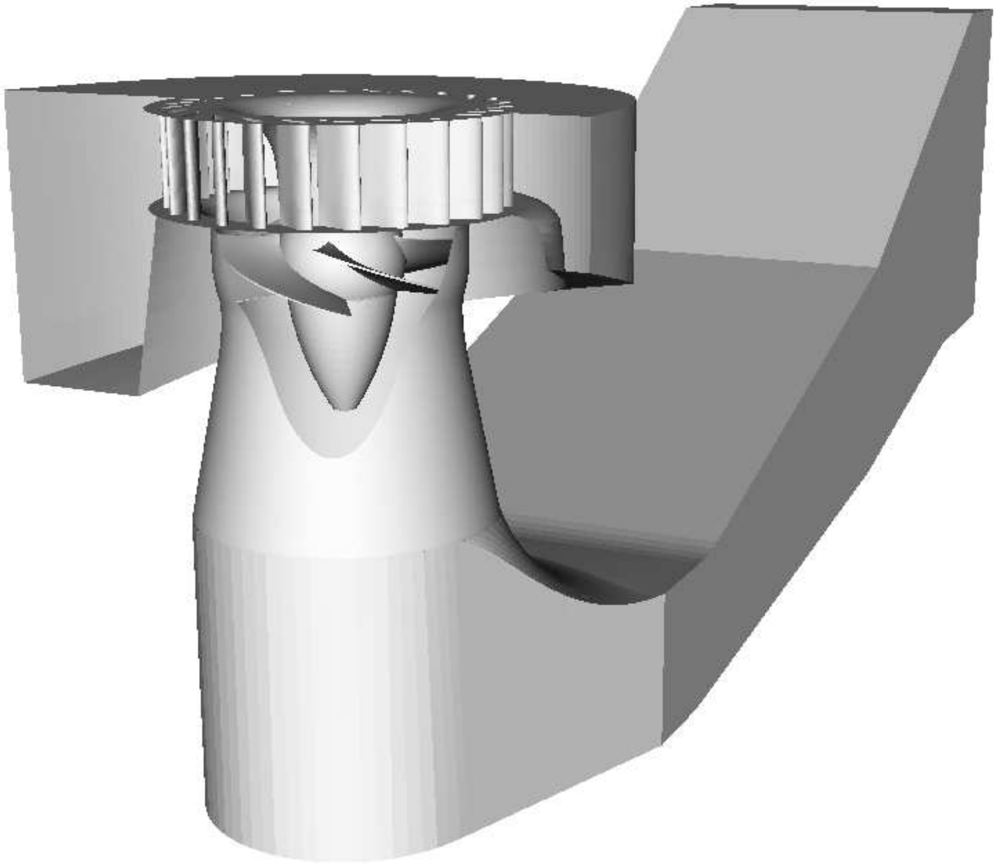
OpenFOAM SIMULATION OF THE FLOW IN THE HÖLLEFORSEN DRAFT TUBE MODEL

Nilsson H. and Page M.
Chalmers / Hydro Quebec



The OpenSource OpenFOAM CFD solver

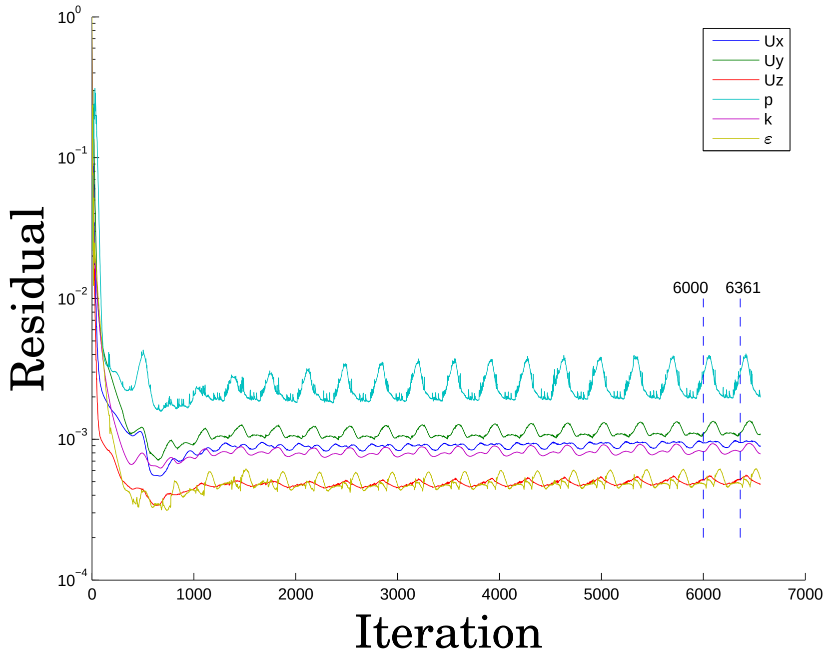
- OpenFOAM = Open Field Operation and Manipulation, www.openfoam.org
  An OpenSource object oriented C++ tool for solving PDE's
- Preprocessing (grid generator, converters, manipulators, case setup)
- Postprocessing (using OpenSource Paraview)
- Many specialized CFD solvers implemented, e.g.
  - simpleFoam: A finite volume steady-state solver for incompressible, turbulent flow of non-Newtonian fluids, using the SIMPLE algorithm
  - turbFoam: A finite volume solver for unsteady incompressible, turbulent flow of non-Newtonian fluids, using the PISO algorithm
  - icoTopoFoam: Sliding grid
- OpenSource = possibility to have insight into the code
  - * Makes development and tailor-made solvers possible
    - simpleUnsteadyFoam: Unsteady SIMPLE solver
    - cavInterFoam: Cavitation using VOF and the Kunz' cavitation model
  - * Makes research implementations available and results reproducable.
- Access to an international community of OpenFOAM users
- Runs in parallel using automatic/manual domain decomposition.

Studied case

- Case 1: Steady calculation
- Prescribed wall-function grid, $1,002,360$ grid points.
- Standard $k-\varepsilon$ turbulence model with wall-functions
- Axi-symmetric inlet boundary conditions using linear interpolation of the measurements (velocities multiplied by a factor of 1.07 to get the correct volume flow), $k=1/2(\overline{u^2}+\overline{v^2}+\overline{w^2})$, $\varepsilon=C_\mu^{3/4}k^{3/2}/l_{turb}$, $C_\mu=0.09$, $l_{turb}=0.1*(R_{wall}-R_{cone})$
- Homogeneous Neumann b.c. at the outlet for velocity, k and ε. Back-flow values of $k=0.4$ and $\varepsilon=3.55$ whenever needed (derived from the inlet average).
- The pressure had a homogeneous Neumann boundary condition everywhere except at the outlet, where it was set to zero.
- No surface roughness used.
- Gamma discretization scheme. A smooth and bounded blend between the second-order central scheme and the first-order upwind scheme.

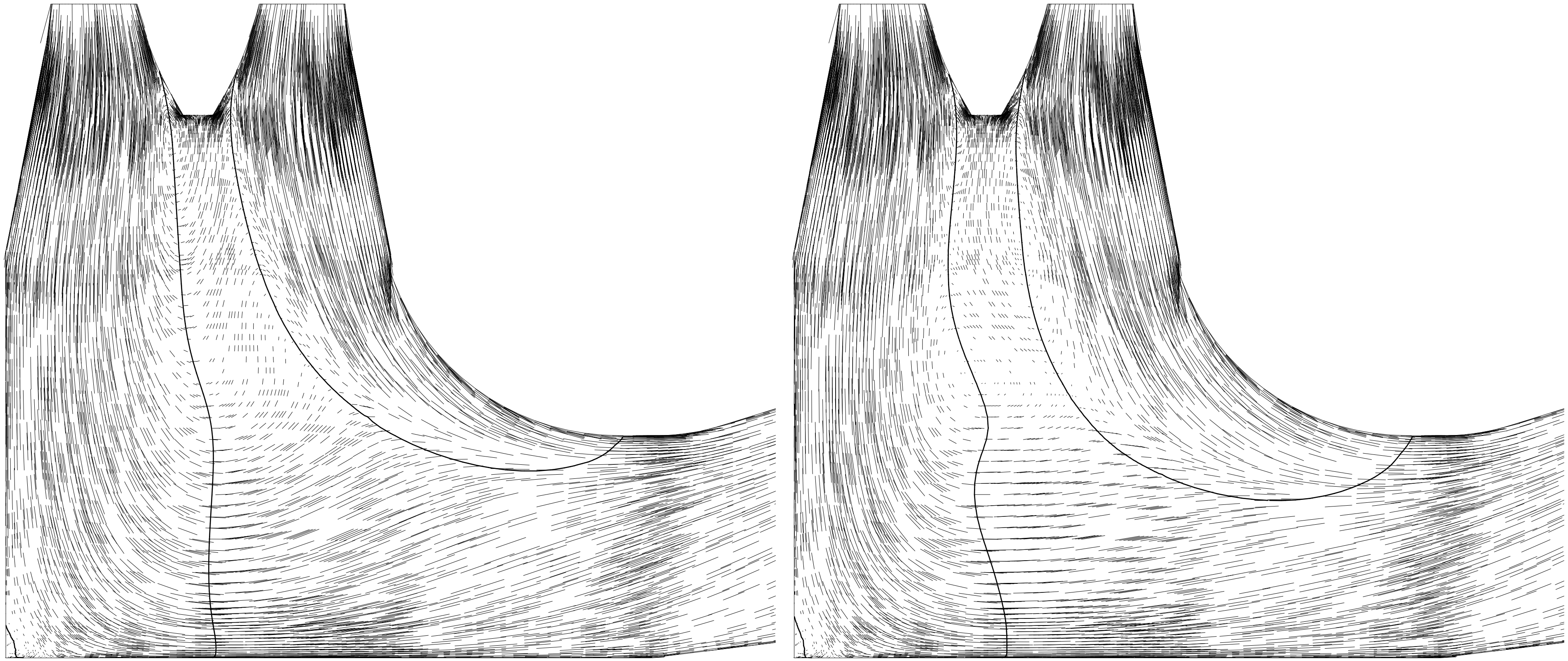
Convergence

- The residuals drop rapidly in the initial phase of the computation, but are then oscillating in a periodic fashion about a mean level.
- Inherent unsteadiness in the flow makes a steady solution impossible with the present level of diffusion (turbulence and numerical).
- One period in the fluctuating residuals correspond to one period in the vortex rope.
- Chosen averaging interval shown with dashed lines, yielding the 'quasi-steady' solution that is presented in the present work.

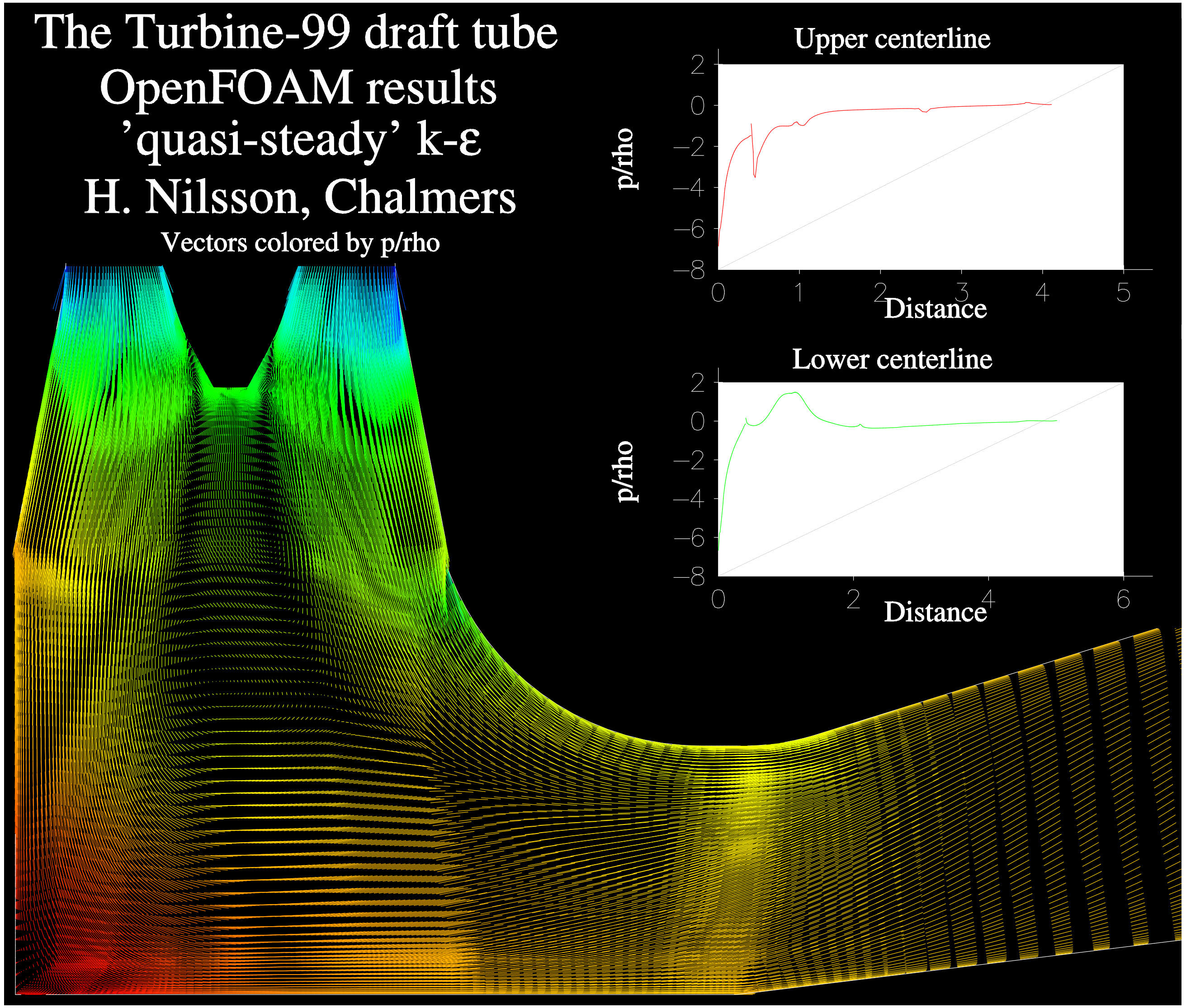


Unsteadiness

- Two snapshots of the flow to show the magnitude of the flow oscillations. The solid lines are where the vertical velocity is zero.



Unsteadiness, the movie

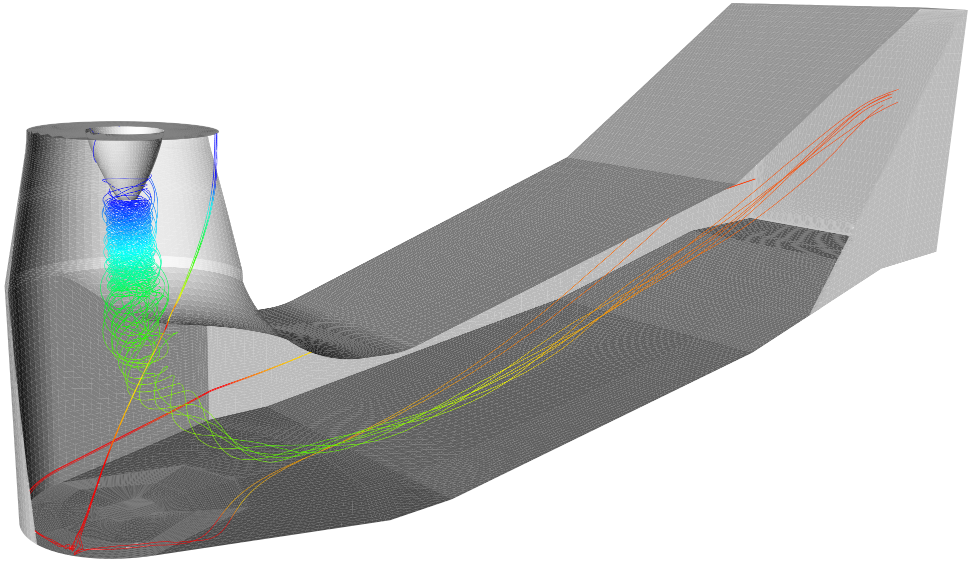


Computational time and parallel performance

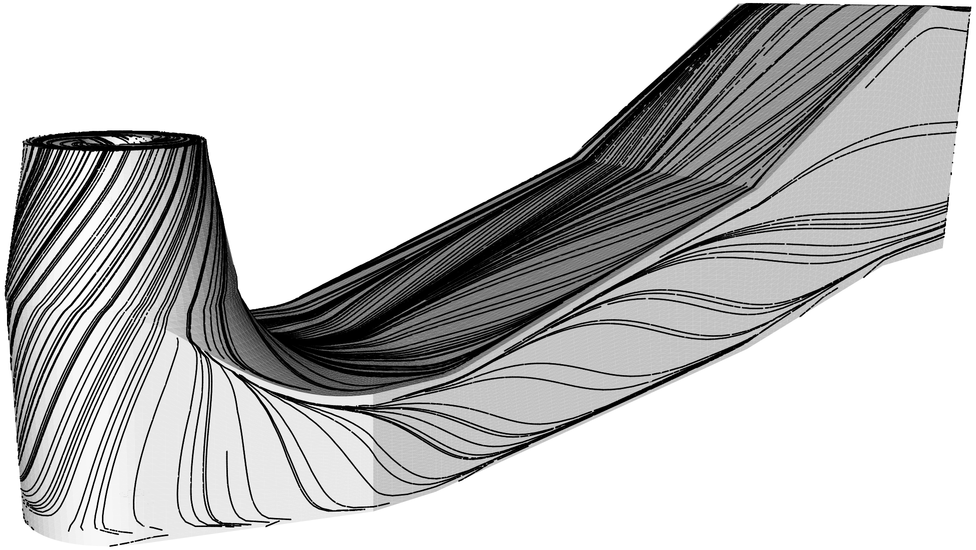
- 15s wall time (including some output) and 7s CPU time per iteration
- 10 CPU's on a dual node AMD Opteron cluster with 2.2GHz CPU's, 500MB RAM/CPU, 1MB cache/CPU and a 100Mbps Ethernet network.
- Default settings used in OpenFOAM $\Longrightarrow$ The parallel efficiency could be improved.

Computational results

- The results are very similar to CFX-5 results by Page et al.
- y^+ range $2 \leq y^+ \leq 150$ ($\overline{y^+} \approx 30$)
- $C_{pr} = 1.0171$ (pressure recovery factor)
- $C_{prm} = 0.9027$ (mean pressure recovery factor)
- $\zeta = 0.173$ (energy loss coefficient)



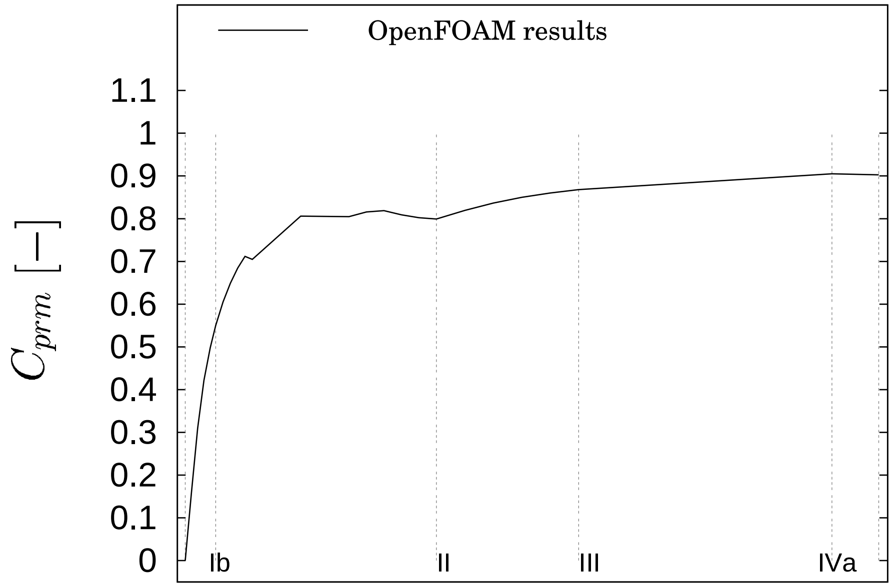
Streamlines colored by static pressure

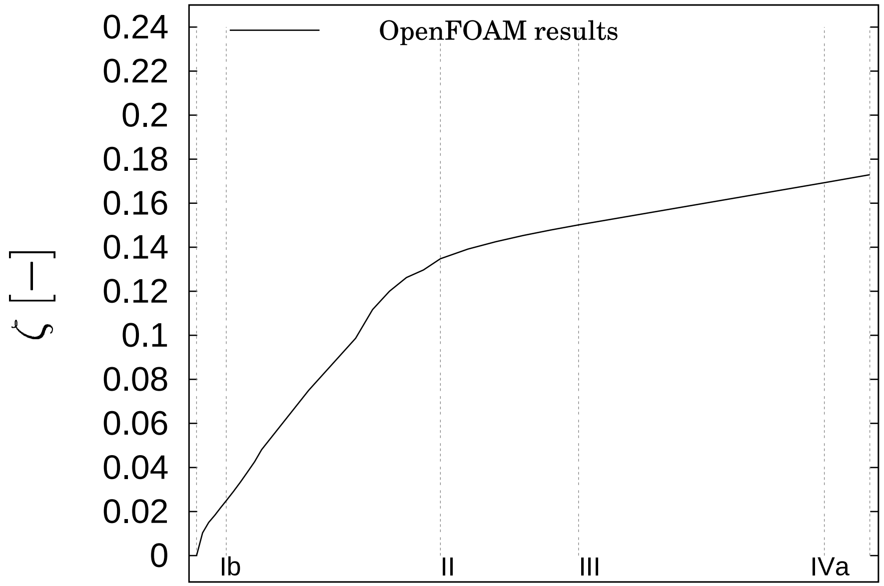


Surface smearlines

Through-flow analysis

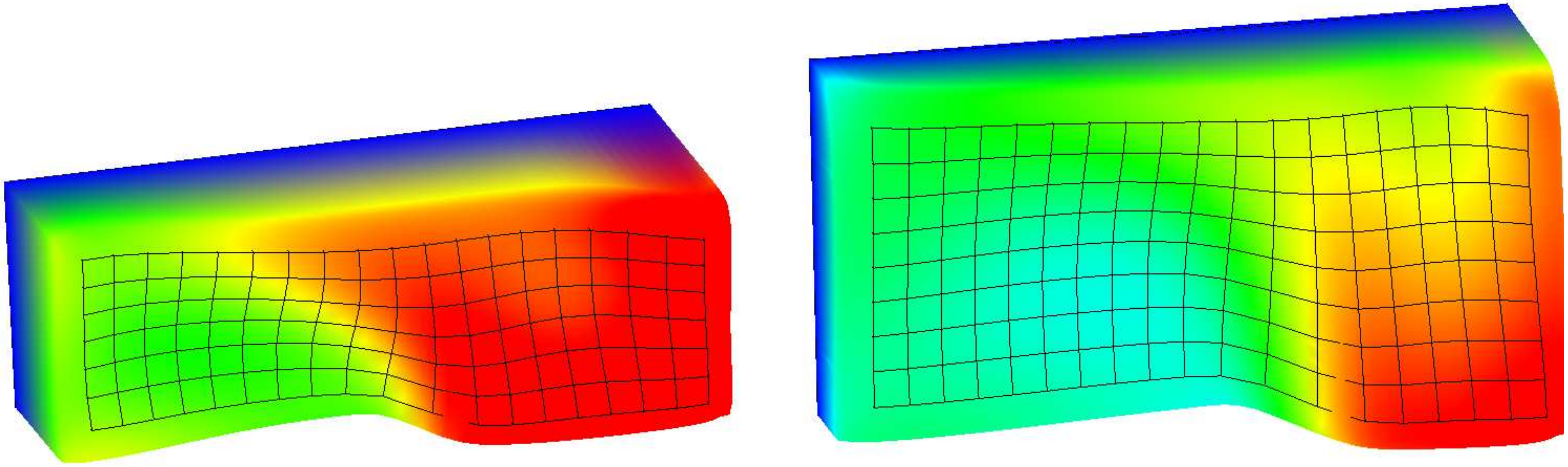
- Through-flow analysis of the mean pressure recovery factor (C_{prm}) and the energy loss coefficient (ζ)





Velocity distributions

To be compared in detail with measurements and other computational results at the workshop.



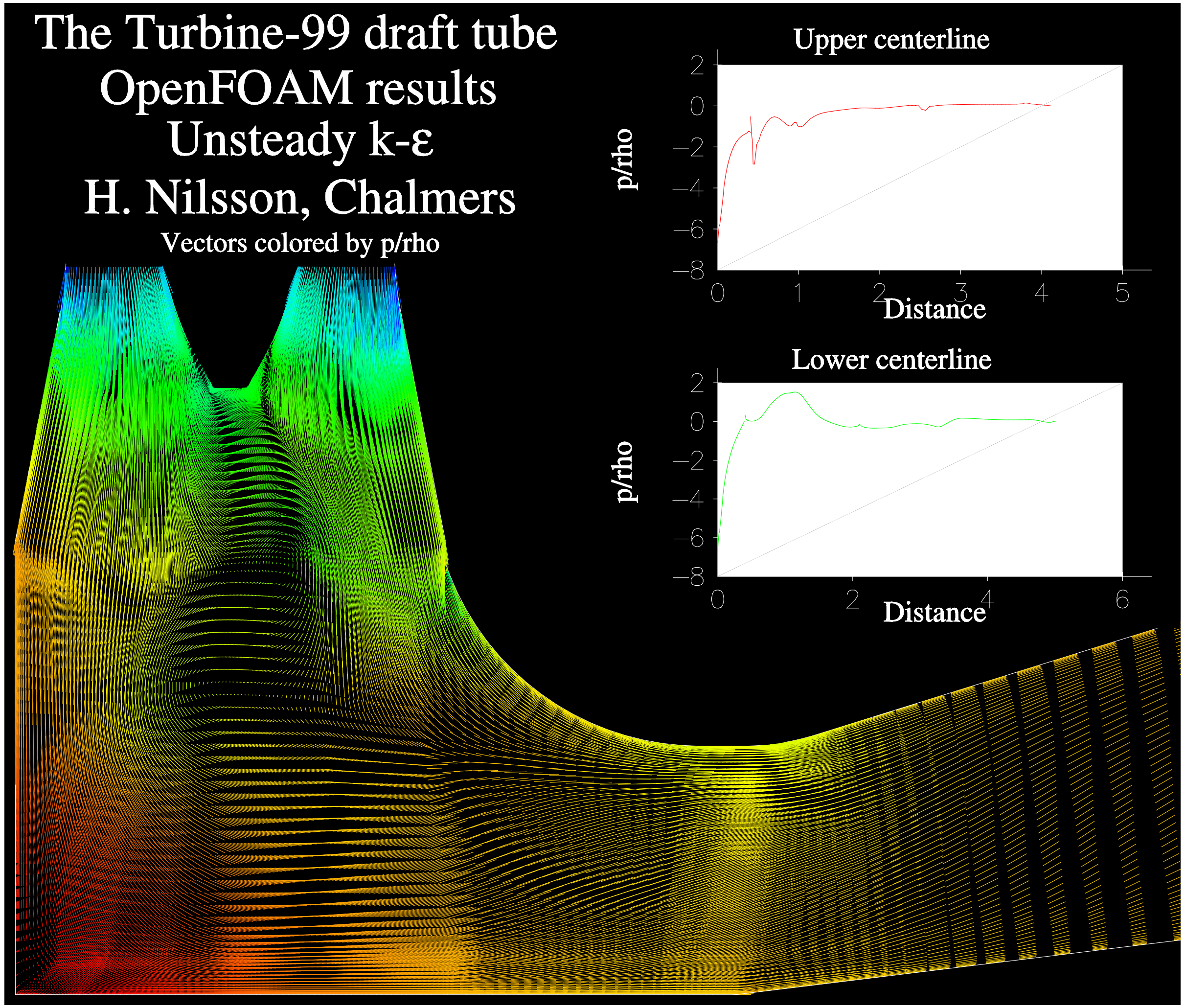
Through-flow velocity distributions at Cross-Sections II and III

Unsteadiness

- The ’quasi-steady’ solution was unsteady
- The time terms are not included in the equations
- A ’false time step’ is given by the iterations and the under-relaxations
- The ’false time step’ is different for different control volumes
- A true unsteady solution is needed to resolve the unsteadiness
- An unsteady $k - \varepsilon$ computation of the same case has been made

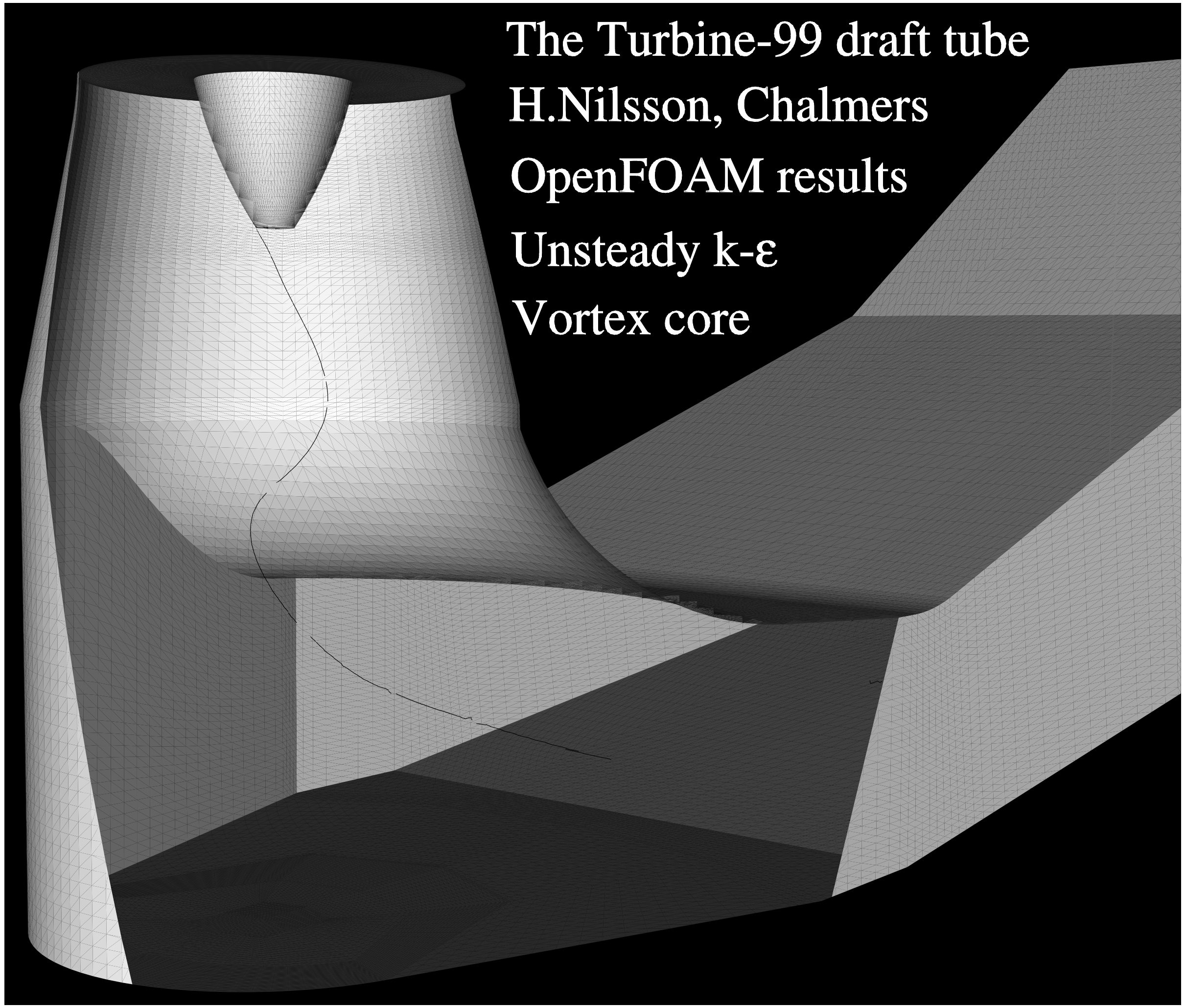
Preliminary unsteady computations

Time step: $3 \cdot 10^{-4}s$, Periodicity in time: $\sim 0.48s$



Preliminary unsteady computations

Time step: $3 \cdot 10^{-4}s$, Periodicity in time: $\sim 0.48s$



Conclusions

- The steady computation was unable to dampen the physical unsteadiness in the flow, i.e. neither the $k - \varepsilon$ turbulence model nor the numerical schemes were diffusive enough to yield a fully steady solution.
- A preliminary unsteady computation yields a highly unsteady flow
- The averaged 'quasi-steady' solution yields results very similar to those of CFX-5 (Page et al.)
- OpenFOAM is able to generate good computational results in an efficient way.
- The OpenFOAM common platform facilitates international collaboration like the one in the present work.

Thank you for your attention!

Acknowledgements

The OpenFOAM developers (www.openfoam.org)
Dr. Niklas Nordin, SCANIA
Dr. Hrvoje Jasak, Wikki Ltd.
Martin Beadoin, Hydro Quebec

Financing

ELFORSK (Swedish Electrical Utilities Research and Development Company)
The Swedish National Energy Administration
GE Energy Sweden AB