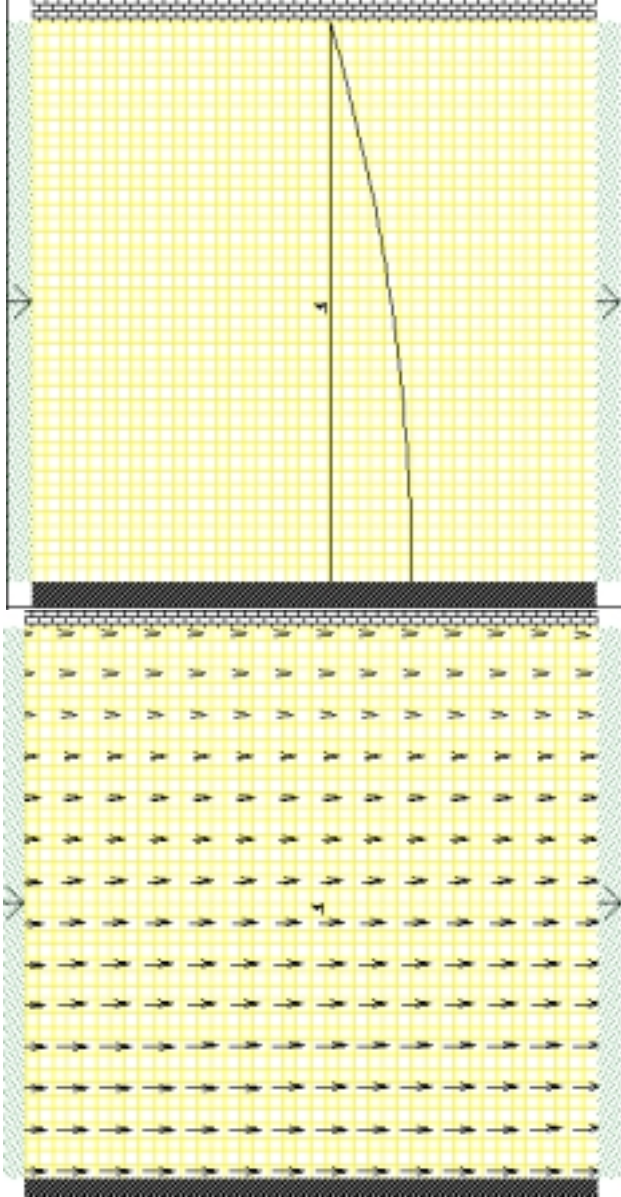


Once you hit the **OK** - button, a domain solution is converged upon until $S_{max} = 0.000001$.

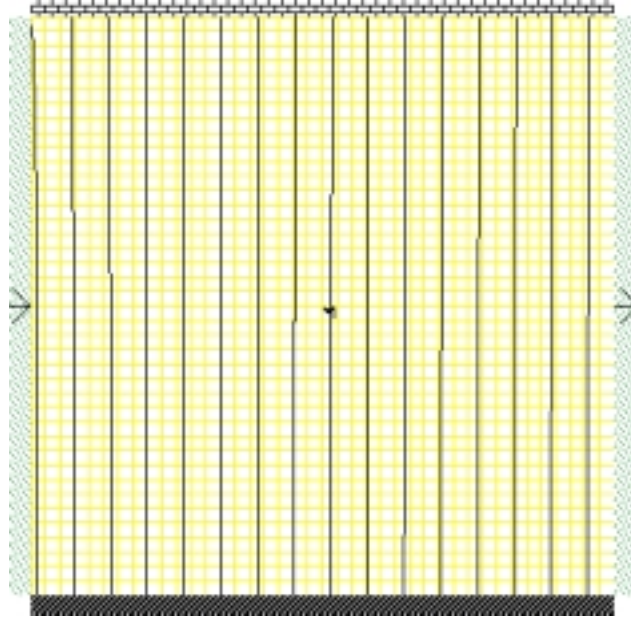


Verify that the solutions appear as below:

Velocity Field Solutions



Pressure Field



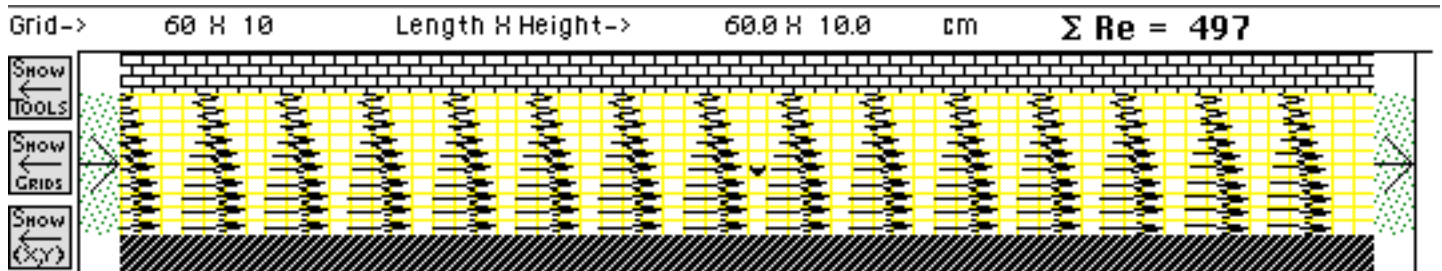
QUESTION: How well is the Plane Poiseuille flow solution satisfied? That is, calculate the error in:

$$v_{\max} = (h^2 / 2\mu) (-\Delta P / L)$$

where h is the half-separation and L is the length over which the pressure change acts.

PROBLEM 1B - Laminar Pipe Flow

Now set up the pipe flow scheme below for water. Notice the domain dimensions of **60 cm x 10 cm**. A half-parabolic input as well as output must be specified because the present code is not particularly stable in problems with cylindrical symmetry. An axis of symmetry is shown at the bottom. The top boundary is, again, a no-slip boundary. Use the same centerline V as above.



QUESTION: How well is the Poiseuille Pipe flow solution satisfied? That is, calculate the error in:

$$v_{\max} = (R^2 / 4\mu) (-\Delta P / L)$$

where R is the pipe radius and L is the length over which the pressure change acts.

QUESTION: Notice that the centerline velocity is reduced by half for the same pressure drop in pipe flow versus planar flow. In one sentence, what do you think is the physical origin of this effect?