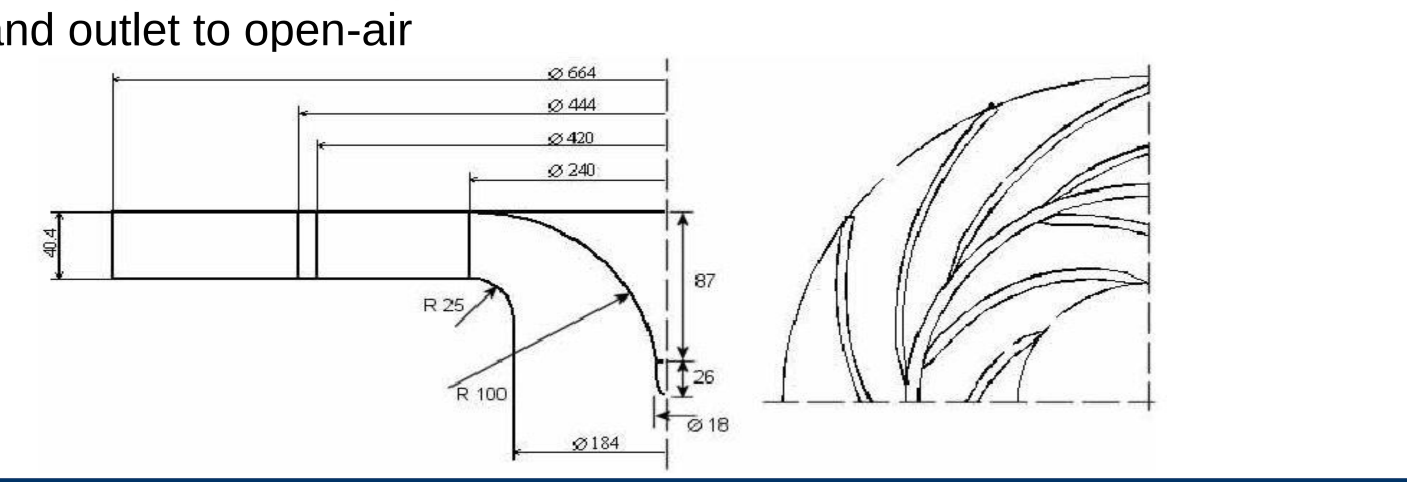
3rd IAHR International Meeting of the WorkGroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Brno, Czech Republic, October 14-16, 2009

The ERCOFTAC Centrifugal Pump OpenFOAM Case-Study

Olivier Petit (1), Maryse Page (2), Håkan Nilsson (1), Martin Beaudoin (2)
(1) Chalmers University of Technology, (2) Hydro-Québec, Institut de recherche

Geometry

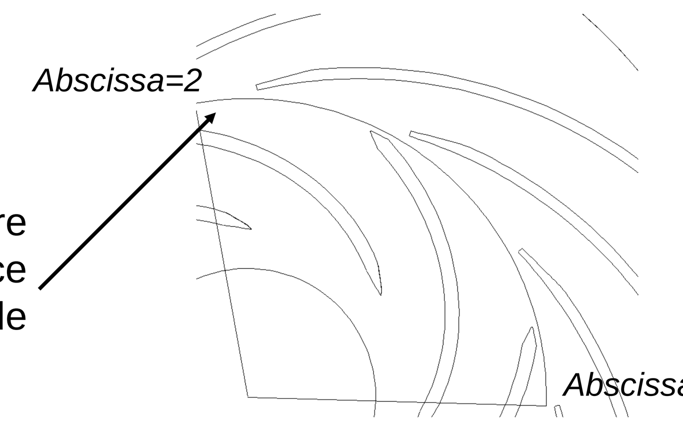
- 7 impeller blades, 12 diffuser vanes
- 6% vaneless gap
- Rotational speed : 2000 rpm
- Inlet and outlet to open-air



Measurements

- Experimental data provided by Ubaldi et al. [1].
- Hot-wire probe measurements at a radial distance 4mm from the trailing edge (Dm/D2=1.02) are used for the validation in the present work.
- More experimental results can be found in the literature.

The computational results are validated at a radial distance Dm/D2=1.02, along two blade passages of the impeller.



Position of the hot-wire probe, and of the sampling of the simulated data

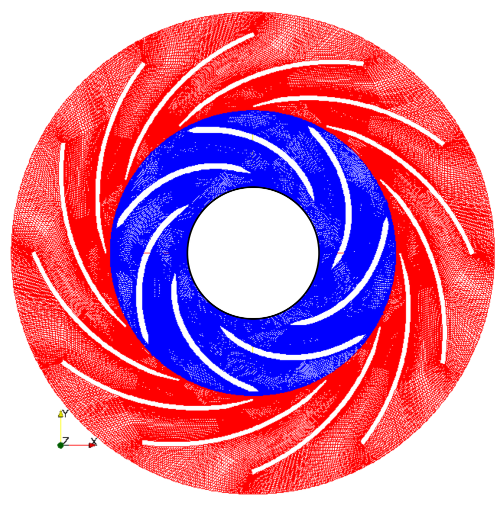
Cases

- The cases are part of the work done by the OpenFOAM Turbomachinery Working Group and can be found at **http://openfoamwiki.net/index.php/Sig_Turbomachinery**
- The computational domain is 2D in the present work
- The rotor-stator interaction is handled by the frozen-rotor, and sliding grid approaches, with two different interface coupling methods.
- The computational parameters are:

Calculated data for the 2D cases		Boundary conditions	
Inlet Diameter	D_0=200 mm	At the inlet	$V_{radial} = U_0$ $\frac{\mu_T}{\mu} = 10$ (viscosity ratio)
Z thickness (OpenFOAM requires one cell thickness in 2D)	Z = 1mm		$k = \frac{3}{2} U_0^2 I^2 = 0.48735 m^2/s^2$ (I=5%)
Flow rate	$Q = \frac{\varphi U_2 \pi D_2^2}{4} = 0.292 m^3/s$		$\varepsilon = \frac{C_\mu \rho k^2}{\mu_T} = \frac{C_\mu \rho k^2}{\mu(\mu_T/\mu)} = \frac{C_\mu k^2}{\nu(\mu_T/\mu)}$
Inlet radial speed	$U_0 = \frac{Q}{A_0} = \frac{Q}{2\pi r_0 z} = 11.4 m/s$	At the outlet	Average static pressure 0

Computational grid

- 2D grid made using ICEM-Hexa
- Block-structured
- 94 000 cells
- Average Y+ value : 35



References

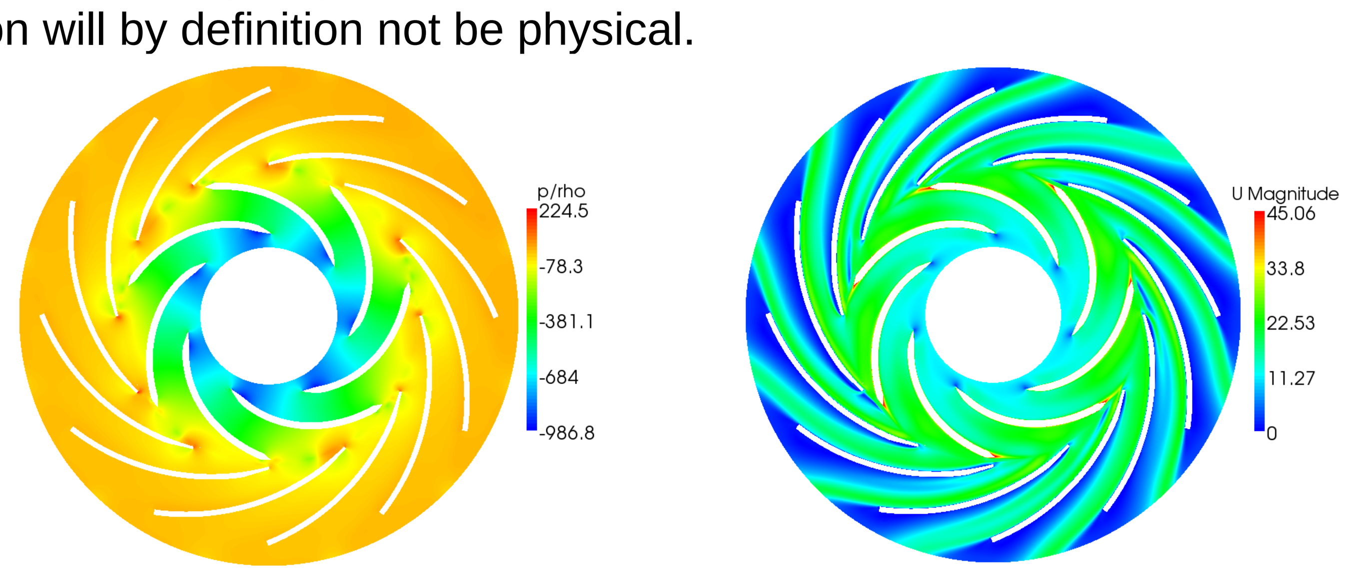
1. Ubaldi M., Zunino P., Barigozzi G. and Cattanei A., ***An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow***, *ASME Journal of Turbomachinery*, vol.118, 41-54, 1996.
2. Beaudoin M. and Jasak H., ***Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM***, *Open Source CFD International Conference 2008*.

Work in progress

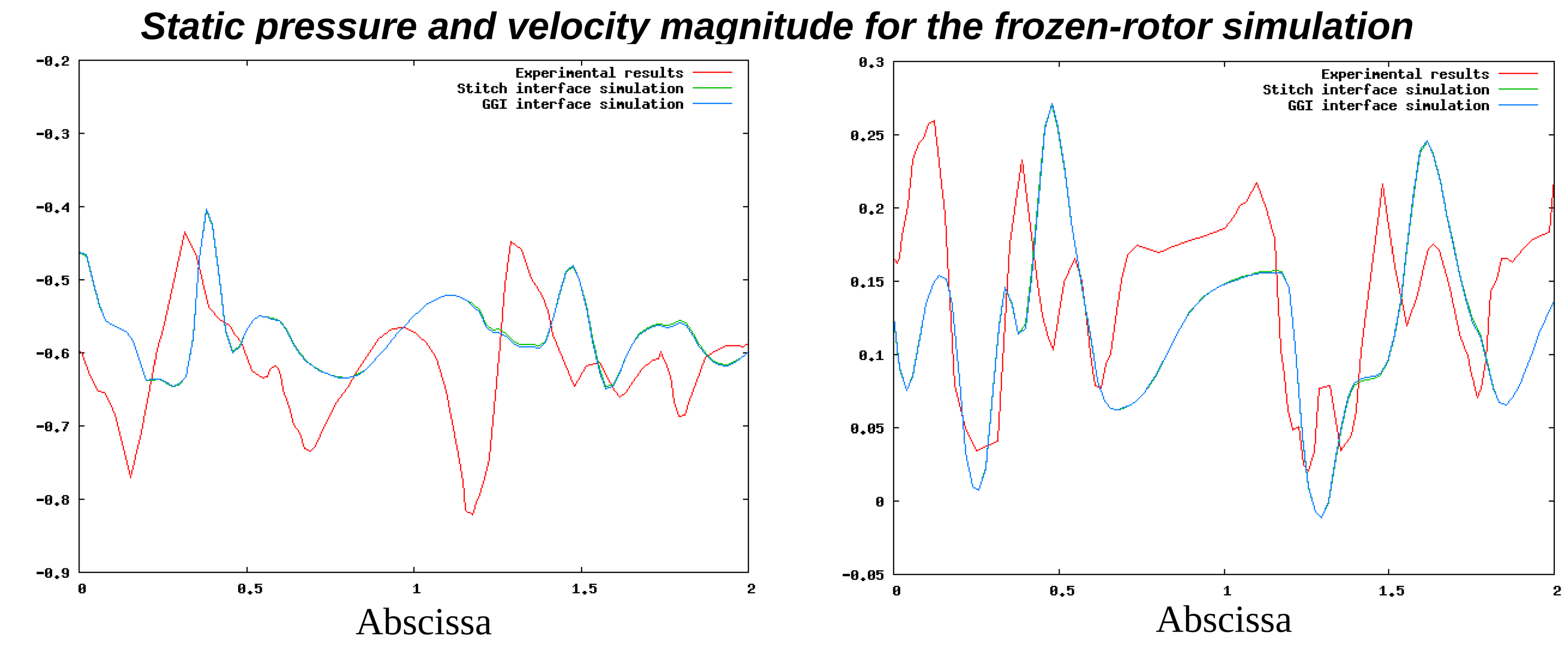
- 3D case-study, steady state and unsteady simulations.
- Tutorial for the coming mixing plane domain coupling interface.

Results – Steady state simulation

The frozen-rotor approach gives something that resembles a snapshot of the real flow in the pump, but the advection of the impeller wakes in the diffuser region will by definition not be physical.



Static pressure and velocity magnitude for the frozen-rotor simulation

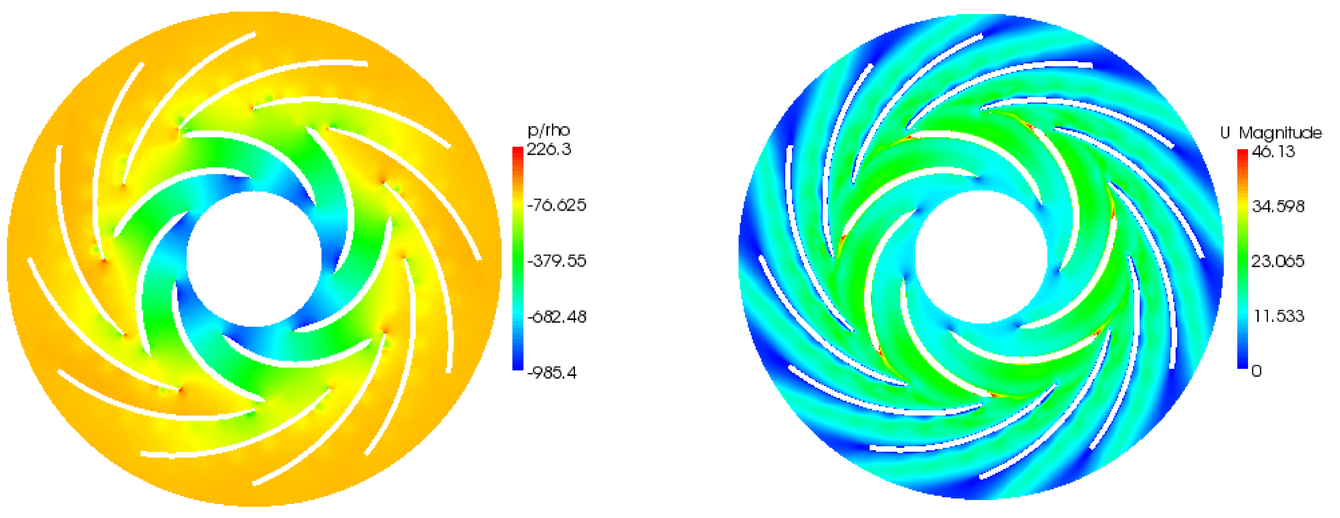


Distributions of the tangential and radial velocities, for the two frozen rotor simulations compared to the experimental data

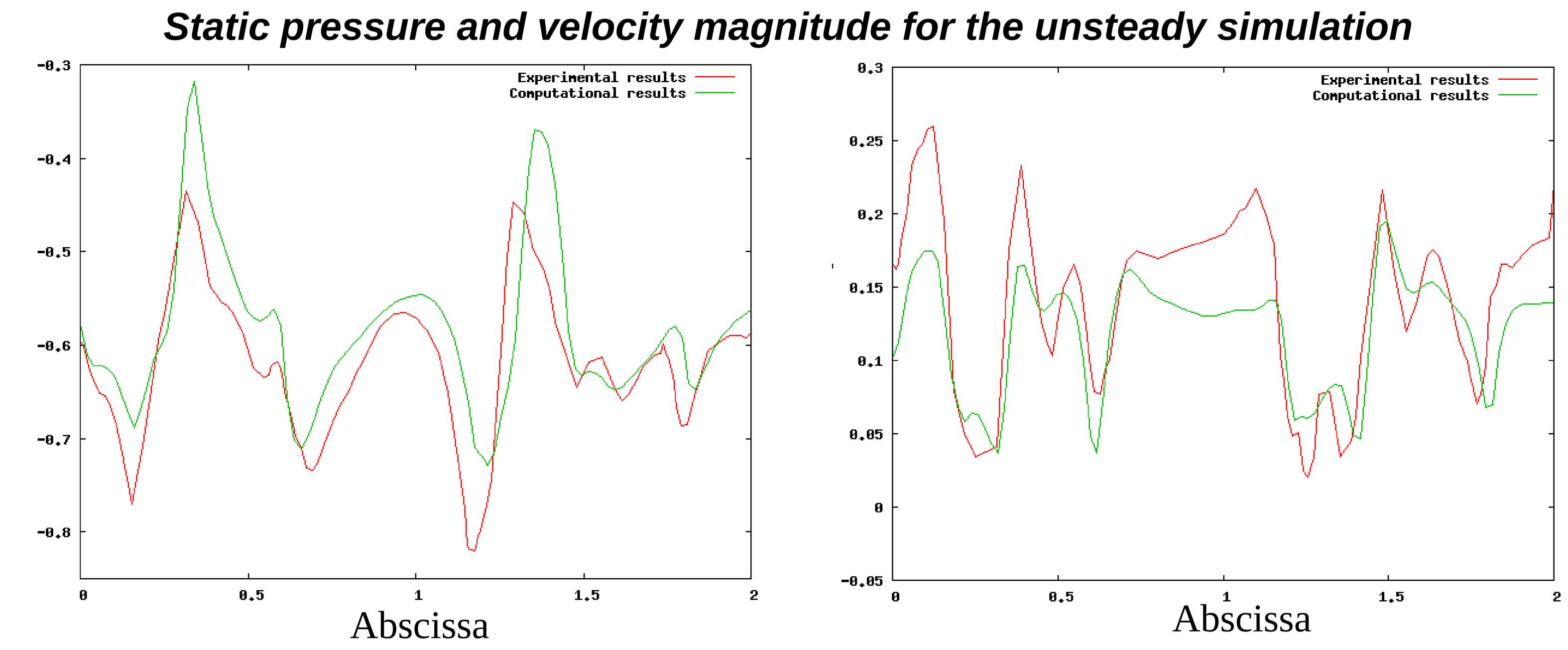
- Most of these differences are likely due to the frozen rotor formulation rather than the OpenFOAM implementation.
- Commercial CFD codes show similar results as OpenFOAM for this case.

Results – Unsteady simulation

With the unsteady simulation (sliding grid), the wakes are more visible, and they are advected properly between the diffuser blades.



Static pressure and velocity magnitude for the unsteady simulation

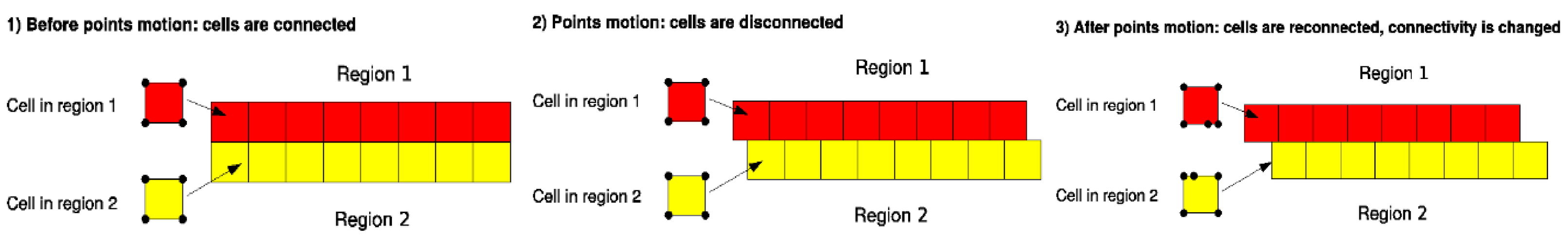


Distributions of tangential and radial velocity for the unsteady simulation, compared to the experimental data

Interface coupling methods

With topological changes :

- Two parts of a mesh with non-conformal faces can be **stitched** to each other by re-meshing locally and re-connecting the faces of both parts. This is done once for frozen-rotor, and at each time step for sliding grid. This can be time-consuming.



Without topological changes :

- By using a **General Grid Interface (GGI)**, no re-meshing is required, hence evaluation across the interface is quicker.
- Developed by M. Beaudoin and H. Jasak [2], the GGI is a coupling interface used to couple multiple non-conformal regions where the patch nodes on each side of the interface do not necessarily match.
- The GGI interface for OpenFOAM is conservative by design, handles cyclic and non-cyclic mesh topologies and runs in parallel.