5th OpenFOAM Workshop, Chalmers University of Technology, Gothenburg, Sweden, June 21-24, 2010

CHALMERS

Four Studies with Moving Meshes in OpenFOAM

H. Nilsson, S. Xie, O. Bergman, A.O. Gonzalez, M. Vanama, M. Jönsson, O. Petit, L. Davidson,
M. Svensson, K. Holmqvist (Chalmers University of Technology), E. Lindén, J. Spång (Husqvarna AB)

CHALMERS

Modelling of Whiplash Trauma -Parametric study of rear-end impacts using FEM and CFD

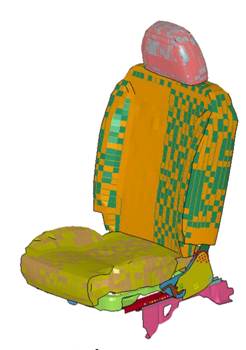
SAFER

Andreu Oliver Gonzalez and Mourya Vanama

This work is a combination of Finite Element (FE) modelling in LS-DYNA and Computational Fluid Dynamics (CFD) in OpenFOAM. It investigates the effect of acceleration pulses, head restraint and seating posture on facet joint loads and spinal canal pressure transient magnitudes ,on the neck injury outcome during whiplash motion.

A parametric study was carried out with the FE human body model THUMS (Total HUman Model for Safety) for a variety of crash pulses (2.5g, 5g and 7.5g) and crash conditions that have a known relative risk of long term neck injuries in rear-end impacts. The injury criterion NIC (Neck Injury Criterion) was calculated.

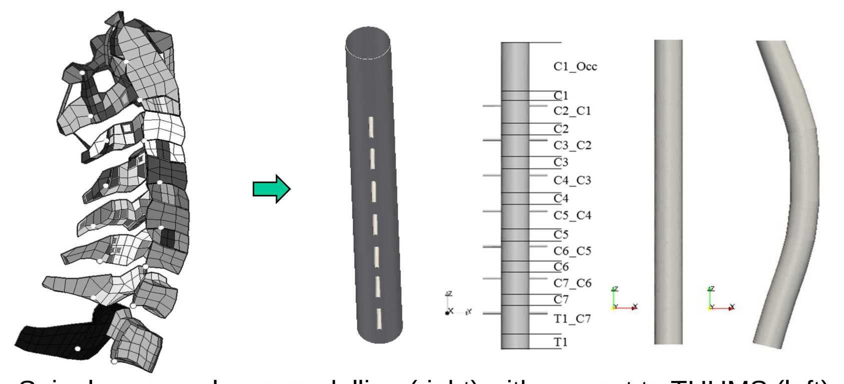
THUMS



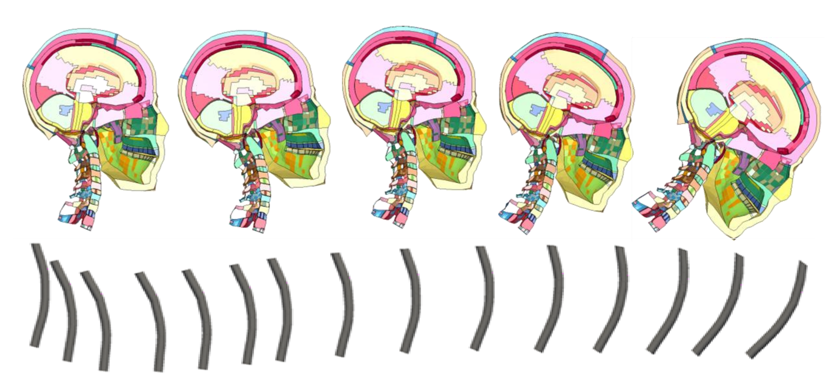
Driver seat

Rear-end impact simulation at 5g acceleration pulse

The THUMS model was used to generate the motion data of the spinal canal. This information was used as an input to simulate the pressure transients in the network of blood vessels (spinal venous plexus) during the whiplash motion by means of CFD simulations using the OpenFOAM CFD toolbox. Using the output of the FE simulations, a moving mesh technique was applied to prescribe the motion of the mesh points in the CFD simulations.

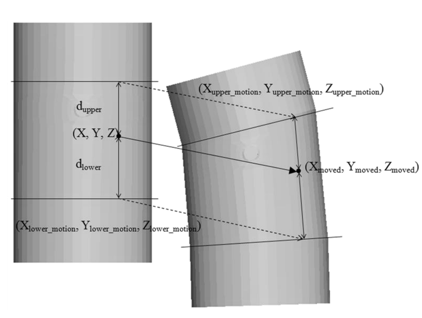


Spinal venous plexus modelling (right) with respect to THUMS (left)



Whiplash motion simulation in FE (above) and CFD (below)

Motion interpolation



The results give new input to proposed injury mechanisms and injury risk assessment criteria concerning long term neck injuries. The effect of the head restraint, and its position with respect to the occupant's head, in mitigating the above injuries is discussed for different crash scenarios.

The full report can be downloaded from Chalmers Publication Library:
http://publications.lib.chalmers.se/cpl/record/index.xsql?pubid=122637

CHALMERS

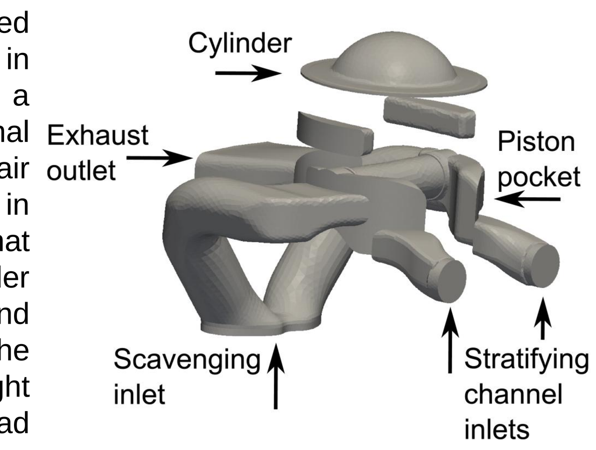
Stratified scavenging computations in two-stroke engines using OpenFOAM

Husqvarna

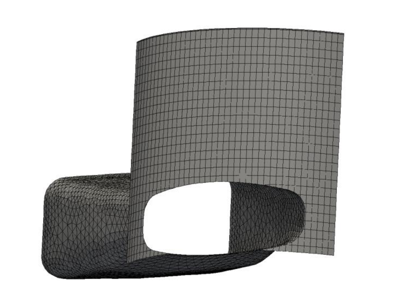
Mikael Jönsson

This work presents the methodology and the results of CFD-simulations in a stratified charged two-stroke engine. The aim is to evaluate the performance of the engine in terms of scavenging and trapping efficiency. One of the main purposes is to evaluate the use of OpenFOAM, and in particular OpenFOAM-1.5-dev with the twoStrokeEngine-library, for this kind of applications.

The stratified charged engine is developed to reduce the amount of unburned fuel in the exhaust gases. This is achieved by a pocket in the piston and additional channels, stratifying channels, for fresh air which will suppress the air fuel mixture in the scavenging channel. The result is that the fresh charge that first enters the cylinder will have a low concentration of fuel and thus reduce the amount of fuel leaving the cylinder unburned. The figure to the right shows the engine geometry at top dead center.



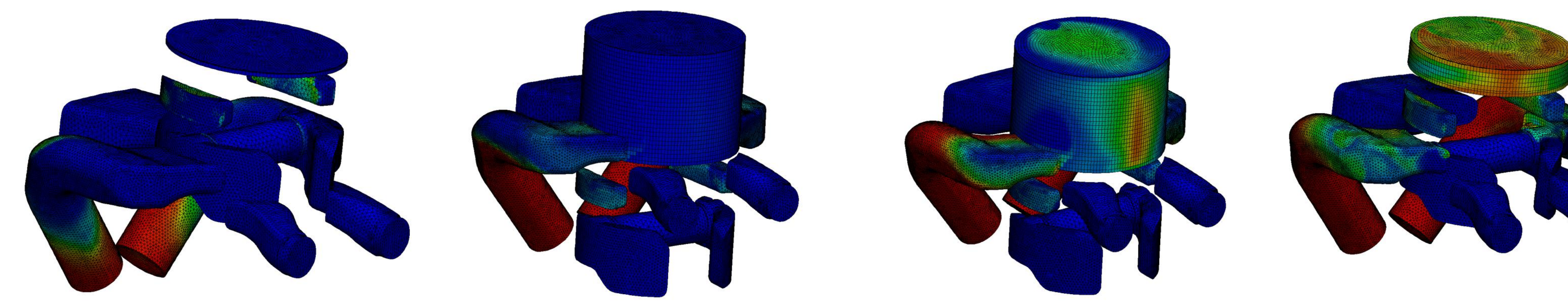
The case involves mesh operations such as cell layer addition and removal and sliding interfaces. An example of a sliding interface between the cylinder and the exhaust port is shown in the figure below.



To visualize the flow, a passive scalar is added at the scavenging inlet. Below, the first figure to the left visualizes the fuel (red) as the piston is at top dead center. The pocket in the piston is creating a pass between the stratifying channel and the scavenging channel. As can be seen, the fresh air vill suppress the fuel in the scavenging channel.

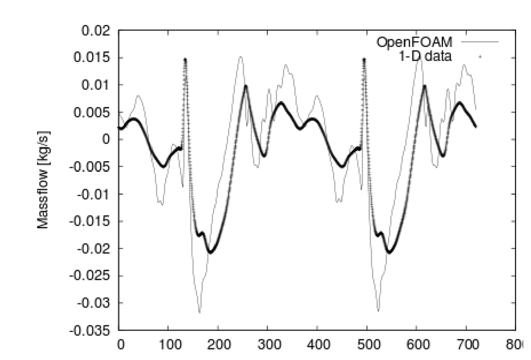
In the second figure the piston has reached the bottom dead center, where the exhaust port and scavenging ports are now open. A small amount of fuel has just started to enter the cylinder. 35 crank angle degrees after bottom dead center, the third figure shows that the fuel has started to flow into the cylinder.

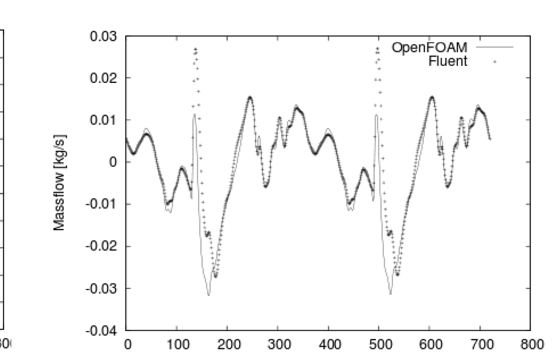
The fourth figure shows the fuel trapped in the cylinder, and the scavenging ports and the exhaust port are now completely closed. It can be seen that the fresh air from the stratifying channel once again has begun to suppress the fuel in the scavenging channel. This is possible since the piston pocket now has provided a pass for the fresh air.



A comparison with 1D simulation results, of the mass flow at the scavenging inlet, is shown in the left plot. The overall shape is similar while OpenFOAM gives a better representation of the dynamics in the system.

The right plot shows a similar comparison with 3D results of the Fluent commercial CFD code. This shows a good correlation in both the shape and magnitudes of the curves.





The full report can be downloaded from Chalmers Publication Library:
http://publications.lib.chalmers.se/cpl/record/index.xsql?pubid=122640

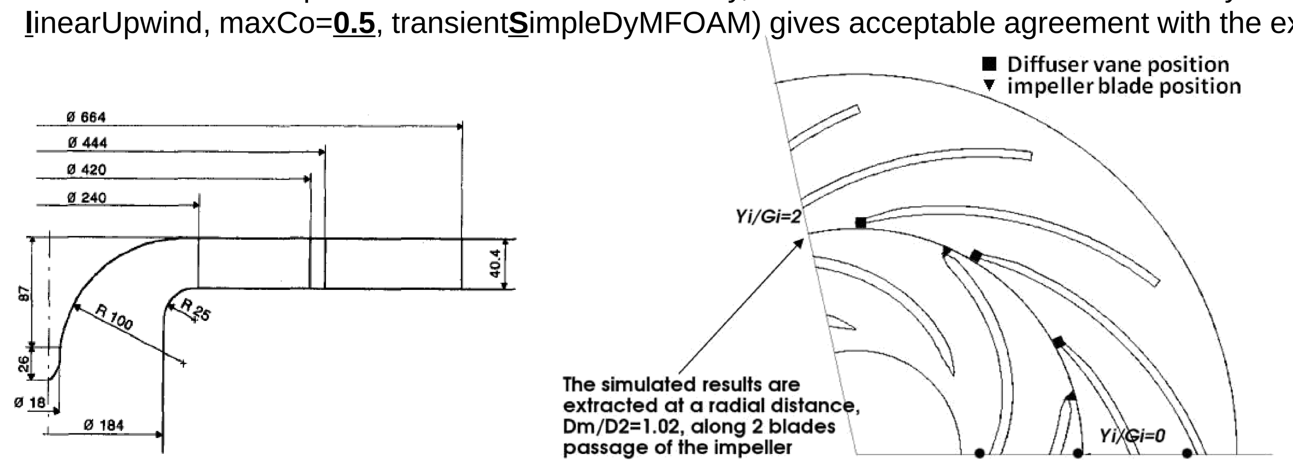
CHALMERS

Studies of the ERCOFTAC Centrifugal Pump with OpenFOAM

SVC

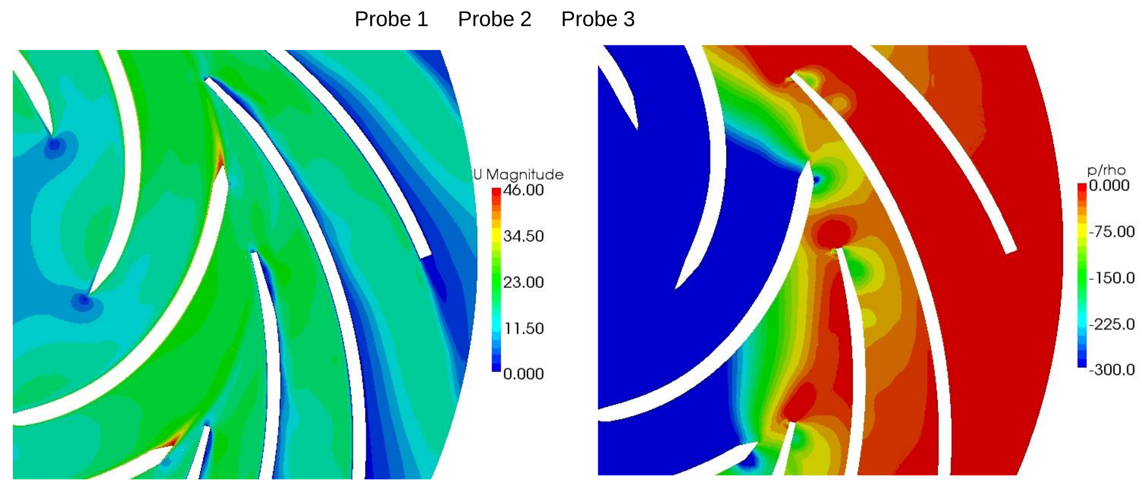
Shasha Xie

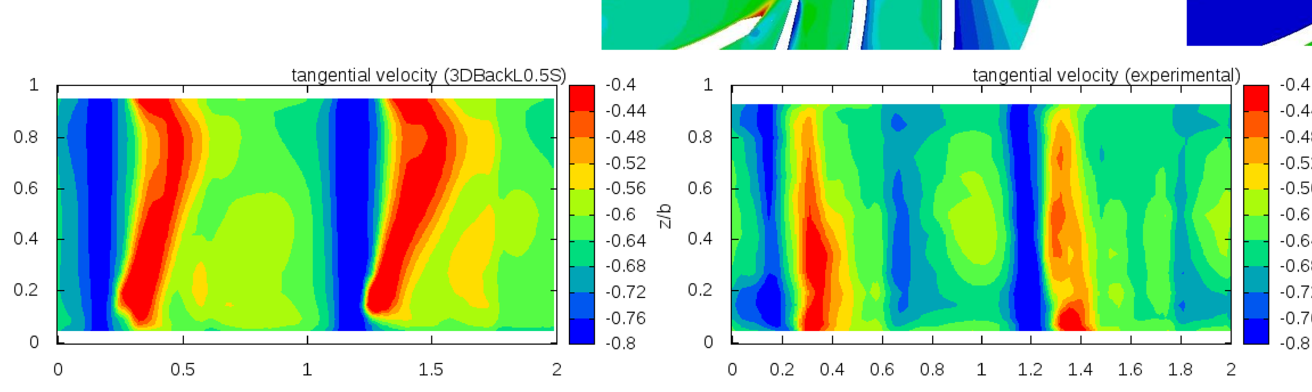
This work investigates the rotor-stator interaction in the ERCOFTAC Centrifugal Pump. 2D and 3D steady-state and unsteady simulations are performed. Several numerical schemes are considered such as Euler, Backward and Crank-Nicholson (with several off-centering coefficients) time discretization, and upwind and linear-upwind convection discretization. Furthermore, the choice of different maximum Courant number and different unsteady solvers have been studied, and the required computational time has been compared for all the cases. Finally, the results from the 3D unsteady case 3DBackL0.5S (**3D**, **back**ward, **l**inearUpwind, maxCo=**0.5**, transient**S**impleDyMFOAM) gives acceptable agreement with the experimental results.



Geometry of the ERCOFTAC Centrifugal Pump and the relative position of the runner and stator in the steady-state simulations. In the unsteady case the rotor mesh is rotating. Detailed comparisons are made along the Yi/Gi = 0-2 arc. The static pressure is probed at Probes 1-3.

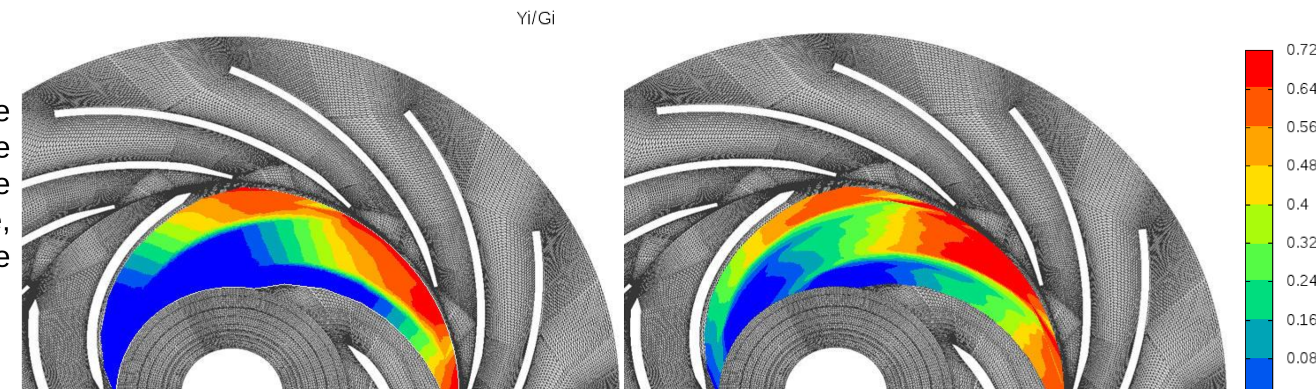
Snap-shot of the relative velocity magnitude (left) and static pressure (right) for the 3D unsteady case 3DBackL0.5S. The advected impeller wakes are visible in the diffusor blade boundary layers.





Snap-shot of the tangential velocitiy for the 3D unsteady 3DBackL0.5S case along Yi/Gi = 0-2, for different spanwise positions, compared to the ensemble averaged experimental data.

Snap-shot of the static pressure coefficient distribution in the impeller blade passage, for the 3DBackL0.5S 3D unsteady case, compared to the ensemble averaged experimental data.



The full report can be downloaded from Chalmers Publication Library:
http://publications.lib.chalmers.se/cpl/record/index.xsql?pubid=122638

CHALMERS

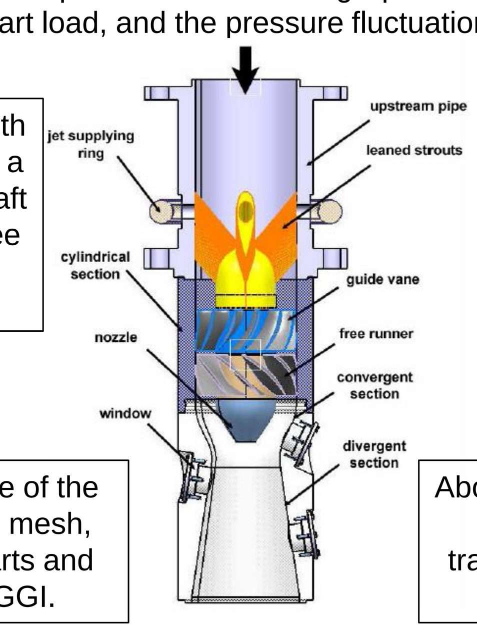
Numerical investigation of the flow in a swirl generator, using OpenFOAM

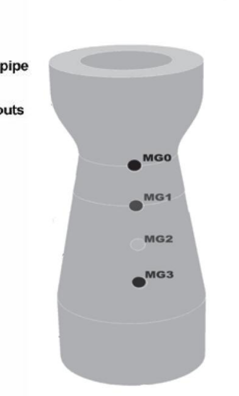
SVC

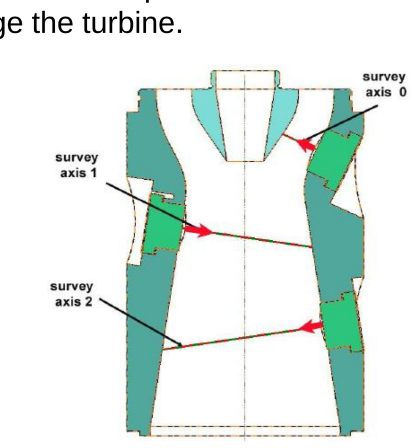
Oscar Bergman

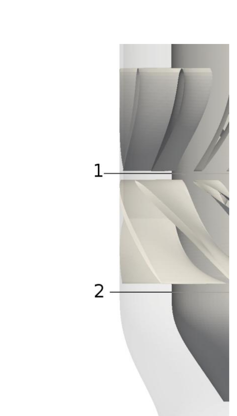
This work presents and validates OpenFOAM results of the flow in a swirl generator. The swirl generator is designed to create a swirling flow with a vortex rope which causes large pressure fluctuations. Vortex ropes are a common problem in Francis water turbines working at part load, and the pressure fluctuations may damage the turbine.

Right: The swirl generator with leaned strouts, guide vanes, a free rotating runner and a draft tube. The draft tube has three windows for LDV measurements.







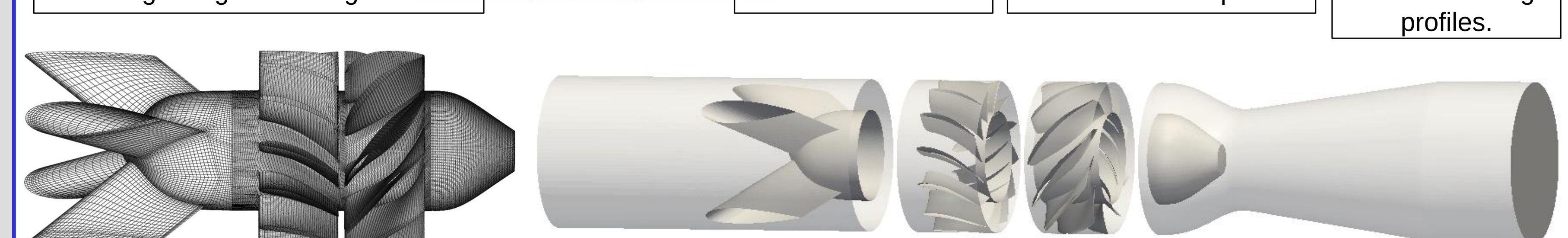


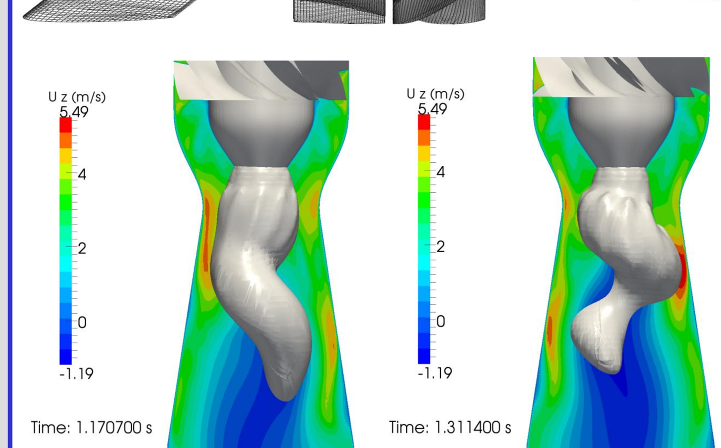
Below, left and right: Outline of the computational domain and mesh, which was made in four parts and merged together using GGI.

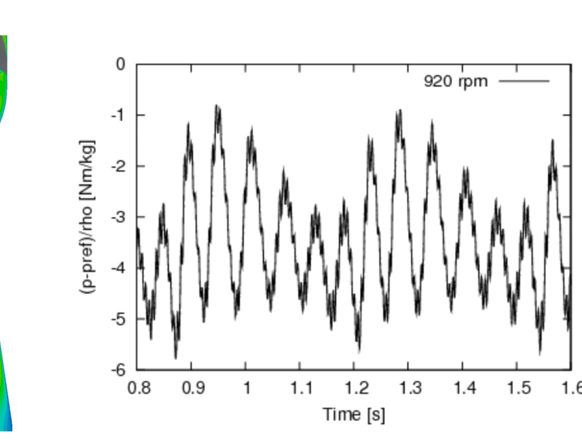
Above: The location of pressure transducers in the draft tube

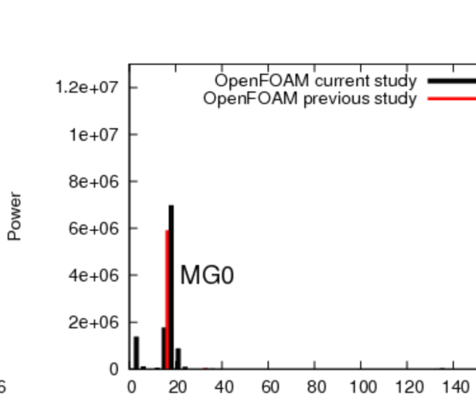
Above: The three LDV survey axes. The arrows indicate the start of the abscissa in the plots.

Above: Two cross-sections for comparisons with theoretical design profiles.







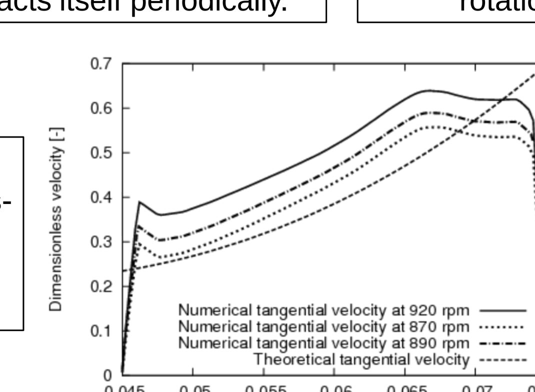


Above: The vortex rope which spins with the runner and extracts/retracts itself periodically.

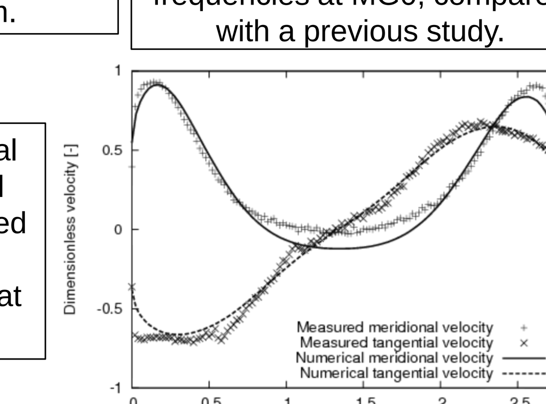
Above: Pressure oscillations at MG0 for a rotational speed of 920 rpm.

Above: Pressure Fourier frequencies at MG0, compared with a previous study.

Right: Tangential velocity profiles at cross-section 2, compared with the theoretical design profile.



Right: Tangential and meridional velocity compared with the LDV measurements at survey axis 2



The full report can be downloaded from Chalmers Publication Library:
http://publications.lib.chalmers.se/cpl/record/index.xsql?pubid=122641