LECTURE 1

Computational Fluid Dynamics (CFD) wb1428

Mathieu Pourquie
m.j.b.m.pourquie@wbmt.tudelft.nl
http://www.ahd.tudelft.nl/∼mathieu/CFD.html
Fluid dynamics group
Stromingsleer
building part 5B
room 1-32
015-2782997

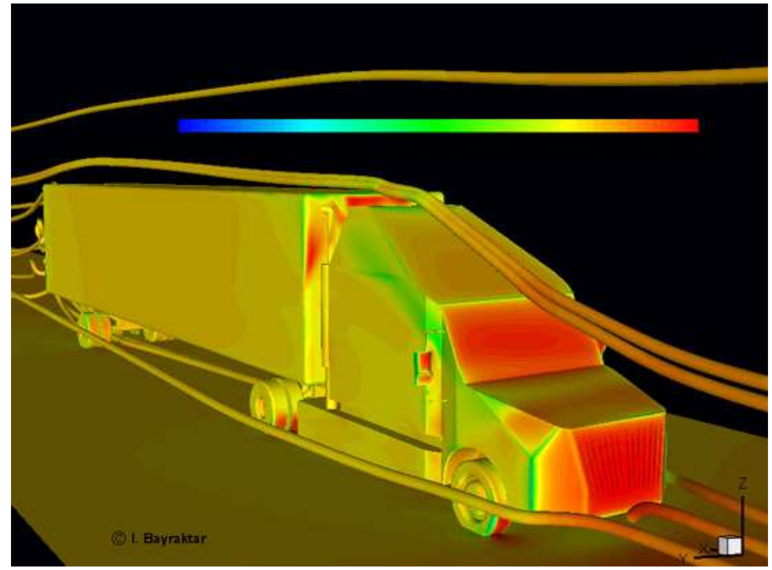
- relevance CFD
- subject of lectures
- examination
- material
- questionnaire

What is CFD about?

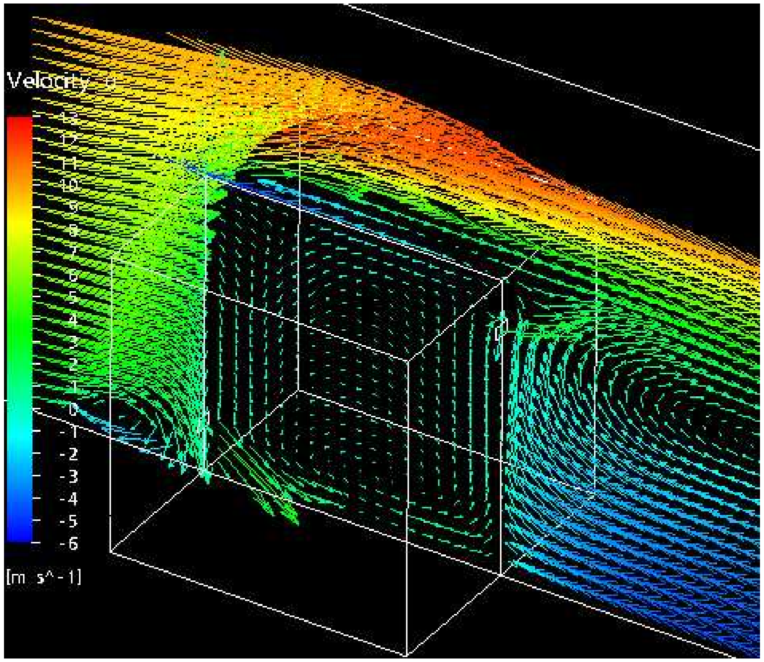
- Fluid dynamics
- Theoretical
- Experimental
- CFD: Computational

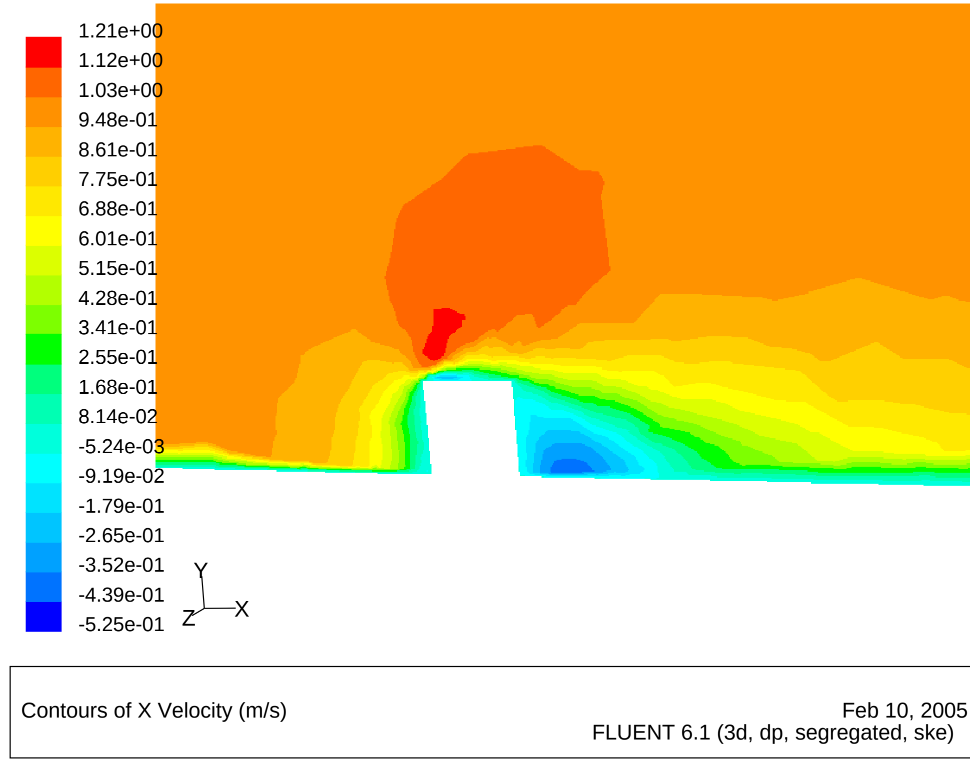
Why (Computational) Fluid Dynamics?

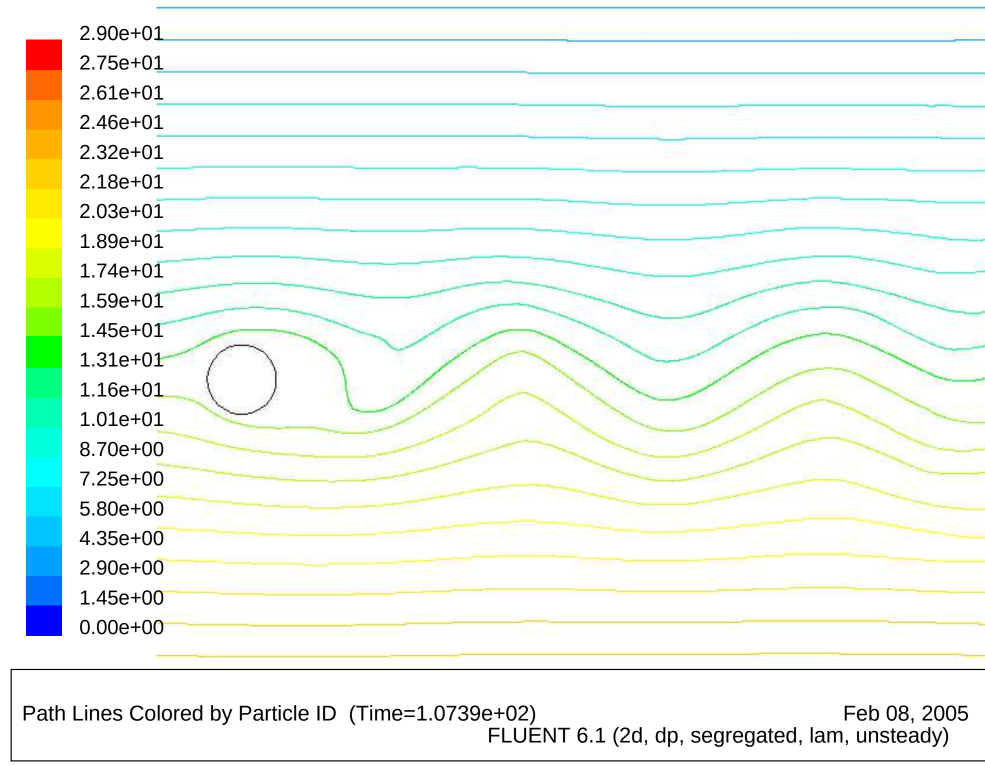
- flows in nature and technology
- flow combined with heat transfer
- flow combined with particle transfer
- flows with free surfaces (water waves)
- flow with chemistry
- flows with moving boundaries

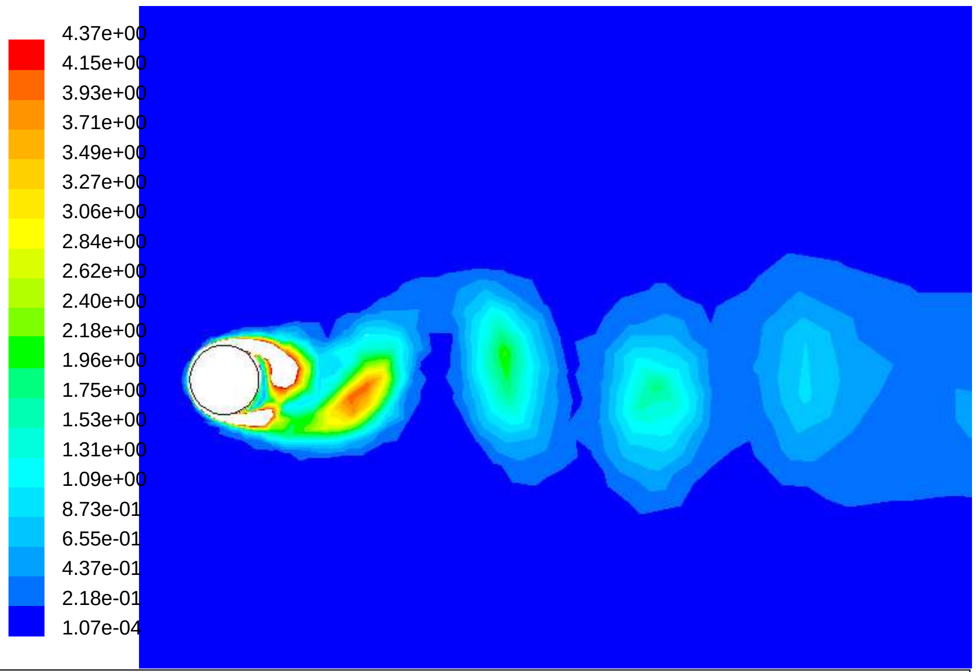


© I. Bayraktar

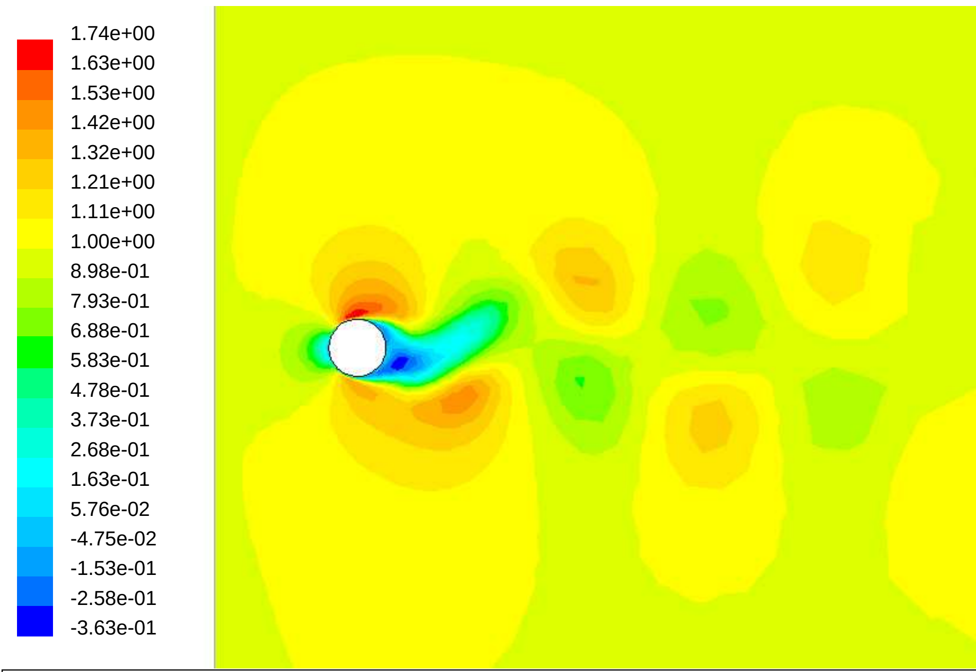

Velocity
-1
-2
-3
-4
-5
-6
[m s^-1]


1.21e+00
1.12e+00
1.03e+00
9.48e-01
8.61e-01
7.75e-01
6.88e-01
6.01e-01
5.15e-01
4.28e-01
3.41e-01
2.55e-01
1.68e-01
8.14e-02
-5.24e-03
-9.19e-02
-1.79e-01
-2.65e-01
-3.52e-01
-4.39e-01
-5.25e-01
Y
X
Z
Contours of X Velocity (m/s)
Feb 10, 2005
FLUENT 6.1 (3d, dp, segregated, ske)

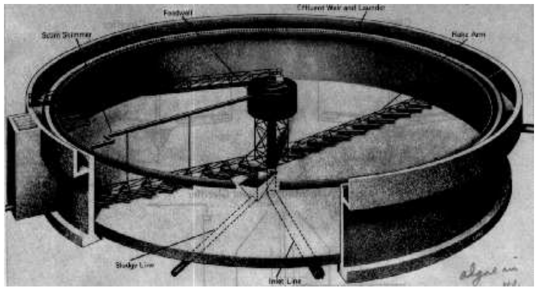

2.90e+01
2.75e+01
2.61e+01
2.46e+01
2.32e+01
2.18e+01
2.03e+01
1.89e+01
1.74e+01
1.59e+01
1.45e+01
1.31e+01
1.16e+01
1.01e+01
8.70e+00
7.25e+00
5.80e+00
4.35e+00
2.90e+00
1.45e+00
0.00e+00
Path Lines Colored by Particle ID (Time=1.0739e+02)
Feb 08, 2005
FLUENT 6.1 (2d, dp, segregated, lam, unsteady)

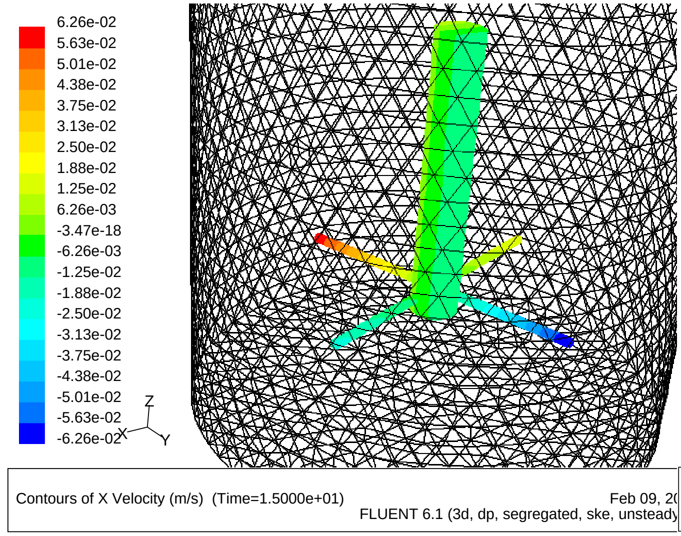


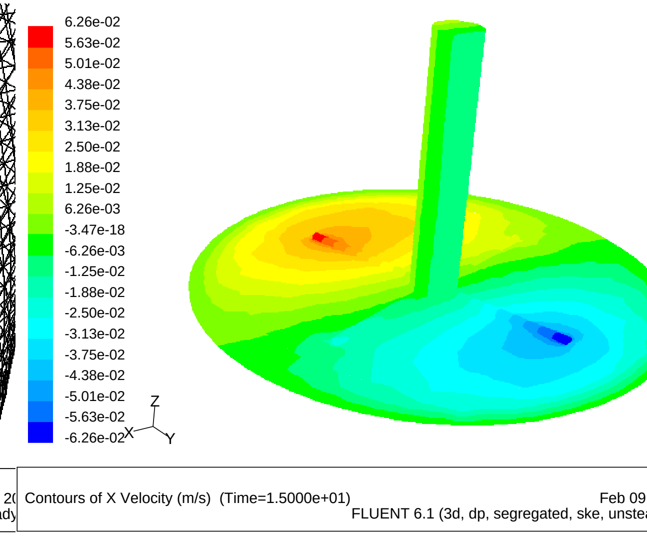
Contours of Vorticity Magnitude (1/s) (Time=1.0739e+02) Feb 08, 2005
FLUENT 6.1 (2d, dp, segregated, lam, unsteady)

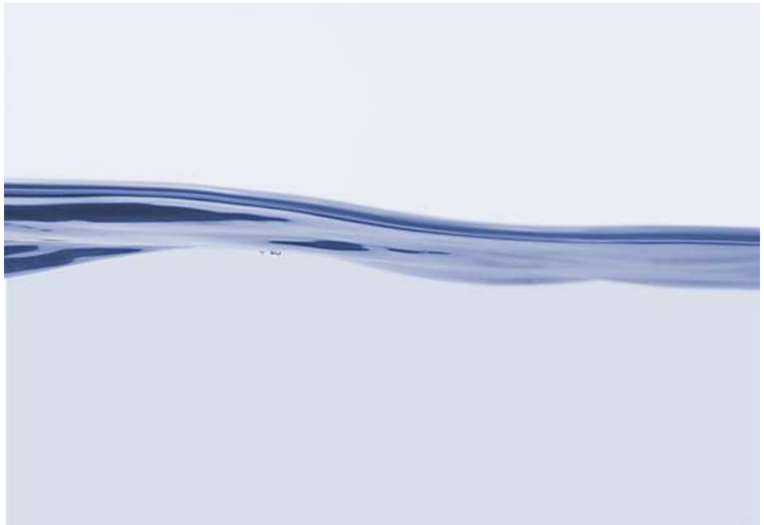


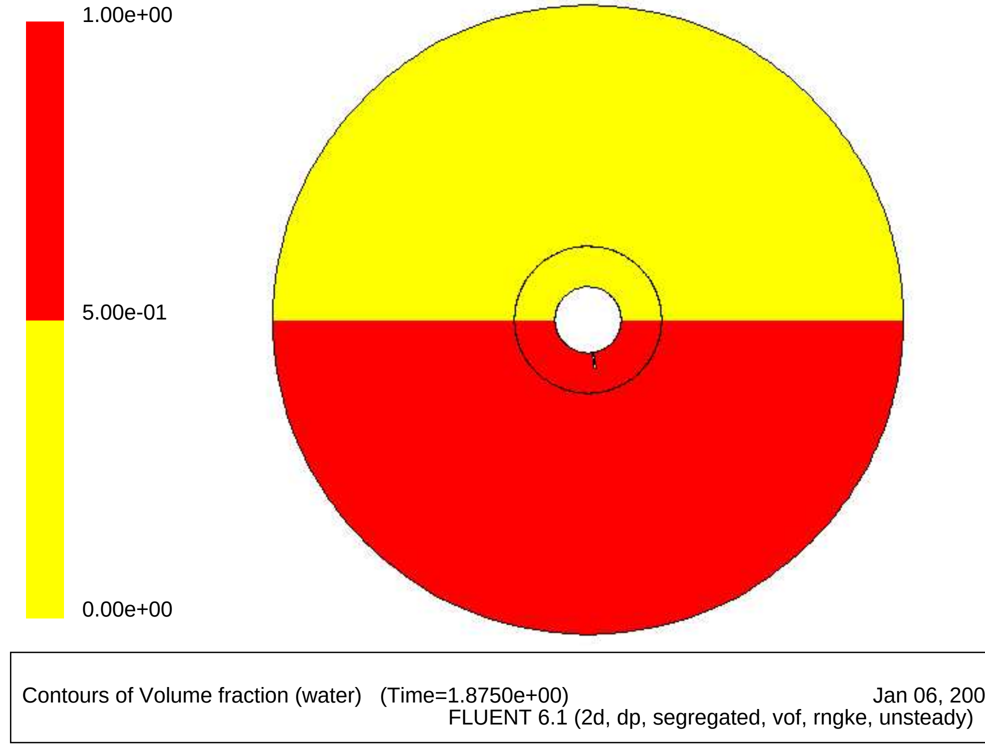
Contours of X Velocity (m/s) (Time=1.0739e+02) Feb 08, 2005
FLUENT 6.1 (2d, dp, segregated, lam, unsteady)

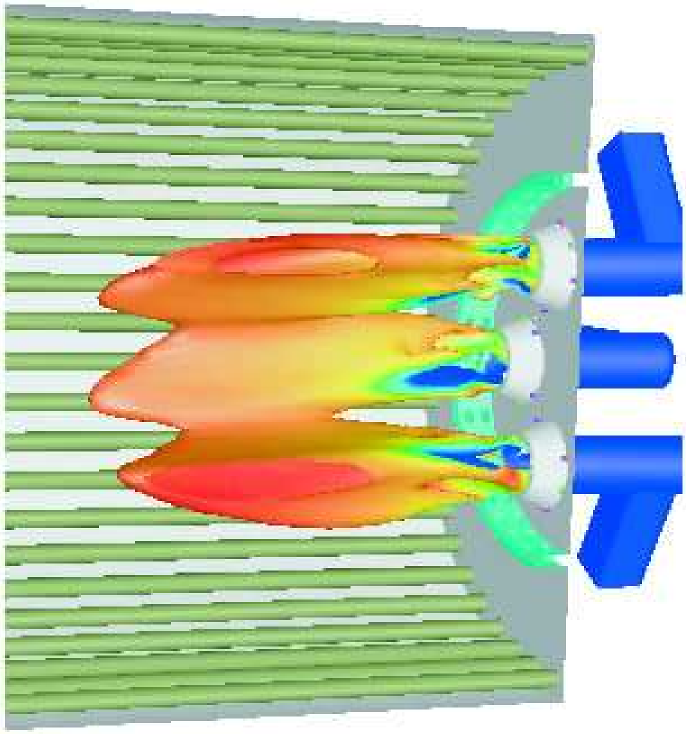


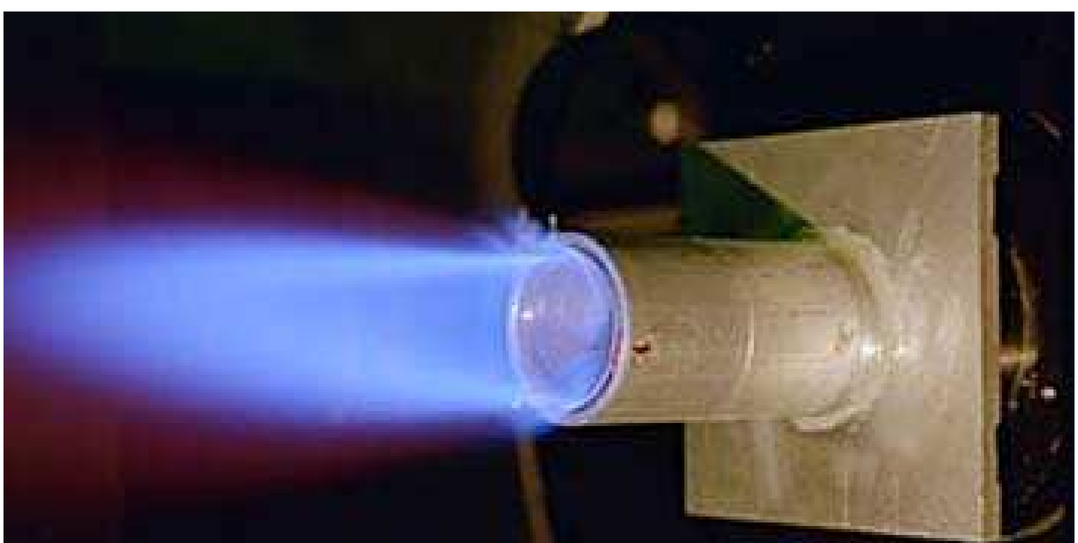

6.26e-02
5.63e-02
5.01e-02
4.38e-02
3.75e-02
3.13e-02
2.50e-02
1.88e-02
1.25e-02
6.26e-03
-3.47e-18
-6.26e-03
-1.25e-02
-1.88e-02
-2.50e-02
-3.13e-02
-3.75e-02
-4.38e-02
-5.01e-02
-5.63e-02
-6.26e-02
Z
X
Y
Contours of X Velocity (m/s) (Time=1.5000e+01)
Feb 09, 2
FLUENT 6.1 (3d, dp, segregated, ske, unsteady


6.26e-02
5.63e-02
5.01e-02
4.38e-02
3.75e-02
3.13e-02
2.50e-02
1.88e-02
1.25e-02
6.26e-03
-3.47e-18
-6.26e-03
-1.25e-02
-1.88e-02
-2.50e-02
-3.13e-02
-3.75e-02
-4.38e-02
-5.01e-02
-5.63e-02
-6.26e-02
Z
X
Y
Contours of X Velocity (m/s) (Time=1.5000e+01)
Feb 09, 2
FLUENT 6.1 (3d, dp, segregated, ske, unstead




1.00e+00
5.00e-01
0.00e+00
Contours of Volume fraction (water) (Time=1.8750e+00)
Jan 06, 200
FLUENT 6.1 (2d, dp, segregated, vof, rngke, unsteady)





Why Computational Fluid Dynamics?

- speed of computers
- price of computers
- data accessible
- complicated geometry
- try out ideal circumstances

What is this lecture about?

- Fluid Dynamics on computer
- Solve fluid flow equations on computer
- Write your own code or
- Use a commercial package: Fluent

Writing your own code:

- takes time
- takes experience
- spend time on non-essential subjects

BUT:

- you see the code
- you can repair bugs

Using a commercial package:

- preprocessor
- postprocessor
- programming done for you
- manual
- help desk
- takes less time
- takes less experience

BUT:

- you do not see the code
- no bug repair

IN PRACTICE:

- you will use a commercial code
- continuity, manual, helpdesk

Do commercial codes always work?

NO.

- commercial code is collection of tools
- commercial codes have bugs

Commercial packages

- fluent http://www.fluent.com
- CFX http://www-waterloo.ansys.com/cfx/
- starcd http://www.cd-adapco.com note same as comet!
- comet http://www.cd-adapco.com note same as starcd!
- femlab http://www.comsol.com/
- flow3D http://www.flow3d.com

- CFD
- Fluid Dynamics
- Computer
- you

questionnaire

- did you do advanced fluid dynamics?
- did you do anything numerical before (v Kan, numerical analysis)
- what programming languages do you know (Fortran, C, C++, Pascal, matlab)

I assume you

- know things presented in first year fluid dynamics (wb1123d2)
- know things presented in second year fluid dynamics (wb1224)
- know things presented in Advanced fluid dynamics (wb1422atu/w1427b)
- do third year Heat and Mass Transfer (wb1321)
- have some numerical background WI3097WB
- Do you? Please answer the questionnaire

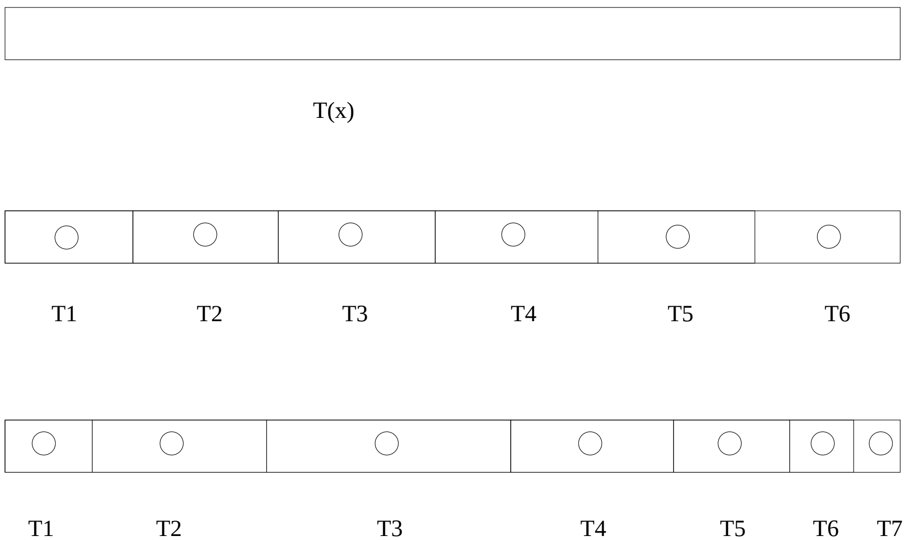
I assume you

- know how to calculate a Taylor series
- know what complex numbers are
- can work with complex numbers
- know what a diffusion or heat transport equation is, know how a solution looks like
- know what a transport or advection equation is, know how a solution looks like
- can use Excel or matlab or programming language (analyze results)
- Do you? Please answer the questionnaire

Objective: learn about flow solver

What does a flow solver do

Discretization



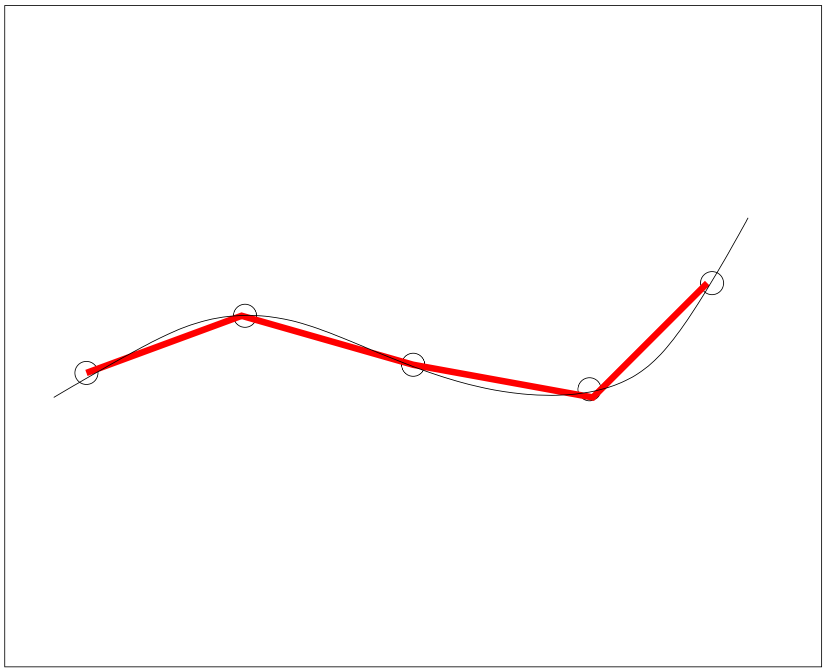
Discretization $\longrightarrow$ equations

Solution of equations

Visualization of solution

Why discretize?

- more equations



- profile assumption
- for profile: solution

Objectives of lectures (in principle)

- simulate fluid flow using commercial CFD package
- numerical method
- numerical grid
- physical model
- discuss and validate results

- background of the package
- simulate model problems
- some model problems in matlab

Plan

- use Fluent
- use matlab
- see relation

Examination

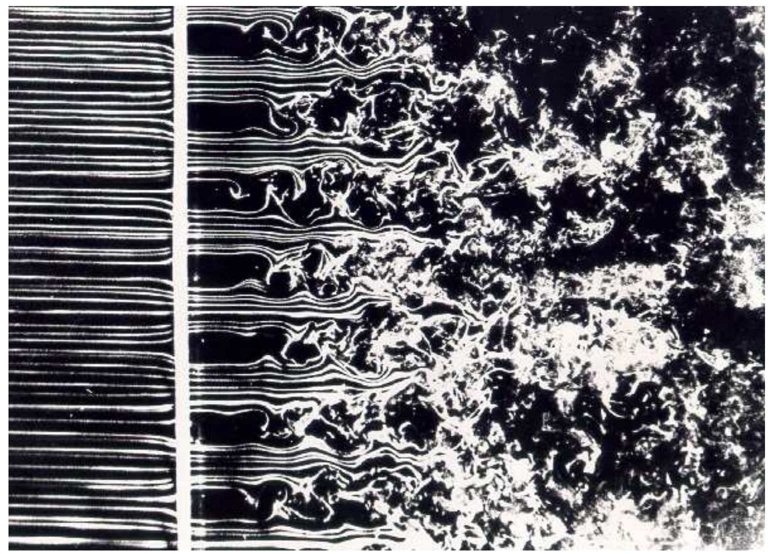
- assignment: fluid flow calculation with Fluent
- assignment: matlab
- combination possible
- suggest your own flow
- suggest your own assignment
- assignment: exercise by hand and with matlab (optional)
- three credit points
- you want more? Come up with a more realistic Fluent problem (1 point)

Material

- hand-outs
- sheets on www
- background material:
  - Ferziger & Peric, Computational methods for fluid dynamics, Springer
  - J. van Kan, Numerieke wiskunde voor technici, DUP
  - Delft Fluent user http://www.ahd.tudelft.nl/∼mathieu/fluent_group/in

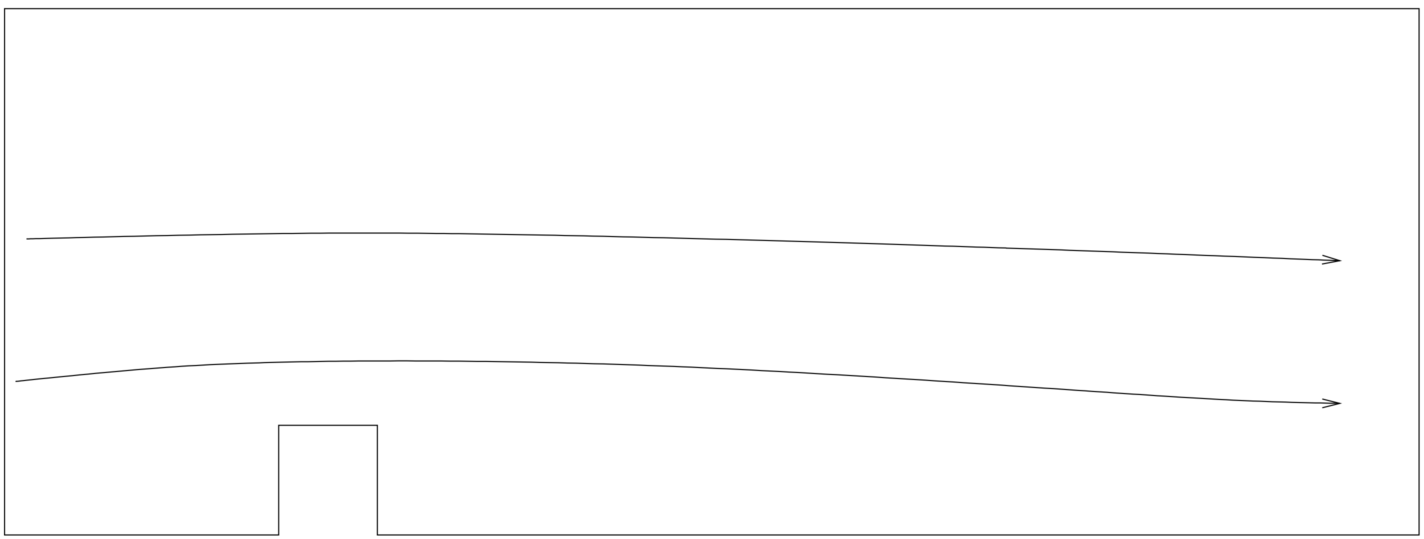
limitations of lectures (in principle)

- incompressible (density = constant)
- Flow, plus possibly
  - physical models for turbulence
  - heat transfer



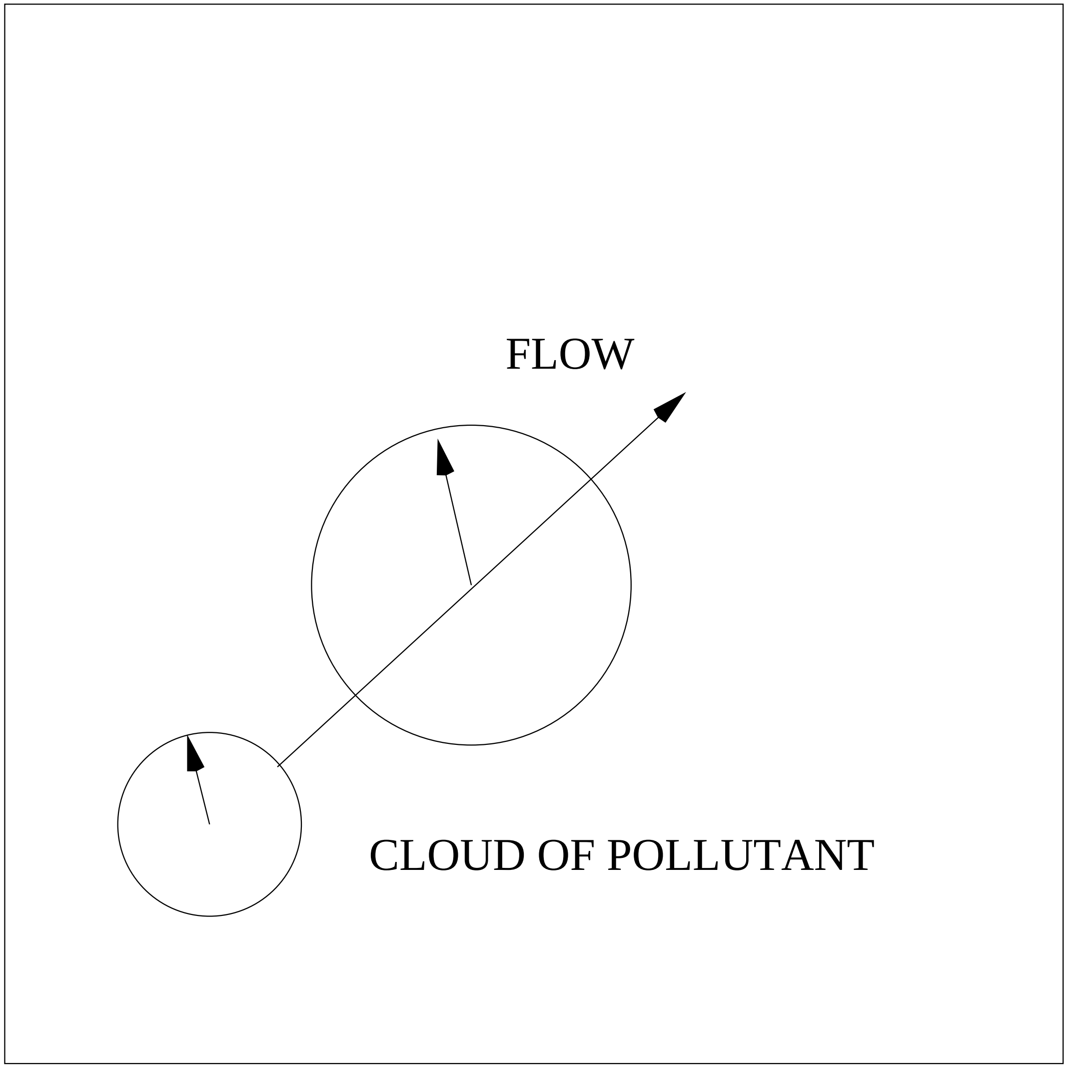
first thing to do (next time)

- use the package
- first exercise
- ASK QUESTIONS
- DISCUSS RESULTS

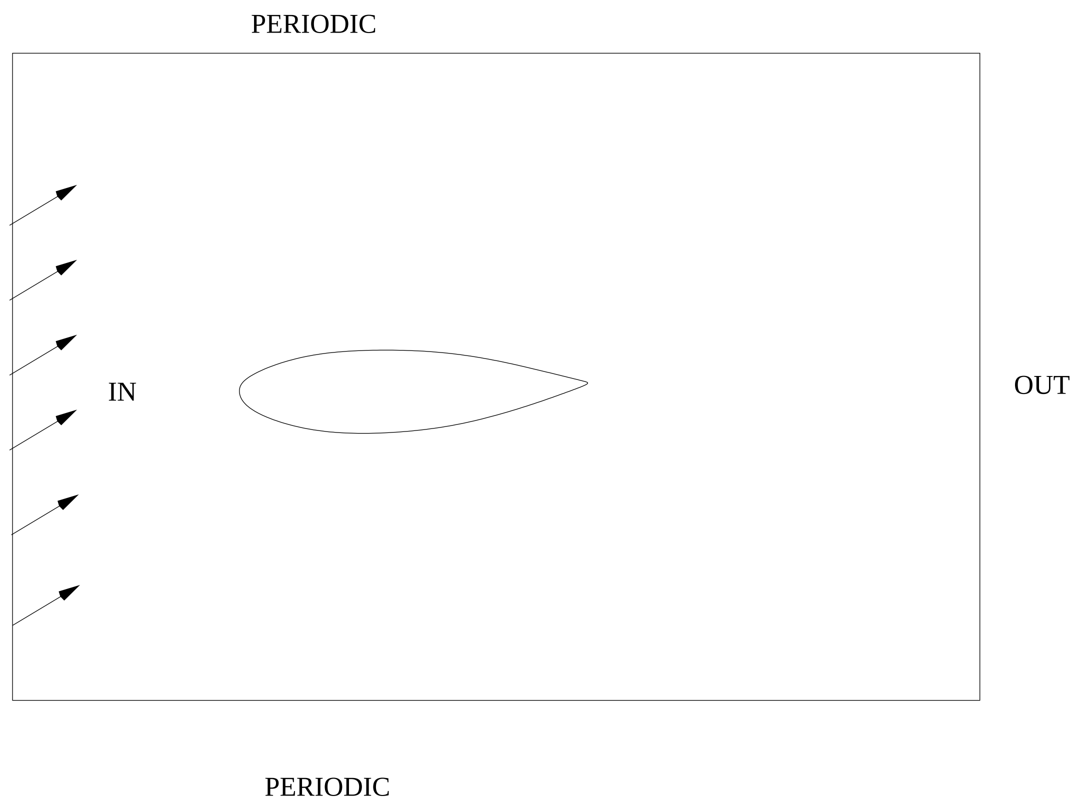


First exercise with the package Fluent

- make the geometry (Gambit)
- discretize (make the grid, Gambit)
- solve equations (flow, heat, Fluent)
- visualize the results (Fluent)



- solve pollution problem
- wind field known
- use several discretisations



- more advanced grids
- periodic boundary conditions
- a turbulence model

Navier-Stokes equations (N-S)

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$\begin{aligned} \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} &= \frac{-1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial x^2} + \nu\frac{\partial^2 u}{\partial y^2} \\ I + ADVECTION &= PRESSURE + DIFFUSION \end{aligned}$$