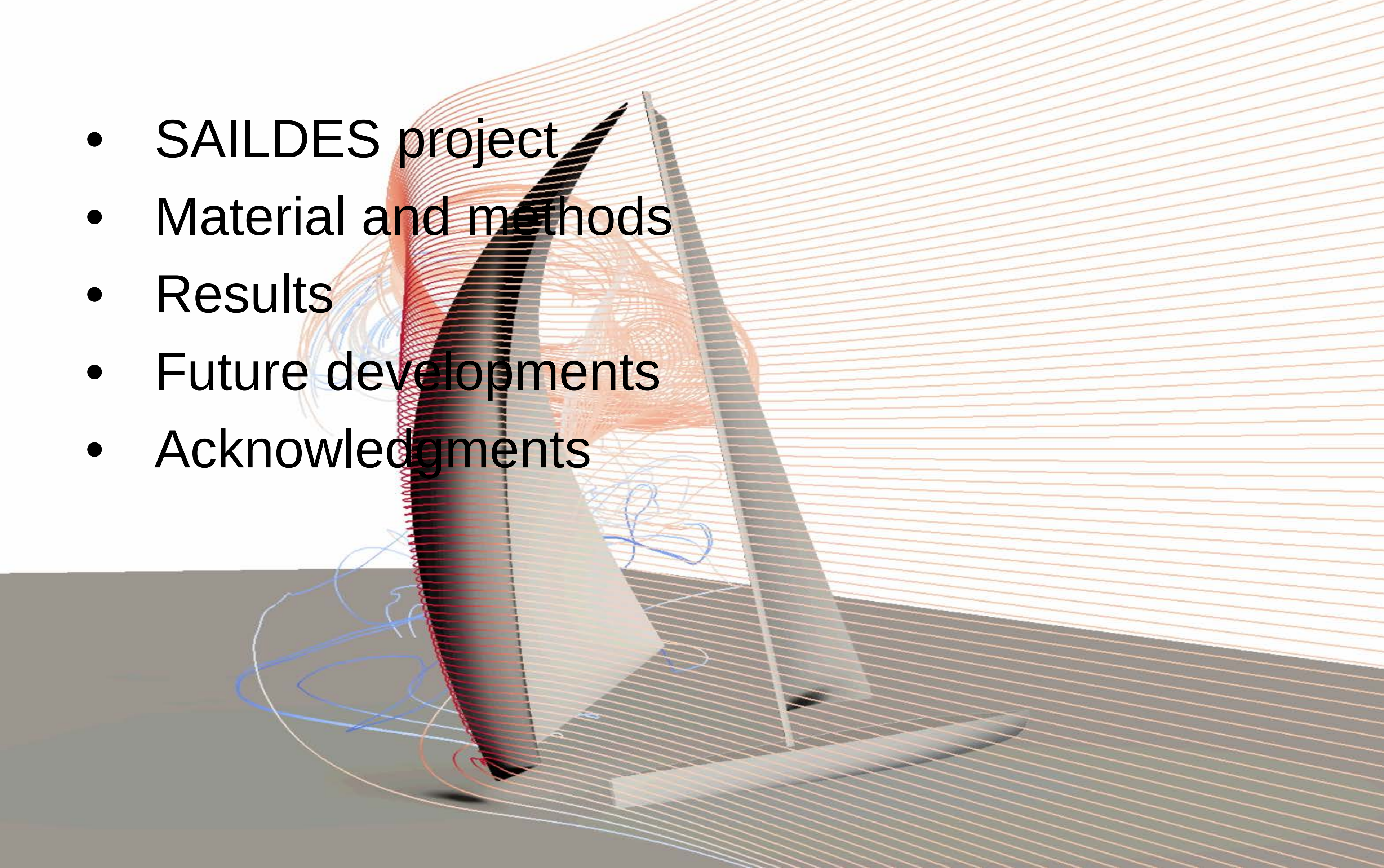
Delayed Detached Eddy Simulation of Yacht Sails with Experimental Validation

Raffaele Ponzini,
CINECA – SCAI Dept.
Italy

PRACE Autumn School 2013 - Industry Oriented HPC Simulations, September 21-27, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, Slovenia

Outline of the presentation



- SAILDES project
- Material and methods
- Results
- Future developments
- Acknowledgments

SAILDES project

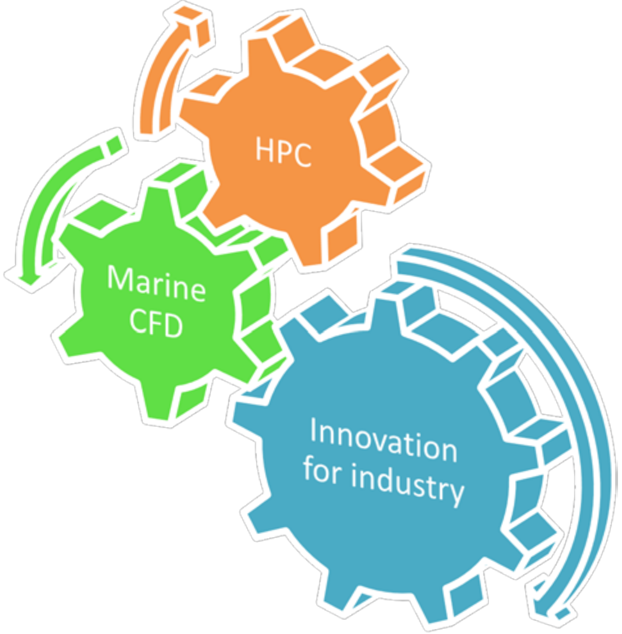
Collaboration between CINECA and The University of Newcastle upon Tyne (UK) through its School of Marine Science and Technology (Prof. I.M. Viola), investigating the use of scientific engineering computations in the marine field.

Student Intership involved actively:

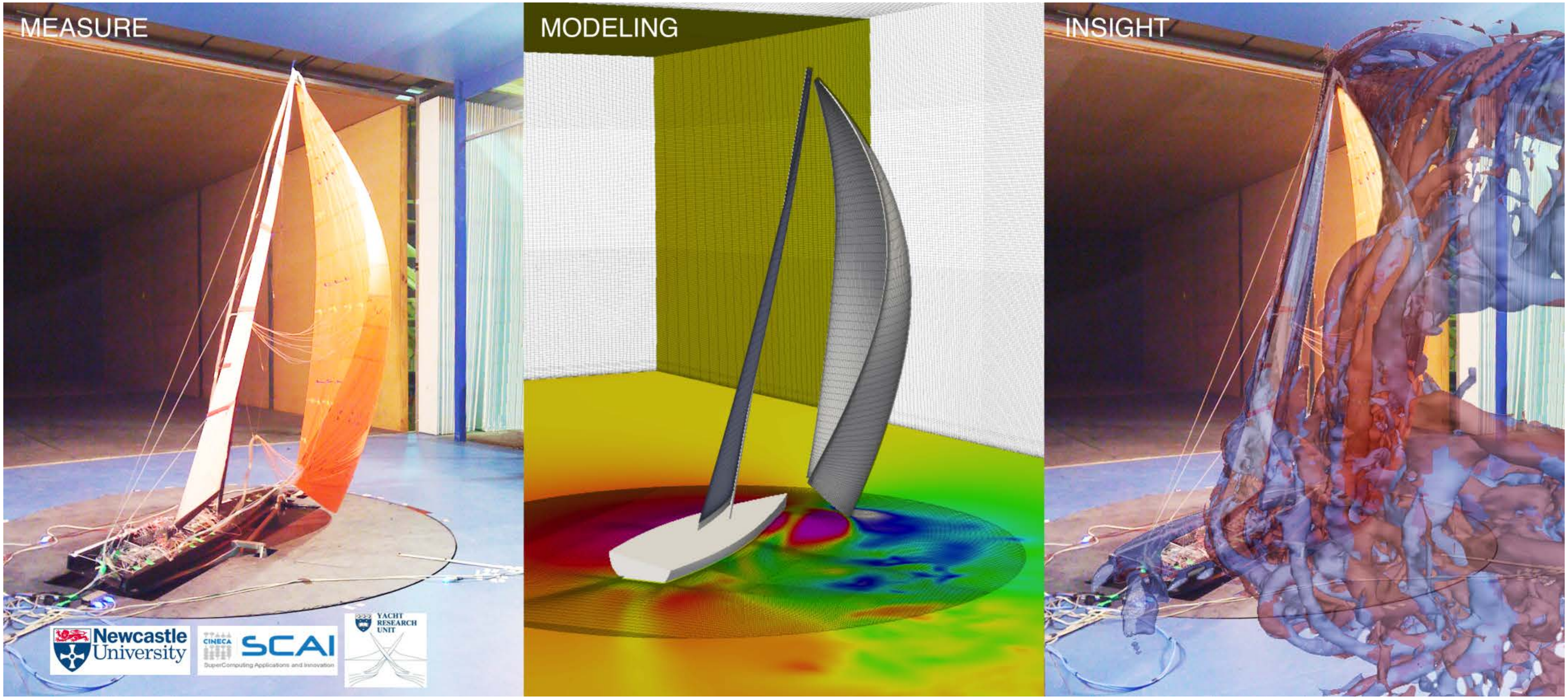
S. Bartesaghi (PhD. Politecnico di Milano, Italy);

T. Van Renterghem (Arts et Métiers ParisTech, France)

First outcome of the SAILDES project in collaboration with Ansys Italy:
Delayed Detached Eddy Simulation of Yacht Sails with Experimental Validation



SAILDES project



Experimental setup

- Sails of a candidate America’s Cup (AC) class, AC90, were designed and tested in the wind tunnel of the Yacht Research Unit, University of Auckland.
- This experimental model was designed, build and instrumented with a specific focus on providing a benchmark for numerical methods and thus the experimental setup was simplified as explained in a previous work from Viola and Flay [1] and in Viola *et al.* (2011) (see [2]).

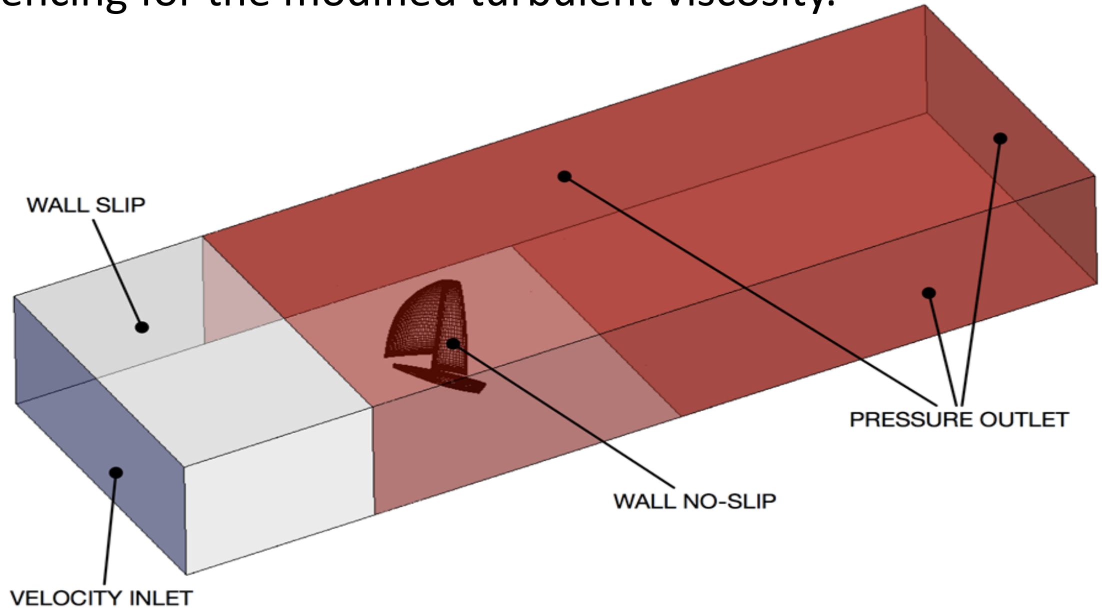
---

[1] Viola I.M. and Flay R.G.J., Pressure Distribution on Modern Asymmetric Spinnakers, International Journal of Small Craft Technology, Trans. RINA, vol. 152, part B1, pp. 41-50, 2010.

[2] Viola I.M., Pilate J., Flay R.G.J., Upwind Sail Aerodynamics: a Pressure Distribution Database for the Validation of Numerical Codes, International Journal of Small Craft Technology, Trans. RINA, vol. 153, part B1, pp. 47-58, 2011.

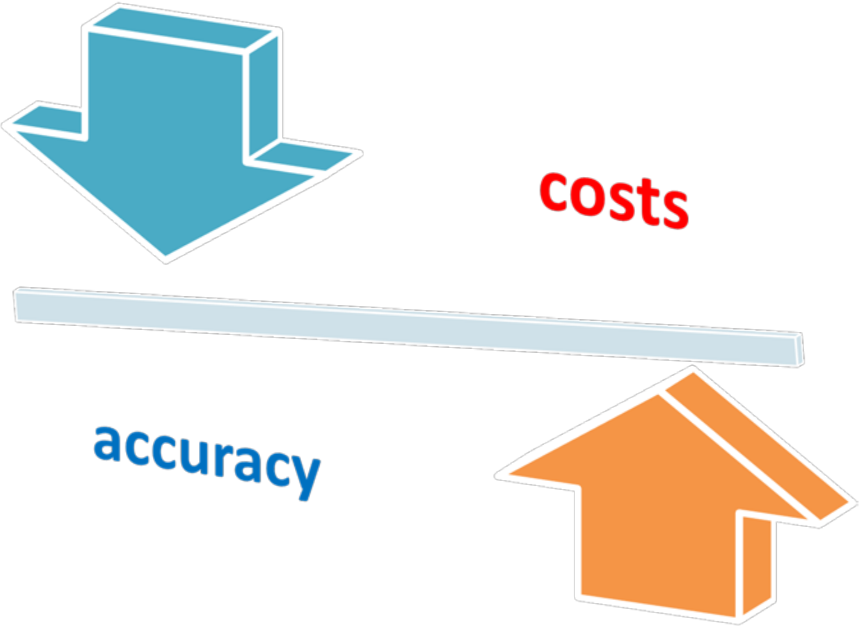
Numerical models

- Incompressible Navier-Stokes equations for Newtonian fluids.
- Delayed Detached Eddy Simulation model implemented in Ansys Fluent, version 13.1.
- Turbulence model: Spalart–Allmaras
- Standard Wall-function
- 3D double precision unsteady pressure based solver
- SIMPLEC pressure-velocity coupling scheme was used with a skewness correction equal to zero.
- The discretization scheme adopted was second order for the pressure, central differencing for the momentum and central differencing for the modified turbulent viscosity.
- Intensity 2% uniform at the inlet
- Flat velocity profile: 3.5 m/s
- Reynolds-spi: 4 x 10^5



DDES model

- Development of hybrid models that attempt to combine the best aspects of RANS and LES methodologies in a hybrid technique: the detached-eddy simulation (DES) approach by Spalart et al (1997)
- This model attempts to treat near-wall regions in a RANS-like manner, and treat the rest of the flow in an LES-like manner.



Numerical setup

- Different mesh sizes - from 4 to 32 million cells – were built and the scalability on up to 256 computational cores was tested.
- About twenty loops (5.2 seconds/loop) of the whole domain using time steps from $\mathbf{5 \cdot 10^{-4}}$ to $\mathbf{2 \cdot 10^{-3}}$ seconds were performed in order to obtain statistically reliable data and to perform a comparison with the experimental results of pressures over the sails surfaces.

Mesh Size	**Wind-Tunnel Loops**	**Time-step**	**RANS**
4 mln	**38**	$\mathbf{2 \cdot 10^{-3}}$	**y**
32 mln	**10**	$\mathbf{5 \cdot 10^{-4}}$	**y**
256 mln	**--**	**--**	**n**

HPC environment

HP x86_64 cluster:

- Using up to 256 cores - X5660 (exa-core)
- Equipped with 2GB of RAM per core
- Infiniband QDR

+

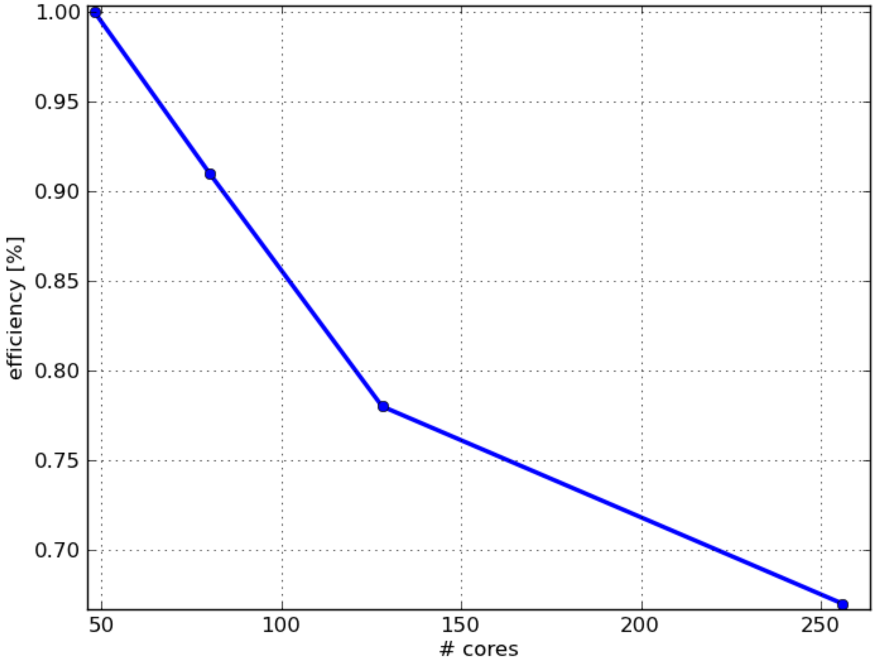
Remote Visualization node:

DL980 (8cpu Intel E5420 - 512GB RAM - with NICE DCV technology (image compression)

Scalability results

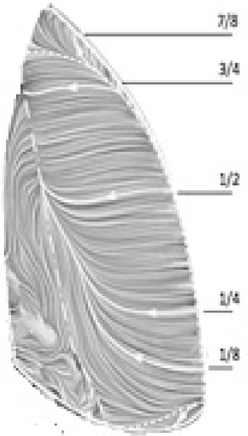
- Speed-up on HPC clusters: efficiency of about 70% on up to 256 cores

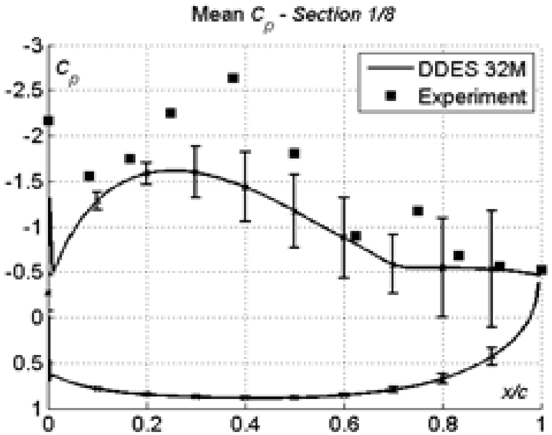
- Feasibility in industry:
  - RANS: ½ day / loop
  - DDES: 2 days / loop

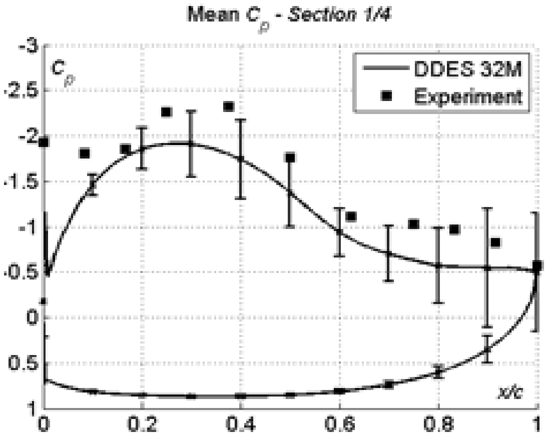


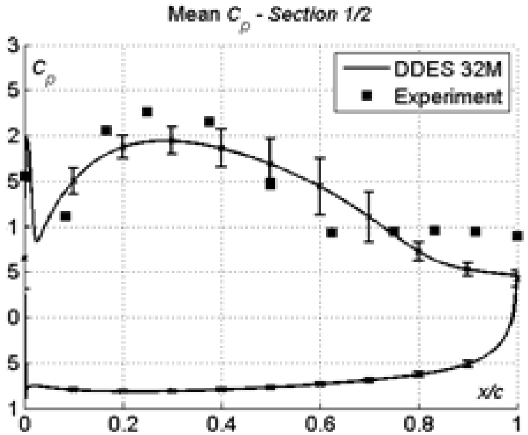
Cp comparisons with uncertainty evaluation

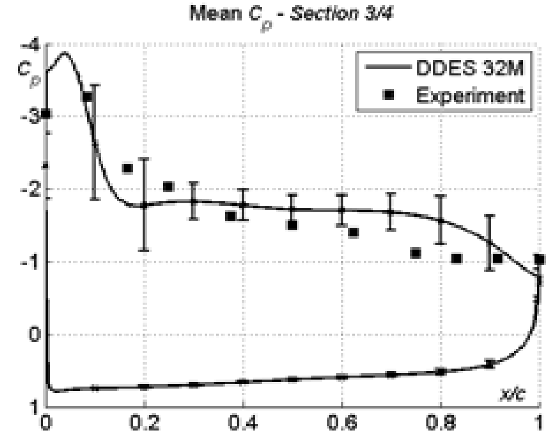
Higher uncertainty located were complex phenomenon occurs (windage estimation due to tip vortex identification)

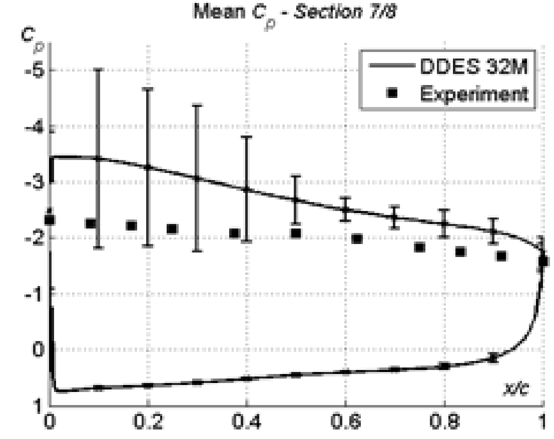






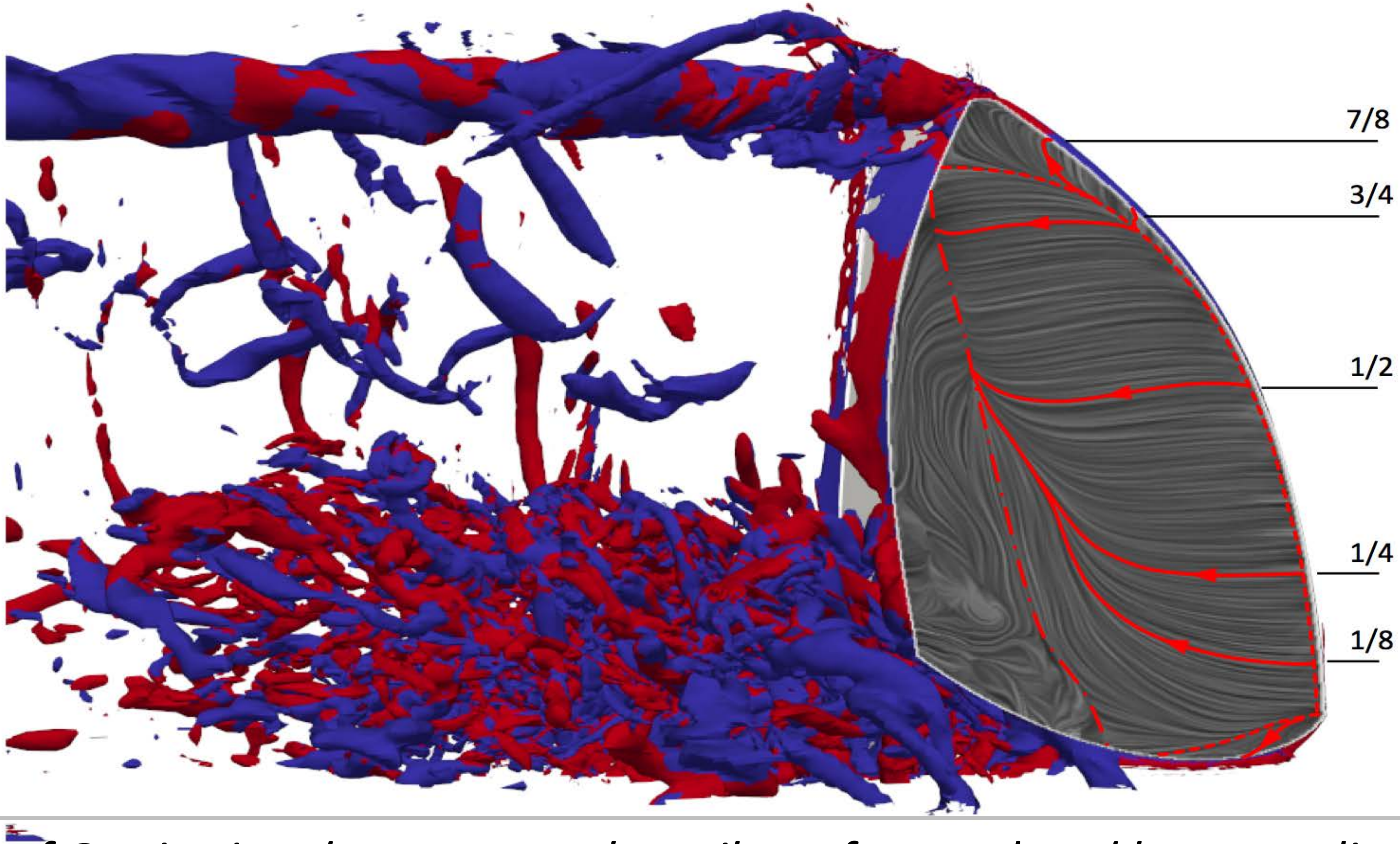




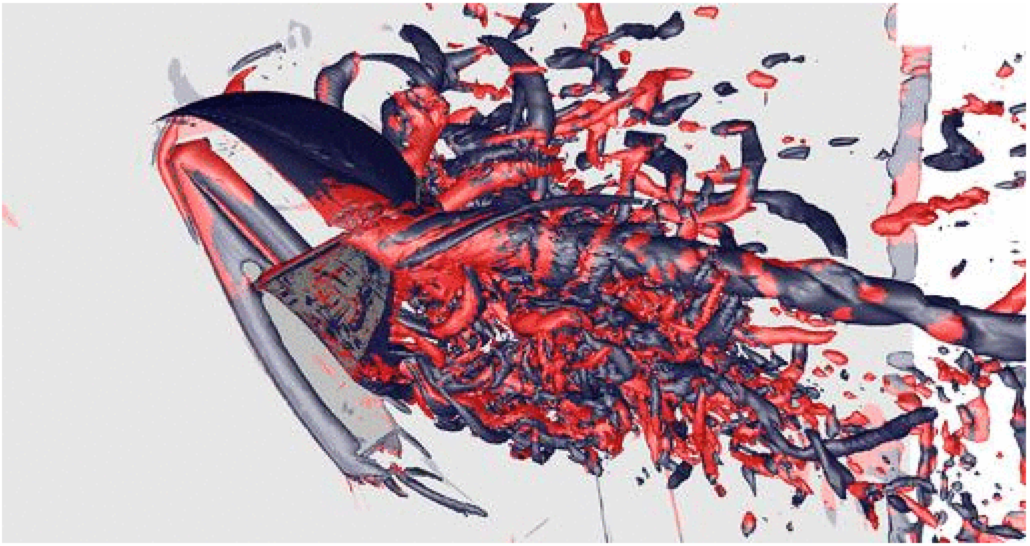


Experimental and numerical Cp on selected chords along the spinnaker surface (1/8h,1/4h, 1/2h,3/4h,7/8h).

Q criterion comparisons: visualization



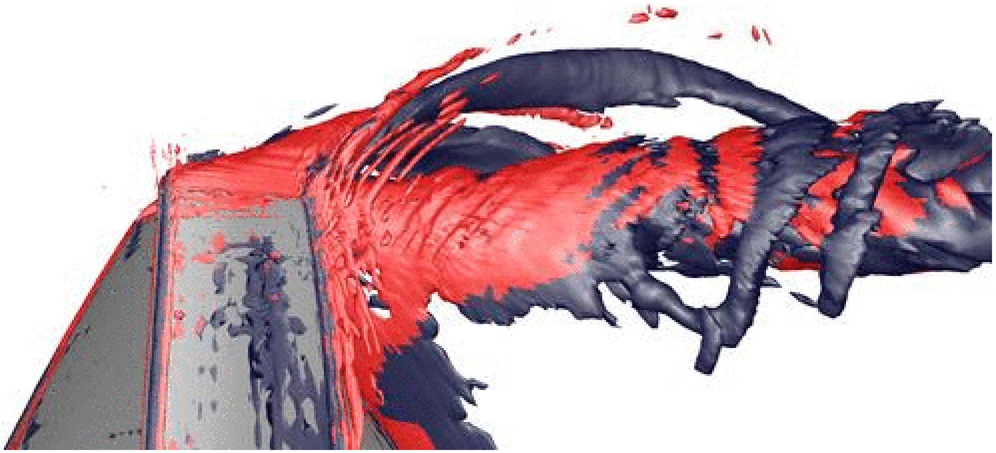
Iso-surfaces of Q-criterion downstream the sails surfaces colored by normalized helicity. Surface shear lines, separation and reattachment lines are also showed on the leeward side of the spinnaker.



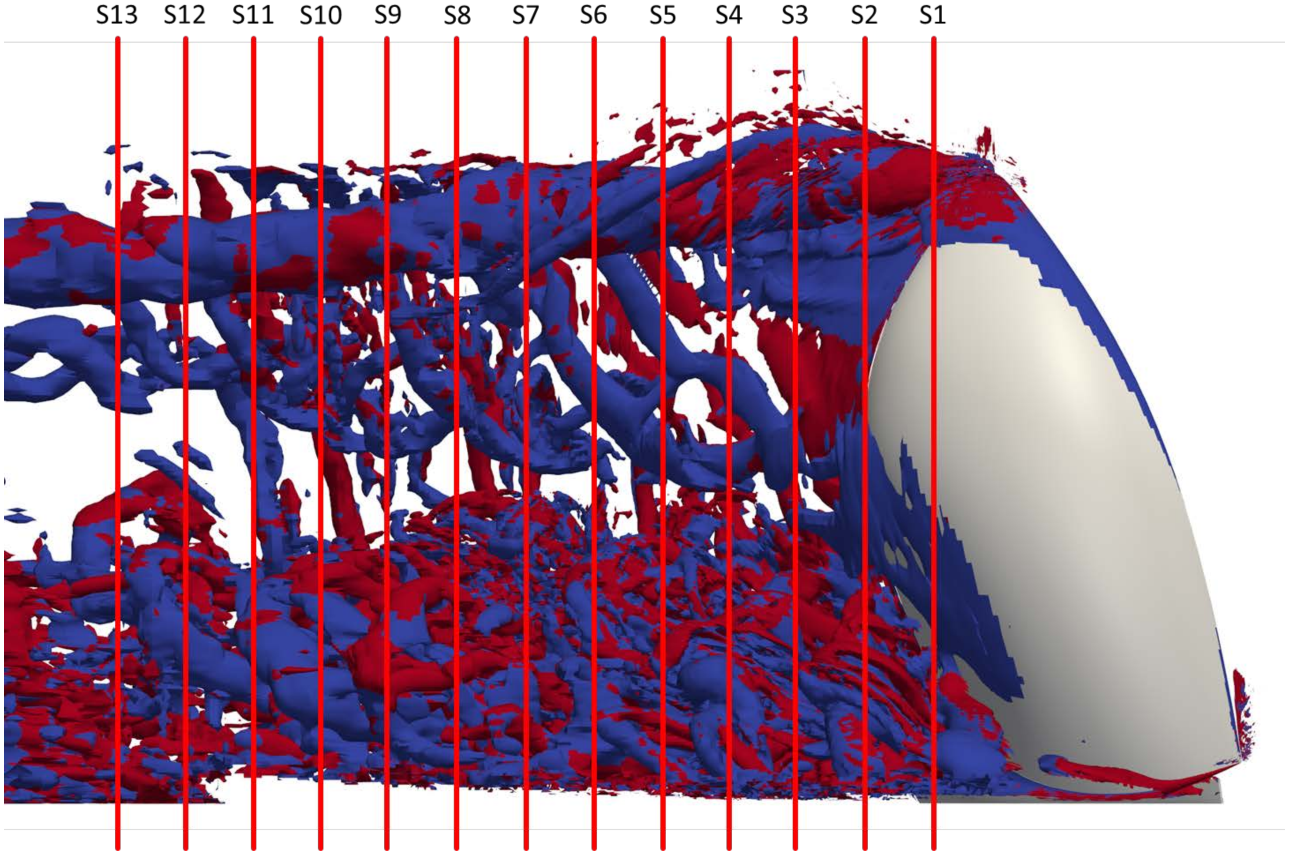
Top view

Side view

Top spinnaker vortex details

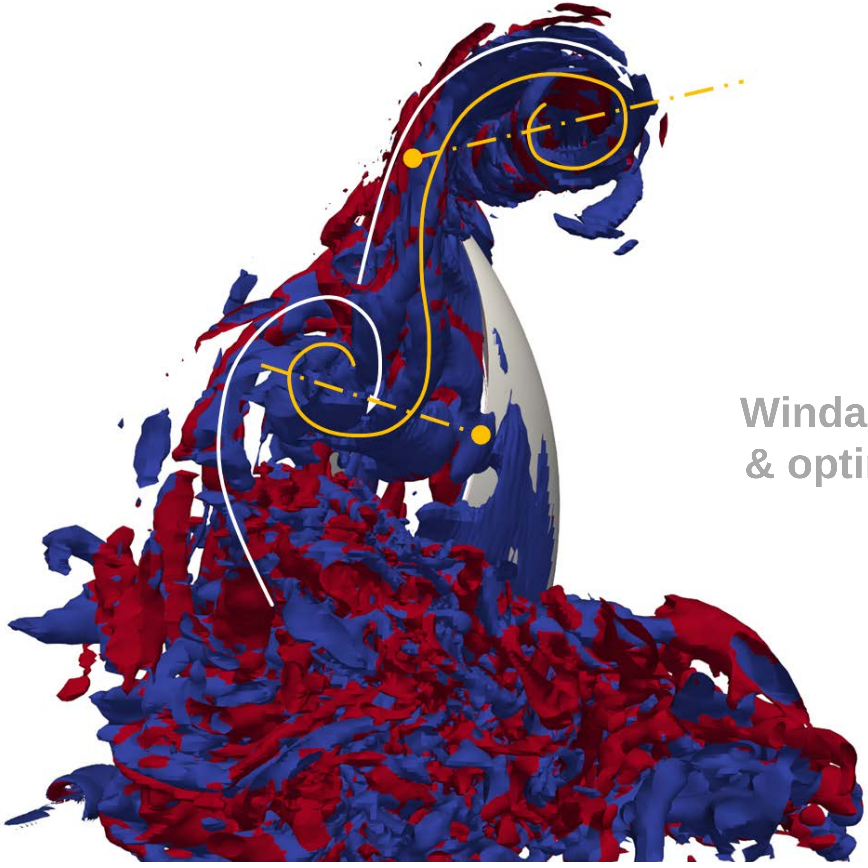


Q criterion comparisons: visualization

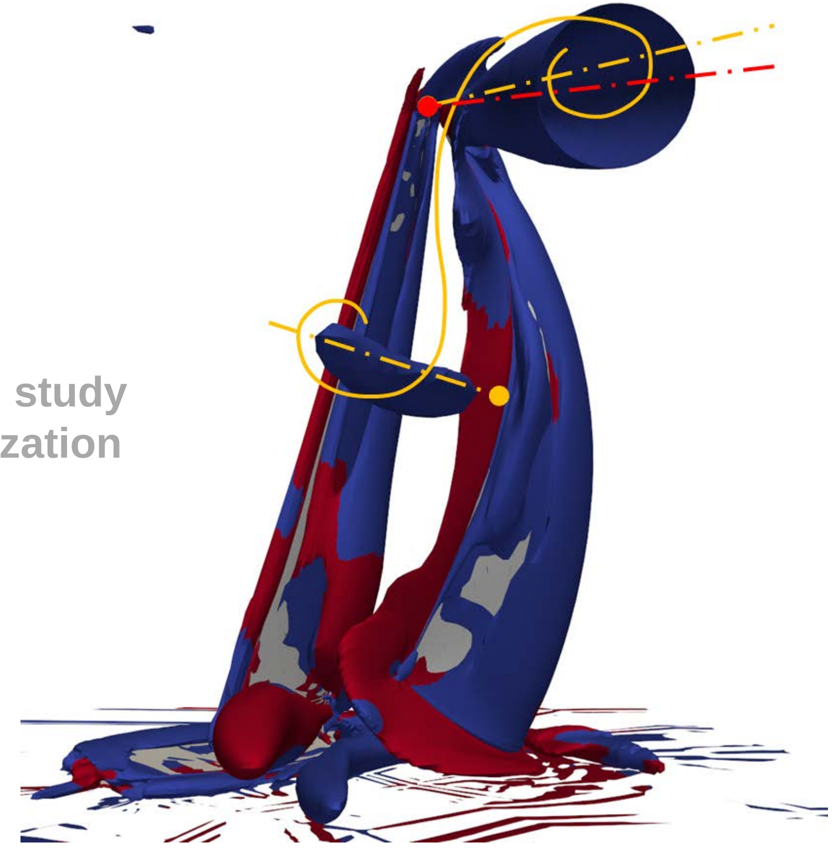


DDES vs RANS

32M_001s_30s – isoQ100_Helicity – S9



4M_RANS – isoQ100_Helicity – S9



Windage study
& optimization

Implications

- The study of turbulent structures around complex three dimensional geometries and curved surfaces under high lift conditions and their interaction with local flow fields where laminar to turbulent transition, separation and reattachment play a major role, is very challenging both numerically and experimentally.
- A synergy between experiments and numerical simulation can be the way to overcome these challenges:
  - experimental setups can be designed tailored to validate numerical models
  - advances on numerical hybrid modelling methods and high performance computing resources can lead to an affordable cost-effective strategy able to shrink cost and time consumption

Future developments

- Technology dependent: x86_64 next generation improvements
- Application dependent: Enlarging the mesh size and enrich the turbulent scale computed vs modeled

Mesh Size	Model Setup	Scalability	Wind-Tunnel Loops
4 mln	X	X	X
32 mln	X	X	X
256 mln	X	X	X

Acknowledgments

Yachts and Super-Yachts research unit, the Newcastle University (UK)

Yacht Research Unit, the University of Auckland (NZ)

Ansys Italy (IT)

Thank you for your attention

Questions?

PRACE

PARTNERSHIP FOR
ADVANCED COMPUTING IN EUROPE

Introduction on CFD in hemodynamics

Raffaele Ponzini, PhD
CINECA – SCAI Dept.
Segrate, Italy

PRACE Autumn School 2013 - Industry Oriented HPC Simulations, September 21-27, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, Slovenia

University *of Ljubljana*
Faculty *of Mechanical Engineering*

CAPACITIES

e-infrastructure

Contents

- CFD introduction
- Blood fluid dynamics:
  theory, equations and examples
- Implementation in Ansys Fluent: CFD model setup
  1. Part A: introduction
  2. Part B: the Ansys Fluent menu
  3. Part C: user defined function Implementation

CFD introduction

- What is CFD
- How is implemented in hemodynamics
- Why is useful in hemodynamics
- How should I use it

Computational fluid dynamics: CFD

- Fluid dynamics: physics of fluids
- Computational: numeric involved in solving the equation describing the motion of the fluid

There is a strong interplay between **math concepts**, **physics knowledge** and **technological tools and environments** used to implement a CFD model

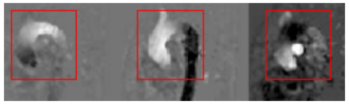
- No general rules for specific model setup
- Need of a-priori knowledge on the fluid behavior
- If possible experimental (or theoretical) data to validate CFD results

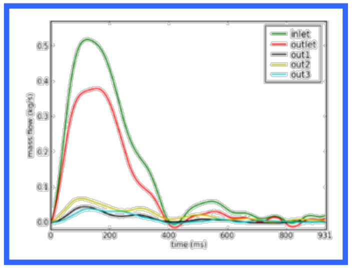
Computer aided engineering workflow

PRE-PROCESSING **COMPUTATION** **POST PROCESSING**

COMPUTATIONAL

PHYSICS





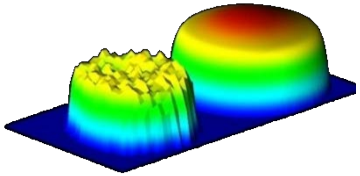
MODEL

SOLVING

HPC

ENVIRONMENT

VISUALIZATION



RESULTS

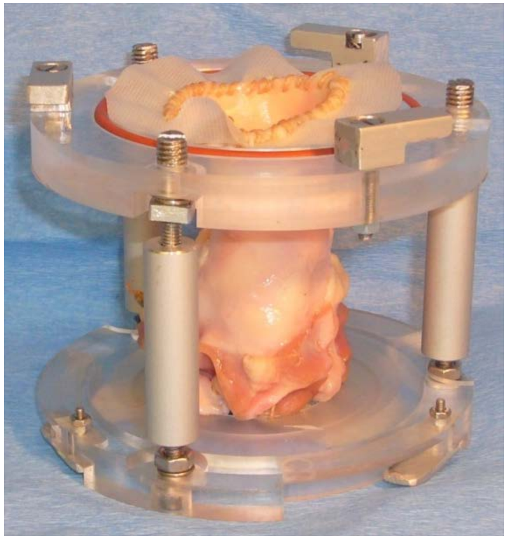
From physics to model by means of measures

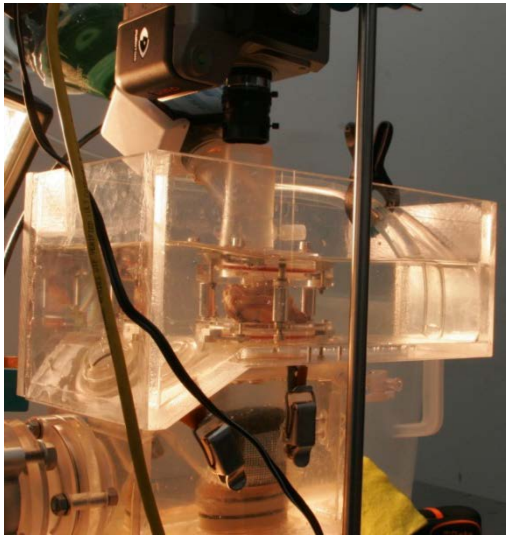
- ☐ In vitro
- ☐ Animal models
- ☐ In vivo

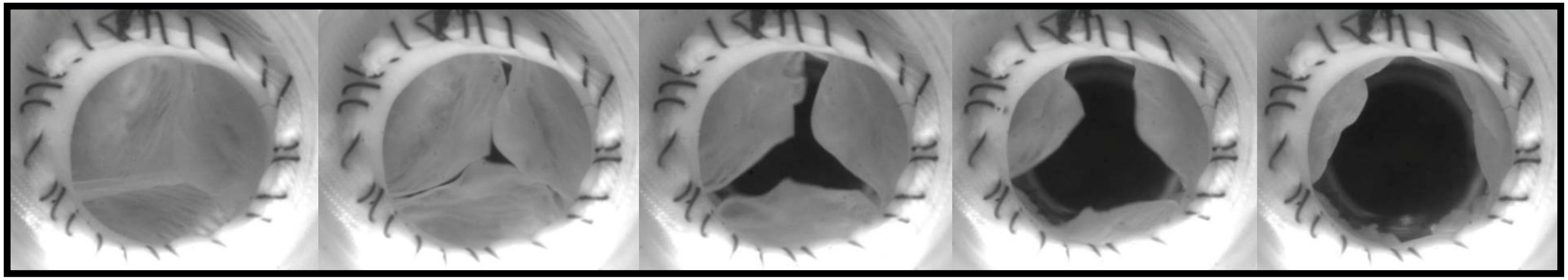
Measures

Measures are necessary to built reliable CAE models

In vitro models



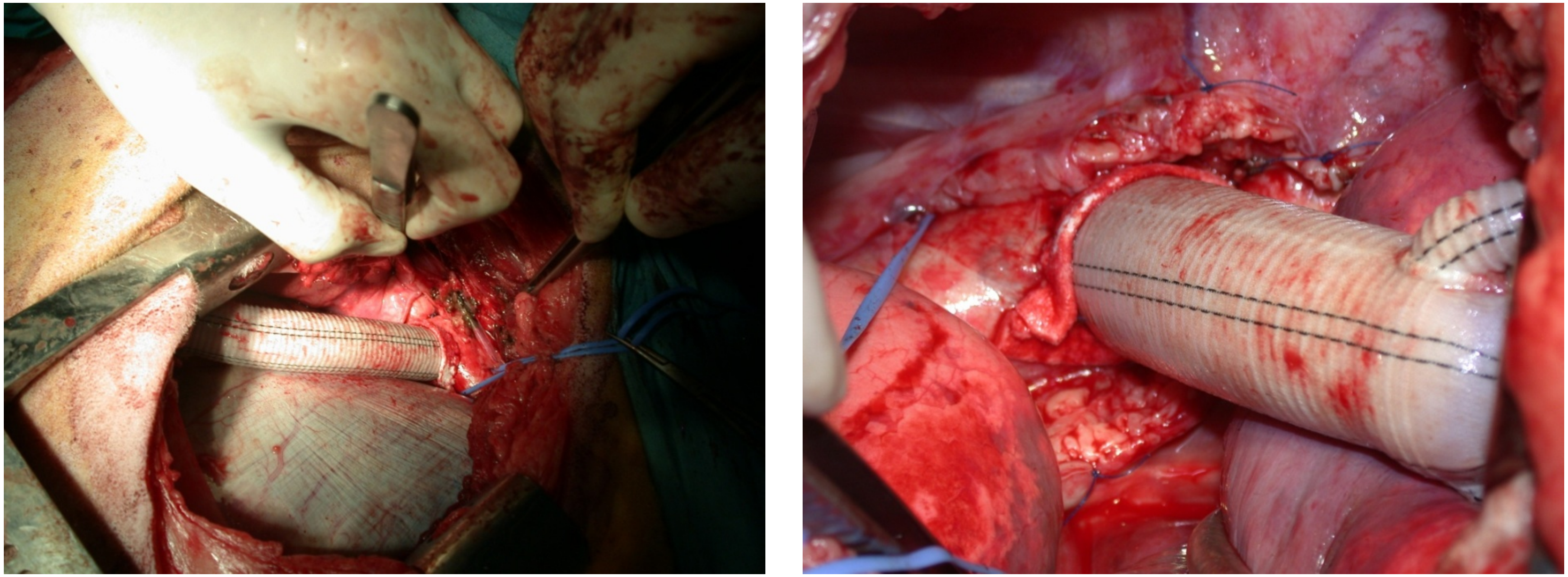




0 ms **19.26 ms** **24.88 ms** **28.09 ms** **33.71 ms**

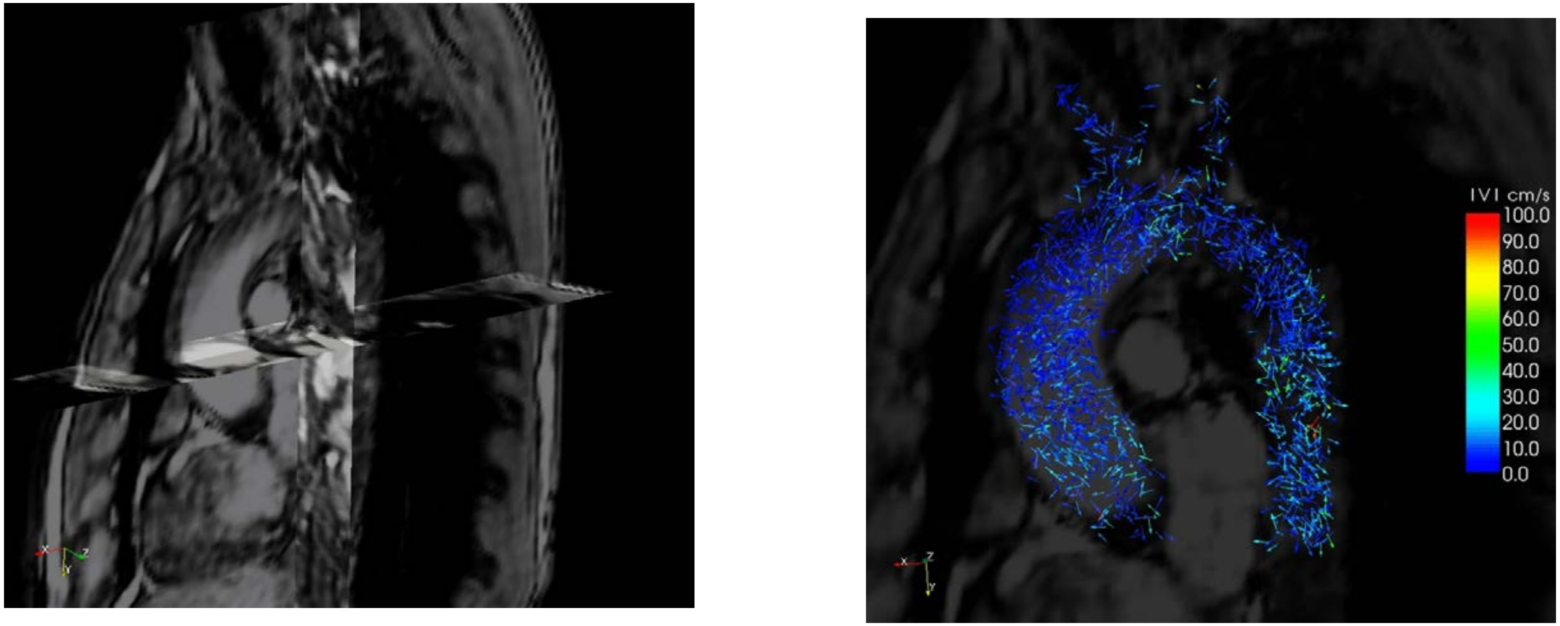
Bicuspid porcine valve setup mock
Performed by Riccardo Vismara (Politecnico di Milano)
at the ForcardioLab
http://www.forcardiolab.it/

Animal models



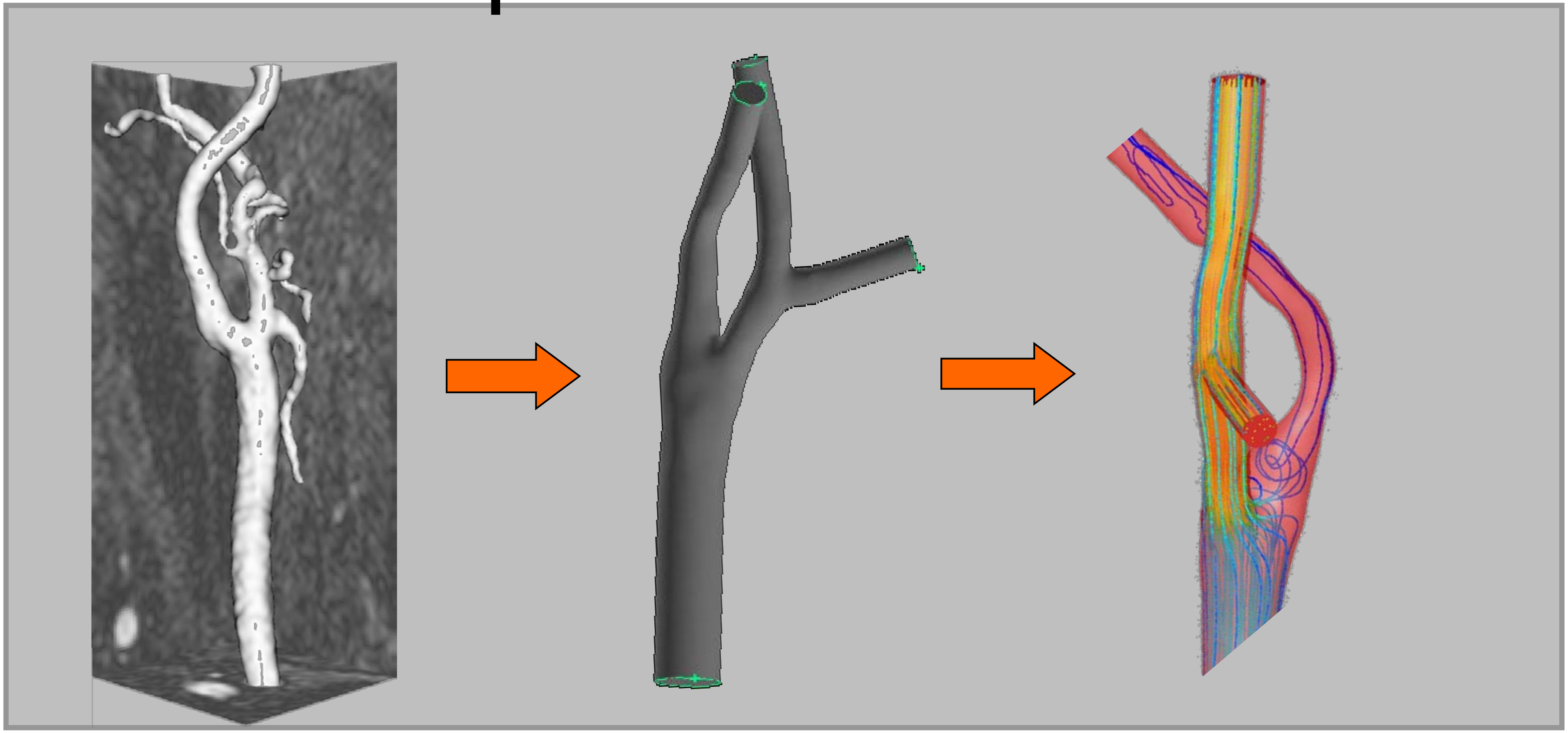
PTFE implant device in sheep
Performed by
Fabio Acocella and Stefano Brizzola
Dipartimento Di Scienze Cliniche Veterinarie
Facolta' Di Medicina Veterinaria
Universita’ Degli Studi Di Milano

In vivo image based measures



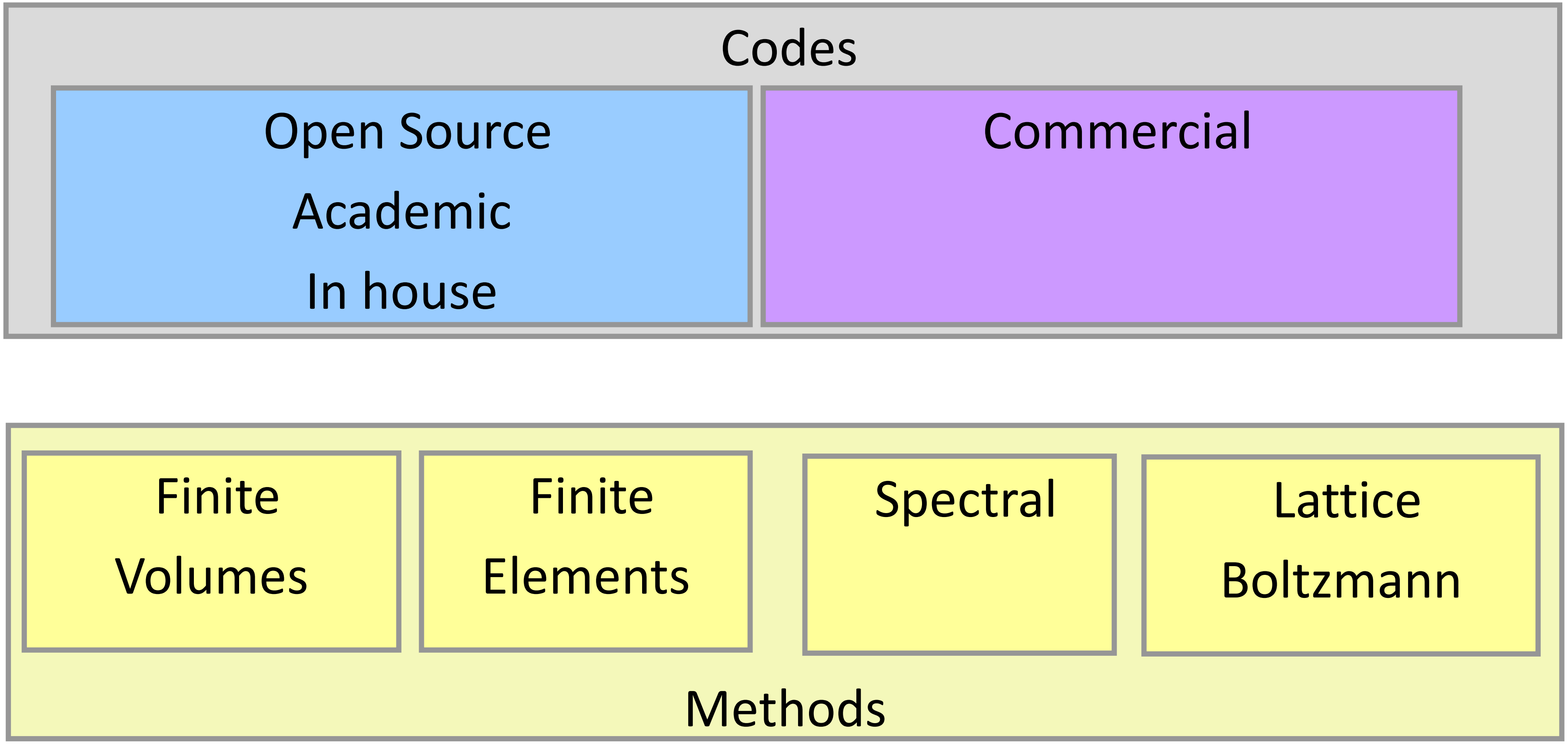
Phase contrast MRI acquisition (3D, 3 velocity ancoding directions)
Performed by
Giovanna Rizzo (IBFM-CNR) and
Marcello Cadioli (Philips Italia)

Computational model



1. Image processing or CAD tools
2. Geometry modeler
3. Meshing tools
4. CFD solvers

CFD solver



Codes

Open Source

Academic

In house

Commercial

Finite

Volumes

Finite

Elements

Spectral

Lattice

Boltzmann

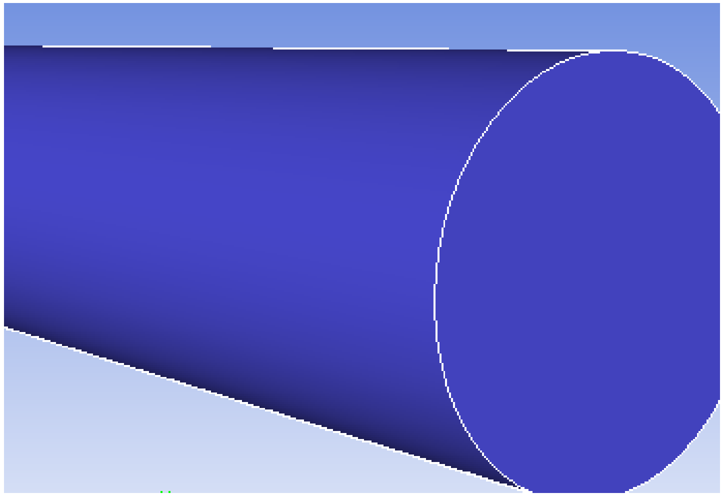
Methods

The CFD approach

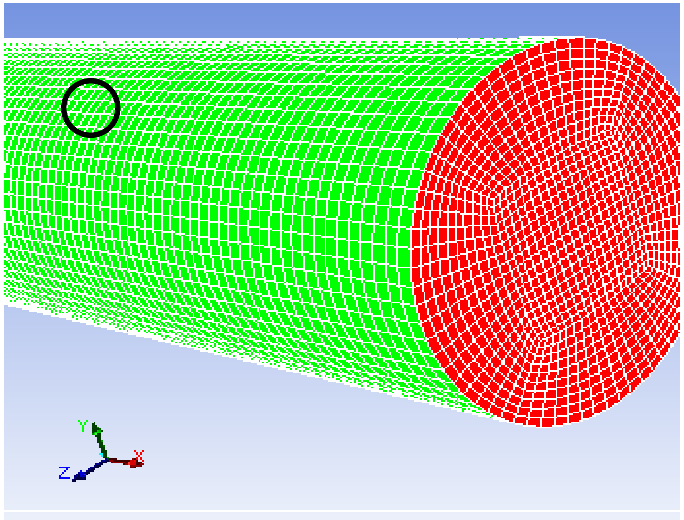
- THE FLUID DYNAMIC APPROACH uses the geometry and mechanical properties of the vasculature and the principles of conservation of mass and momentum to obtain the blood flow-pressure relation.
- The solution of such equations yields information on instantaneous velocity and pressure distributions.

Domain-mesh-cell

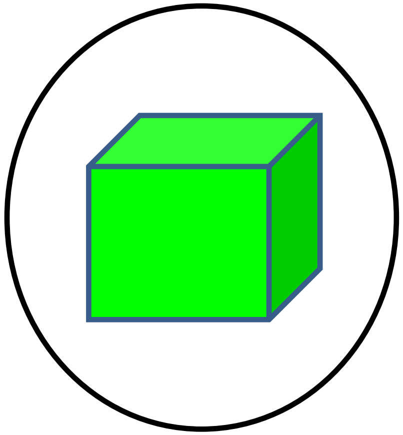
Computational domain



Mesh/grid/discretized domain



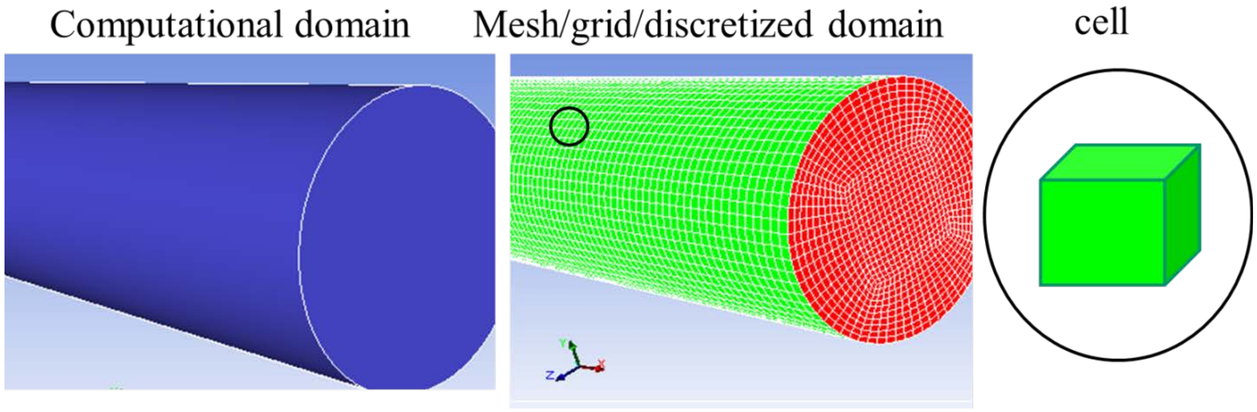
cell



Discretization

- Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the “grid” or the “mesh.”
- General conservation equations for mass and momentum are discretized into algebraic equations and solver for each and all cells in the discretization

(physic --> math --> numeric --> sw)



CFD concept-1

In theory:

If #cells → ∞ then the numerical solution →'exact'

and this will be independent from the numerical scheme adopted.

In practice:

#cells if finite then the numerical solution → 'OK'

and this will be dependent from the properties of the numerical scheme adopted

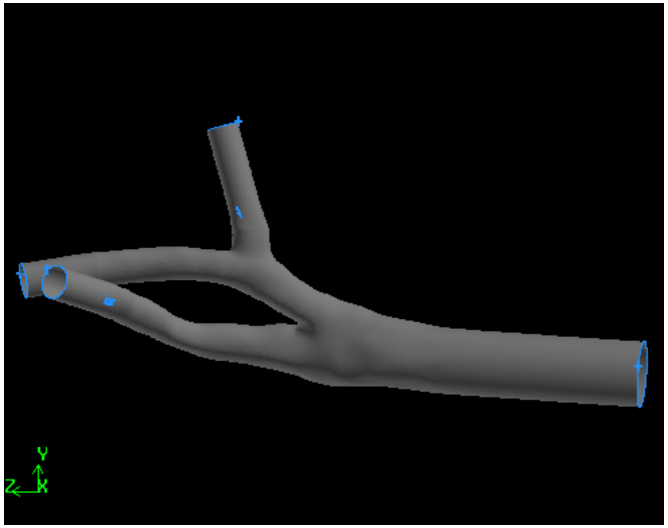
CFD concept-2

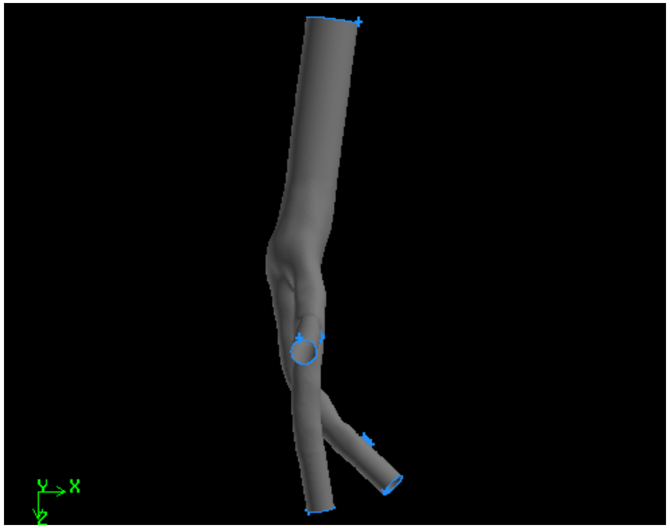
(Exact) Analytical solution is not available:

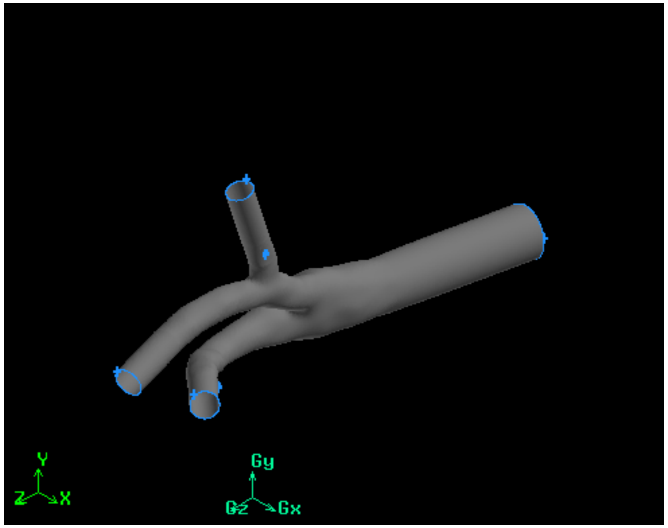
Numerical/Algebraic (Approximated) solution of the problem

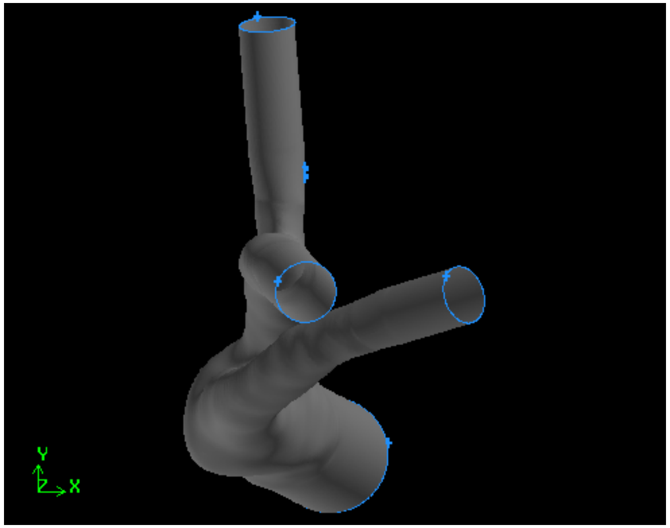
a tasking environment

A tasking environment: geometry

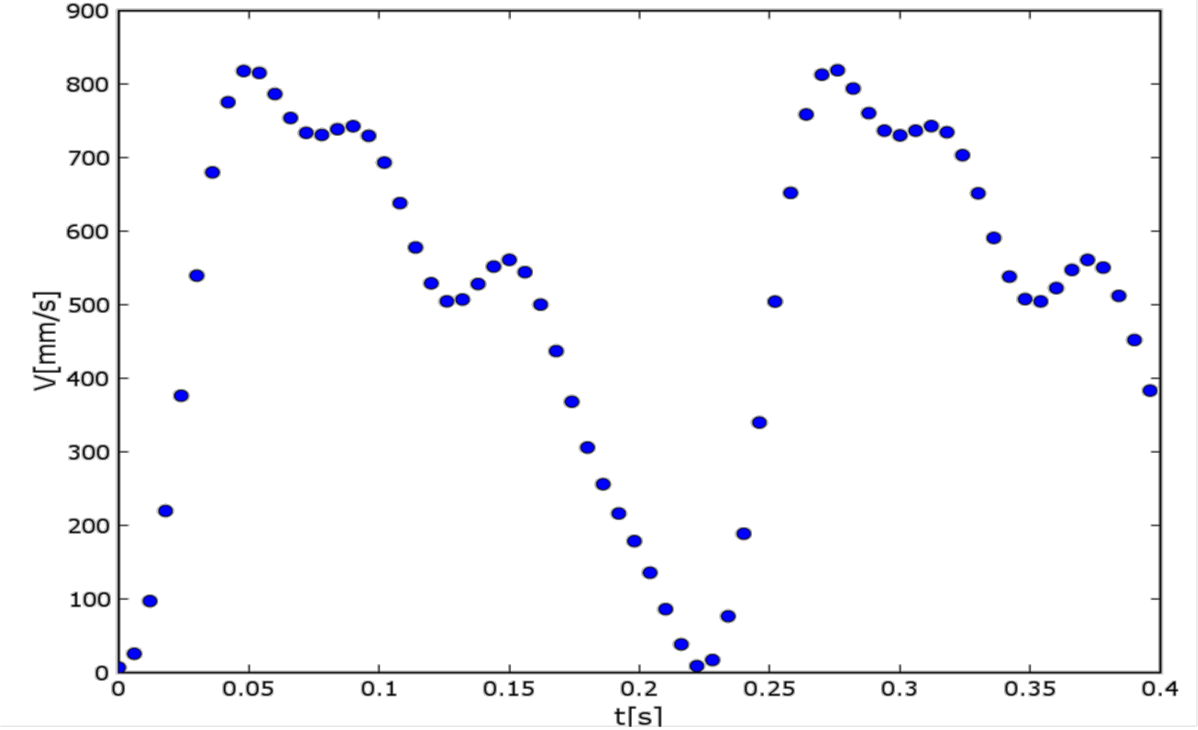




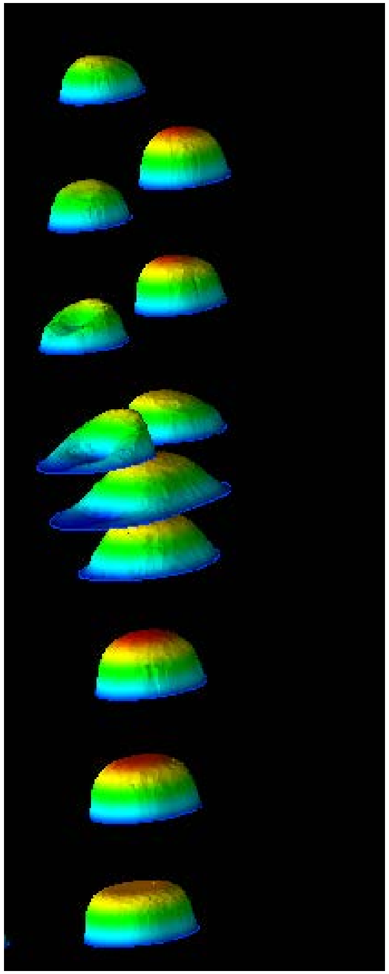




A tasking environment: physics



Time dependent flow waveforms



Spatial/temporal dependent velocity profiles

A tasking environment: knowledge

Limited ‘experimental’ knowledge on:

- ☐ Wall-properties (Young’s modulus?)
- ☐ Fluid-to-wall interaction (wall displacements?)
- ☐ Spatial/temporal velocity profiles distribution (accuracy of the measures in space & time?)

A tasking environment: math/numeric and tech

❑Convergence

❑ Consistency

❑ Stability

❑ Boundedness

❑ Conservativeness

❑ Transportiveness

Math **||** **Numeric**

How should I use my computational tools

CFD can be helpful (and economically inexpensive) for:

- System analysis (and optimization) (example blood filter)

  Fiore GB, Morbiducci U, Ponzini R, Redaelli A. Bubble tracking through computational fluid dynamics in arterial line filters for cardiopulmonary bypass. ASAIO J. 2009; 55:438-44.

- Detailed measures (3D, high resolution space/time) (CFD Arch AO compared to PC MRI resolution)

  Morbiducci U, Ponzini R, Grigioni M, Redaelli A. Helical flow as fluid dynamic signature for atherogenesis risk in aortocoronary bypass. A numeric study. J Biomech. 2007;40(3):519-34.

- Hypothesis testing

  Morbiducci U, Gallo D, Ponzini R, Massai D, Antiga L, Montevecchi FM, Redaelli A. Quantitative Analysis of Bulk Flow in Image-Based Hemodynamic Models of the Carotid Bifurcation: The Influence of Outflow Conditions as Test Case. Ann Biomed Eng. 2010; 38(12):3688-705.

- Validation of clinical practice (Doppler series)

  Ponzini R, Vergara C, Redaelli A, Veneziani A. Reliable CFD-based estimation of flow rate in haemodynamics measures. Ultrasound Med Biol. 2006; 32:1545-1555.

  Ponzini R, Lemma M, Morbiducci U, Montevecchi FM, Redaelli A. Doppler derived quantitative flow estimate in coronary artery bypass graft: a computational multiscale model for the evaluation of the current clinical procedure. Med Eng Phys. 2008; 30:809-16.

  Ponzini R, Vergara C, Rizzo G, Veneziani A, Roghi A, Vanzulli A, Parodi O, Redaelli A. Womersley number-based estimates of blood flow rate in Doppler analysis: in vivo validation by means of phase-contrast MRI. IEEE Trans Biomed Eng. 2010, 57:1807-15.

- Implantable devices (FSI)

  Nobili M, Morbiducci U, Ponzini R, Del Gaudio C, Balducci A, Grigioni M, Maria Montevecchi F, Redaelli A. Numerical simulation of the dynamics of a bileaflet prosthetic heart valve using a fluid-structure interaction approach. J Biomechanics, 2008; 41:2539-50.

  Morbiducci U, Ponzini R, Nobili M, Massai D, Montevecchi FM, Bluestein D, Redaelli A. Blood damage safety of prosthetic heart valves. Shear-induced platelet activation and local flow dynamics: a fluid-structure interaction approach. J Biomech. 2009 Aug 25;42(12):1952-60.

Blood fluid dynamics: theory, equations and examples

- Physic of blood
- Working hypothesis in vessels
- Conservation laws
- Implementation in Ansys Fluent
- Notes on convergence
- Notes on source of errors

Physic of blood

- The main functions of the **blood** are the transport, and delivery of oxygen and nutrients, removal of carbon dioxide and waste products of metabolism, distribution of heat and signals of immune system.
- The **blood** flow resistance is influenced by the complicated architecture of the vascular network and flow behaviour of **blood** components - **blood** cells and plasma.
- At a macroscopic level the **blood** appears to be a liquid material, but at a microscopic level the **blood** appears to be a material with microscopic solid particles of varying size - various **blood** cells.

Blood density

Density is a fluid property and is defined as:

ρ = Mass/Volume

Usually blood density is assumed to be similar (equal) to water density at $T=300$ K and $P=10^5$ Pa

ρ = 1060 [Kg/m^3]

Blood viscosity

For viscous flow if the relationship between viscous forces (tangential component) and velocity gradient is linear then the fluid is called Newtonian and the slope of the line is a measure of a fluid property called viscosity (dynamic).

$S_y = \mu \, dv/dx$

Also a cinematic viscosity can be defined by:

$\upsilon = \mu/\rho$

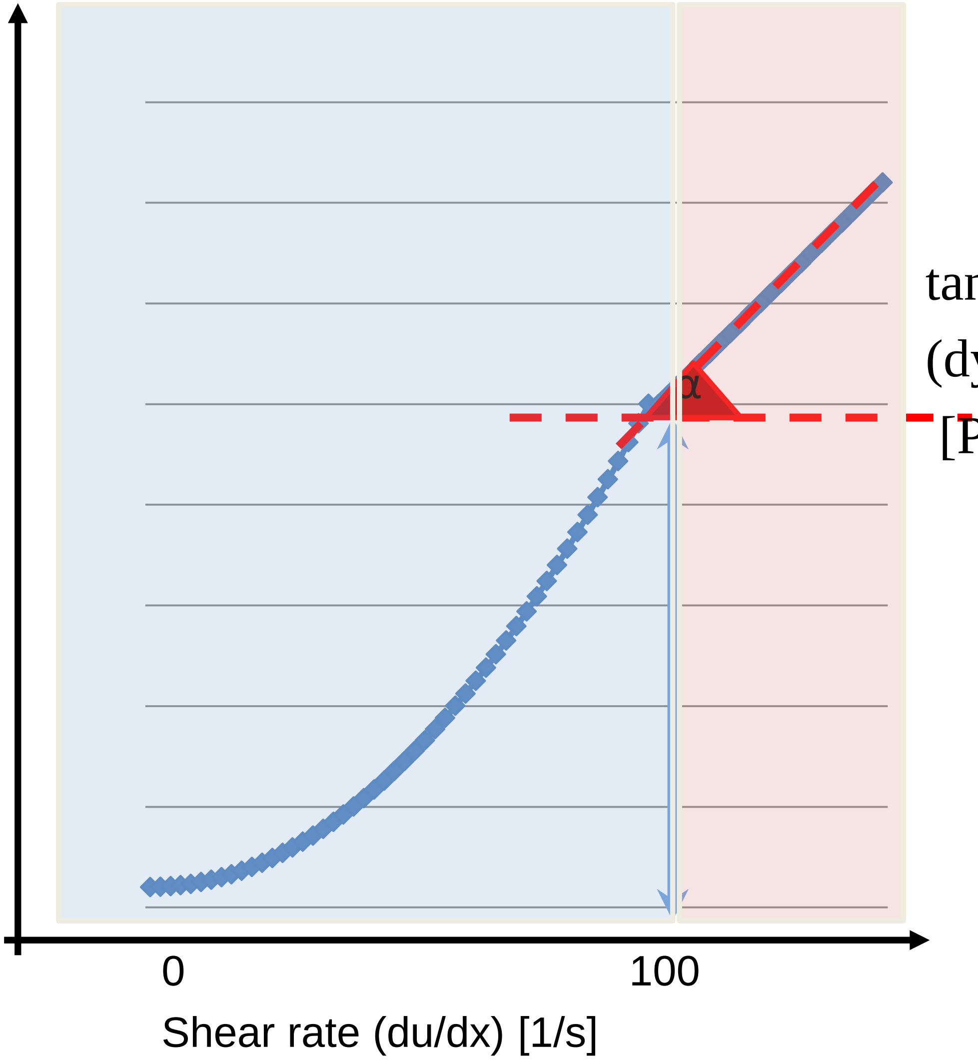
Typical values are:

$\mu_{blood} = 0.0035$ [kg/ms] ($\upsilon_{blood} = 3.3 \; 10^{-6}$ [m^2/s])

Blood is Newtonian and non Newtonian

Shear stress

[Pa]



tang(α) = μ

(dynamic viscosity)

[Pa s]

Reynolds number

- The Reynolds number Re is defined as:

$$\mathrm{Re} = \mathrm{r}\, V L / \mathrm{m}.$$

Here:

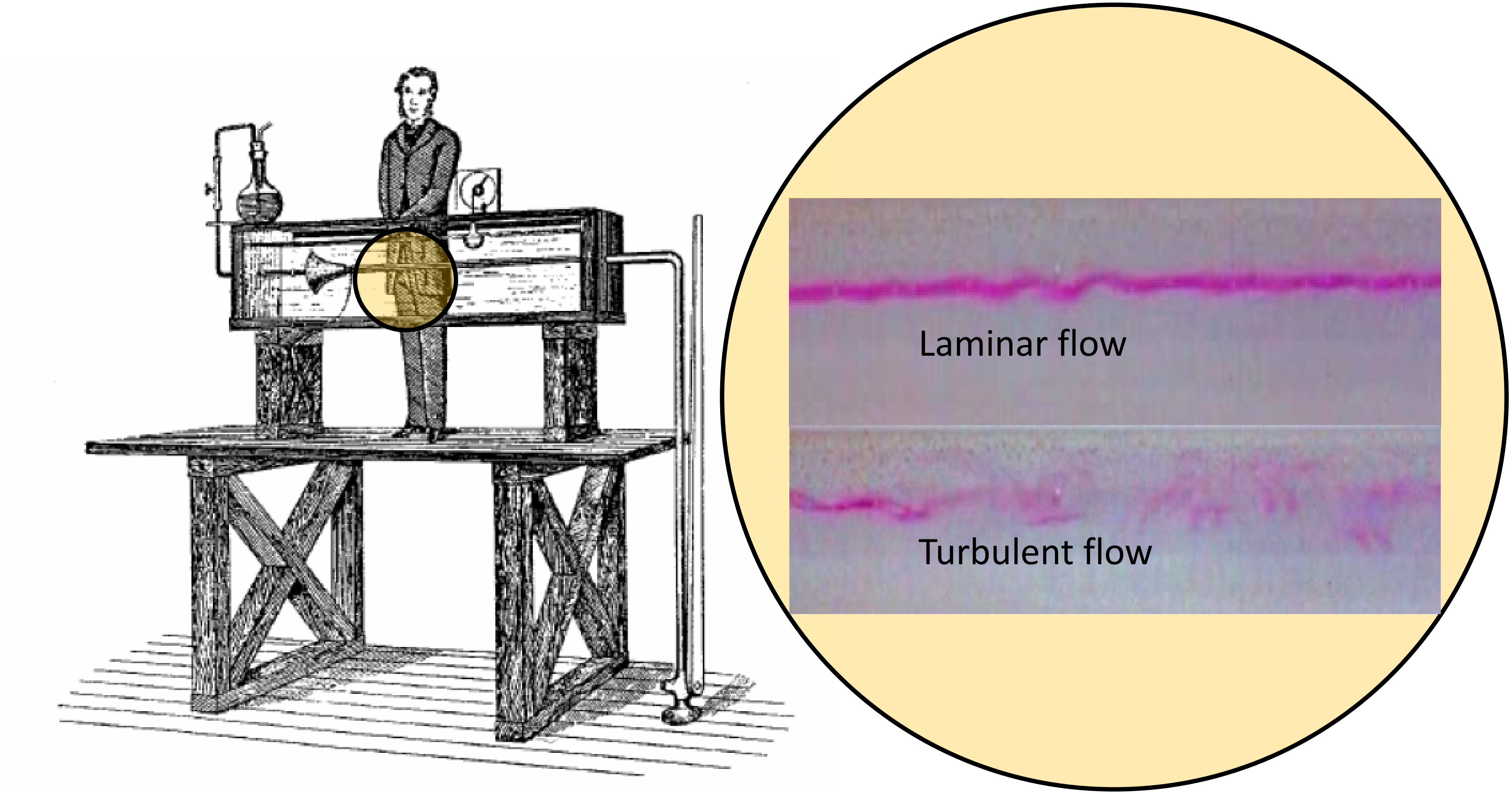
L is a characteristic length (say D in tubes)

V is the mean velocity over the section(*Q/Area*)

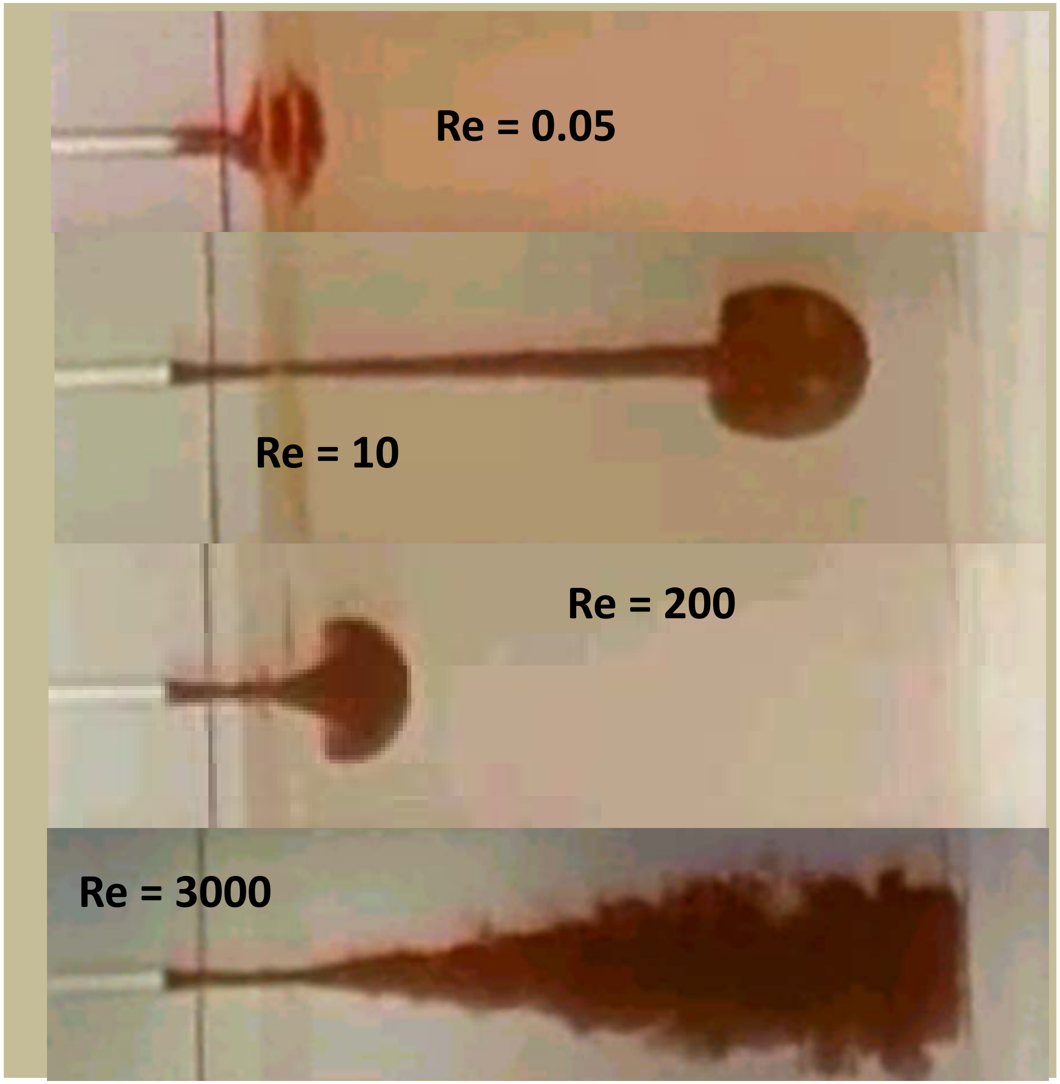
density and viscosity are: r, m

- If Re >> 1 the flow is dominated by inertia.
- If Re << 1 the flow is dominated by viscous effects.

Effect of Reynolds number



Effect of Reynolds number



Blood flow regimen

Womersley number

- The Womersley number W (or Wo or α) is defined as:

$$W = L(2\pi f\ r/m)^{1/2.}$$

Here:

L is a characteristic length (say $D/2$ in tubes)

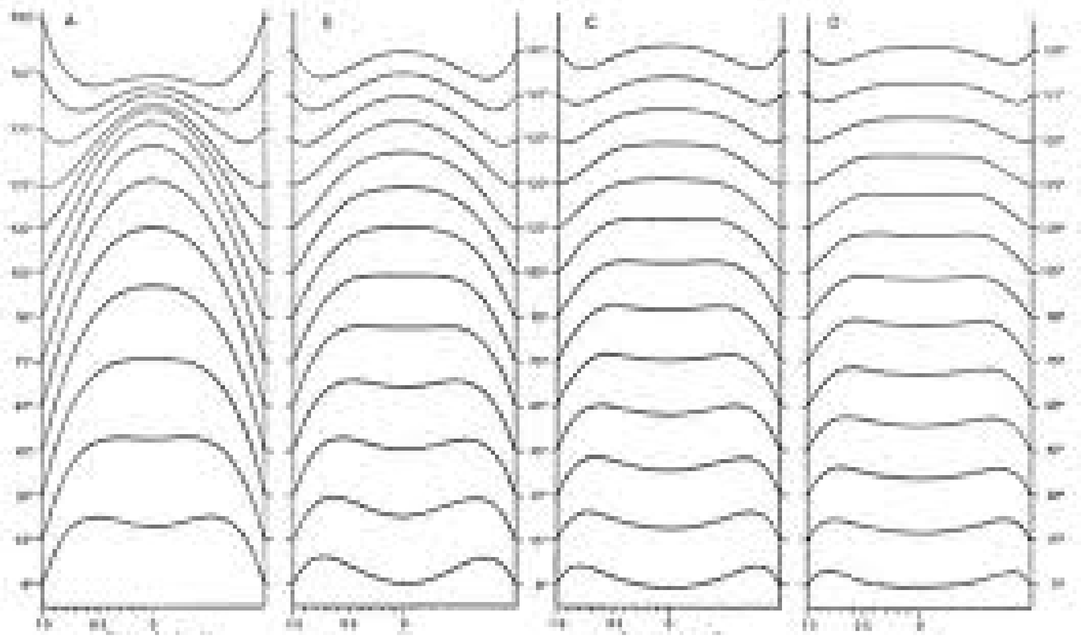
f is the frequency of the flow waveform (1/period)

density and viscosity are: r, m

- If $W \neq 0$ (< 1) the flow is dominated by viscous effect (similar to a poiseuille flow).
- If $W >> 1$ the flow is dominated by transient effect

Effect of Womersley number

Some typical values for the Womersley number in the cardiovascular system for a canine at a heart rate of 2Hz are:



Ascending Aorta -- 13.2
Descending Aorta -- 11.5
Abdominal Aorta -- 8
Femoral Artery -- 3.5
Carotid Artery -- 4.4
Arterioles --.04
Capillaries -- 0.005
Venules -- 0.035
Interior Vena Cave -- 8.8
Main Pulmonary Artery -- 15

Womersley VS Reynolds

$$\mathrm{Re} = \frac{convective\ inertia\ force}{viscous\ friction\ force}$$

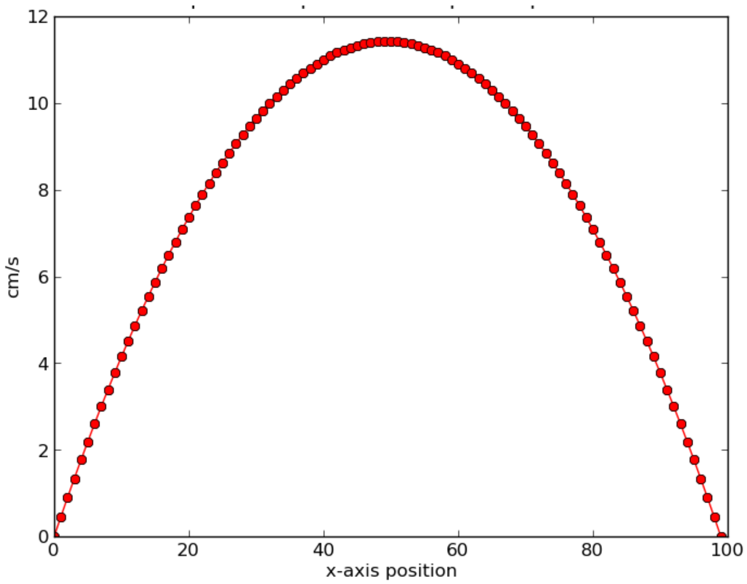
$$\mathrm{W} = \frac{transient\ inertia\ force}{viscous\ friction\ force}$$

Flow classifications

- Laminar vs. turbulent flow.
  - Laminar flow: fluid particles move in smooth, layered fashion (no substantial mixing of fluid occurs).
  - Turbulent flow: fluid particles move in a chaotic, “tangled” fashion (significant mixing of fluid occurs).
- Steady vs. unsteady flow.
  - Steady flow: flow properties at any given point in space are constant in time, e.g. $p = p(x,y,z)$.
  - Unsteady flow: flow properties at any given point in space change with time, e.g. $p = p(x,y,z,t)$.

Steady laminar flow in a cylinder

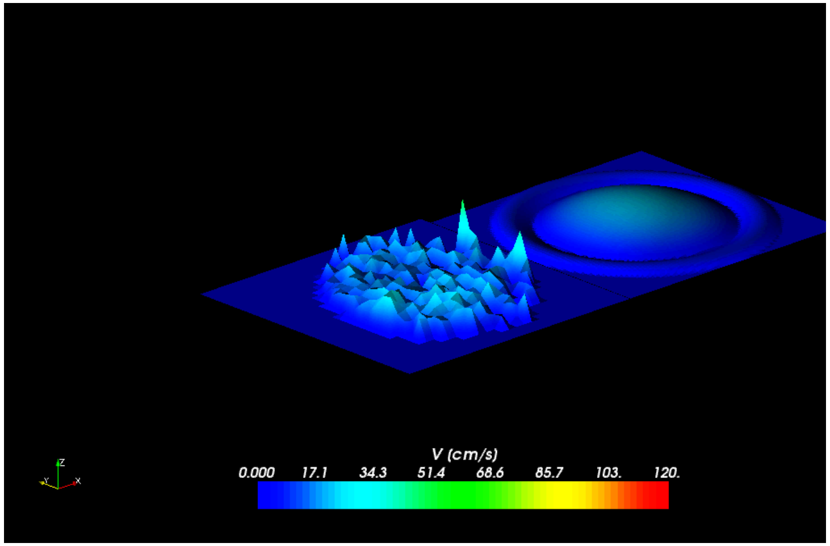
- Steady viscous laminar flow in a horizontal pipe involves a balance between the pressure forces along the pipe and viscous forces.
- The local acceleration is zero because the flow is steady.
- The convective acceleration is zero because the velocity profiles are identical at any section along the pipe.



- The shape of the spatial velocity profile is a parabola centered on the axis of the cylinder.
- The peak value is proportional to the pressure drop actin on the cylinder.

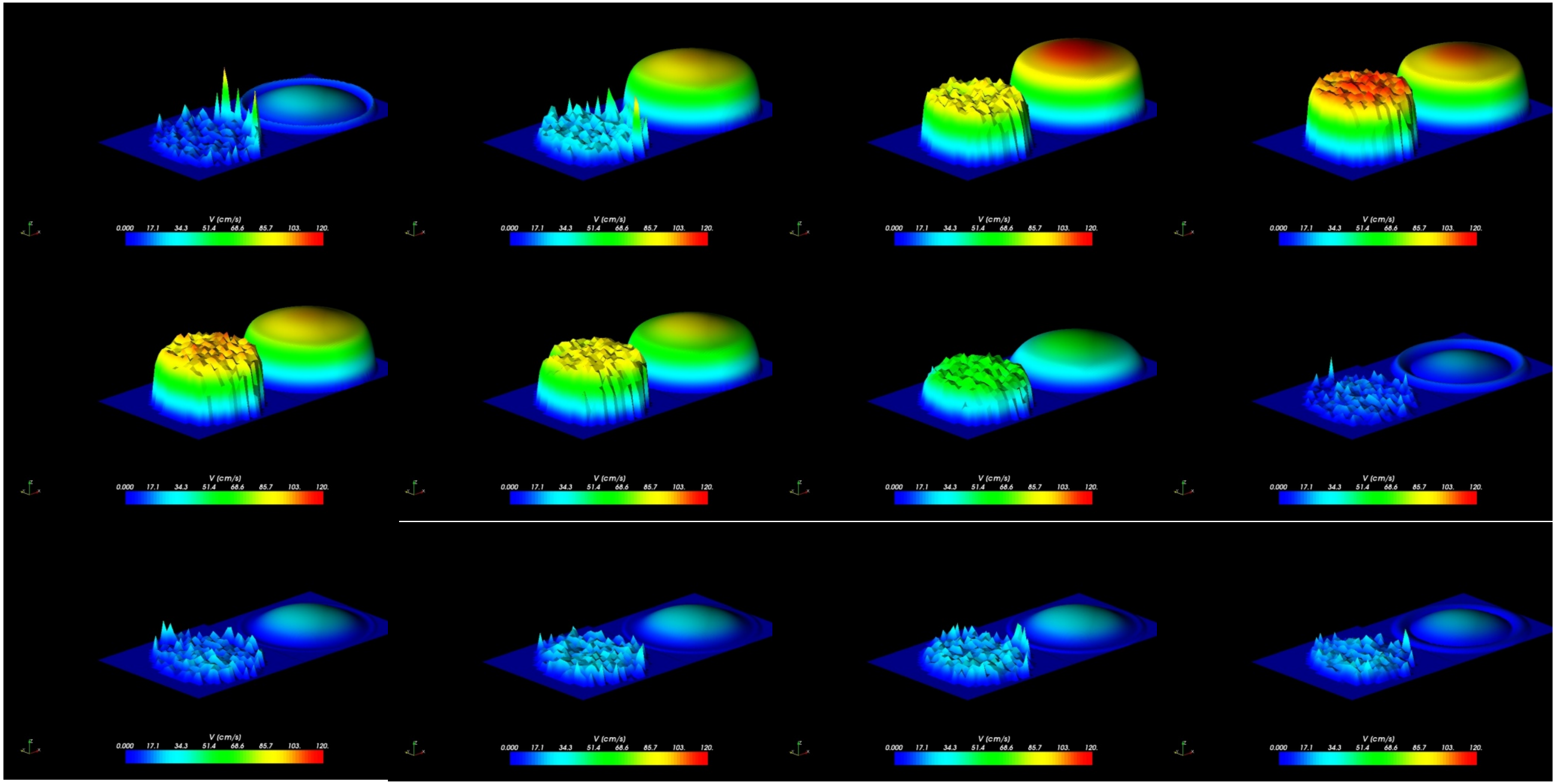
Unsteady flow in a cylinder

- Unsteady flow in a cilinder is governed by a balance among:
  - local acceleration,
  - convective acceleration,
  - pressure gradients
  - viscous forces
- In syntheys all the factors in the Navier-Stokes equations are relevant to determine the flow evolution.
- Thanks to the symmetry properties of the domain the solution of this problem can be analytically be determined (Womersley solution)
- In general due to the shape of the domain this is not possible



Womersley solution
VS
PC MRI acquisition in a straigth abdominal aorta section

Womersley solution
VS PC MRI acquisition in a straigth abdominal aorta section



Incompressible vs. compressible flow

- Incompressible flow: volume of a given fluid particle does not change.
  - Implies that density is constant everywhere.
  - Essentially valid for all liquid flows.
- Compressible flow: volume of a given fluid particle can change with position.
  - Implies that density will vary throughout the flow field.

Single phase vs. multiphase flow & homogeneous vs. heterogeneous flow

- Single phase flow: fluid flows without phase change (either liquid or gas).
- Multiphase flow: multiple phases are present in the flow field (e.g. liquid-gas, liquid-solid, gas-solid).
- Homogeneous flow: only one fluid material exists in the flow field.
- Heterogeneous flow: multiple fluid/solid materials are present in the flow field (multi-species flows).

Blood working hypothesis

1. Laminar flow (Reynolds < 2300)
2. Incompressible fluid
3. Unsteady behavior (Womersley ≠ 0)

The problem is described exactly by:

- three Navier-Stokes equations
- the equation of continuity

- BUT:
  - A general solution of such a system of nonlinear partial differential equations has not been achieved;
  - The physiological quantities which would arise in a treatment of blood flow in large arteries are not well known.

For both reasons it is necessary **to work in terms of approximate models**, <u>which include the important features of the system under consideration and neglect unimportant features.</u>

Important features of blood as fluid

1. Continuum hypothesis (molecular scales and or suspended particles are not relevant to study large arteries (d > 1 μm)
2. Laminar (Reynolds < 2000)
3. Unsteady (Womersely >> 1)
4. Incompressible (density ρ is constant ≈ ρwater)
5. Isotropic (same behaviour in all directions)
6. Newtonian (viscosity (μ and λ) are constant and do not depends on the shear rate)

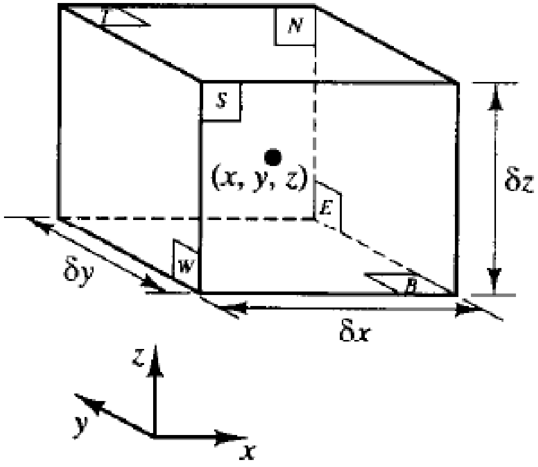
Unimportant features of blood as fluid

- Viscosity depends on:
  - Temperature (energy eqn can be neglected)
  - % hematocrit
- Compressible (around 10^{-11}[m^2/N] for low pressure)
- Transitional to turbulent under particolar conditions and for certaind vascular disctricts (valves, etc.)

Equations of conservation

Two general conservation equations:

1. Mass (divergence free)
2. Momentum: Newton’s second law: the change of momentum equals the sum of forces on a fluid particle



Control-volume