CFD for water power using OpenFOAM

Håkan Nilsson
Chalmers University of Technology
Applied Mechanics / Fluid Dynamics / CFD

Outline of the presentation:

- Presentation of SVC
  (the Swedish Water Power Center)
- Presentation of OpenFOAM
- Presentation of the use of OpenFOAM in the water turbine field, including validation against results from CFX-5 and measurements
- Presentation of other people and activities within SVC using CFD

SVC – Svenskt VattenkraftCentrum (Swedish Water Power Center, www.svc.nu)

Two main parts:

- Civil engineering in water power constructions, LTU/KTH
- Turbines and generators, CTH/LTU/UU

The second part is divided into four parts:

- Tribology (lubrication of bearings), LTU
- Generators, UU
- Rotor dynamics, LTU
- Fluid dynamics, CTH/LTU

Collaboration partners: Swedish Energy Agency, Hydro Power companies[a] (through Elforsk AB), GE Energy (Sweden) AB, Waplans Mekaniska Verkstad AB, Chalmers, LTU, KTH, Uppsala university.

[a] Vattenfall AB Vattenkraft, Fortum Generation AB, Sydkraft Vattenkraft AB, Skellefteå Kraft AB, Graninge Kraft AB, Jämtkraft AB, Sollefteåforsens AB, Karlstads Energi AB, Gävle Energi AB, Öresundskraft AB

SVC – Ph.D. projects within Computational Fluid Dynamics

Ongoing and starting ph.D. projects within computational fluid dynamics:

- Unsteady flow in draft tubes, CTH
- Full scale study of cavitational damage and numerical investigation, CTH/LTH/WAPLANS
- Development of two-phase flow and cavitation models for full scale water turbine runner flow, CTH/LTH
- CFD modeling of the flow in the U9 model and prototype using OpenFOAM, CTH

Examples of collaboration projects:

- Coupling between rotor dynamics and the water flow, Martin Karlsson, LTU.
- Air cooling of water turbine generators, (planned), Samir Saaidi, VRD.

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
  - icoDyMFoam: Sliding/moving grid
- OpenSource = possibility to have insight into the code
  * Makes development and tailor-made solvers possible
    - simpleUnsteadyRotatingFoam: Unsteady SIMPLE solver with Coriolis and centrifugal terms
    - 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.

OpenFOAM, further information

The official OpenFOAM homepage:
http://www.openfoam.org

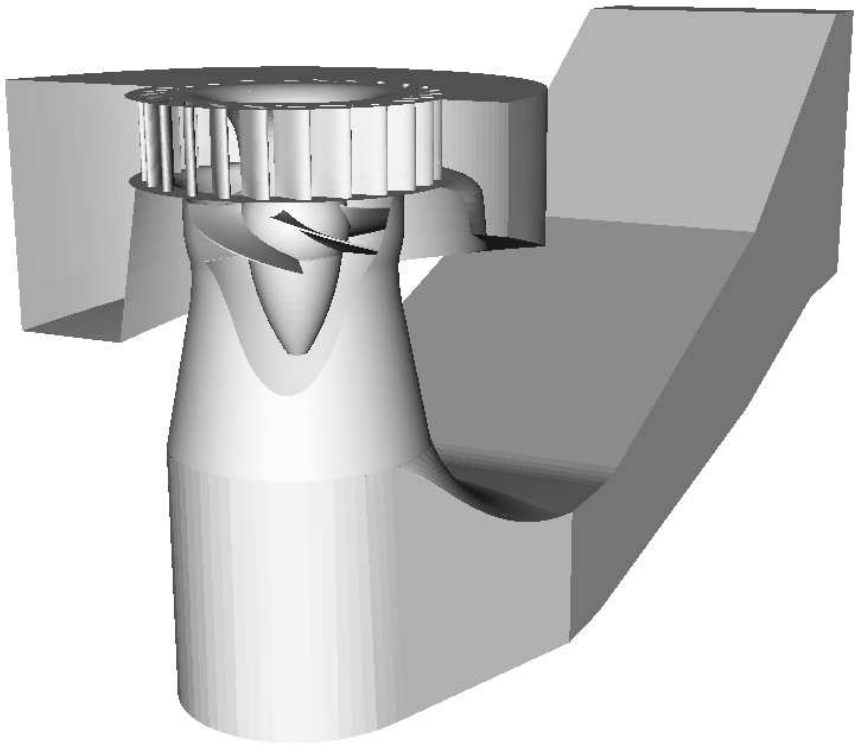
The OpenFOAM workshop homepage:
http://www.openfoamworkshop.org

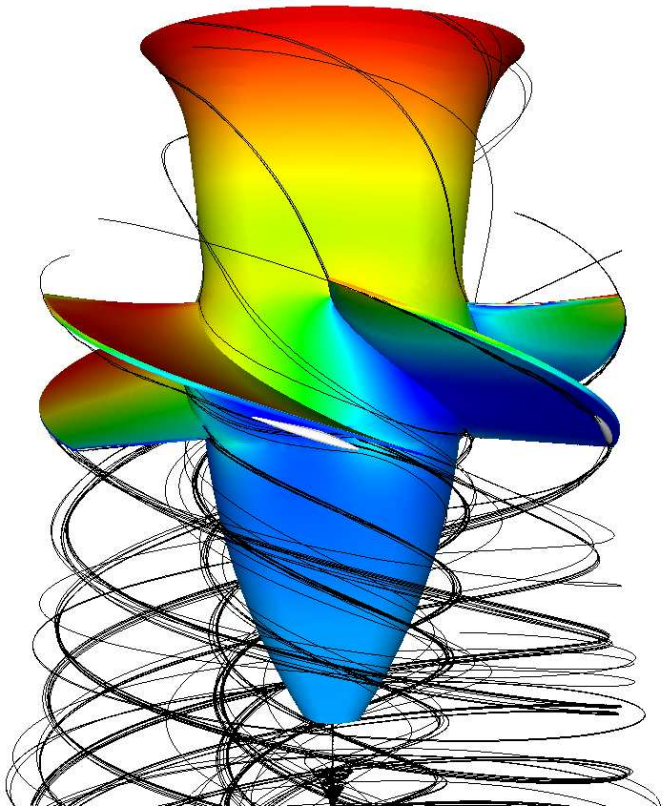
OpenFOAM extensions at SourceForge:
http://sourceforge.net/projects/openfoam-extend/

OpenFOAM ph.D. course at Chalmers, Q2 2007:
http://www.tfd.chalmers.se/ hani/kurser/OF_phD_2007/
(send me an e-mail if you are interested: hani@chalmers.se)

OpenFOAM seminar at Chalmers/C3SE, 070515:
http://www.tfd.chalmers.se/ hani/pdf_files/OF-seminar_070515.pdf

OpenFOAM for water turbine applications, and a validation





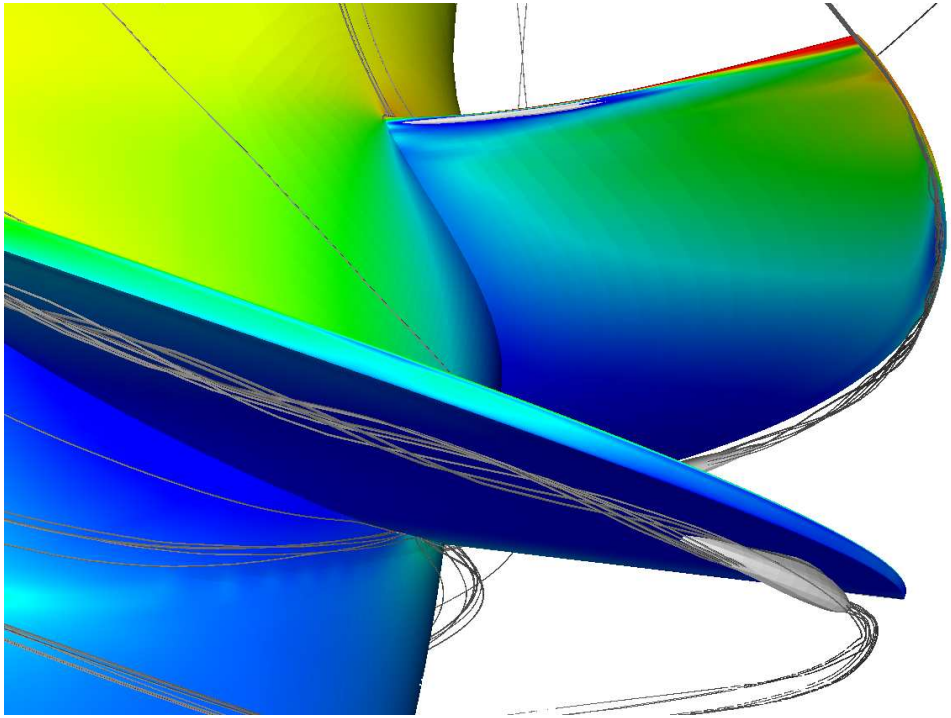
Financed by SVC (www.svc.nu):
Swedish Energy Agency, ELFORSK, Svenska Kraftnät, [a]
Chalmers, LTU, KTH, UU

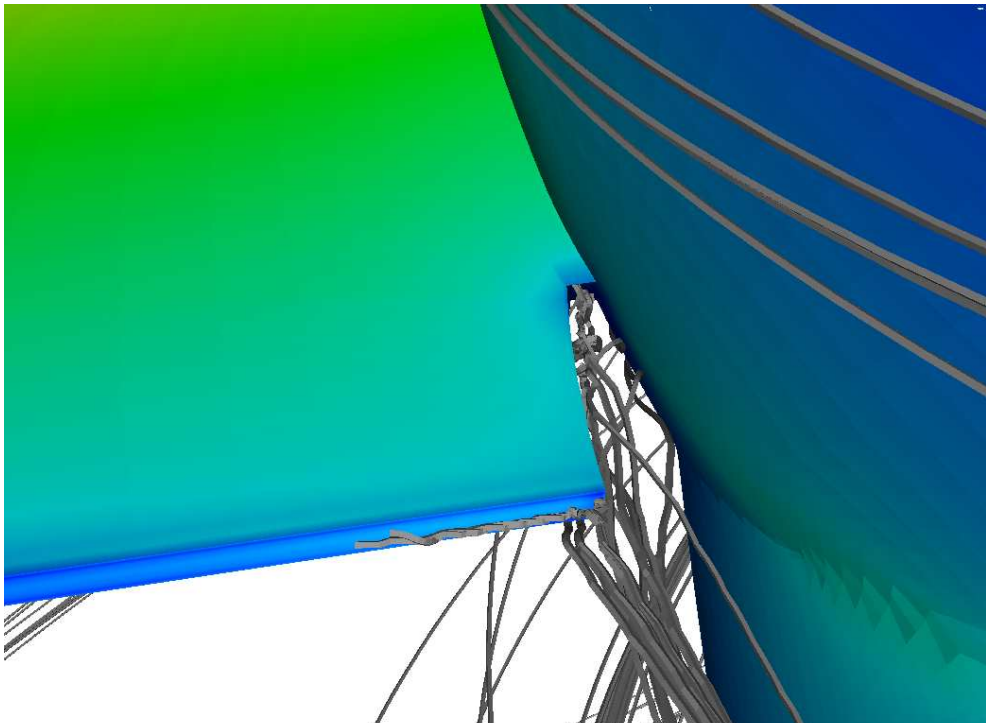
[a]Companies involved: CarlBro, E.ON Vattenkraft Sverige, Fortum Generation, Jämtkraft, Jönköping Energi, Mälarenergi, Skellefteå Kraft, Sollefteåforsens, Statoil Lubricants, Sweco VBB, Sweco Energuide, SweMin, Tekniska Verken i Linköping, Vattenfall Research and Development, Vattenfall Vattenkraft, Waplans, VG Power and Öresundskraft

Håkan Nilsson, Chalmers / Applied Mechanics / Fluid Dynamics

Runner tip and hub clearances

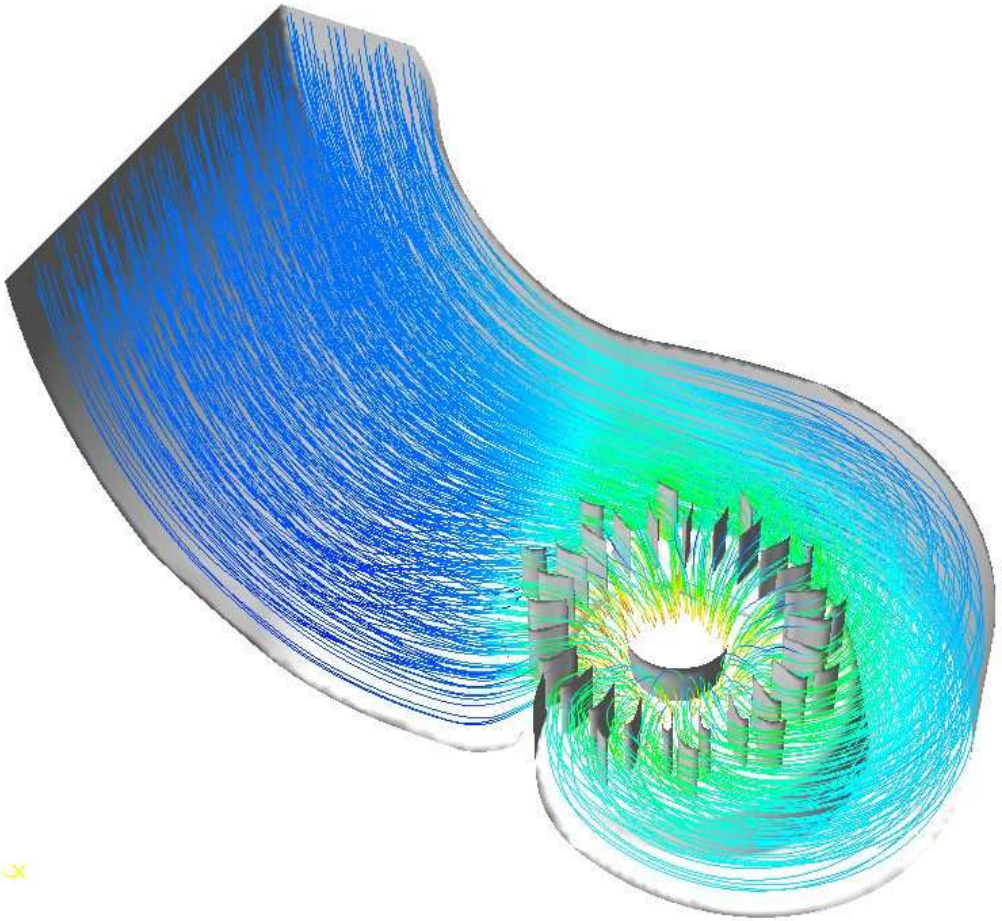
(small geometrical scales)

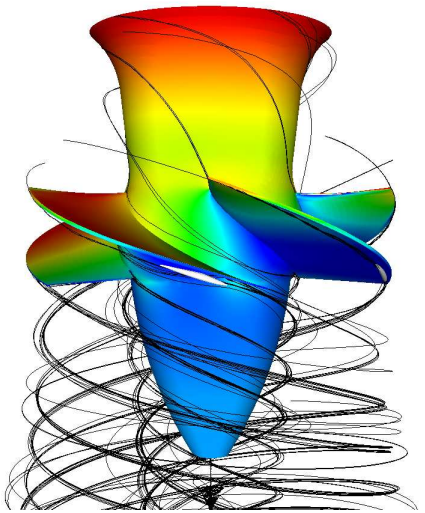


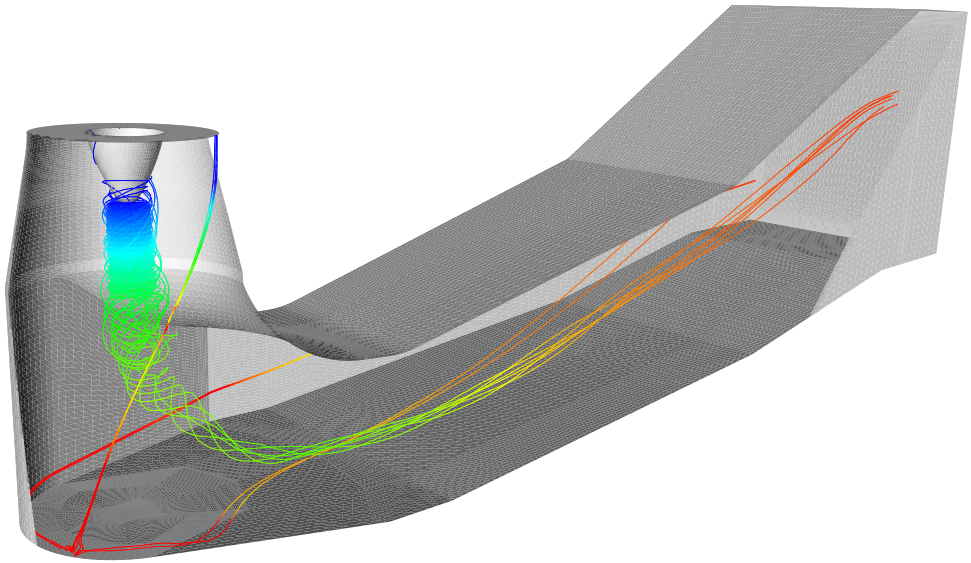


Coupling between flow in different parts of the system

(large geometrical scales)







Håkan Nilsson, Chalmers / Applied Mechanics / Fluid Dynamics

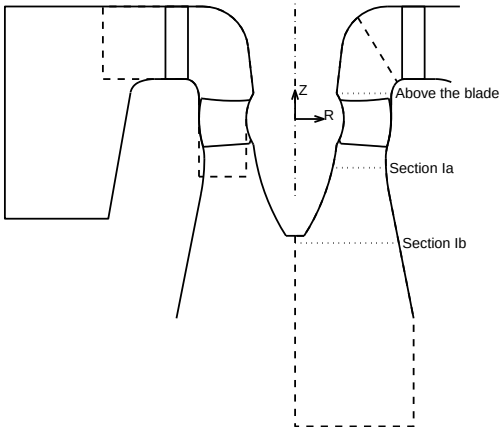
Special needs for water turbine applications

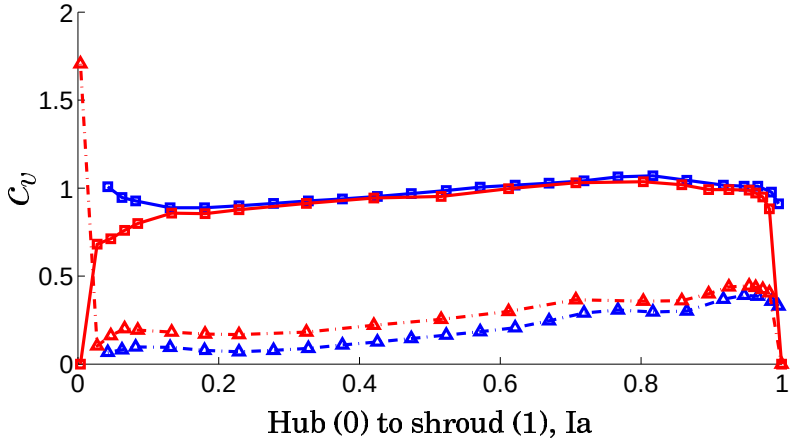
- Rotating coordinate systems / multiple frames of reference
- Sliding grid / mixing plane / GGI
- Rotational periodic boundaries (conformal / non-conformal)
- Cavitation models / two phase methods
- Fine resolution in large domains (parallel computations)
- Moving mesh (rotor dynamics)
- Unsteady SIMPLE solvers (skew grids)
- ...

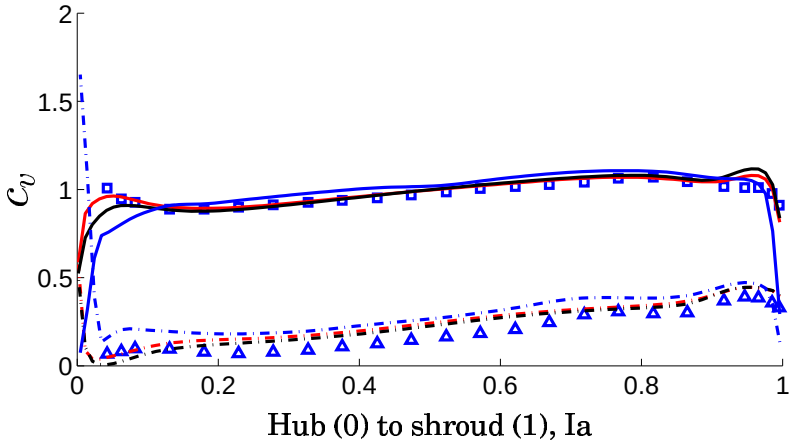
Continous development of these features:

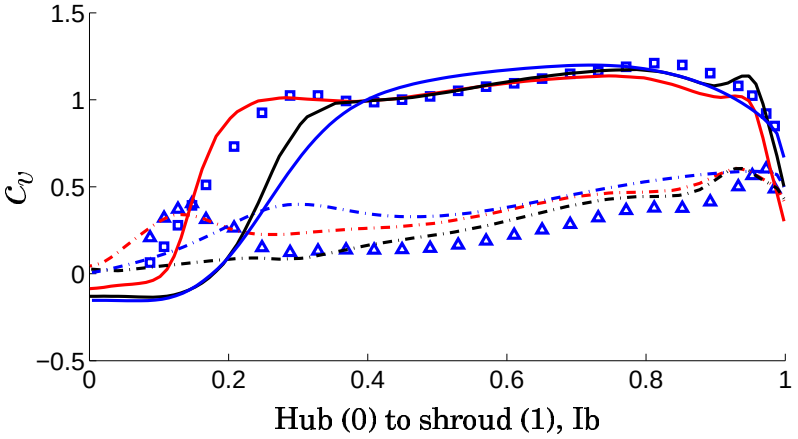
- The OpenFOAM Turbomachinery Working Group (Chalmers, Hydro Quebec, VA TECH ...)
- OpenFOAM Extentions at sourceForge

Validation of OpenFOAM in the Hölleforsen runner (velocity profiles at cross-sections Ia and Ib)



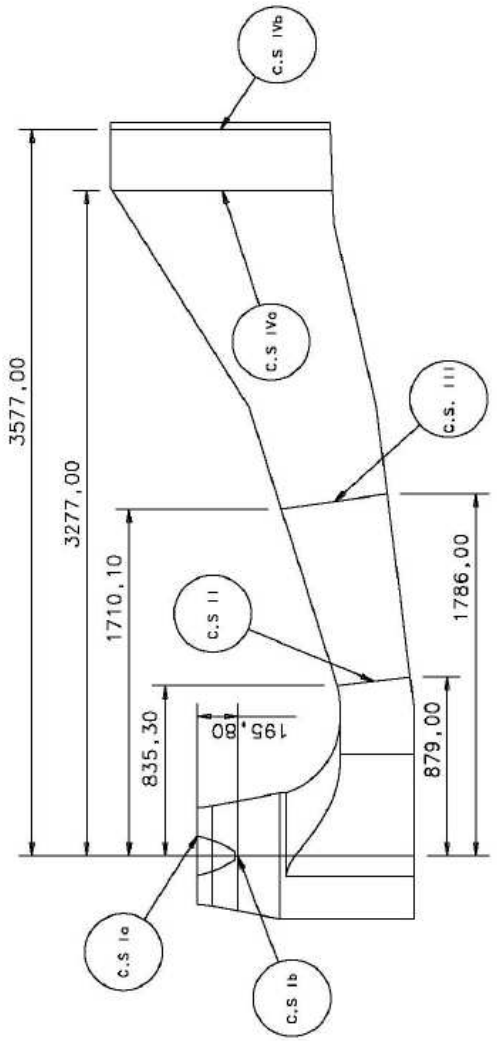




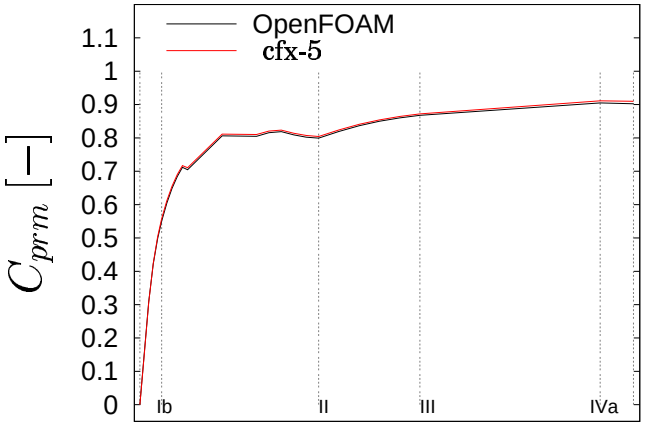


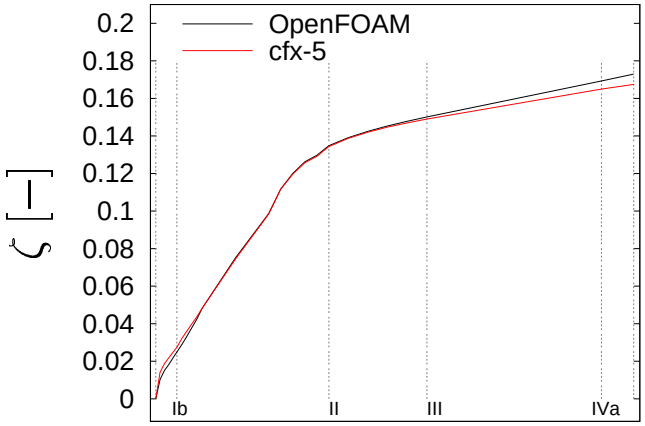
Squares: measured axial velocity. Triangles: measured tangential velocity. In (a) the colors correspond to two different measurements. In (b) and (c): Blue curve: quasi-steady draft tube, Black curve: runner without hub clearance, Red curve: runner with hub clearance.

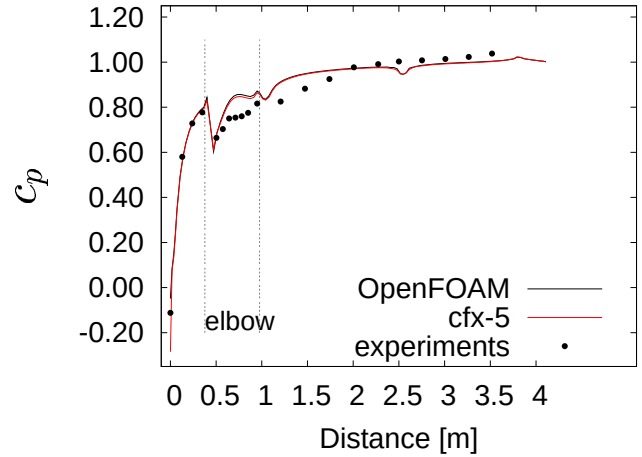
Validation of OpenFOAM in the Hölleforsen draft tube (development of engineering quantities in the flow direction)

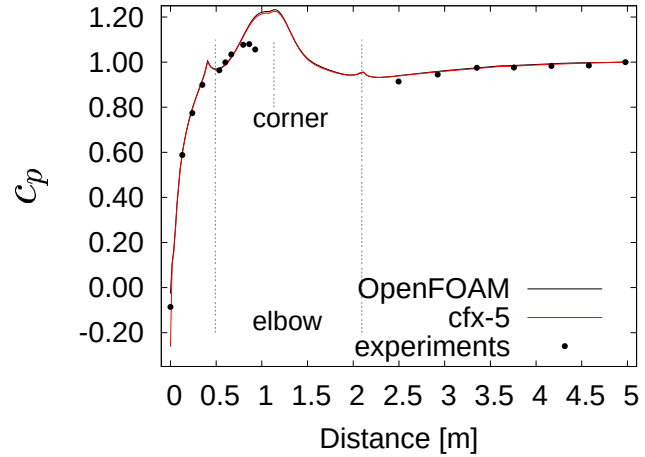


Quasi-steady draft tube computation



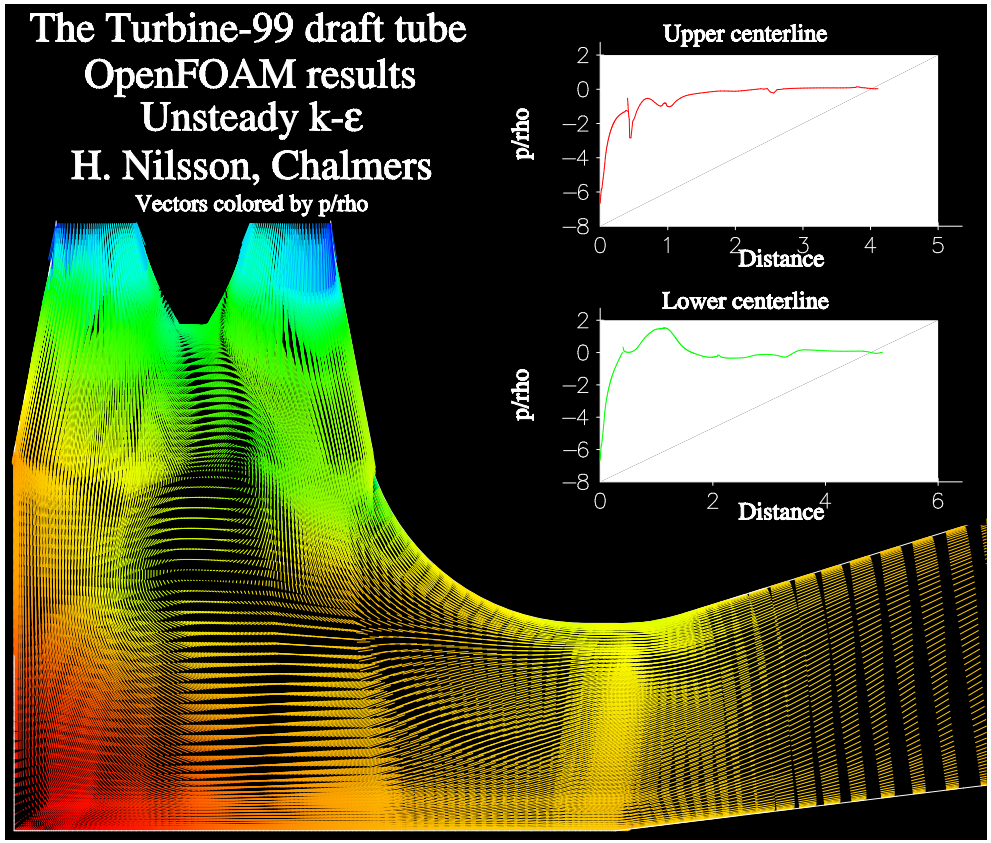


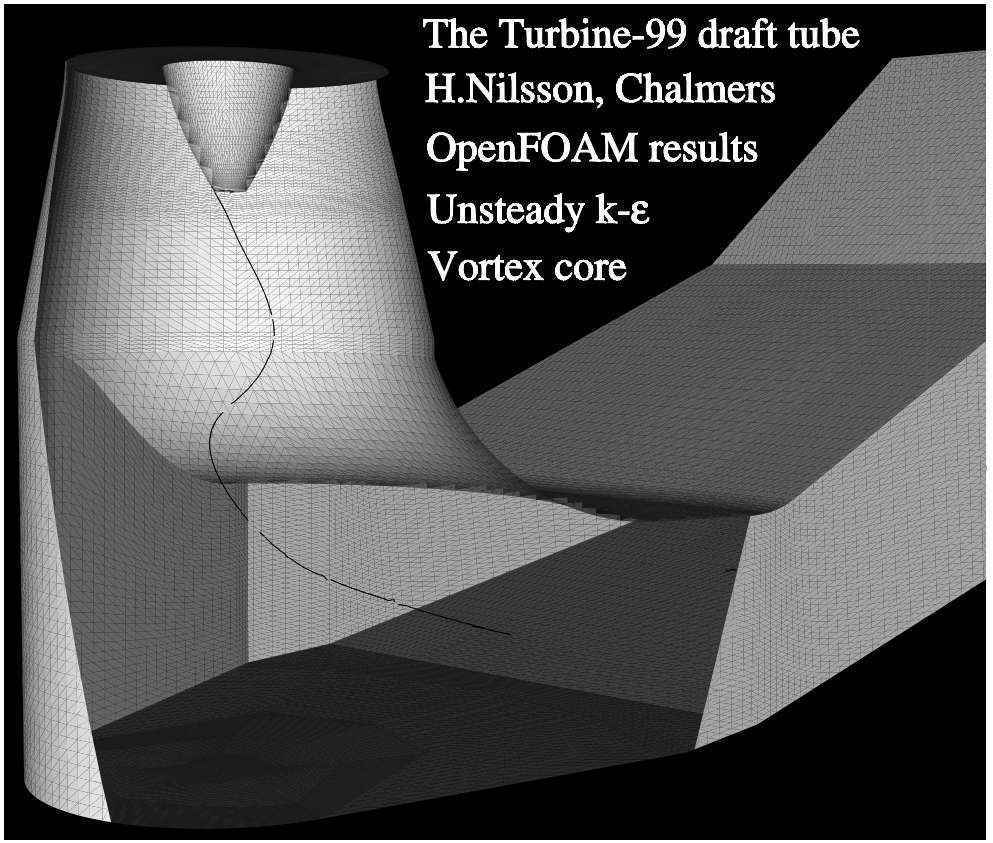




Vortex rope of the unsteady draft tube computation

Periodicity in time: $\sim 0.48s$ (same as CFX-5)

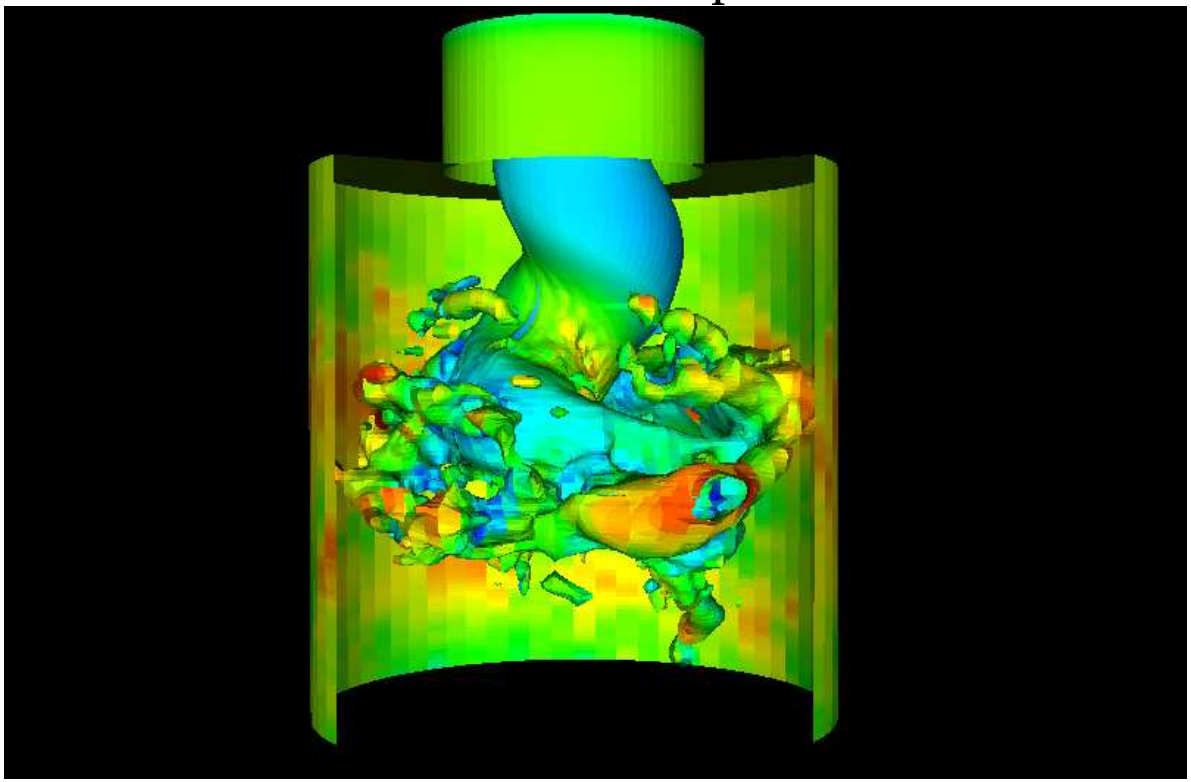




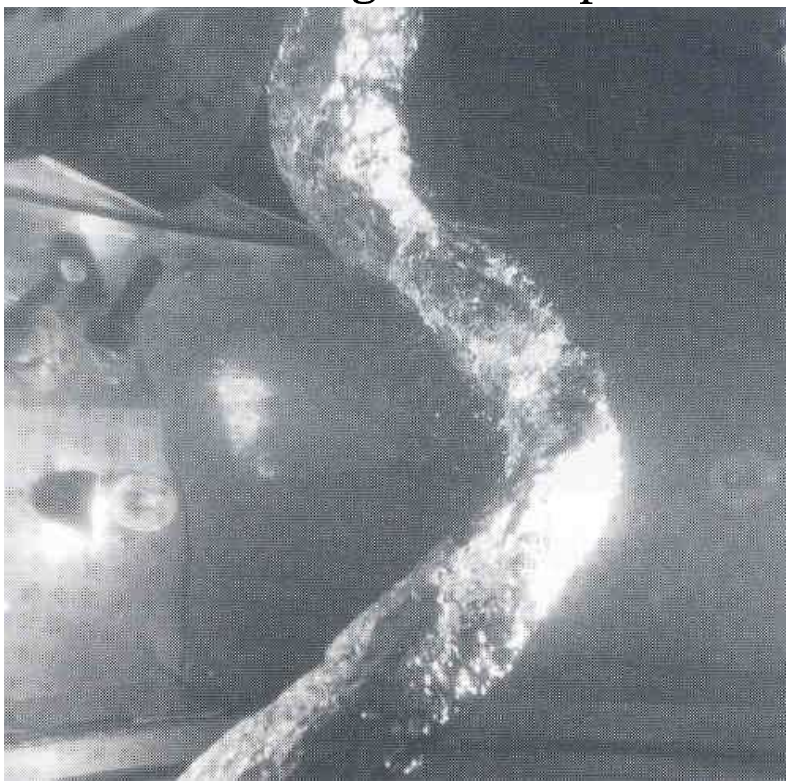
People and projects within SVC/CFD

Walter Gyllenram, Chalmers, supervised by Håkan Nilsson.
Unsteady flow in draft tubes (In-house code CALC-PMB, Matlab and Fluent)

Snapshot of single-phase LES computation.
Iso-surface of static pressure.

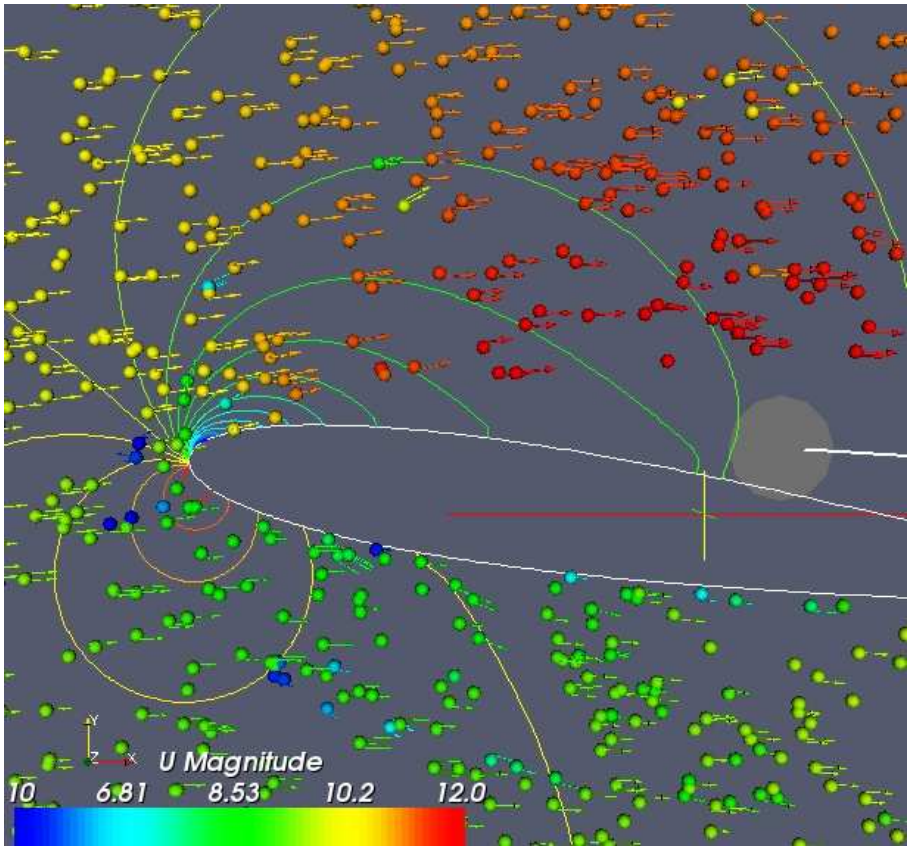


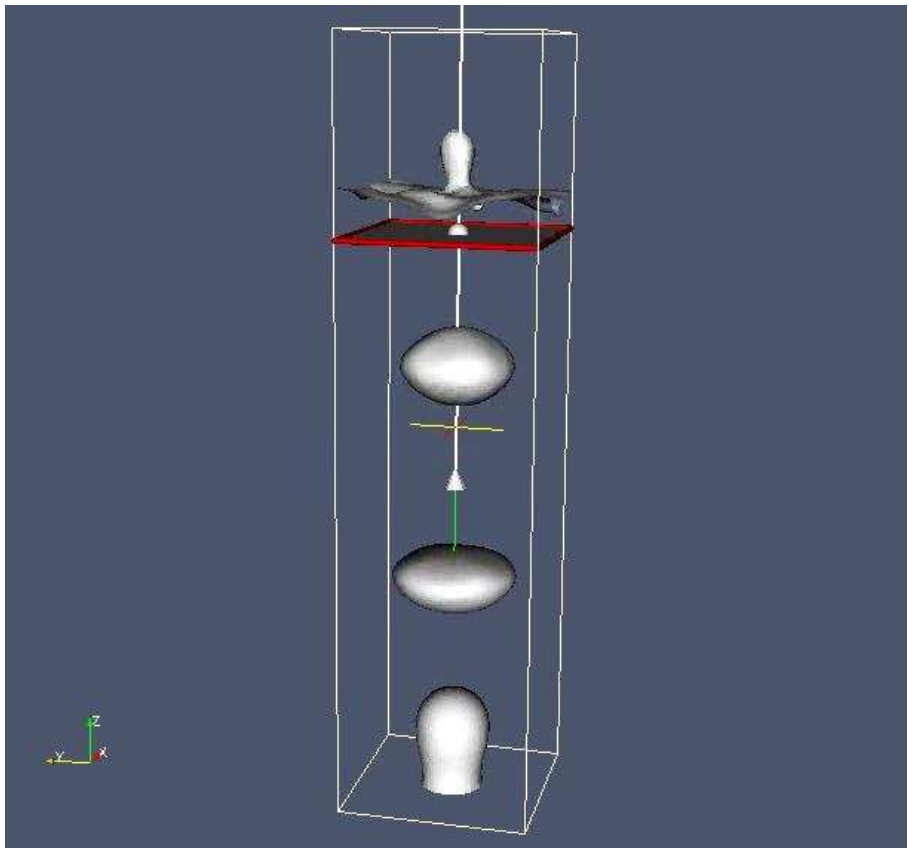
Cavitating vortex rope.



People and projects within SVC/CFD

Aurelia Cure (LTH, Co-supervised by Håkan Nilsson, Chalmers)
Henrik Lindsjö (WAPLANS/LTH, Co-supervised by Håkan Nilsson, Chalmers)
Projects: Cavitation in water turbines
Lagrangian particle tracking at the NACA0015 profile
Rising bubbles using the VOF method



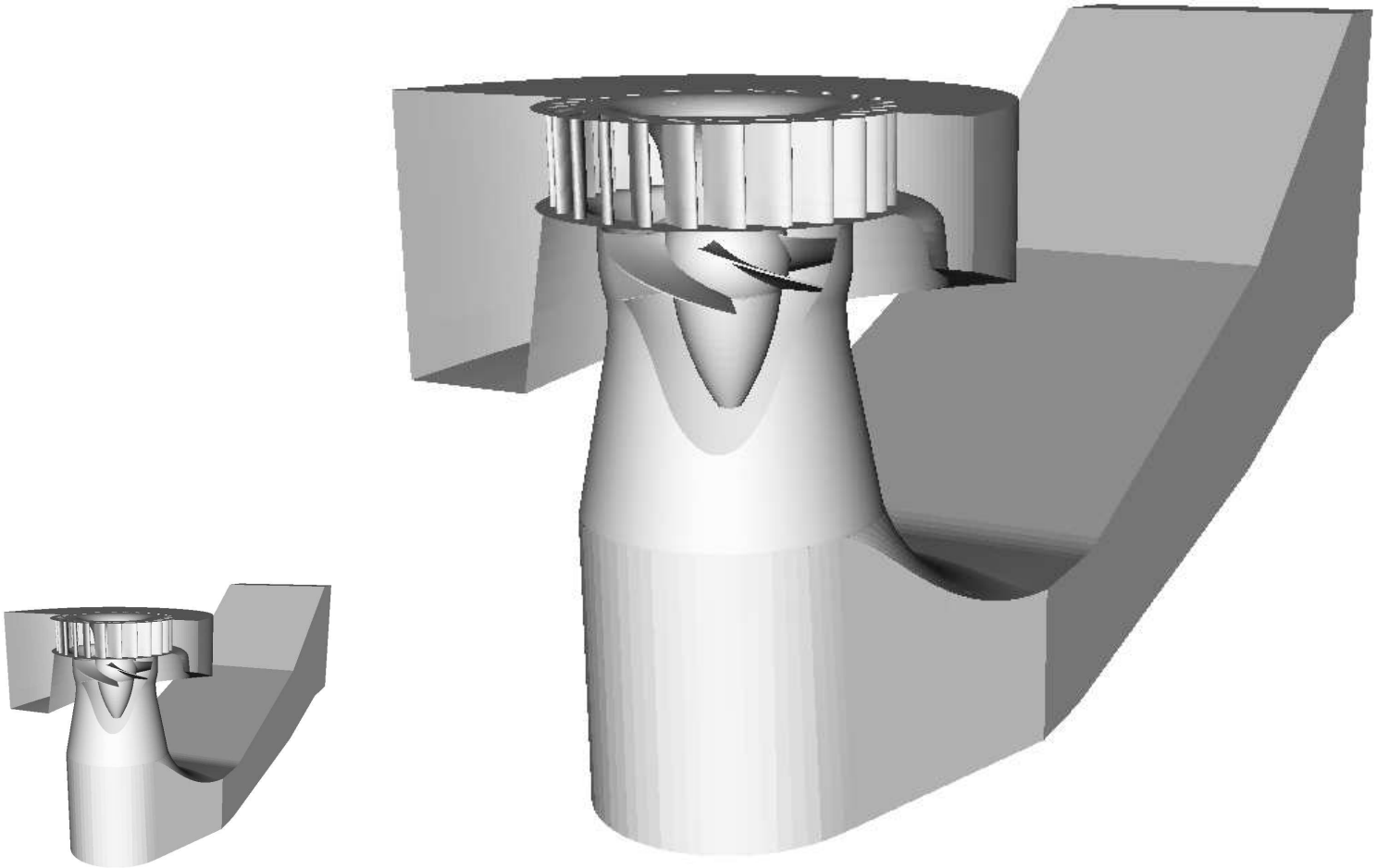


People and projects within SVC/CFD

Samir Saaidi, VRD, supervised by Håkan Nilsson, Chalmers
Planned project: Cooling of water power generators

People and projects within SVC/CFD

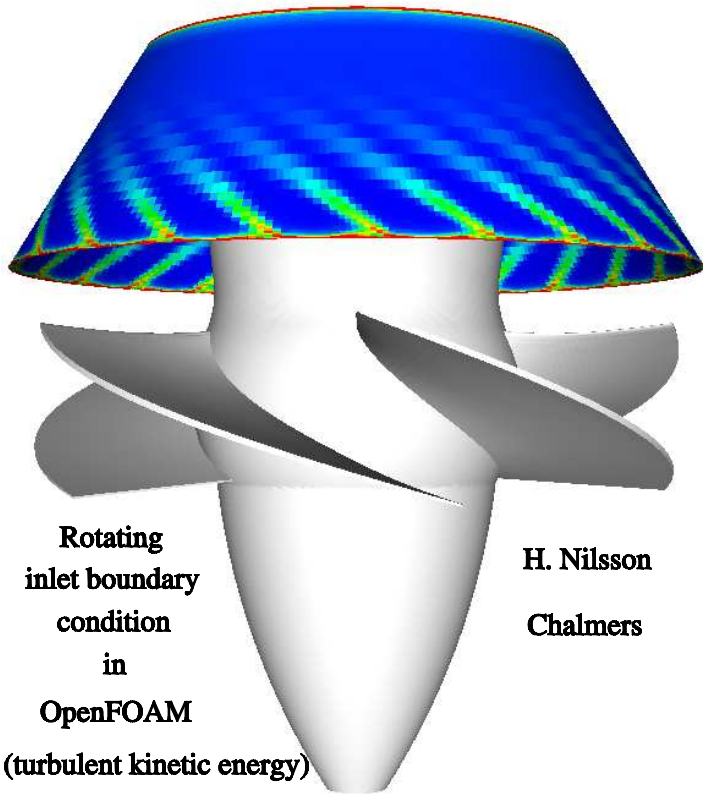
CFD of the flow in the U9 model and prototype using OpenFOAM
Looking for ph.D. student candidates

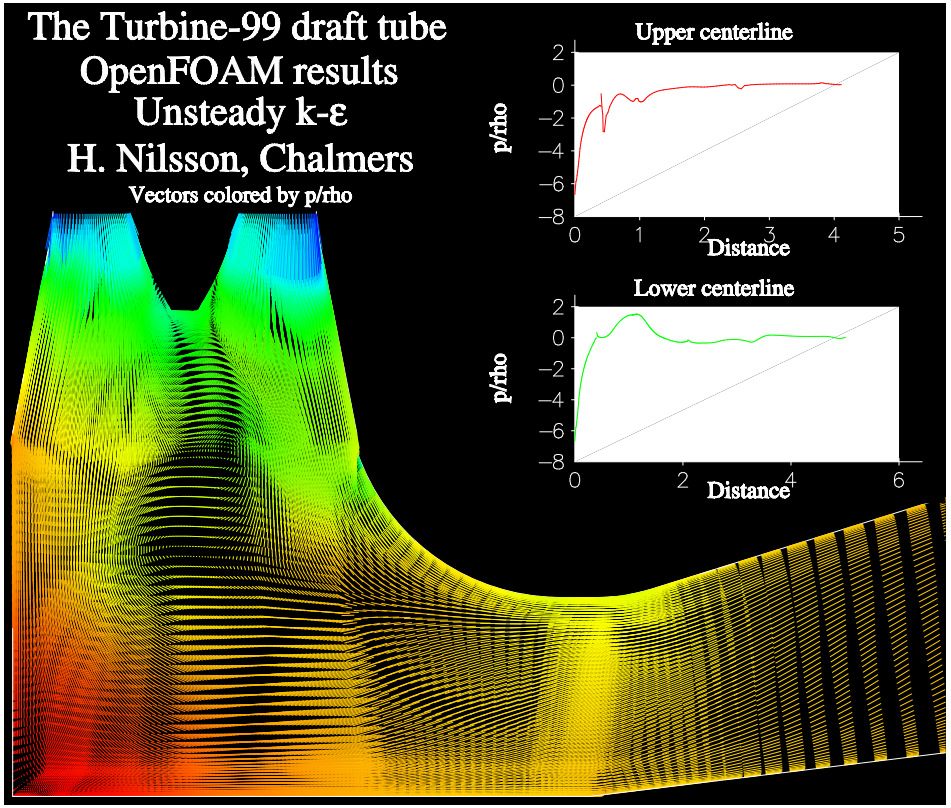


Håkan Nilsson, Chalmers / Applied Mechanics / Fluid Dynamics

People and projects within SVC/CFD

Martin Karlsson, LTU, Collaborating with Håkan Nilsson, Chalmers
Rotor dynamics -
non-axisymmetric inlet b.c., varying rotational velocity and moving runner (deforming mesh)





Thank you for your attention!

Links:
http://www.openfoam.org
http://www.openfoamworkshop.org
http://sourceforge.net/projects/openfoam-extend/
http://www.tfd.chalmers.se/ hani/kurser/OF_phD_2007/
http://www.tfd.chalmers.se/ hani/pdf_files/OF-seminar_070515.pdf

Acknowledgements
The OpenFOAM developers (www.openfoam.org)
Dr. Hrvoje Jasak, Wikki Ltd.

Håkan Nilsson is partly financed by SVC (www.svc.nu):
Swedish Energy Agency, ELFORSK, Svenska Kraftnät, [1]
Chalmers, LTU, KTH, UU

[1]Companies involved: CarlBro, E.ON Vattenkraft Sverige, Fortum Generation, Jämtkraft, Jönköping Energi, Mälarenergi, Skellefteå Kraft, Sollefteåforsens, Statoil Lubricants, Sweco VBB, Sweco Energuide, SweMin, Tekniska Verken i Linköping, Vattenfall Research and Development, Vattenfall Vattenkraft, Waplans, VG Power and Öresundskraft