Oblig 1

Greger Svenn

March 9, 2016

i)

We are asked to verify the analytical solution for flow in ducts of different shapes.

$$\mu \frac{d^2u}{dy^2} = \frac{d\hat{p}}{dx} = \text{constant} \tag{1}$$

Transforming it into its variational form we get

$$\begin{aligned}
(u'', v) &= (\frac{d\hat{p}}{dx}, v) \\
u'v|_{d\Omega} - \int_\Omega u'v'dy &= \int_\Omega \frac{d\hat{p}}{dx} v dy \\
-\int_\Omega u'v'dy &= \int_\Omega \frac{d\hat{p}}{dx} v dy
\end{aligned} \tag{2}$$

Implemented in fenics as

```
-mu*inner(grad(u),grad(v))*dx == dpdx * v * dx
```

Ellipse

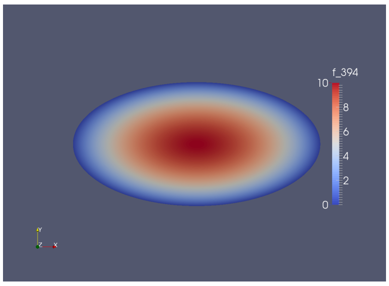
```
meshsize: 0.0591501982186
P 1 : error: 0.198612991237
      Qdif:  0.297069394929
P 2 : error: 0.210289258749
      Qdif:  0.262444333081
P 3 : error: 0.209726175255
      Qdif:  0.261816464212

meshsize: 0.0286080707886
P 1 : error: 0.0490186164831
      Qdif:  0.0742216200784
P 2 : error: 0.0523161573248
      Qdif:  0.0653951481331
P 3 : error: 0.0522316313421
      Qdif:  0.0653017153615

meshsize: 0.0139846868179
P 1 : error: 0.012118367695
      Qdif:  0.0183633262084
P 2 : error: 0.0129971279264
      Qdif:  0.016265804744
P 3 : error: 0.0129865902472
      Qdif:  0.0162541180776
```

The mesh refinement does make the errornorm and flowrate-difference better, for all chosen 'CG' elements. Comparing P1, P2 and P3 at the same meshsize shows that there isn't much of a difference.

Figure 1: numerical velocity field. Fine mesh



Triangle

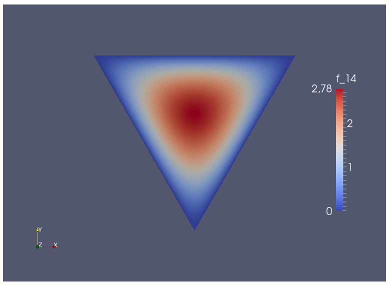
```
meshsize: 0.0350377086466
P 1 : error: 0.00171802938519
      Qdif:  0.00516271982613
P 2 : error: 1.5715194884e-05
      Qdif:  1.57157366643e-06
P 3 : error: 3.77584863776e-07
      Qdif:  1.82520665248e-13

meshsize: 0.0172032083643
P 1 : error: 0.000457344678688
      Qdif:  0.0012954806937
P 2 : error: 2.09535710775e-06
      Qdif:  9.82156328488e-08
P 3 : error: 3.56580191718e-07
      Qdif:  7.22977233636e-13

meshsize: 0.00840202867406
P 1 : error: 0.000121216065957
      Qdif:  0.000325354790669
P 2 : error: 2.73151840825e-07
      Qdif:  6.13381312409e-09
P 3 : error: 3.54823796132e-07
      Qdif:  2.97228908153e-12
```

Comparing the numerical solution with the analytical solution for the triangle gives us great results. Not much difference when we refine for P3. P1 gives us a convergence rate of 2 and P2 gives us a convergence rate of 3.

Figure 2: numerical velocity field. Fine mesh



Annulus

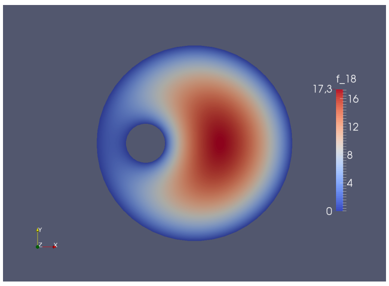
```
meshsize: 0.0588455214472
P 1 Qdif: 0.856400783432
P 2 Qdif: 0.696032328298
P 3 Qdif: 0.692098040142

meshsize: 0.0283145494553
P 1 Qdif: 0.214641713003
P 2 Qdif: 0.171826383154
P 3 Qdif: 0.17126495232

meshsize: 0.0136809542295
P 1 Qdif: 0.0535579359806
P 2 Qdif: 0.0425287440364
P 3 Qdif: 0.042454583381
```

For the annulus we can only compare the flow rate. We can see that we get better results when refining our mesh, even though the results aren't that good.

Figure 3: numerical velocity field. Fine mesh



ii)

148

$$F''' + FF'' + 1 - (F')^2 = 0 \tag{3}$$

By using the hint, $F' = H$, we get

$$H'' + FH' + 1 - H^2 = 0 \tag{4}$$

Then the boundary conditions become

$$\begin{aligned} F(0) = F'(0) = 0 \rightarrow F(0) = H(0) = 0 \\ F'(\infty) = 1 \rightarrow H(\infty) = 1 \end{aligned} \tag{5}$$

Using two test functions, vh and vf, we may display our problem as follows

$$(H, vf) = (F', vf) \tag{6}$$

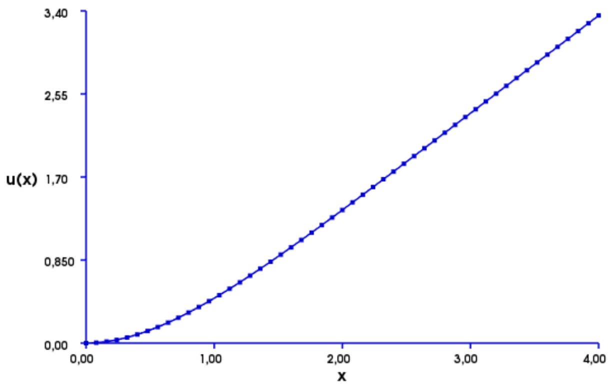
$$-(H', vh') + (FH', vh) + (1, vh) - (H^2, vh) = 0 \tag{7}$$

We then use Newton- and Picard iteration because of the non linearity.

Newton

Newton solver finished in 6 iterations and 6 linear solver iterations.

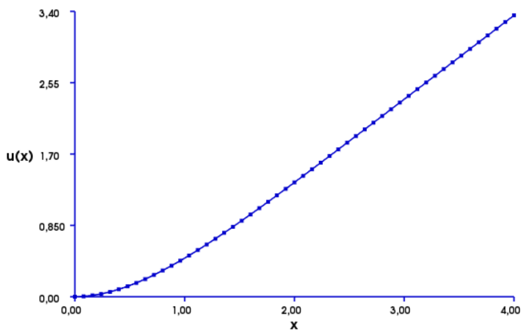
Figure 4: plot of F, using Newtons method



Picard

```
iter=1: norm=9.1337
iter=2: norm=7.14
iter=3: norm=3.97771
        .
        .
        .
iter=64: norm=2.85202e-12
iter=65: norm=1.92189e-12
iter=66: norm=4.34496e-13
```

Figure 5: plot of F, using Picard iteration



Both methods give similar result. Newtons method was however much faster

150

$$F''' + 2FF'' + 1 - (F')^2 = 0 \tag{8}$$

As above we substitute, get two equations, and use two test functions. We end up with

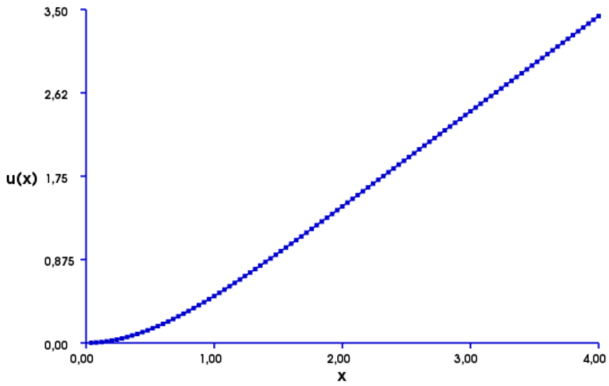
$$(H, vf) = (F', vf) \tag{9}$$

$$-(H', vh') + 2(FH', vh) + (1, vh) - (H^2, vh) = 0 \tag{10}$$

Newton

Newton solver finished in 6 iterations and 6 linear solver iterations

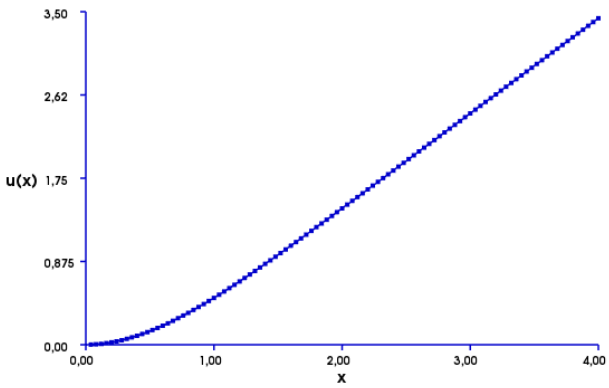
Figure 6: plot of F, using Newtons method



Picard

```
iter=1: norm=2.59437
iter=2: norm=2.20915
iter=3: norm=0.272425
        .
        .
        .
iter=16: norm=1.75043e-11
iter=17: norm=2.37214e-12
iter=18: norm=3.51235e-13
```

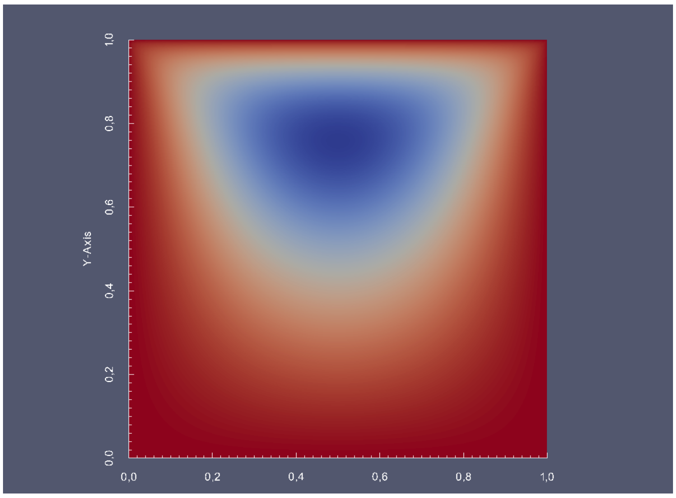
Figure 7: plot of F, using Picard iteration



iii)

```
Meshsize: 0.0141421356236
Vortex center at x: 0.5 and y: 0.76
```

Figure 8: Plot of stream function



iv)

I've chosen $L = 2$. With $Re = 0.01$ I get $\mu = 200$

a)

```
meshsize: 0.111027, minvalue: -0.0579006617083
Vortex center at x: 0.0 and y: 0.2

meshsize: 0.055514, minvalue: -0.0547620168121
Vortex center at x: 0.0 and y: 0.2

meshsize: 0.027757, minvalue: -0.0537767674796
Vortex center at x: 1.06551379861 and y: 0.0778896071092

meshsize: 0.013878, minvalue: -0.0533669982984
Vortex center at x: 1.07097328183 and y: 0.077088741035

meshsize: 0.006939, minvalue: -0.0531659519101
Vortex center at x: 1.06824354022 and y: 0.0806141740721
```

b)

Figure 9: Figure 3-37b in White shows the recirculation bubble in flow past a step

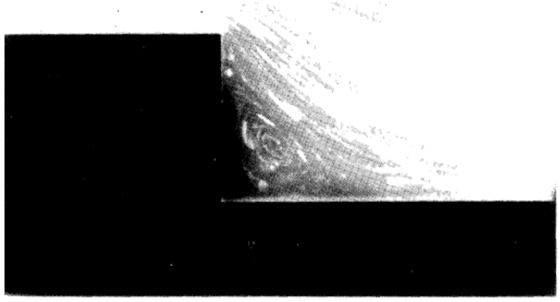
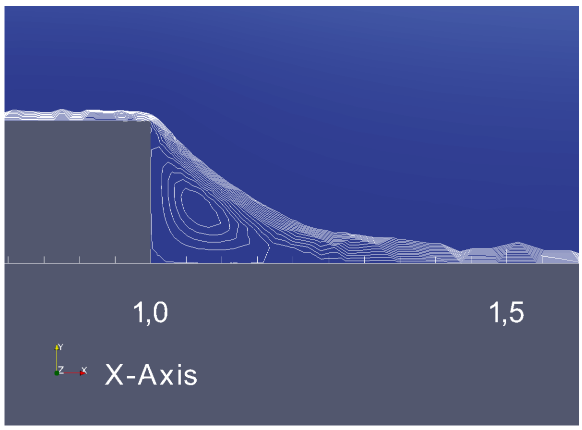


Figure 10: Numerical recirculation bubble in flow past step



c)

```
Flux inn: 0.425219807767
Flux out: 0.425219807767
Flux diff: 6.66133814775e-16
```

Since the difference is approximately zero, we have conservation of mass.

d)

The vortex center is at exactly the same point when direction of the flow is reversed. Vortex location is not dependent on flow direction, it's determined by the geometry.

e)

Normal stress on the bottom wall for flow i positive x-direction:

```
normal stress: 243.204770541
```

Normal stress on bottom wall for flow i negative x-direction:

```
normal stress: -243.204770541
```