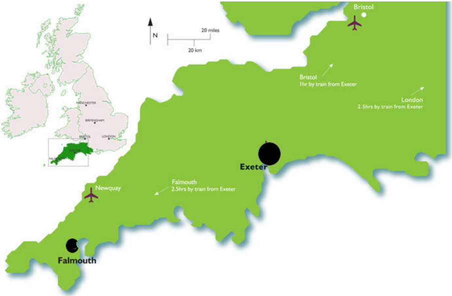
Computational Fluid Dynamics Research at the University of Exeter

Dr Gavin Tabor

CEMPS, University of Exeter

University of Exeter

- Founded 1922 – university status 1955
- "Top 10" non-specialist UK university



UNIVERSITY OF EXETER

CEMPS, CWS

College of Engineering. Maths and Physical Sciences

- One of 6 Colleges in University
- Formed from Schools of Physics, Engineering+Computer Science, CSM (Cornwall)
- Teaching and Research covering all aspects of Mathematical, Physical and Applied Sciences (Climate modelling, Renewable Energy, Materials, Biomechanics, Astrophysics...)

Several autonomous research groupings including the Centre for Water Systems

- Cross-disciplinary group in hydroinformatics and urban water systems
- Expertise includes hydraulics, systems analysis, EC, data mining ...

Research – GRT

CFD research in a number of areas :

- Basics of CFD – turbulence modelling, code development (OpenFOAM)
- Applications of CFD
  - Engineering applications – Urban Water Drainage, renewable energy
  - Biomedical applications – blood flow, air flow, Image Based Meshing

OpenFOAM

OpenFOAM is an Open Source CCM code/code library :

- Written in C++ – Based on FVM on arbitrary unstructured (polyhedral cell) meshes
- Originally developed by Henry Weller and others at IC (1990 – 2000); Nabla Ltd (2000 – 2004) as FOAM (Field Operation And Manipulation)
- Now released (2004 –) under Gnu GPL by OpenCFD Ltd. (http://www.opencfd.co.uk/)
- Fully featured – covers turbulent flow (LES,RANS), multiphase flow (Lagrangian, Eulerian, FSF), combustion, FSI, parallelisation . . .
- Extensive user community – Academic and commercial usage.

Strictly, OpenFOAM is not a CFD code – it is a C++ library of classes for writing CFD codes.

OpenFOAM uses the full range of the C++ language – inheritance, polymorphism, templating, operator overloading etc – where appropriate :

- Class mechanism – define new "types" for CFD
- Interface vs implementation : segregation of effort.
- Operator Overloading – provides standard mathematical syntax
- Inheritance, polymorphism etc – encodes relationships between conceptual entities in code

Effective result is a high level "language" for encoding CFD.

Advantages

Open Source software development encourages code sharing and information exchange. OpenFOAM's structure and use of C++ is ideal for this and provides a common platform for CFD research.

Advantages :

- Complete transparency and code availability (vs. "grey box" approach for commercial codes)
- Information exchange with other practitioners at a fundamental level
- CFD use not rationed by license fees.

Basic code designed to make implementing new models simpler (eg. overloading mathematical operators) and more rigorous (physical dimension checking)

Example : Burgers equation

1d Burgers equation :

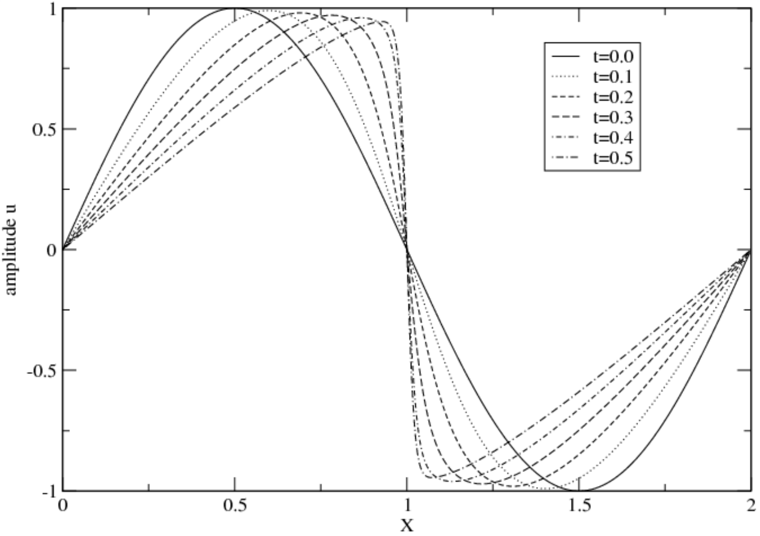
$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} = \nu\frac{\partial^2 u}{\partial x^2}$$

3d version (conservative form) :

$$\frac{\partial \underline{u}}{\partial t} + \nabla.(\frac{1}{2}\underline{u}\ \underline{u}) = \nu\nabla^2\underline{u}$$

```
fvVectorMatrix UEqn
(
    fvm::ddt(U)
  + 0.5*fvm::div(phi, U)
  - fvm::laplacian(nu, U)
);

UEqn.solve();
```



t=0.0
t=0.1
t=0.2
t=0.3
t=0.4
t=0.5
amplitude u
X
1
0.5
0
-0.5
-1
0
0.5
1
1.5
2

Outline of talk

1. Urban Drainage modelling (work within CWS) – Vortex Flow controls, flood runoff
2. Biomedical flow and Image Based Meshing
3. Microscale flow within porous media – packed beds and open-celled foams

Urban Drainage Modelling

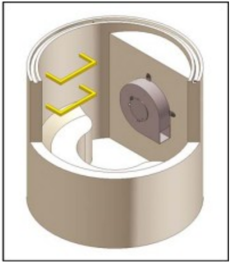
Two projects:

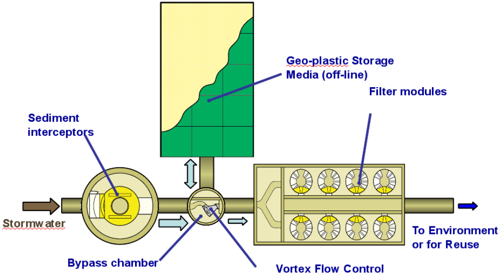
1. Vortex Flow Control
    - Industrial PhD project with UK company – Hydro International

2. Surface water runoff from road surface
    - Part of FRMRC-II project – large multi-centre research project in flood risk modelling

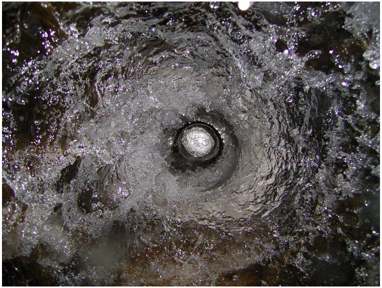
Vortex Flow Controls

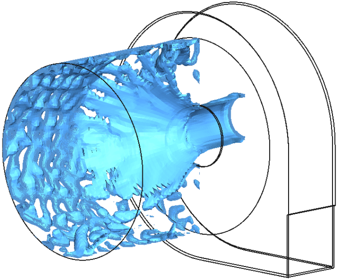
- KTP project with Hydro International
- VFC's used to switch stormwater flow to temporary storage





Vortex behaviour important – complex computational problem – free surface, turbulence modelling.





Examine surcharged and free-surface vortexing behaviour with CFD.

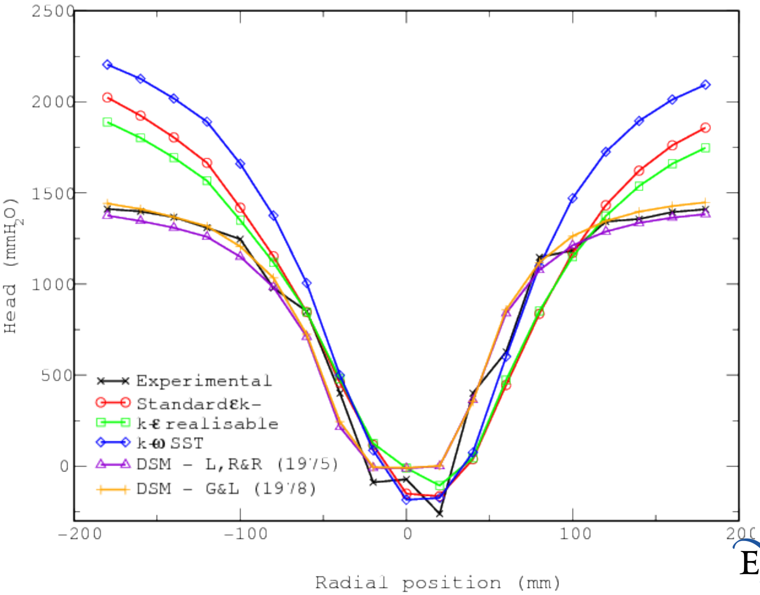
CFD

- Mesh refinement study – 100,000 to 4.5M cells
- Flow rates varied between $4l/s$ – $8l/s$
- Range of turbulence models used; $k - \epsilon$, $k - \omega$, DRSM
- Compared against experimental data (head loss, pressure tappings).
  - DRSM models give best results (turbulence significantly anisotropic)
  - EVM exhibit unphysical inflation of turbulent viscosity
- Transient behaviour important and captured through DRSM
- Both surcharged and free-discharge behaviour can be captured.

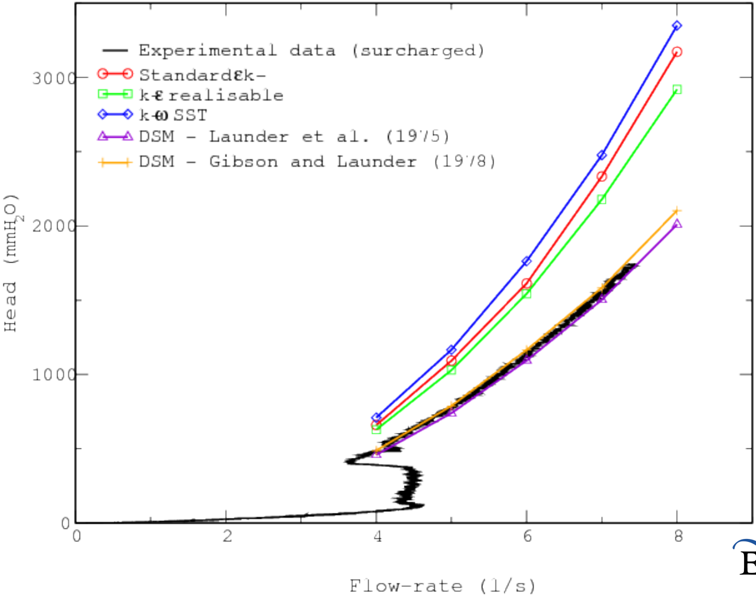
CFD

- Mesh refinement study – 100,000 to 4.5M cells
- Flow rates varied between $4l/s$ – $8l/s$
- Range of turbulence models used; $k-\epsilon$, $k-\omega$, DRSM
- Compared against experimental data (head loss, pressure tappings).
  - DRSM models give best results (turbulence significantly anisotropic)
  - EVM exhibit unphysical inflation of turbulent viscosity
- Transient behaviour important and captured through DRSM
- Both surcharged and free-discharge behaviour can be captured.

Pressure comparisons



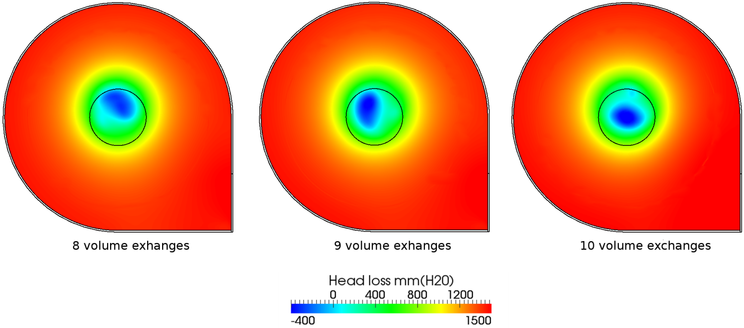
Head loss



UNIVERSITY OF EXETER

Pressure distribution

Flow instability present (vortex precession)



Summary

- Mesh refinement study – 100,000 to 4.5M cells
- Flow rates varied between $4l/s$ – $8l/s$
- Range of turbulence models used; $k - \epsilon$, $k - \omega$, DRSM
- Compared against experimental data (head loss, pressure tappings)
    - DRSM models give best results (turbulence significantly anisotropic)
    - EVM exhibit unphysical inflation of turbulent viscosity
- Transient behaviour important and captured through DRSM
- Both surcharged and free-discharge behaviour can be captured.

UNIVERSITY OF EXETER

Surface water runoff

Simulation of surface water runoff (FRMRC-II project) from roadway to gully pot.

- Critical point of coupling between 2d (surface flow) and 1d (pipe network) models
- Experimental data (Sheffield) to validate CFD models
- Vary all aspects of geometry + flow conditions
- Intention to develop empirical models for energy loss

CFD modelling

Problem must deal with both shallow depth effects and deep flow. Three approaches investigated :

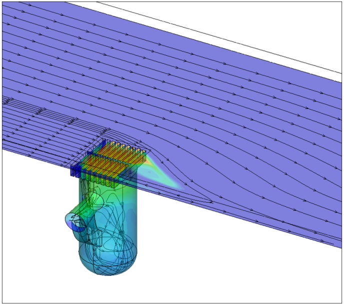
- Volume of Fluid method – indicator function taking values $[0, 1]$
- Investigating G-equation

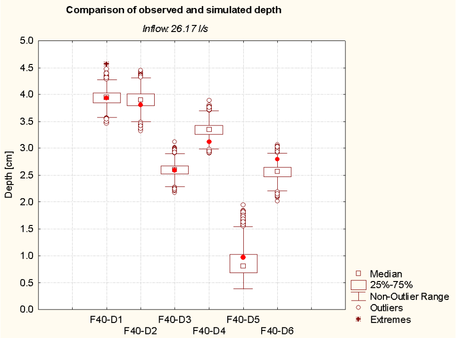
Problem; mesh resolution for thin surface flow makes mesh requirements prohibitive. Therefore:

- Finite Area approach (surface flow) coupled to VOF (gully pot).

Numerical details

- Solver: Implicit pressure based
- Multiphase model: Implicit VOF, level set
- Pressure velocity coupling: PISO
- Pressure discretization: PRESTO
- Turbulence closure: $k - \omega$ model
- Momentum discretization: 1st & 2nd order upwind
- Volume fraction discretization: 1st & 2nd order upwind




Comparison of observed and simulated depth
Inflow: 26.17 l/s
Depth [cm]
5.0
4.5
4.0
3.5
3.0
2.5
2.0
1.5
1.0
0.5
0.0
F40-D1
F40-D2
F40-D3
F40-D4
F40-D5
F40-D6
Median
25%-75%
Non-Outlier Range
Outliers
Extremes

Biofluid Mechanics

CCM – CFD, FE stress analysis – useful in biomedical field

- generate data that would be difficult to gather otherwise
- perform "experiments" that would be difficult (impossible, unethical) otherwise

Eg;

- CFD – blood flow in heart, arteries; air flow in lungs, trachea
- CSA – stress in bones, implants, impact analysis, soft tissue

Main problem – definition of geometry. Biomedical geometries

- complex over range of scales – possibly fractal
- patient-specific (individual differences possibly important)
- difficult to characterise mathematically
- ... and unknown – no blueprints available.

We need some way of determining the geometry. Various medical imaging techniques exist – MRI, CT etc.

Then need techniques to convert from image to mesh.

Image Based Meshing

IBM – automated methodology for generating geometry (+ mesh) from medical scans.

Many groups utilise 'CAD'-type approaches :

- create bounding geometry from stack of images
- then use existing automatic meshing techniques

e.g.

H. M. Ladak, J. S. Milner, and D. A. Steinman. ‘Rapid three-dimensional segmentation of the carotid bifurcation from serial mr images.’ *J. Biomech. Eng.*, 122(1):96 – 99, February 2000.

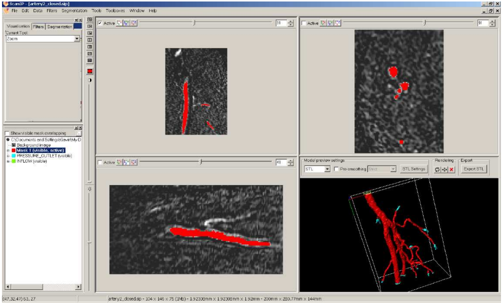
L. Antiga, B. Ene-Iordache, L. Caverni, G. P. Cornalba, and A. Remuzzi. ‘Geometric reconstruction for computational mesh generation of arterial bifurcations from ct angiography.’ *Computerized Medical Imaging and Graphics*, 26:227 – 235, 2002.

ScanIP/ScanFE

More direct approach to combine geometry detection and mesh creation – Voxel based methods.

ScanIP – Segments different volumes of interest from 3D data

- semi-automated and manual techniques
- includes various filters and thresholding tools



ScanFE

Base Cartesian mesh generated by marching cubes algorithm

- tetrahedronalised at boundaries based on cutting planes
- some adjustment to smooth boundary
- produces mixed tet/hex mesh
- adaptive meshing to reduce mesh density by agglomeration

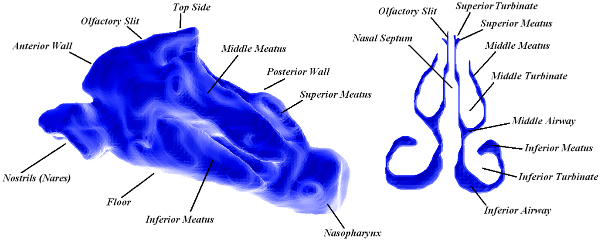
L. Zunarelli and P. G. Young. 'Analytical and numerical modelling of head injury mechanisms'. *Simulation and Modelling Techniques Applied to Medicine*, Institute of Physics, November 2-3, London, 1999.

Originally developed for FE stress analysis in bones – however FV CFD and FE techniques very similar (dual meshes).

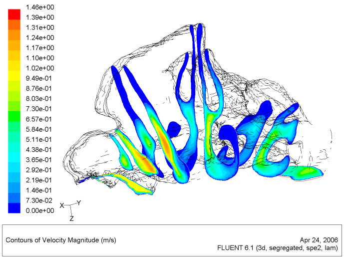
Develop FV output as cells and faces (Fluent mesh format) rather than points and edges (FEA).

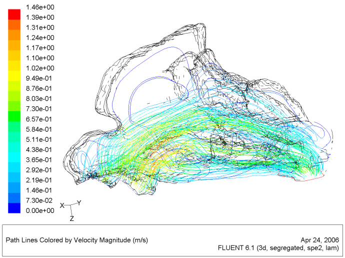
Example – nasal air flow

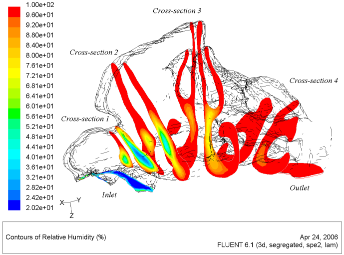
Nasal geometry highly complex – nose acts to filter, humidify and warm air entering body.

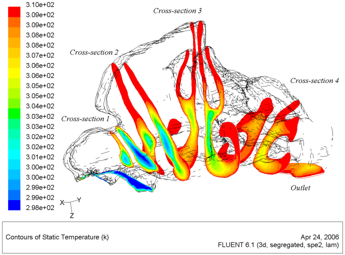


Scan airways using MRI; segment using ScanIP; mesh using ScanFE. Calculations performed for constant inhalation (steady state) and single intake (transient). Compute air flow, temperature, relative humidity.

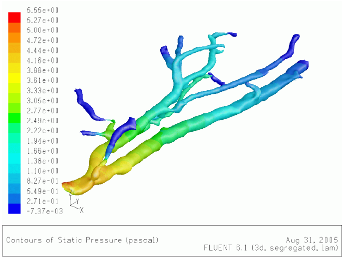

1.46e+00
1.39e+00
1.31e+00
1.24e+00
1.17e+00
1.10e+00
1.02e+00
9.49e-01
8.76e-01
8.03e-01
7.30e-01
6.57e-01
5.84e-01
5.11e-01
4.38e-01
3.65e-01
2.92e-01
2.19e-01
1.46e-01
7.30e-02
0.00e+00
X Y Z
Contours of Velocity Magnitude (m/s)
Apr 24, 2006
FLUENT 6.1 (3d, segregated, spe2, lam)

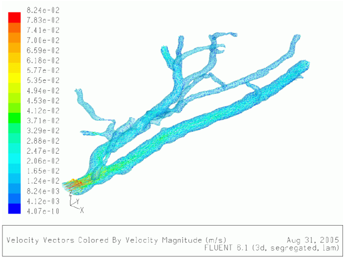

1.46e+00
1.39e+00
1.31e+00
1.24e+00
1.17e+00
1.10e+00
1.02e+00
9.49e-01
8.76e-01
8.03e-01
7.30e-01
6.57e-01
5.84e-01
5.11e-01
4.38e-01
3.65e-01
2.92e-01
2.19e-01
1.46e-01
7.30e-02
0.00e+00
X Y Z
Path Lines Colored by Velocity Magnitude (m/s)
Apr 24, 2006
FLUENT 6.1 (3d, segregated, spe2, lam)

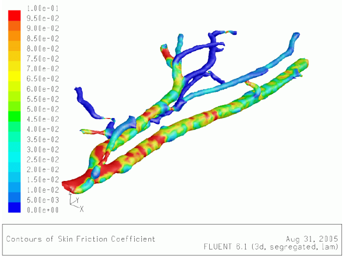

1.00e+02
9.60e+01
9.20e+01
8.80e+01
8.40e+01
8.00e+01
7.61e+01
7.21e+01
6.81e+01
6.41e+01
6.01e+01
5.61e+01
5.21e+01
4.81e+01
4.41e+01
4.01e+01
3.61e+01
3.21e+01
2.82e+01
2.42e+01
2.02e+01
Cross-section 3
Cross-section 2
Cross-section 4
Cross-section 1
Inlet
Outlet
X Y Z
Contours of Relative Humidity (%)
Apr 24, 2006
FLUENT 6.1 (3d, segregated, spe2, lam)

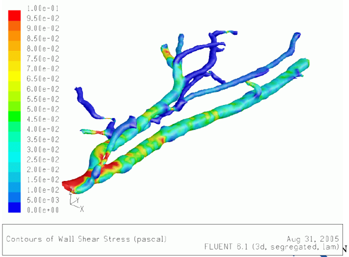

3.10e+02
3.09e+02
3.09e+02
3.08e+02
3.08e+02
3.07e+02
3.06e+02
3.06e+02
3.05e+02
3.05e+02
3.04e+02
3.03e+02
3.03e+02
3.02e+02
3.02e+02
3.01e+02
3.00e+02
3.00e+02
2.99e+02
2.99e+02
2.98e+02
Cross-section 3
Cross-section 2
Cross-section 4
Cross-section 1
Outlet
X Y Z
Contours of Static Temperature (k)
Apr 24, 2006
FLUENT 6.1 (3d, segregated, spe2, lam)

Blood flow – femoral artery









Packed Beds and Columns

Packed Bed – structure comprising a number of macroscopic particles packed into a particular volume

- Creates a network of space through which fluid can move
- Provides large surface area of contact between solid and fluid
- Particles can be random or uniform size/shape; structured/unstructured packing
- Valuable for heat exchange, combustion, catalysis, filtration etc.

Packed Column – packed bed enclosed within walls

Macroscopic flow properties determined by microstructural flow and geometry

UNIVERSITY OF EXETER

Empirical correlations

Most important: relation between flow speed and pressure drop – easilly measured experimentally.

Ergun equation (1950's) :

$$\frac{\Delta P}{L} = \frac{150\mu\left(1-\varepsilon\right)^2 U_S}{\varepsilon^3 d_P^2} + \frac{1.75\left(1-\varepsilon\right)\rho U_S^2}{\varepsilon^3 d_P}$$

Evaluation of coefficients and corrections for non-spherical particles determined experimentally. Eg. Eisfeld and Schnitzlein (2001) – compilation of 2300 experimental data points from 27 datasets.

Ergun equation limited to high aspect ratios : $(N = D/d_P) > 50$

Theoretical/computational analysis

Theoretical analysis – confined to simplified models eg. "bundle of straws" model, simplistic unit cells.

Computational analysis :

- Use CFD to probe microstructural flow
- Often restricted to simplified unit cells
- Lattice Boltzmann approach often used – increased speed, "voxelised" (non-body-fitted) geometry.

Challenges

Aim of project : to apply, validate and use FV CFD methods to compute flows through realistic bed geometries.

Challenges :

- **Geometry** Bed constructed and scanned (IBM) or synthesised using digital packing software (MacroPac)
- **Meshing** Resulting domains meshed using IBM or separate automatic mesh generation (Gambit)
- **Validation** Experimental measurements performed using equivalent or identical beds – eg using ALM to synthesise actual bed

Digital packing

Digital packing algorithms for packing objects (spheres) – eg. MacroPac – uses Monte Carlo methods to pack (includes simulated shaking).

Resulting output – list of sphere centres; imported into AutoCAD and used to reconstruct ACIS format for meshing.

Meshing strategies :

Gambit Import ACIS; use TGrid mesh generator and Tet/Hybrid options. TGrid uses wrapping technique to create triangular surface mesh; propagated into volume using Delauney triangulation.

IBM Create STL file for export to ScanFE via ScanCAD – alternative meshing strategy

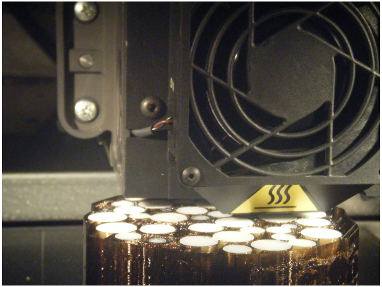
UNIVERSITY OF EXETER

Experimentation

Parallel line of experiment – measure superficial velocity, pressure drop in air for equivalent beds.

Three approaches for making the bed :

- Use actual bed (when available)
- Manufacture equivalent bed eg. from marbles.
- Manufacture actual bed using ALM



Summary of cases investigated :

Model	Bed	Geometry	Meshing	Experiment
A	Spheres, diameter 14 mm	MacroPac	Gambit	Equivalent bed constructed of marbles
B	Cylinders, diameter 22 mm, length 30 mm	Physical packed bed	MRI Scan of bed + ScanFE	Physical packed bed
C	Spheres, diameter 16 mm	Physical packed bed	MRI scan of bed + ScanFE	Physical packed bed
D	Spheres, diameter 15 mm and 10 mm	MacroPac	ScanFE	Bed synthesised via RP

CFD

Computations carried out using Fluent

Unclear whether flow is turbulent or laminar;

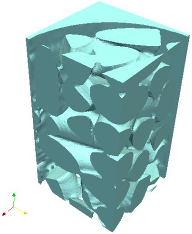
- Flow speeds cover range of $\mathcal{R}e$ – laminar to turbulent
- Mesh resolution fine – beyond Kolmogorov scale
- Distinction between laminar/transient and turbulent/steady state?

Calculations performed laminar, turbulent with $k - \omega$ and $k - \epsilon$ models; steady state (SIMPLE) and transient (PISO). Little difference between results.

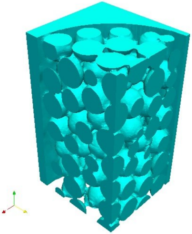
Geometry construction

Model B.

Model B.



Model C.

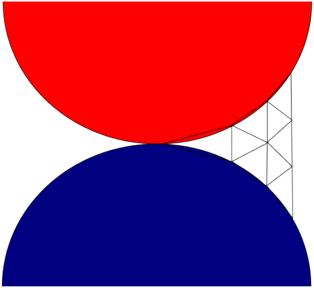


Mesh Construction

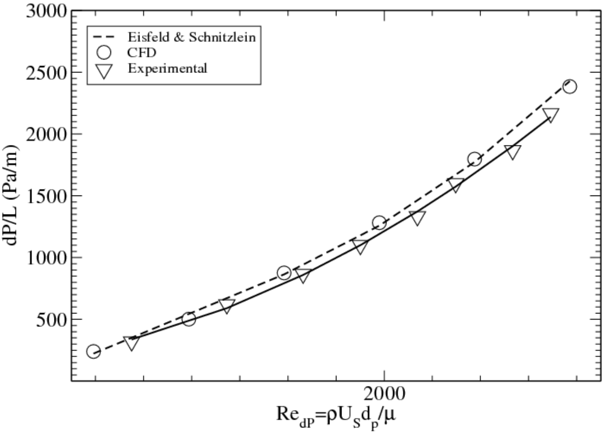
CAD-based meshing very challenging :

- Spheres meet at contact points – badly distorted cells
- Only small number of distorted cells; but dominate solution

IBM-based meshing broadens contact points – eliminates need for distorted cells. Flow in this region insignificant; no impact on solution accuracy.

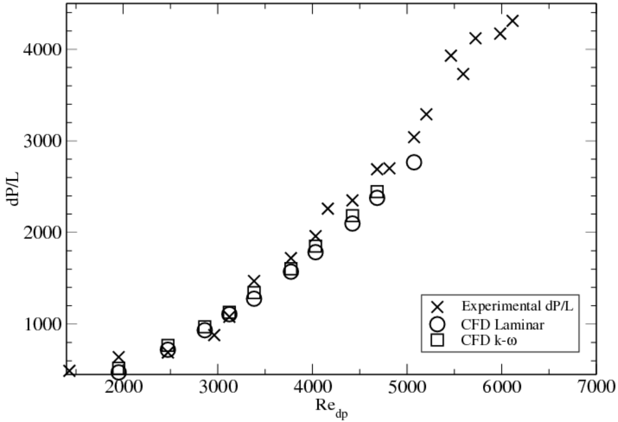


Results – Model A

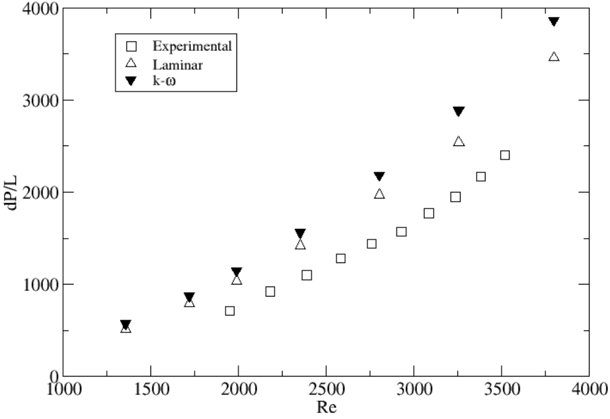


UNIVERSITY OF EXETER

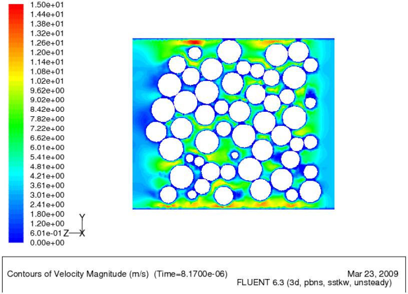
Results – Model B

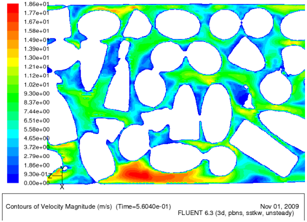


Results – Model C



Velocities


1.50e+01
1.44e+01
1.38e+01
1.32e+01
1.26e+01
1.20e+01
1.14e+01
1.08e+01
1.02e+01
9.62e+00
9.02e+00
8.42e+00
7.82e+00
7.22e+00
6.62e+00
6.01e+00
5.41e+00
4.81e+00
4.21e+00
3.61e+00
3.01e+00
2.41e+00
1.80e+00
1.20e+00
6.01e-01
0.00e+00
Y
Z
X
Contours of Velocity Magnitude (m/s) (Time=8.1700e-06)
Mar 23, 2009
FLUENT 6.3 (3d, pbns, sstkw, unsteady)


1.86e+01
1.77e+01
1.67e+01
1.58e+01
1.49e+01
1.39e+01
1.30e+01
1.21e+01
1.12e+01
1.02e+01
9.30e+00
8.37e+00
7.44e+00
6.51e+00
5.58e+00
4.65e+00
3.72e+00
2.79e+00
1.86e+00
9.30e-01
0.00e+00
Contours of Velocity Magnitude (m/s) (Time=5.6040e-01)
Nov 01, 2009
FLUENT 6.3 (3d, pbns, sstkw, unsteady)

Acknowledgements

Urban Drainage Dan Jarman, Dr Mike Faram (Hydro International), Prof David Butler, Istvan Galambos, Prof Slobodan Djordjevic

Biofluid Mechanics Simpleware Ltd, Prof Philippe Young, Prof X.Y.Xu.

Packed Beds Matt Baker

General Prof Hrvoje Jasak

Papers

"Computational fluid dynamics as a tool for urban drainage system analysis: A review of applications and best practice", D.S.Jarman, M.G.Faram, D.Butler, G.Tabor, V.R.Stovin, D.Burt, E.Throp. *11th International Conference on Urban Drainage, Edinburgh, Scotland* (2008)

"Comparison of LES of steady transitional flow in an idealised stenosed axisymmetric artery model with a RANS transitional model", F.P.P.Tan, N.B.Wood, G.Tabor, X.Y.Xu. To appear in :*J.Biomech.Engng* **133** (2011)

"Computational analysis of transitional air flow through packed columns of spheres using the finite volume technique", M.J.Baker, G. Tabor.*Computers and Chemical Engineering* **34** pp.878 – 885 (2010)

"OpenFOAM: An Exeter perspective", G.Tabor. *V.European Conference on Computational Fluid Dynamics* ECCOMAS CFD 2010, Lisbon,Portugal (2010)

UNIVERSITY OF EXETER