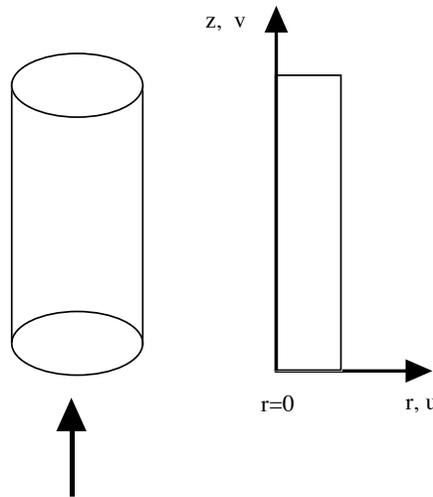


### Navier-Stokes Equation Solved in Comsol 4.1.

Copyright Bruce A. Finlayson, 2010

See also *Introduction to Chemical Engineering Computing, Wiley (2006)*.

Next consider entry flow into a pipe as illustrated in Fig. 10-1. The flow is taken as uniform in radius at the entrance (taking the value 1.0 at each radial position), and the velocity has to develop into its fully developed profile. The pipe radius is taken as 0.5. How far downstream do you have to go to get a fully developed solution? (Another way to pose this question is: how much error do you make if you assume fully developed flow when it isn't?)



**Figure 10-1. Entry flow into a pipe.**

**Step 1: choose the problem.** Open Comsol and choose 2D axisymmetric, next; then Laminar Flow, next; then Stationary, finish.

**Step 2: define the geometry.** Right click on Geometry and choose Rectangle. Set the width to 0.5 and the height to 2.

**Step 3: define the problem.** Click on Laminar Flow and change the Compressibility option to Incompressible flow. Right click on Laminar Flow and select Fluid Properties. Click on the domain, the + to add the domain to the Selection Box. Change the density and viscosity to User defined, with  $\rho = 10$ , viscosity = 1.

**Step 4: set the boundary conditions.** Right click Laminar Flow and choose Inlet. Select the boundary 2 and + to add it to the Selection box; set the normal velocity to 1.0. Right click Laminar Flow and choose Outlet. Select the boundary 2, then + to add it to the box; and change the boundary condition from Pressure, no viscous stress (we know there is viscous stress in fully developed flow) to Pressure,  $p_0 = 0$ .

**Step 5: set the mesh.** Choose mesh and Build All.

**Step 6: solve the problem.** Right click Study = and you get the plot of the solution. What is plotted is the value of  $U = \sqrt{u^2 + w^2}$ .

**Step 7: examine the solution.** To obtain more specific information, right click Results, choose 2D Plot Group, right click 2D Plot Group 2 and choose surface arrows. Click the plot icon to see the arrow plot. Also, right click Results, choose 2D Plot Group, right click 2D Plot Group 3 and choose streamline; click the plot icon to get the streamline plot. See Figures 10.2a, b, c.

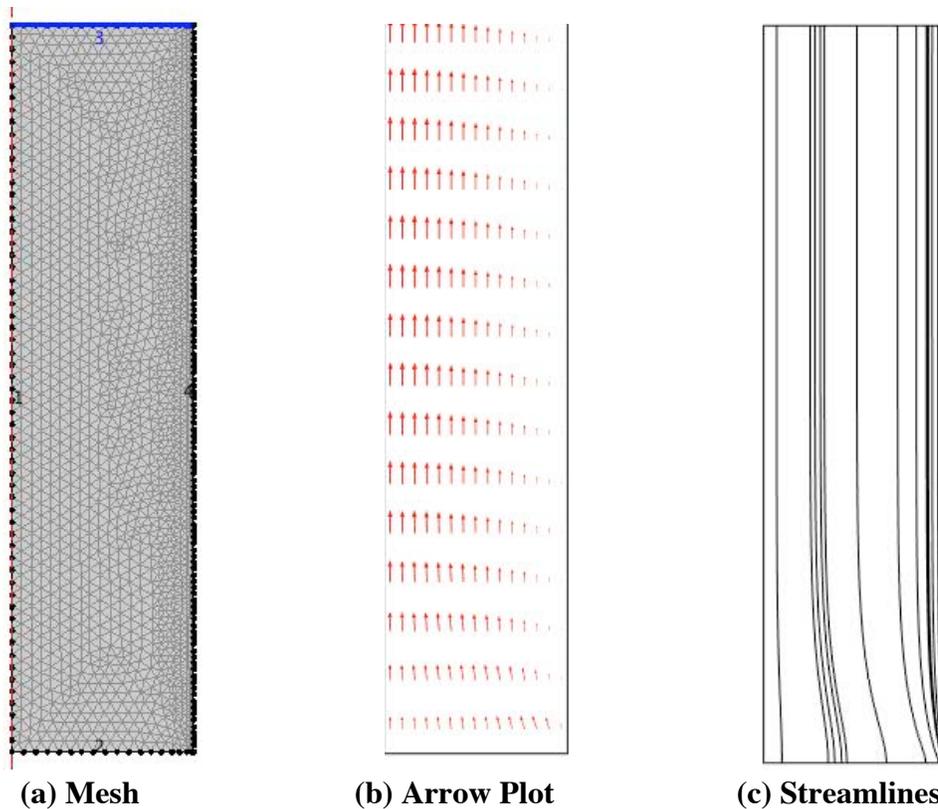
**Step 8: make more plots.** The inlet velocity is found by right clicking Results, choosing 1D Plot Group, selecting the boundary 2, changing the X-Axis Data Expression to  $r$ , and clicking the plot icon. The result is a step function. Change the title and axis labels to Inlet Velocity,  $r$  and  $w$ . Repeat this procedure but choose the outlet boundary. To see the approach to fully developed flow, plot the axial velocity along the centerline. Right click Results, choose 1D Plot Group. Right click 1D Plot Group and select Line Graph, then select boundary 1, the centerline, then the + to add it to the Selection box. Change the X-Axis Parameter to Expression,  $z$ . Change the titles and axis labels and click the plot icon.

**Step 9: improve the solution.** There is a small increase in the centerline velocity near the exit; this is not a valid solution. The reason is that the finite element functions are linear functions. For laminar flows, quadratic functions usually give better results. Look at the top of the Model Builder; on the right there is a down arrow.



Choose More options. Then go back to Laminar Flow; there is now an option for Discretization of fluids. The default value is P1+P1, which means that both the velocity and pressure are approximated with linear elements. Change that to P2+P1; then the velocity is approximated with quadratic elements. Right click Study = to get the new solution. Select the Line Graph that has the centerline velocity plotted and you see that the uptick at the end has disappeared.

**Step 10: calculate average properties.** To calculate average quantities, right click on Derived Values (under Results) and select Line Integration. Click on the inlet boundary 2, then + to add it to the Selection box, then = (at the top) to get the value. It is listed in Results as 0.4997. If you wonder whether the integration has been done using  $rdr$  or just  $r$ , change the expression to  $r \cdot \text{spf} \cdot U$  and click the = sign. The value is 0.1249, which is correct if one integrates  $1 \cdot rdr$  from 0 to 0.5. Thus, the  $rdr$  is included only if you include it in the expression. The small departure from 0.125 is due to numerical error and can be decreased if more elements are used. Doing the same thing at the outlet gives 0.1249. If you want the  $2\pi$  included, you can take out the  $r$  and choose Integration Settings, Compute surface integral. The value is 0.7846, which is  $2\pi$  times 0.125.

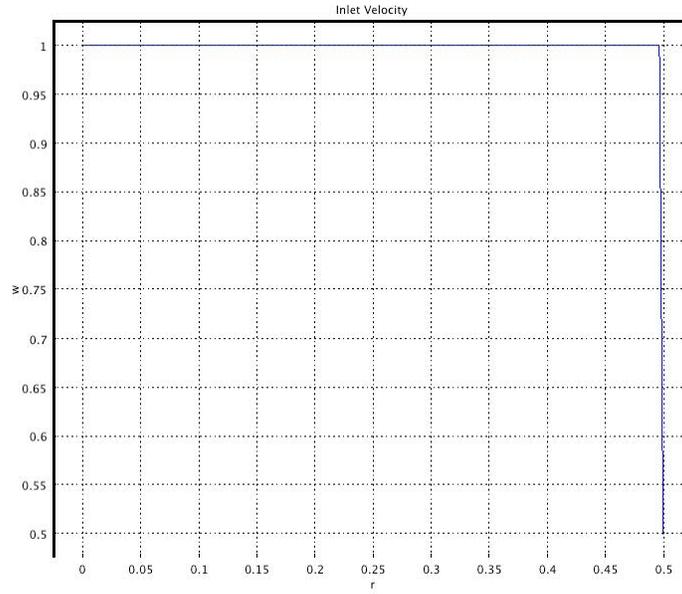


**Figure 10-2. Mesh, arrow plot, and streamlines for entry flow into a pipe**

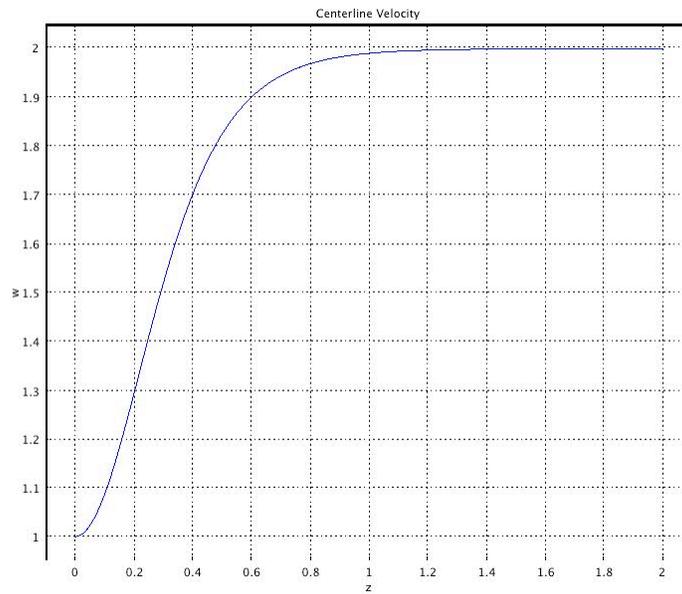
**Step 11: plot more properties.** It is useful to plot the solution along a boundary surface. Right click Results and choose 1D Plot Group. Right click 1D Plot Graph and choose Line Graph. Select the inlet boundary, +, to add it to the Selection box. Click the plot icon to see the inlet velocity, Figure 10-3. The velocity is not exactly 1.0 for all radii at the inlet, because there is a discontinuity in the boundary conditions. The boundary conditions use  $v = 1$  on one boundary and  $v = 0$  on an adjoining boundary, and the program has to choose which one to use at the corner. Comsol does a good job of satisfying these conflicting conditions. (It's not possible to achieve those boundary conditions physically.)

**Step 12: plot along the centerline.** Return to the Line/Extrusion plot and plot along the points  $(r,z) = (0,0)$  and  $(0,2)$  to obtain Fig. 10.4. This figure shows how quickly the center velocity approaches its asymptotic value as  $z$  increases. This gives you an indication of the “entry length.” Figure 10.4 was obtained using quadratic functions for velocity.

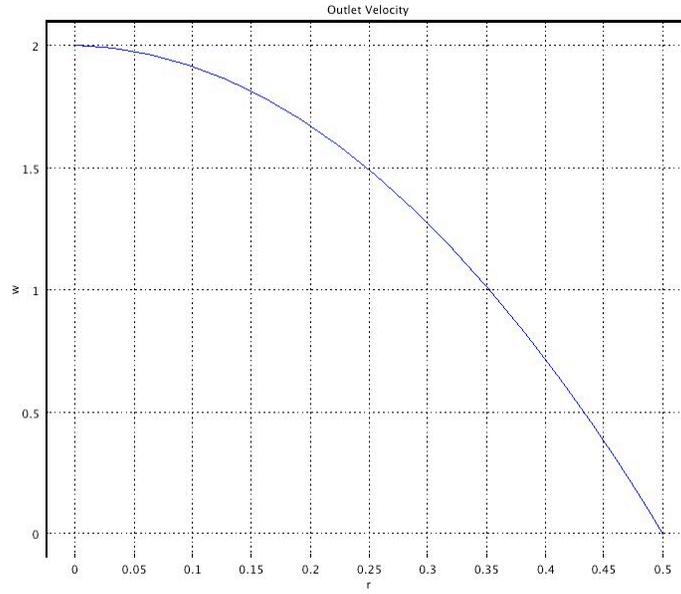
**Step 13: plot exit velocity profile.** Right click Results and choose 1D Plot Group. Right click 1D Plot Group and select the outlet boundary. Click + to add it to the Selection box. Click the plot icon to see the exit velocity. The exact solution is a quadratic function of position, which is satisfied in Fig. 10-5.



**Figure 10-3. Velocity at inlet to pipe**



**Figure 10-4. Centerline velocity in pipe flow**



**Figure 10-5. Velocity profile at exit**

Next, change the problem slightly and take the inlet velocity as the fully developed velocity profile.

$$v(r,0) = 2 \left[ 1 - \left( \frac{r}{0.5} \right)^2 \right] \quad (10.12)$$

**Step 14:** Use a fully developed inlet velocity. This is inserted in Laminar Flow, as the  $v$  velocity.

$$2 * (1 - (r/0.5)^2) \quad (10.13)$$

The streamlines are now straight.