Steady Incompressible Flow Simulations

- A Two-Dimensional Benchmark Problem -

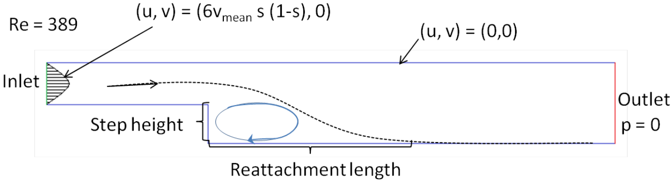
M. Baumann, G. Malenova, S. Milovanovic,
N. Priebe, D. Staubach

Project in Applied Numerical Methods 1

December 06, 2011

Outline

We consider the following benchmark problem:



- flow over backward facing step (BFS),
- well-known experimental data, but no analytic solution,
- parabolic velocity profile at the inlet,
- constant static pressure $p = 0$ at the outlet.

The stationary incompressible Navier-Stokes equations (NSE)

The flow behavior of a fluid can be modeled by the NSE

$$
\begin{aligned}
-\eta \Delta \mathbf{u} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} + \nabla p &= \mathbf{F} \\
\nabla \cdot \mathbf{u} &= 0
\end{aligned}
$$

The following physical quantities are introduced:

- flow velocity $\mathbf{u}$,
- pressure p,
- dynamic viscosity η,
- constant density of the fluid ρ,
- force field $\mathbf{F}$ - we consider only $\mathbf{F} = 0$.

An one-dimensional analogon is given by:

Non-linear ODE of the second-order

Consider the non-linear second-order ODE with Dirichlet boundary conditions:

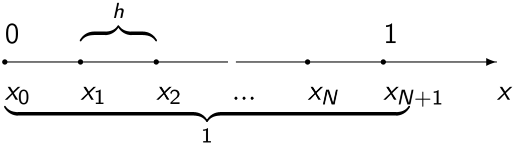
$$-\eta\frac{d^2u}{dx^2} + u\frac{du}{dx} = 1,\ \ \eta \in (0,1) \tag{1}$$

$$u(0) = 1,\ \ u(1) = 0 \tag{2}$$

Numerical methods:

- discretize in space with 2nd order finite differences,
- use Newton's method to treat non-linearity,
- comparison with Matlab's built-in solver `bvp4c.m`

Discretization of the grid:



Discretization of the ODE (1) and boundary conditions (2):

$$-\eta \overbrace{\left(\frac{u_{i+1} - 2u_i + u_{i-1}}{h^2}\right)}^{=:(A\mathbf{u})_i} + u_i \overbrace{\left(\frac{u_{i+1} - u_{i-1}}{2h}\right)}^{=:(B\mathbf{u})_i} - 1 = 0, \quad i = 1, 2, ..., N$$

$$u_0 = 1, \qquad u_{N+1} = 0$$

Matrix form:

$$-\eta A\mathbf{u} + U \cdot B\mathbf{u} - \mathbf{1} = \mathbf{0}, \text{ with } U := diag(u)$$

Define the function $f : \mathbb{R}^N \to \mathbb{R}^N$

$$f(u) := -\eta \frac{d^2u}{dx^2} + u\frac{du}{dx} - 1.$$

Newton's method reads:

- Choose u_0, `TOL`
- WHILE `err` $>$ `TOL`

$$u_{k+1} = u_k - J_f^{-1}(u_k)f(u_k), \text{ where } (J_f)_{i,j} := \frac{\partial f_i}{\partial u_j}$$

$$\texttt{err} = \|u_{k+1} - u_k\|_2$$

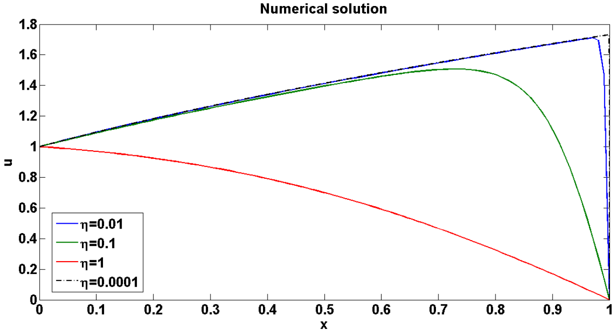
$$k = k + 1$$

Solve the $N \times N$ linear system

$$J_f(u_k)\tilde{u} = -f(u_k) \;\Rightarrow\; u_{k+1} = \tilde{u} + u_k$$

in each step.

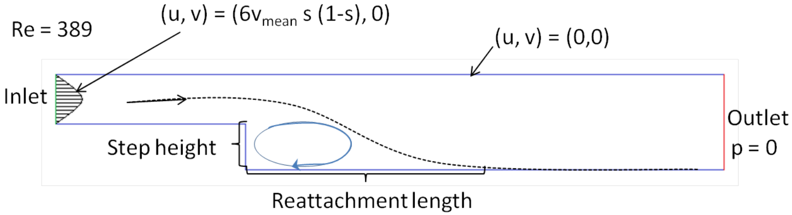
Numerical test calculations for different values of η:



- for small η a finer discretization is needed,
- results have been verified by `bvp4c.m`

Outline

2D case - Modeling with COMSOL

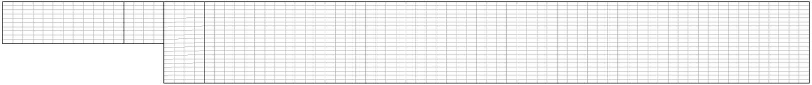


The stationary incompressible Navier-Stokes equations in COMSOL

Model Navigator $\rightarrow$ 2D
COMSOL Multiphysics $\rightarrow$ Fluid Dynamics $\rightarrow$ Incompressible Navier-Stokes $\rightarrow$ Steady-state analysis

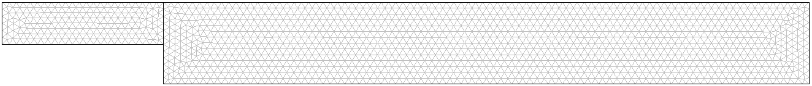
$$\begin{aligned} -\eta\Delta\mathbf{u} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p &= \mathbf{0} \\ \nabla\cdot\mathbf{u} &= 0 \end{aligned}$$

COMSOL Multiphysics – Meshing



Mapped structured mesh

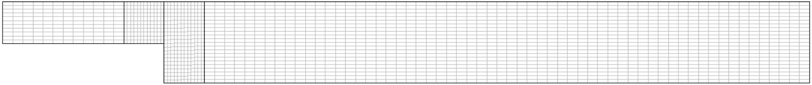
- whole domain: 80 elements in x-, 20 in y-direction
- number of elements: $1,440 \Rightarrow$ *DOFs*: $13,463$



Triangular unstructured mesh

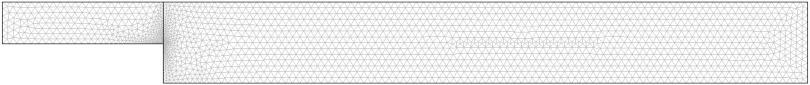
- max element size uniformly: $9e-4$
- number of elements: $2,900 \Rightarrow$ *DOFs*: $13,673$

COMSOL Multiphysics – Meshing



Locally refined structured mesh

- element number in x direction around step: factor 3 larger
- number of elements: $1,967 \Rightarrow$ *DOFs*: $17,225$



Locally refined unstructured mesh

- max element size around step corner: factor 10 smaller
- number of elements: $3,666 \Rightarrow$ *DOFs*: $17,220$

Backward facing step solution

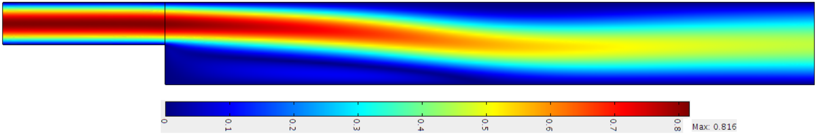


Figure: Velocity magnitude surface plot

Computation times for solver

mesh type	structured	unstructured
standard	$7.648s$	$6.691s$
refined	$9.675s$	$8.426s$

- structured mesh: longer solution time for approximately the same number of *DOFs*
- reason: stronger coupled system $\Rightarrow$ less sparse matrices

Backward facing step solution

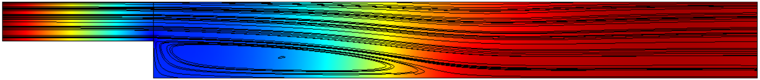


Figure: Velocity streamline and pressure surface plot

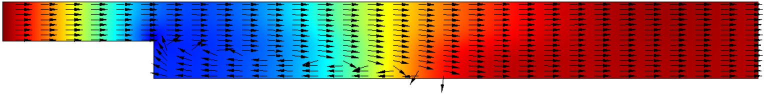
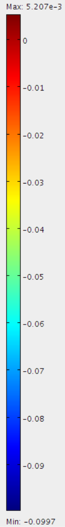
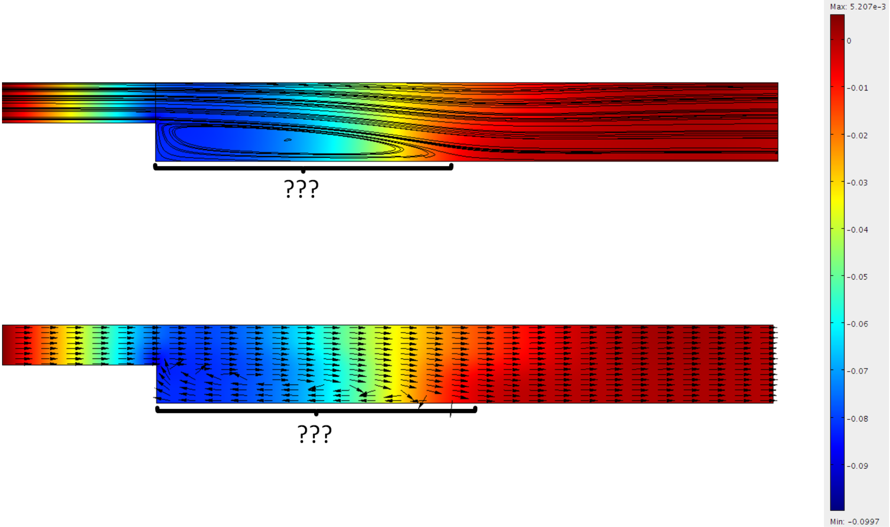


Figure: Velocity arrows and pressure surface plot



Benchmark: How to obtain the accuracy of our solution?

Backward facing step solution



⇒ measure **reattachment length** and **compare** to references

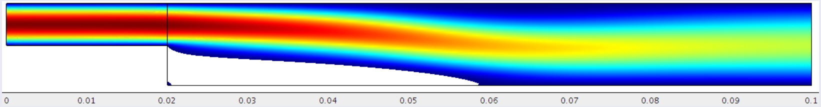
Can we measure the reattachment length exactly?

- "distance from step to *stagnation point* where flow reattaches to the lower wall"
- dimensionless quantity: $\frac{\text{reattachment length}}{\text{step height}}$
- reference experiments for $Re = 389$ for $DOFs \rightarrow \infty$ show: **7.93**

Can we measure the reattachment length exactly?

- "distance from step to *stagnation point* where flow reattaches to the lower wall"
- dimensionless quantity: $\frac{\text{reattachment length}}{\text{step height}}$
- reference experiments for $Re = 389$ for $DOFs \rightarrow \infty$ show: **7.93**

Surface plot for velocity in x-direction ≥ 0



reattachment length: $0.0392 \Rightarrow \frac{0.0392}{0.0052} \approx \mathbf{7.54}$ for $17,225$ $DOFs$

Conclusion

- For the numerical treatment of the two-dimensional Navier-Stokes equations, it may be helpful to consider an analogous one-dimensional ODE,
- Newton's method has been used to treat the non-linearity in this ODE,
- The simulation results obtained by COMSOL Multiphysics show reasonable physical behavior,
- The determination of the reattachment length is a crucial part of the benchmark.

Thanks for your attention!

Are there any questions ?