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

http://www.ahd.tudelft.nl
info for students
wb1428 Computational Fluid Dynamics

Fluid dynamics group
Stromingsleer
building part 5B
room 1-32
015-2782997

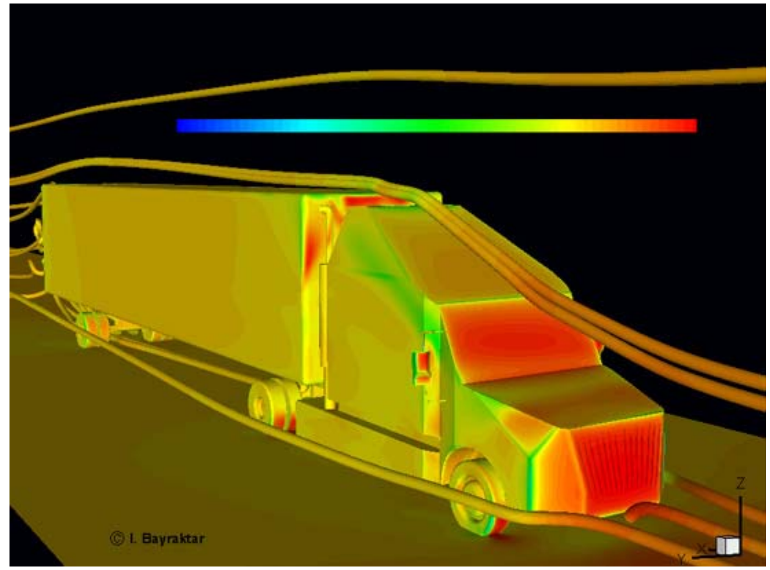
- relevance CFD
- preview of CFD
- subject of lectures
- examination
- material
- questionnaire

What is CFD about?

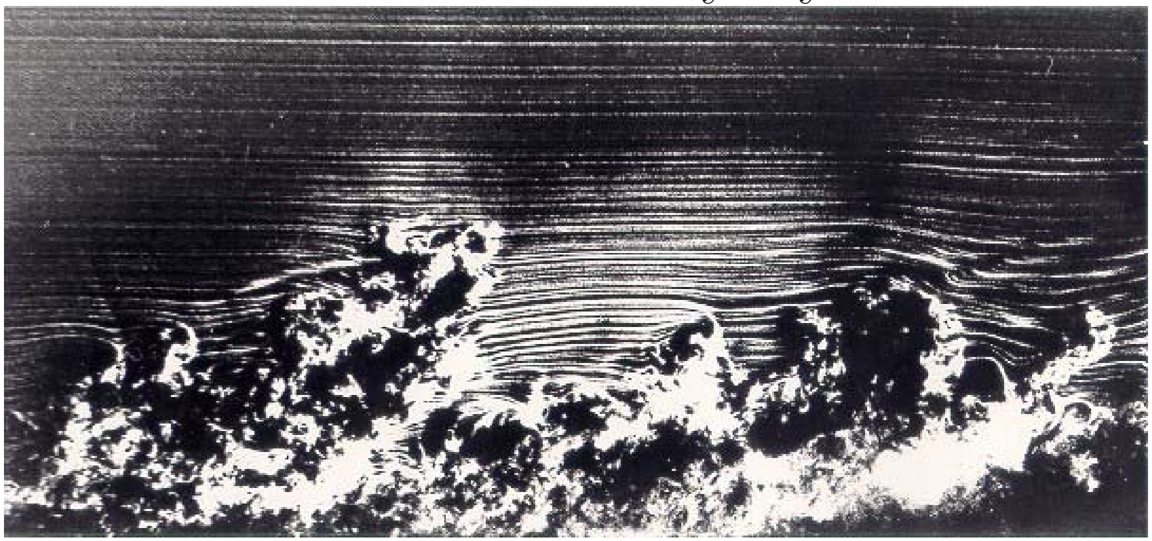
- Fluid dynamics
- Theoretical
- Experimental
- CFD: Computational

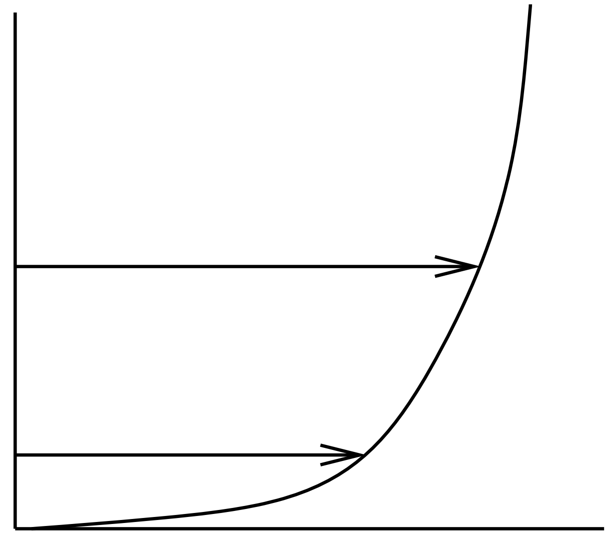
Why (Computational) Fluid Dynamics?

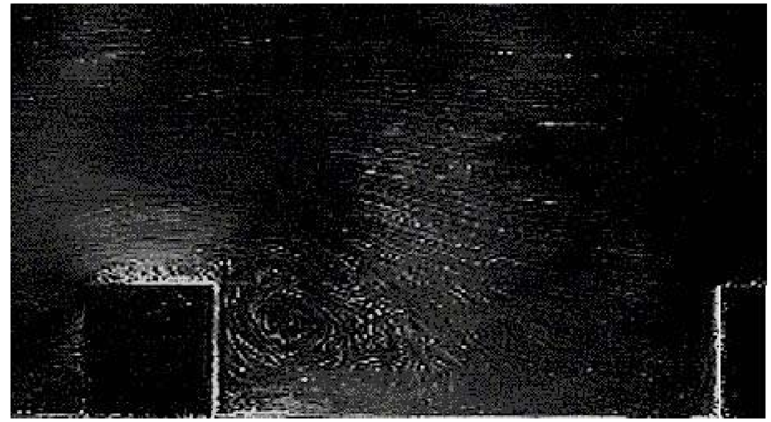
- flows in nature and technology
- flows combined with heat transfer
- flows combined with particle transfer
- flows with free surfaces (water waves)
- flows with free surfaces (oil-water droplet)
- flows with chemistry
- flows with moving boundaries

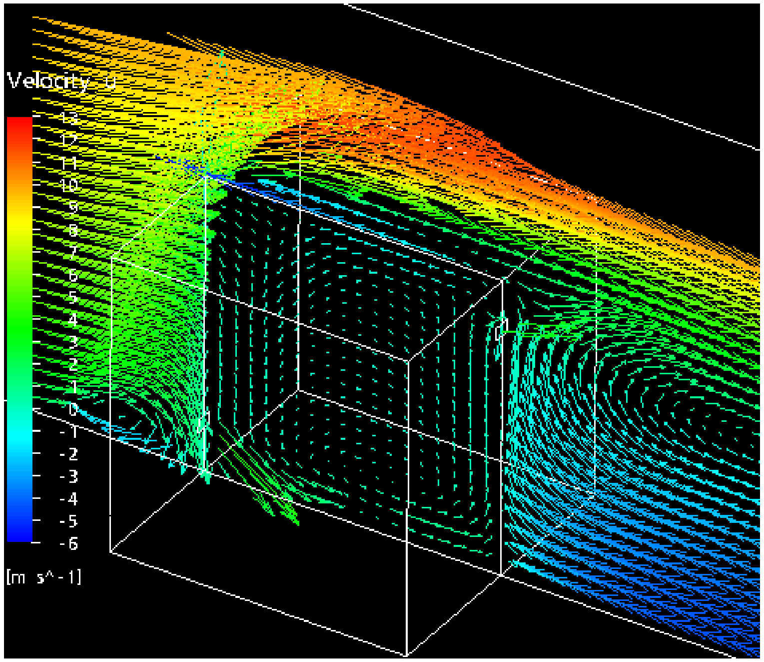

© I. Bayraktar
Z
X
Y

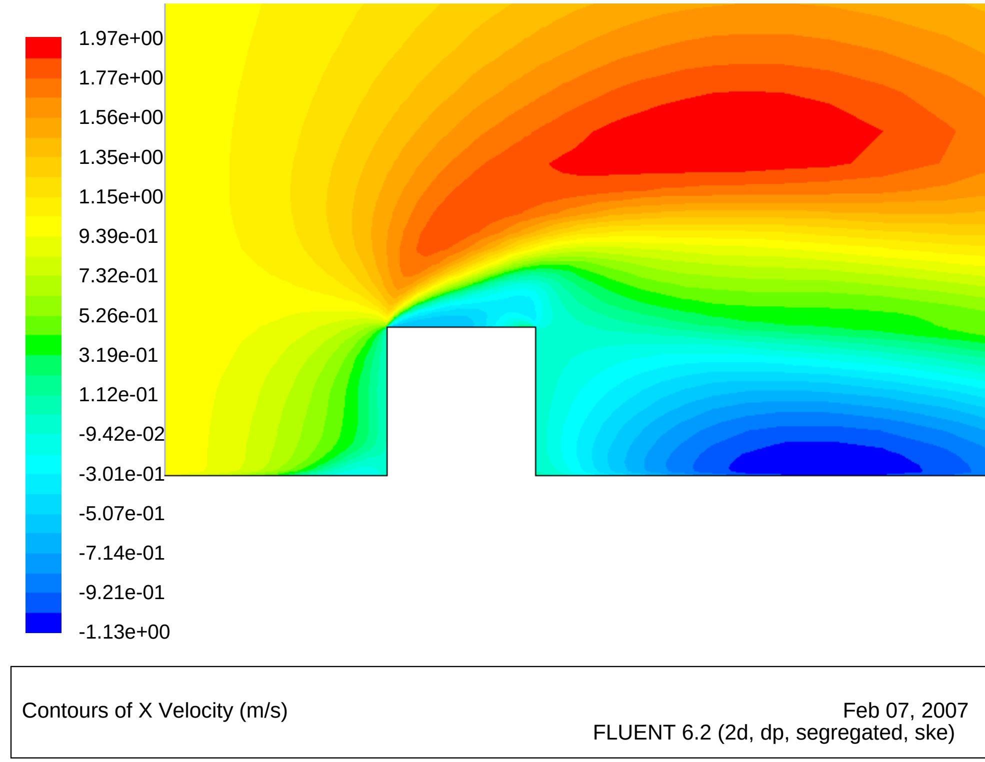
Near the surface: boundary layer flow.

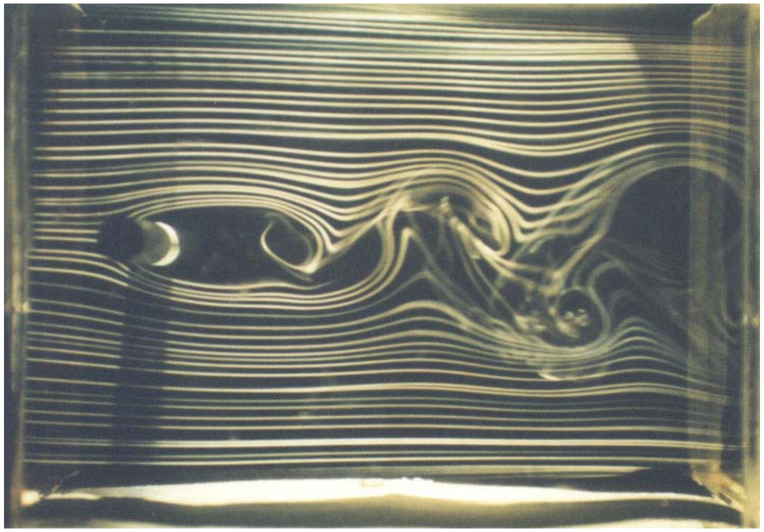


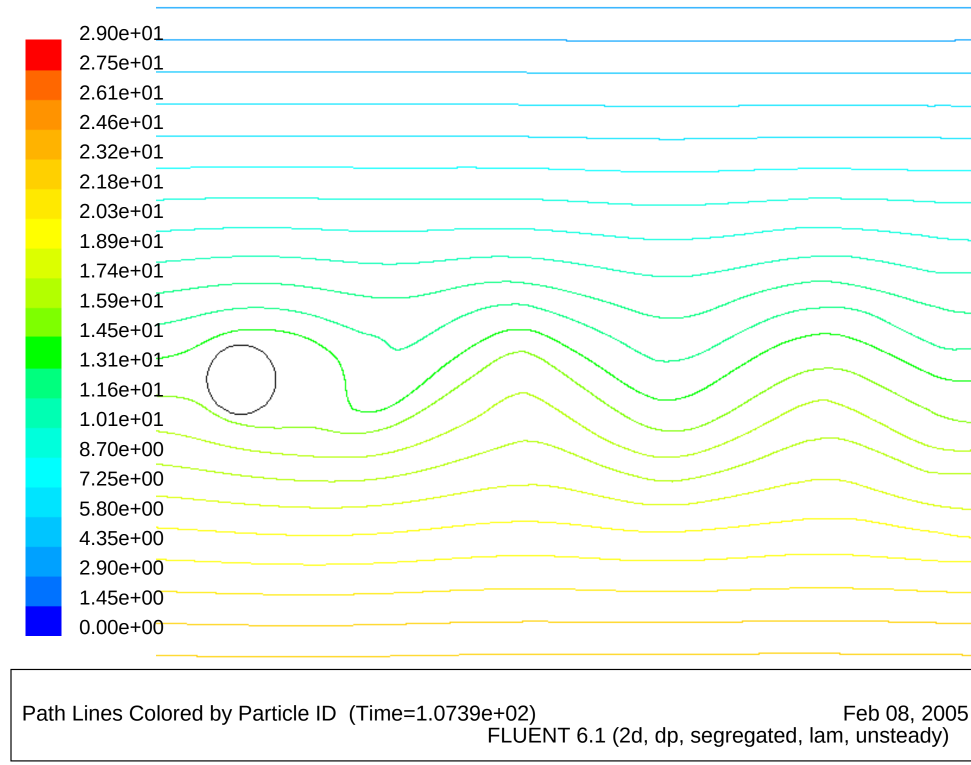


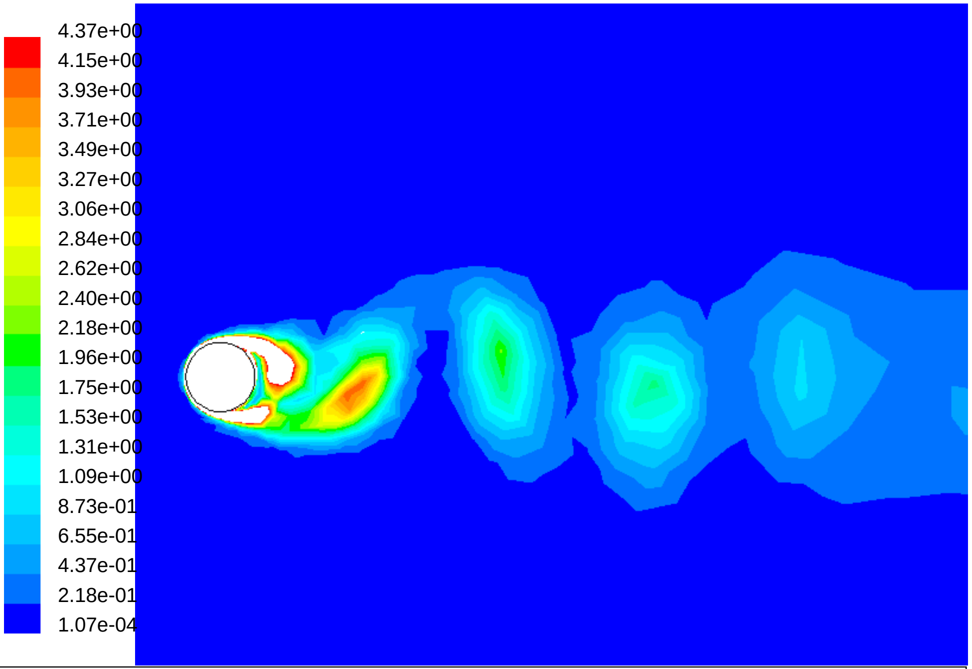



Velocity u
-1
-2
-3
-4
-5
-6
[m s^-1]

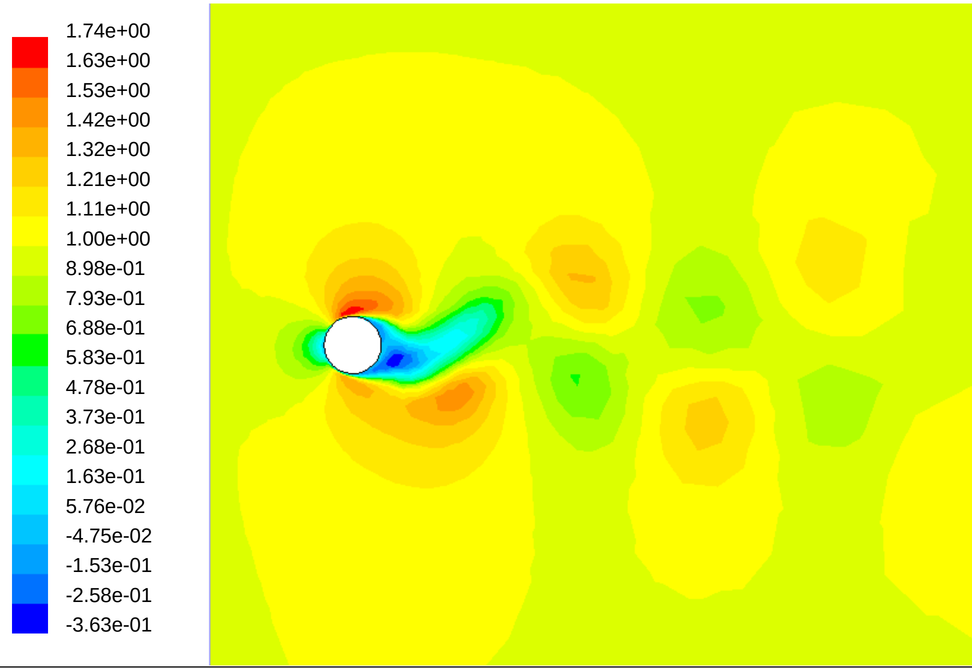

1.97e+00
1.77e+00
1.56e+00
1.35e+00
1.15e+00
9.39e-01
7.32e-01
5.26e-01
3.19e-01
1.12e-01
-9.42e-02
-3.01e-01
-5.07e-01
-7.14e-01
-9.21e-01
-1.13e+00
Contours of X Velocity (m/s)
Feb 07, 2007
FLUENT 6.2 (2d, dp, segregated, ske)




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)

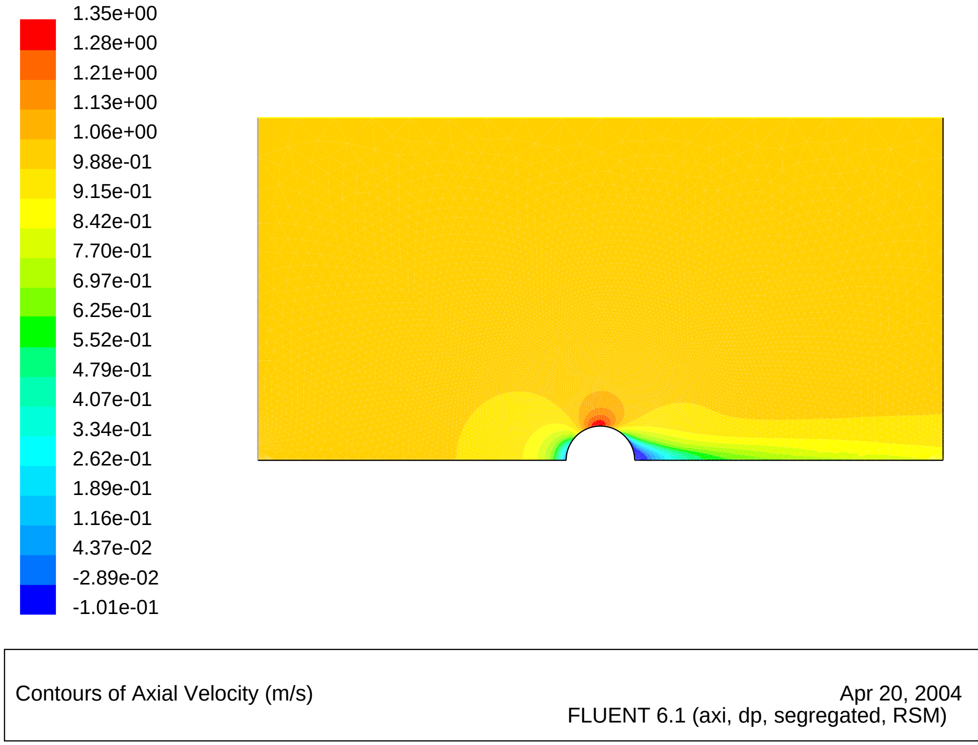


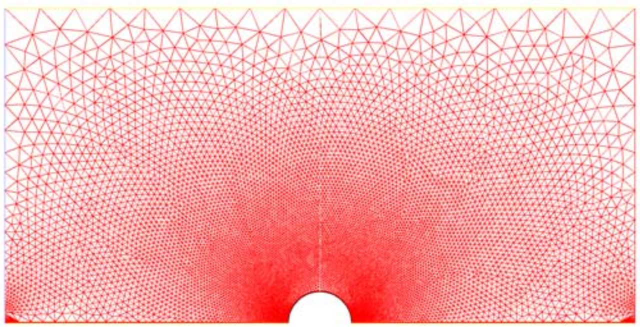
Contours of Vorticity Magnitude (1/s) (Time=1.0739e+02) Feb 08, 2005
FLUENT 6.1 (2d, dp, segregated, lam, unsteady)



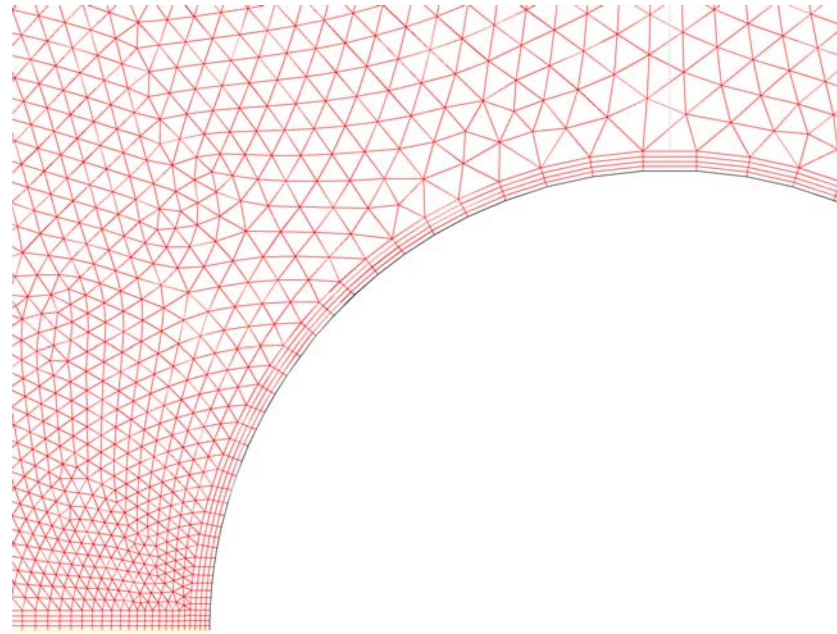
Contours of X Velocity (m/s) (Time=1.0739e+02) Feb 08, 2005
FLUENT 6.1 (2d, dp, segregated, lam, unsteady)

A sphere

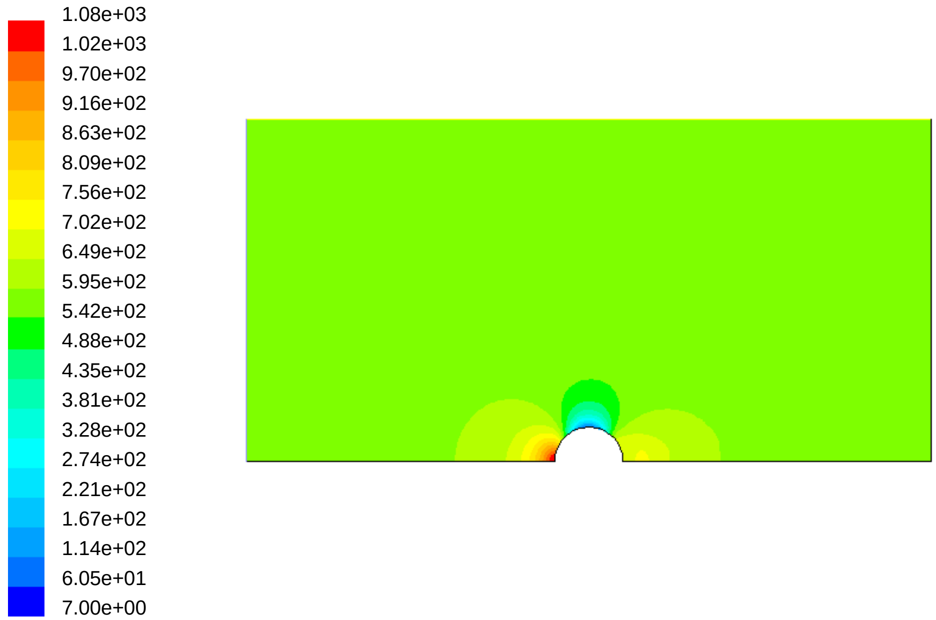

1.35e+00
1.28e+00
1.21e+00
1.13e+00
1.06e+00
9.88e-01
9.15e-01
8.42e-01
7.70e-01
6.97e-01
6.25e-01
5.52e-01
4.79e-01
4.07e-01
3.34e-01
2.62e-01
1.89e-01
1.16e-01
4.37e-02
-2.89e-02
-1.01e-01
Contours of Axial Velocity (m/s)
Apr 20, 2004
FLUENT 6.1 (axi, dp, segregated, RSM)



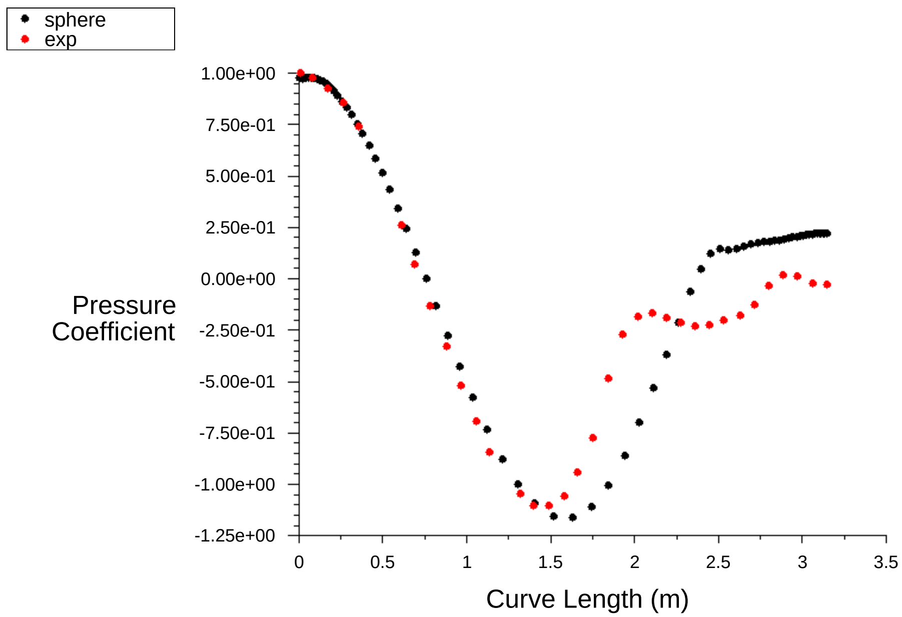
Grid Apr 20, 2004
FLUENT 6.1 (axi, dp, segregated, RSM)



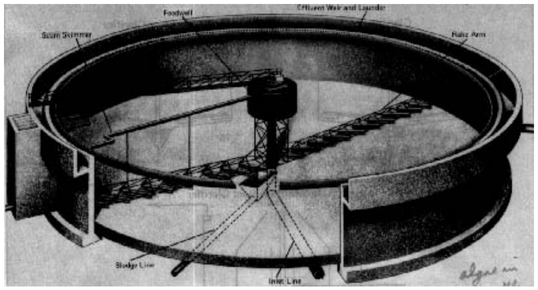
Grid Apr 20, 200
FLUENT 6.1 (axi, dp, segregated, RSM

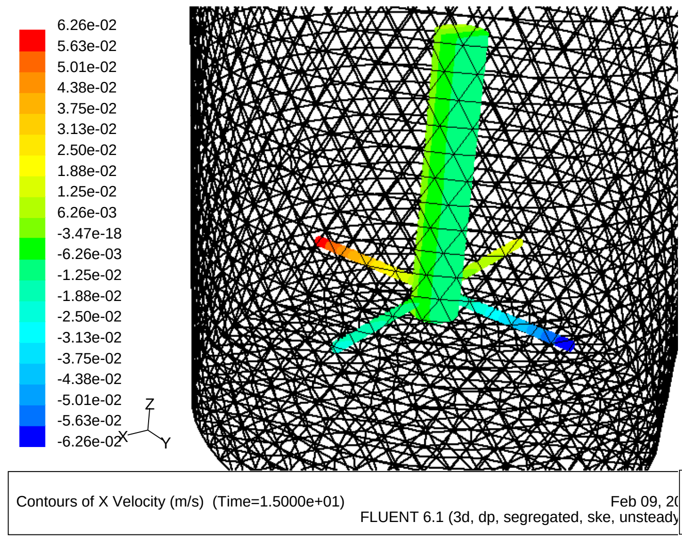


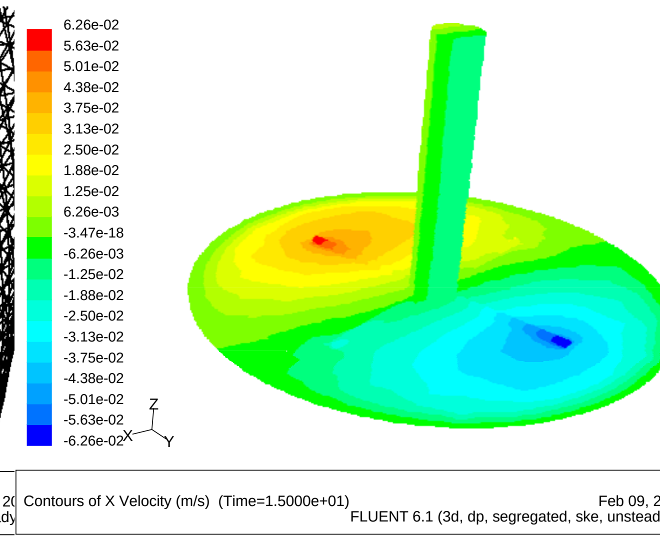
Contours of Static Pressure (pascal) Apr 20, 2004
FLUENT 6.1 (axi, dp, segregated, RSM)

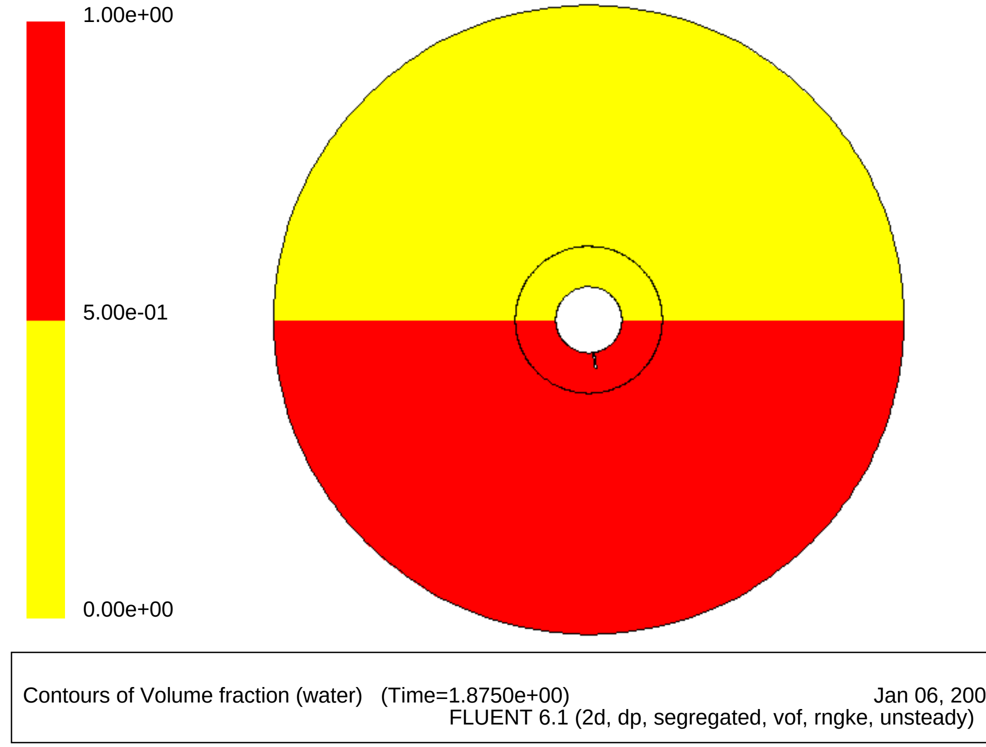


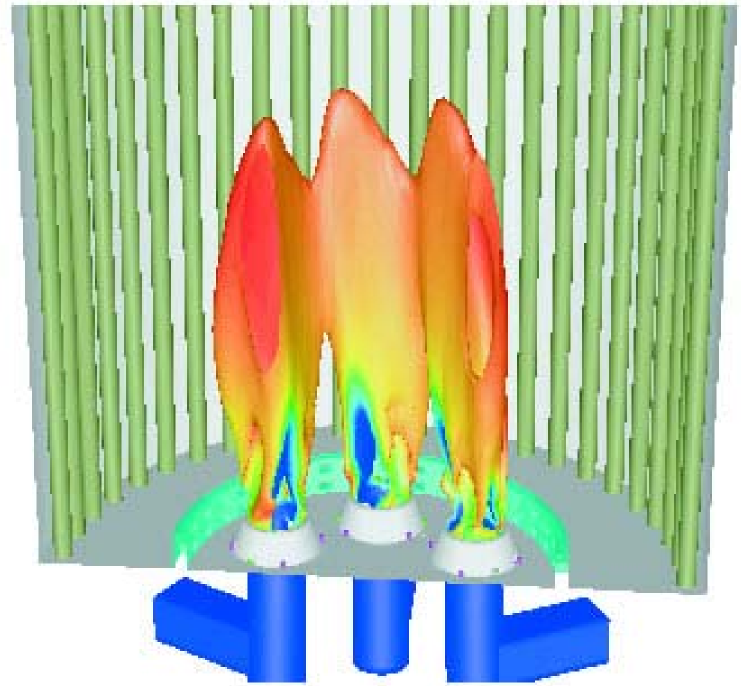
Pressure Coefficient vs. Curve Length Apr 20, 2004
FLUENT 6.1 (axi, dp, segregated, RSM)


Scum Skimmer
Feedwell
Effluent Weir and Launder
Rake Arm
Sludge Line
Inlet Line


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
- speed of numerical algorithms
- data accessible
- change geometry at little cost
- try out ideal circumstances

Activities of a CFD modeler

Writing a big code

- write code (10 %)
  - paper work 8%
    - writing out an integral balance
    - interpolation
  - implementation 2%
- find bugs (80 %)
- validate (10%)

Activities of a CFD modeler
Calculating realistic cases

- Clean the geometry (80 %)
- Make a grid (15 %)
- Validate the code (3%)
- Run the problem (2%)

VALIDATION????

- codes contain bugs
- user chooses a method
- user determines flow regime (turbulent, laminar)
- grid quality

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 understand what the code does
- 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
- you do not always understand what the code does
- no bug repair

IN PRACTICE:

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

Do commercial codes always work?

NO.

- commercial codes have bugs
- numerical algorithms do not always work
- commercial code is collection of tools
- you still need to understand the tools

Commercial packages

- fluent http://www.fluent.com
- CFX http://www-waterloo.ansys.com/cfx/
- ansys http://www-waterloo.ansys.com/
- 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

Objectives of lectures (in principle)

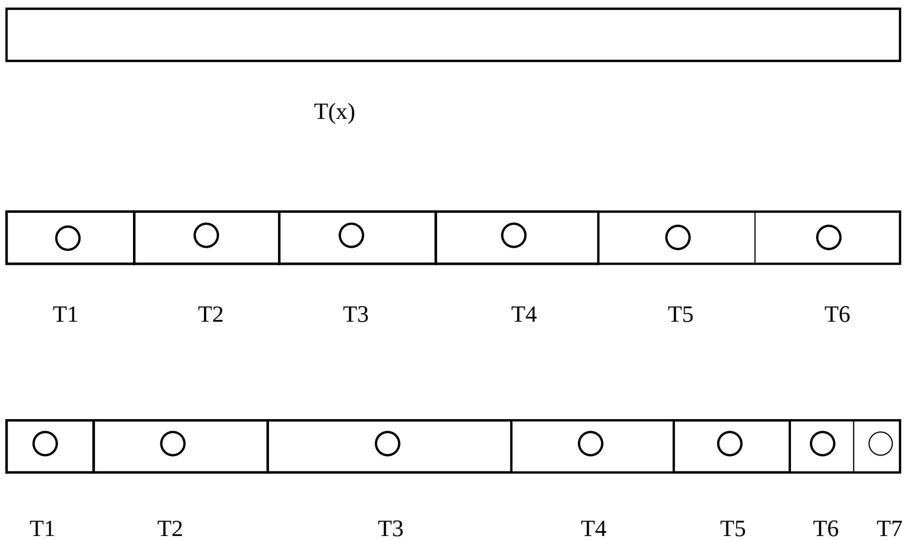
- background of the package
- simulate model problems
- some model problems in matlab

- simulate fluid flow using commercial CFD package
- use/choose numerical method
- make numerical grid
- use/choose physical model
- discuss and validate results
- why validation?
  - rounding errors
  - approximation errors
  - modeling errors
  - CFD package errors

Objective: learn about flow solver

What does a flow solver do

Discretization



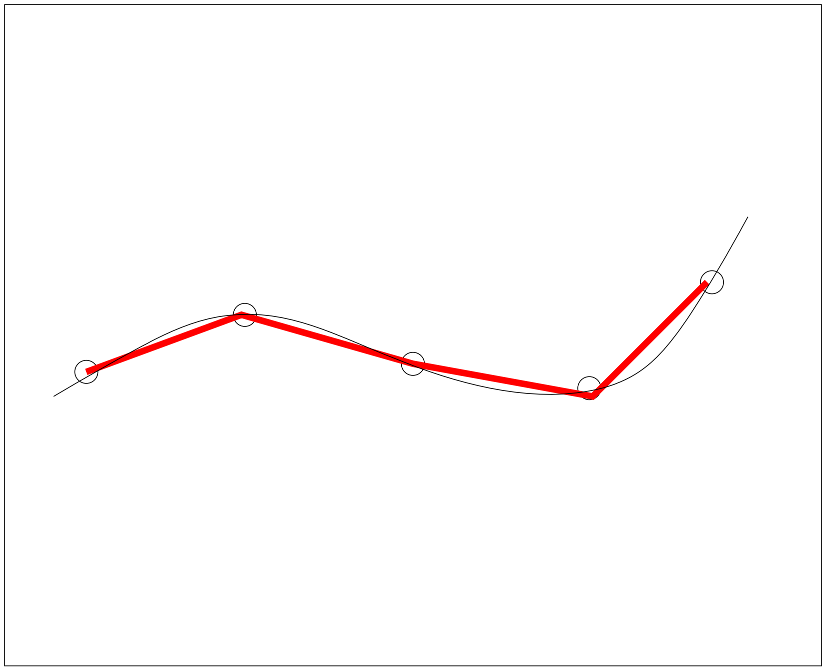
Discretization $\rightarrow$ equations

Solution of equations

Visualization of solution

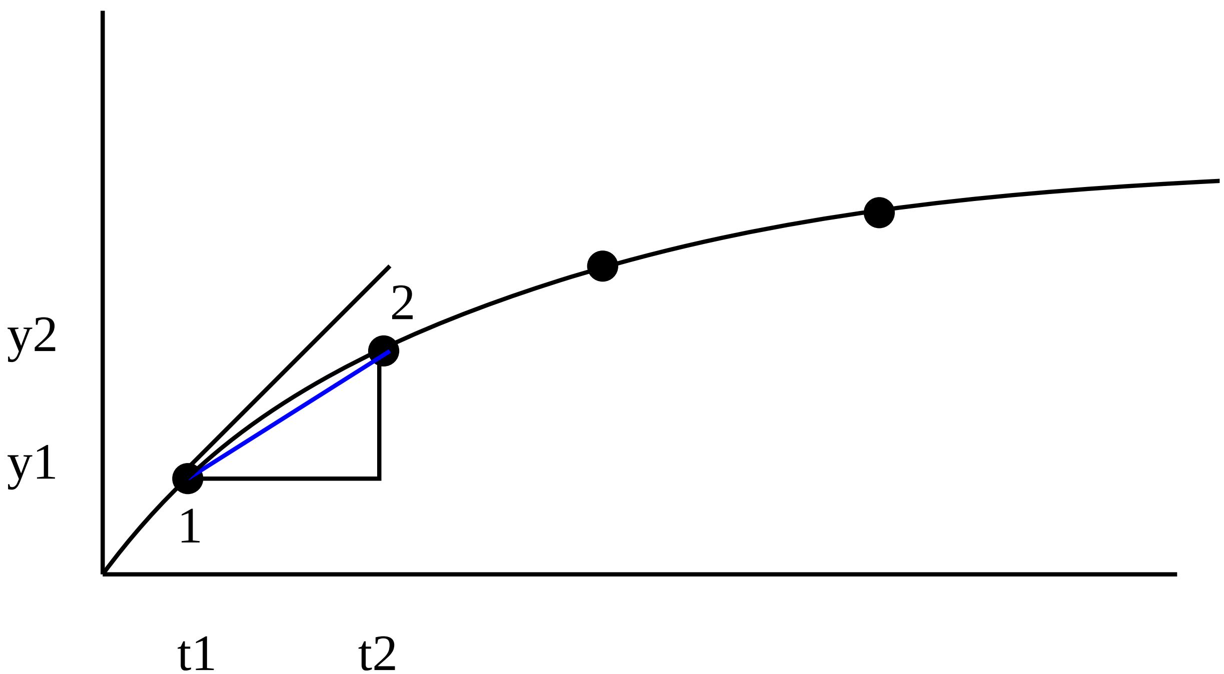
Why discretize?

- more equations



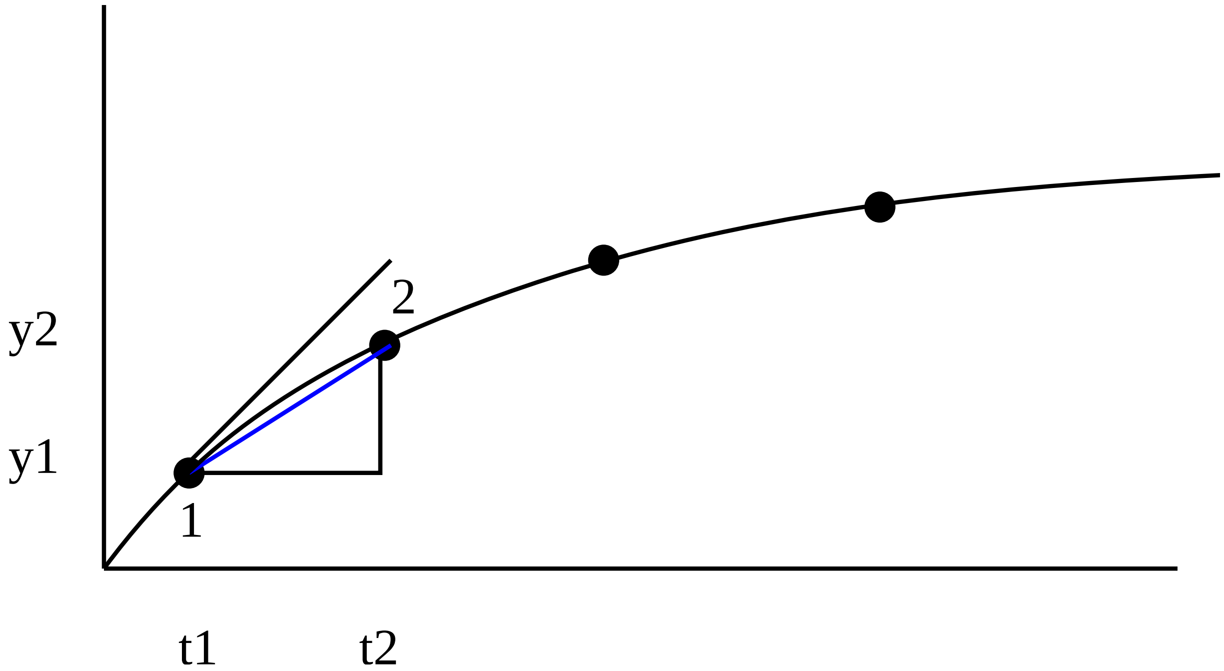
- profile assumption
- when profile: ”easy” equation/solution (computer is dumm!)
- x (space) or t (time)

- linear profile assumption in t between points
- derivative $\frac{dy}{dt} \sim \frac{\Delta y}{\Delta t}$
- derivative can be calculated (approximated) IF y known in points
- how about a profile assumption in a differential equation?



$$\frac{dy}{dt} = -Ky$$
$$\frac{dC}{dt} = -KC$$

- radioactive decay
- chemical reaction species C with abundant other species
- solve with finite difference method



$$\frac{dy}{dt} = -Ky$$

$$\frac{dy}{dt} \sim \frac{\Delta y}{\Delta t} = -Ky$$

$$\frac{y2 - y1}{\Delta t} = -Ky$$

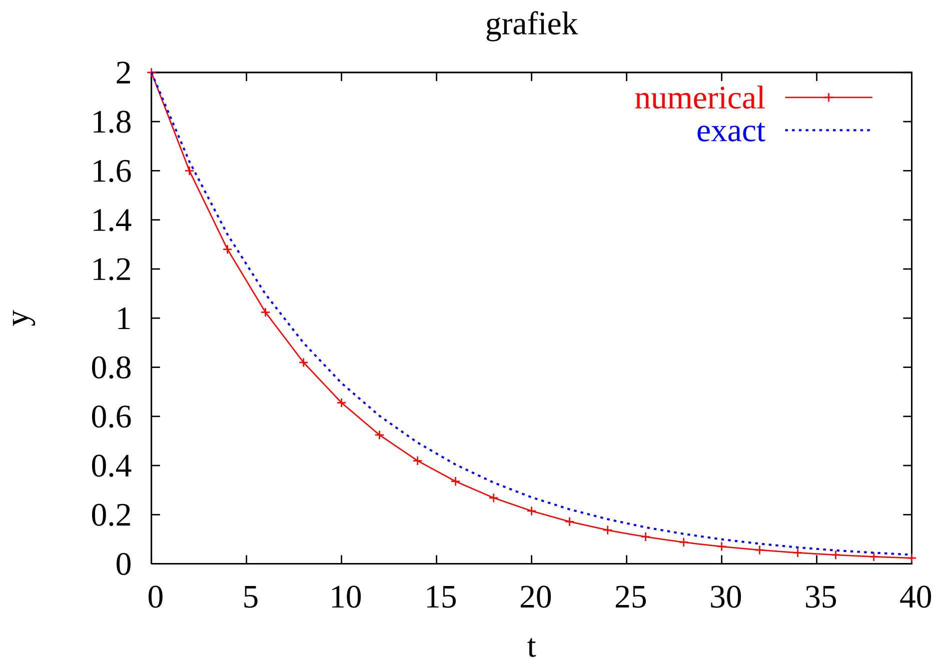
$$\frac{y2 - y1}{\Delta t} = -Ky1$$

$$
\begin{aligned}
\frac{y2 - y1}{\Delta t} &= -Ky1 \\
y2 &= y1 - \Delta t K y1
\end{aligned}
$$

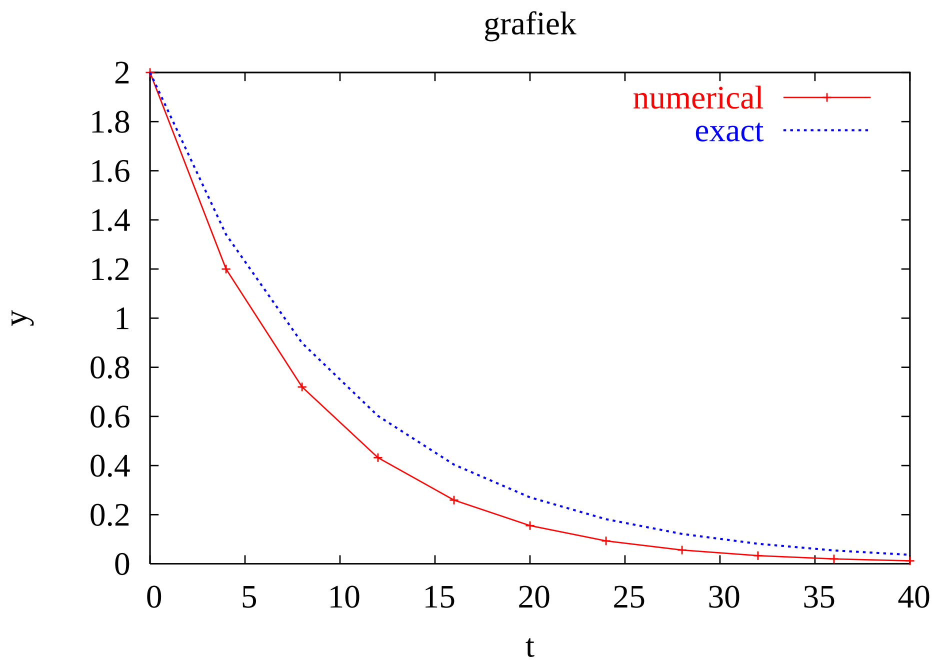
- if you know y1, you get y2
- we have assumed constant derivative, linear profile between points!

Observations:

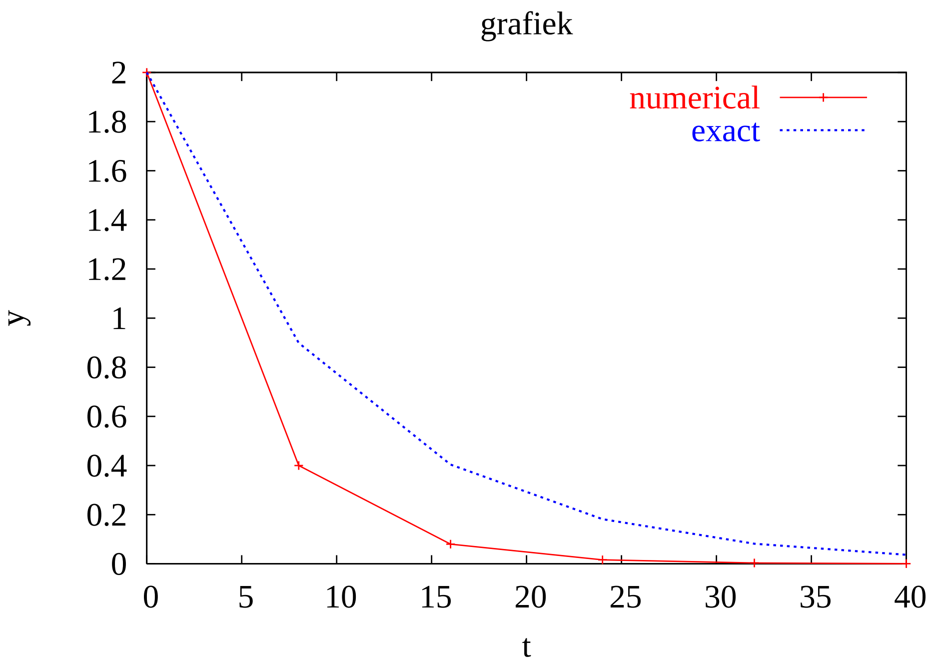
- t1 and t2 closer $\rightarrow$ y1 and y2 closer and approximation derivative more accurate
- and vice versa



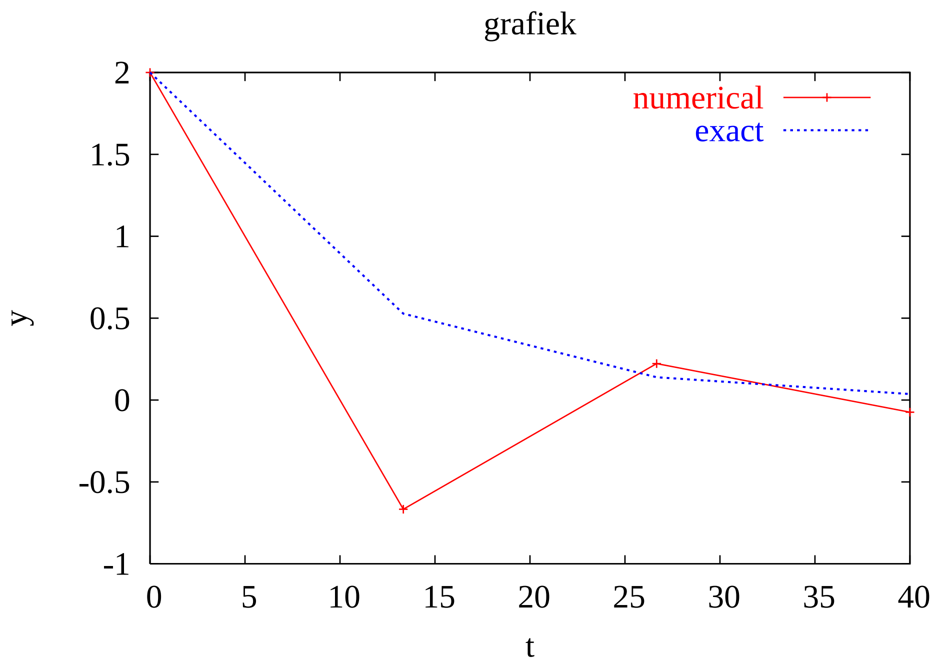
grafiek
numerical
exact
2
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
y
0
5
10
15
20
25
30
35
40
t



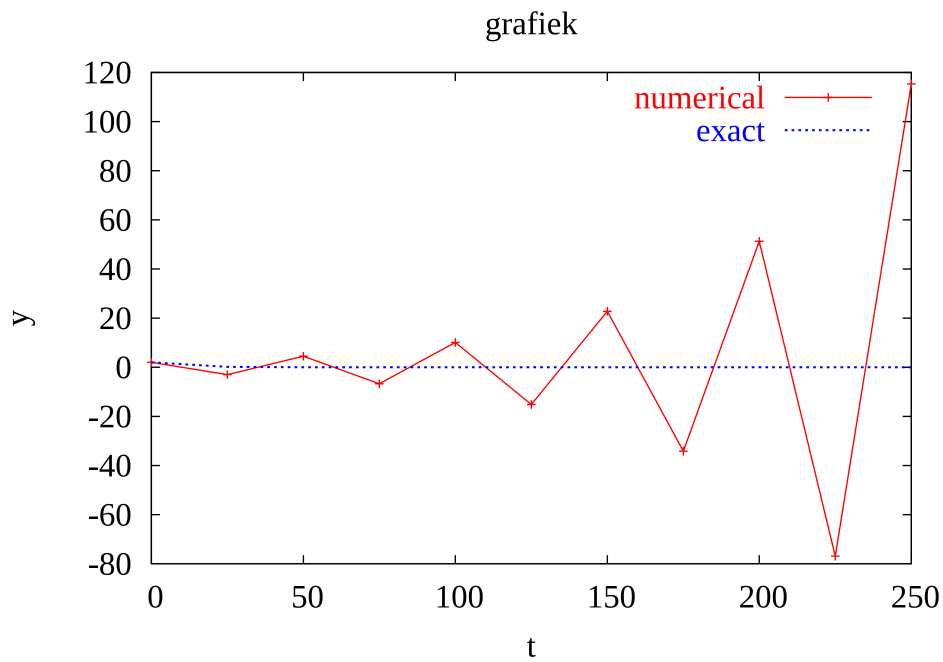
grafiek
numerical
exact
2
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
y
0
5
10
15
20
25
30
35
40
t



grafiek
numerical
exact
y
t
2
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
0 5 10 15 20 25 30 35 40



grafiek
numerical
exact
y
t
2
1.5
1
0.5
0
-0.5
-1
0 5 10 15 20 25 30 35 40



grafiek
numerical
exact
120
100
80
60
40
20
0
-20
-40
-60
-80
0
50
100
150
200
250
y
t

Observations:

- the smaller the timestep the more accurate
- the bigger the timestep the less accurate
- the interpolated exact solution also becomes less accurate
- the calculated solution becomes even less accurate and also becomes non-physical
  - errors accumulate
  - errors can grow: instability

too big timestep, NO programming errors, still:

- non-physical values
- instability

$$y2 = y1 - \Delta t K y1$$
$$y2 = (1 - \Delta t K) y1$$

$$
\begin{aligned}
y2 &= y1 - \Delta t K y1 \\
y2 &= (1 - \Delta t K) y1 \\
y2 &= M y1
\end{aligned}
$$

For the physical solution

- $y2 < y1$

Solution decays.

- $y > 0$

Solution stays positive.

- $y \to 0$

Solution goes to 0 for long times.

- $M < 1$
- $\Delta t < 1/K$

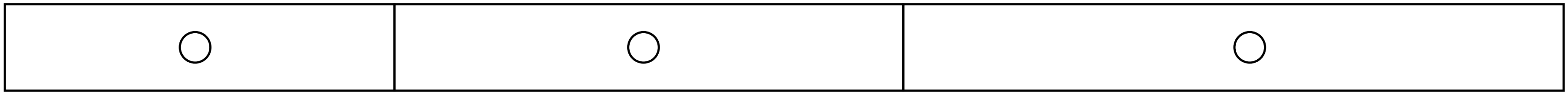
THIS FORMULA WILL BE ENCOUNTERED MANY TIMES

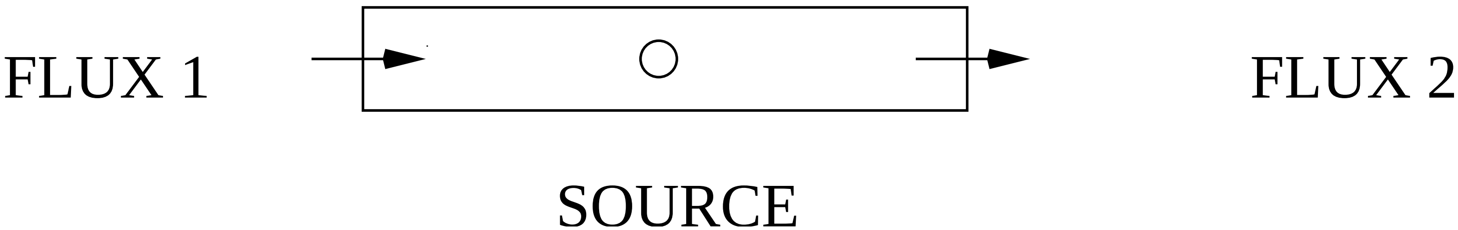
$$\boxed{y2 = My1}$$

$$\boxed{M < 1}$$

- problems in a simple model calculation
- commercial codes can suffer from similar problems
- this can happen in a (much) more complicated situation
- study numerical effects (errors) in simplified situations
- study numerical effects (errors) in building-block flows
- where does the error come from?
  - physical model
  - numerical error
  - programming error

The finite volume method, an integral balance





- volume ΔV
- side surface A
- length Δx
- Fouriers law: net flux $q = -k\frac{\partial T}{\partial x}$

- Midpoint rule: source $= \int s dV = Smp * \Delta V$

Result: $k * \frac{T1-2*T2+T3}{\Delta x^2} = S2$

Working with a package. Flow around buildings.

OUTLINE

- Fluid flow: Navier-Stokes
- Model equations: diffusion, advection, advection-diffusion, wave equation
- finite difference, finite volume (finite element)
- model eqns: Poisson, diffusion, advection, wave
- discretisation, numerical error, stability, explicit, implicit
- matrix equation solvers
- mass conservation (pressure correction)
- Navier-Stokes (incompressible, compressible), heat transfer
- fluent solver structure, boundary conditions, setting up a problem in fluent
- grid generation
- turbulence

- building block flows
  - boundary layer
  - square cylinder
  - round cylinder
  - airfoil

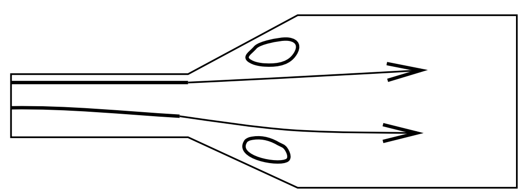
Lecture form

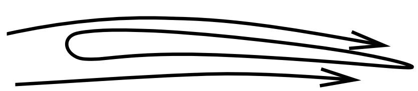
- lectures
- exercises: matlab
- exercises: fluent

LOOK on the www for announcements!
computer exercise PC room Pallas on Thu 21 Feb!

Examination

- assignment: fluid flow calculation with matlab/Fluent
- suggest your own flow
  - (in practice) 2D, axi-symmetric
  - you should have some qualitative and quantitative info
- suggest your own assignment
- assignment: exercise by hand and with matlab (optional)
- discussion of report (2 page, pointwise, plus figures/results)





Examination

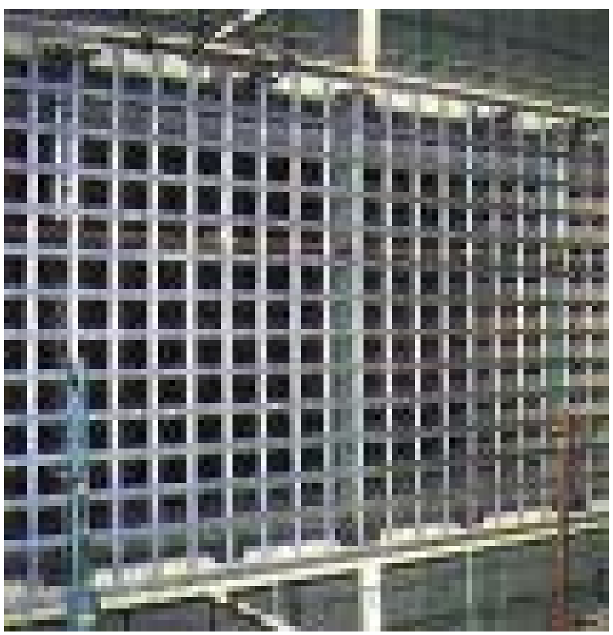
- assignment plus disscussion 60%
- some exercises 40%
  - determine order of method by taylor series
  - write out a discretisation/interpolation (equation, BC)
  - evaluate grids
  - make a stability analysis
  - interpret numerical errors in a simulation
- three credit points
- you want more? Come up with a more realistic (matlab/Fluent) problem (1 point)

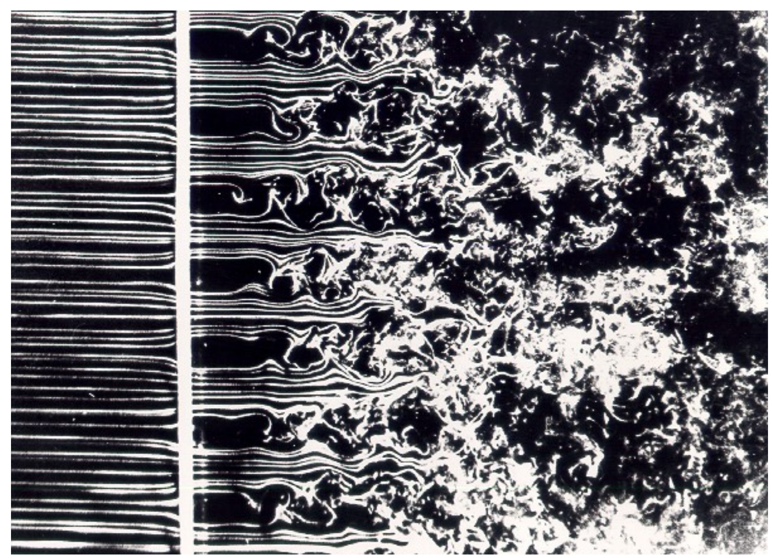
Material

- sheets on www
- background material:
  - Ferziger & Peric, Computational methods for fluid dynamics, Springer
  - J. van Kan, Numerieke wiskunde voor technici, DUP
  - Delft Fluent User Group
    http://www.ahd.tudelft.nl/~mathieu/fluent_group/index.html
  - Pre-requisites:
    - Heat transfer: Winterton, "Heat transfer", OUP (90 pages)
    - Advanced Fluid dynamics
      - Lecture notes FREE notes,http://www.ahd.tudelft.nl, education
      - Batchelor "Fluid Dynamics"
      - Kundu and Cohen, "Fluid Dynamics"
      - Tritton, "Physical fluid dynamics".

limitations of lectures (in principle)

- incompressible (density = constant)
- low Ma compressible (density NOT constant), perhaps some compressible
- Flow, plus possibly
  - physical models for turbulence
  - heat transfer, mass transfer





- 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)