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

The ERCOFTAC centrifugal pump OpenFOAM case-study

Olivier PETIT

Chalmers University of Technology, Göteborg, Sweden

Maryse PAGE

Hydro-Québec, Institut de recherche, Varennes, Canada

Martin BEAUDOIN

Hydro-Québec, Institut de recherche, Varennes, Canada

Håkan NILSSON

Chalmers University of Technology, Göteborg, Sweden

ABSTRACT

This work investigates the rotor-stator interaction features of OpenFOAM-1.5-dev, such as frozen rotor and sliding grid. The case studied is the ERCOFTAC *Test Case U3: Centrifugal Pump with a Vaned Diffuser*, a testcase from the ERCOFTAC Turbomachinery Special Interest Group. The case was presented by Combès at the *ERCOFTAC Seminar and Workshop on Turbomachinery Flow Prediction VII*, in Aussois, 1999. It is a valid test case for evaluation of rotor-stator interaction features, as detailed experimental data is available.

The investigation shows that OpenFOAM gives results that are comparable to the experimental data, in particular for the sliding grid case. The results are less accurate in the frozen rotor simulation due to the improper treatment of the impeller wakes that is part of the frozen rotor formulation.

The ERCOFTAC centrifugal pump OpenFOAM case-study was developed as a contribution to the OpenFOAM Turbomachinery Working Group, and was presented and discussed at the Fourth OpenFOAM Workshop in Montréal, 2009. The complete set-up of the case-study is available from the OpenFOAM-extend project at SourceForge, and instructions and comments are available from the OpenFOAM Wiki.

KEYWORDS

CFD, OpenFOAM, Turbomachinery, Frozen rotor, Sliding grid, GGI, ERCOFTAC centrifugal pump

1 INTRODUCTION

OpenFOAM is an Open Source library written in C++ [1]. It is a well-structured code, mostly used to implement CFD solvers, although it is also used in other applications. OpenFOAM is based on the finite volume method, but there are also implementations of the finite area and finite element methods. The code accepts fully unstructured meshes and polyhedral cells. Many advanced features can be found in OpenFOAM, such as moving meshes and conjugate heat transfer. With regards to basic features, such as turbulence models and discretization schemes, OpenFOAM is a serious and high quality CFD tool that is constantly evolving.
The community-driven OpenFOAM Turbomachinery Working Group [2] develops and validates OpenFOAM for turbomachinery applications. The ERCOFTAC centrifugal pump was chosen as a validation test case for this investigation of the rotor-stator interaction features in OpenFOAM. The results of the initial studies of this case were presented at the Fourth OpenFOAM Workshop in June 2009, in Montréal, Québec. All the files are available at the OpenFOAM-extend project at SourceForge, for anyone who would like to learn OpenFOAM, or become familiar with the turbomachinery features in OpenFOAM. Some of the important features, such as the GGI (General Grid Interface), multiple frames of reference, and moving meshes are described in the present paper. The GGI forms the base for many useful features for turbomachinery.

2 METHOD

In this work, the flow through a centrifugal pump is investigated. The incompressible Reynolds-Averaged Navier-Stokes equations are solved, using the finite volume method and a standard k-ε turbulence model closure with wall-functions. The convection discretization uses a second-order linearUpwind scheme. Two different rotor-stator approaches are used:

- The *frozen rotor* approach is a steady-state formulation where the rotor and stator are fixed with respect to each other, and different reference frames are used in the rotating and stationary parts. This is also referred to as Multiple Reference Frames (MRF), and it allows taking into account the effect of the rotation of the impeller, although no transient rotor-stator interaction is included. It is nevertheless a fast preliminary method, for use as initial conditions for sliding grid simulations.
- The *sliding grid* approach is a transient method where the rotor mesh actually rotates with respect to the stator mesh. The interaction between the rotor and stator are thus fully resolved. This requires a sliding grid interface between the rotor and stator domains. In OpenFOAM there are two alternatives for sliding interfaces, GGI and topology changes.

Details on the two approaches can be found in section 4.

3 THE ERCOFTAC CENTRIFUGAL PUMP CASE-STUDY

The ERCOFTAC centrifugal pump is a simplified model of a centrifugal turbomachine. The original test case was presented by Combès [3] at a Turbomachinery Flow Prediction ERCOFTAC Workshop in 1999.

3.1 Geometry

The simplified model of the centrifugal pump has 7 impeller blades, 12 diffuser vanes and 6% vaneless radial gap as shown in Fig. 1. The geometric data and operating conditions are shown in Tab. 1. The test rig was built by M.Ubaldi et al.[4]. The purpose was to study the flow unsteadiness generated by rotor-stator interaction in a turbomachine.

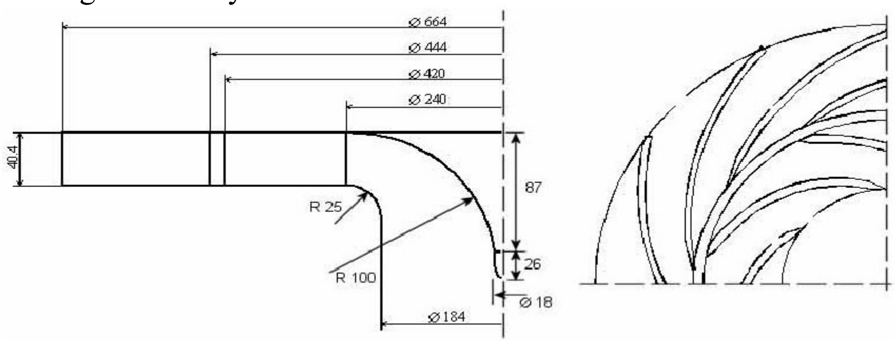

Fig. 1: Impeller and vaned diffuser geometry (Image taken from Ubaldi et al. [4]).

Impeller		**Diffuser**		**Operating conditions**	
Leading edge diameter	$D_1 = 240mm$	Leading edge diameter	$D_3 = 444mm$	Rotational speed	$n = 2000rpm$
Trailing edge diameter	$D_2 = 420mm$	Trailing edge diameter	$D_4 = 664mm$	Impeller tip speed	$U_2 = 43.98m/s$
Number of blades	$z_i = 7$	Number of vanes	$z_d = 12$	Flow rate coefficient	$\varphi = \frac{4Q}{U_2 \pi D_2^2} = 0.048$
Blade span	$z = 40.4mm$	Outlet diameter	$D_5 = 750mm$	Total pressure rise coefficient	$\psi = \frac{2(p_{out} - p_{in})}{8\rho U_2^2} = 0.65$
				Reynolds number	$Re = \frac{U_2 l}{\nu} = 6.5 * 10^5$
				Air density	$\rho = 1.2kg/m^3$

Tab. 1: Geometric data and operating conditions [4].

3.2 Measurements

The experimental data was provided by Ubaldi et al. [4]. The model operates in an open circuit with air directly discharged into the atmosphere from the radial diffuser. The pump operates at the nominal operating condition, at a constant rotational speed of 2000 rpm. (Reynolds number: $6.5x10^5$, incompressible flow regime). Phase locked ensemble averaged velocity components have been measured with hot wire probes at the impeller outlet. The data includes the distribution of the ensemble averaged static pressure at the impeller front end, taken by means of miniature fast response transducers mounted at the stationary casing of the impeller. LDV measurements were also performed in the impeller and in the diffuser.
The measurements made by Ubaldi were made using a hot-wire probe at a radial distance 4 mm from the blade trailing edge (D_m/D_2=1.02). The results from the simulations were extracted in a similar way, as shown in Fig. 2.

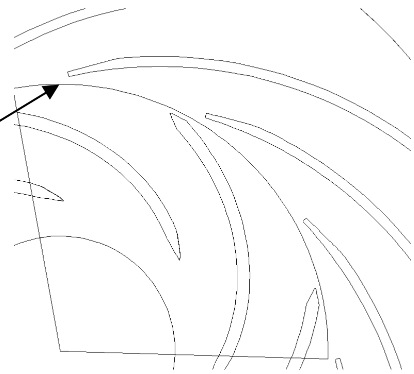

Fig. 2: Position of the hot-wire probe, and of the sampling of the simulated data.

3.3 Case-study set-up and cases

The ERCOFTAC centrifugal pump was used as a case study for the Fourth OpenFOAM Workshop, held in Montréal, Québec, in June 1-4 2009. It can be found in the turbomachinery Special Interest Group web page at the OpenFOAM Wiki (http://openfoamwiki.net/index.php/Sig_Turbomachinery), and in the OpenFOAM-extend SourceForge project (http://sourceforge.net/projects/openfoam-extend/). It includes block-structured meshes, scripts that automatically set up the cases, and automatic post-processing of the results and comparison with the measurements. There are also written descriptions of the OpenFOAM features that are used for the case-study.

In the present work, a 2D representation of the geometry is used. Previous simulations have been done by Bert, Combès and Kueny [5], who showed that relevant information could be recovered from 2D simulations although the real flow has 3D features. The 2D mesh was made using ICEM-HEXA, and the rotor and the impeller were meshed separately. The mesh is block-structured, and consists of about 94 000 cells, with an average Y+ value of 35.
The boundary conditions are shown in Tab. 2.

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

Tab. 2: Computational parameters for the 2D cases.

The ERCOFTAC centrifugal pump (ECP) cases that are currently available are listed here:

- Steady state frozen rotor cases, using MRF (Multiple Reference Frames)
  - *ECPGgi2D*: The frozen rotor approach, and the GGI between the impeller and the diffuser.
  - *ECPStitchMesh2D*: The frozen rotor approach, where the rotor and stator meshes have been stitched together at the interface, forming a single mesh with hanging nodes at the interface.
- Unsteady sliding grid cases
  - *ECPMixerGgiFvMesh2D*: The sliding grid approach, where the GGI is applied between the impeller and the diffuser at each time step.
  - *ECPMixerFvMesh2D*: The sliding grid approach, where the rotor and stator meshes are stitched together at the interface at each time step, forming a single mesh with hanging nodes at the interface at each time step.

3.4 Results

3.4.1 Steady-state simulation – frozen rotor

The simulation was stopped after 5000 iterations, since all the residuals were below 10^{-5}. 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 (see Fig. 3).

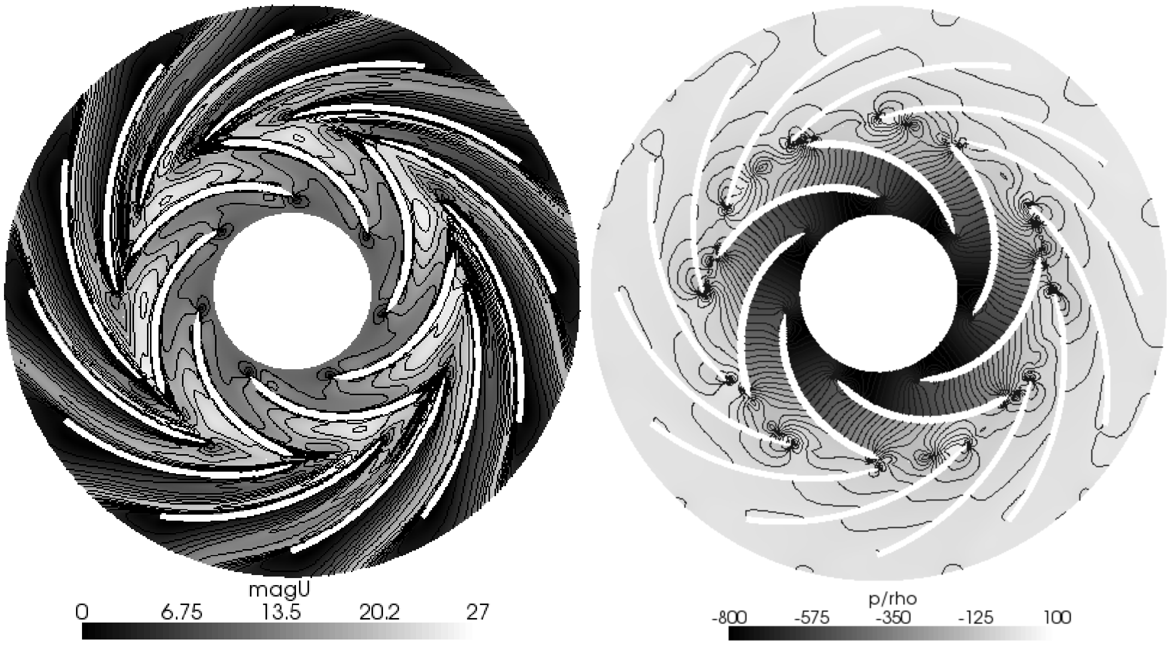

Fig. 3: Velocity magnitude and static pressure for the pump, for the frozen rotor simulation.

The computed velocities show some similarity with the experimental data, as shown in Fig. 4, however, the results do not agree perfectly with the experimental ones. Most of these differences are likely due to the frozen rotor formulation rather than the OpenFOAM implementation. A comparison with preliminary results from a commercial CFD code [6] shows that OpenFOAM gives similar results as commercial CFD codes for frozen rotor simulations.

The GGI and the stitched cases give exactly the same results as can be seen in Fig. 4, and it can thus be concluded that the interface coupling works as it should.

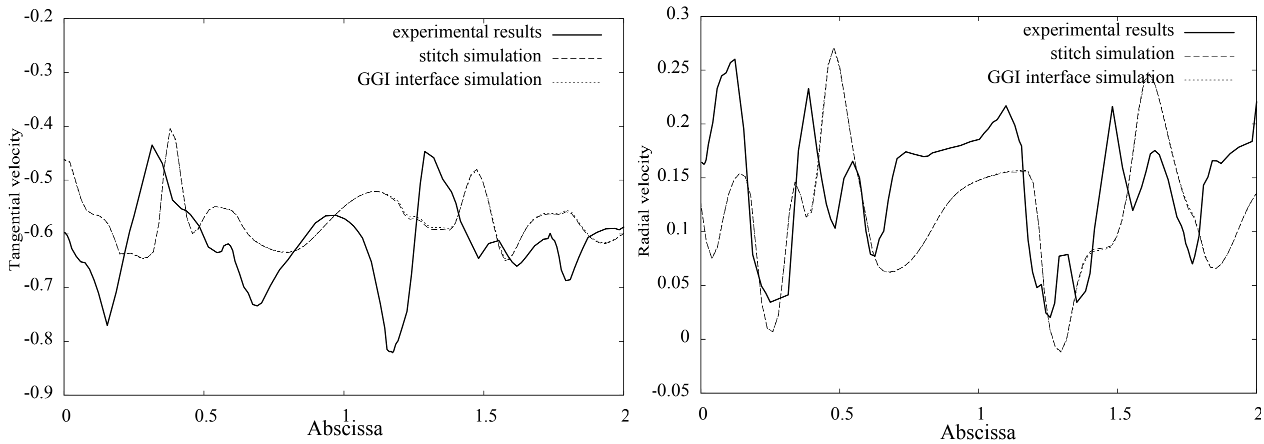

Fig. 4: Distributions of the tangential and radial velocities, for the two frozen rotor simulations compared to the experimental data. The two simulation results are identical.

3.4.2 Unsteady simulation, using the sliding grid approach

Both computations (*ECPMixerGgiFvMesh2D* and *ECPMixerFvMesh2D*) give the same results. Therefore, only the GGI approach will be discussed here.
With the unsteady simulation, the wakes are more visible, and they are advected properly between the diffuser blades, as shown in Fig. 5. There is a better, although not perfect, agreement between the simulations and the experimental data, compare to the frozen rotor simulation, see Fig. 6. However, the tangential velocity is slightly over-predicted.

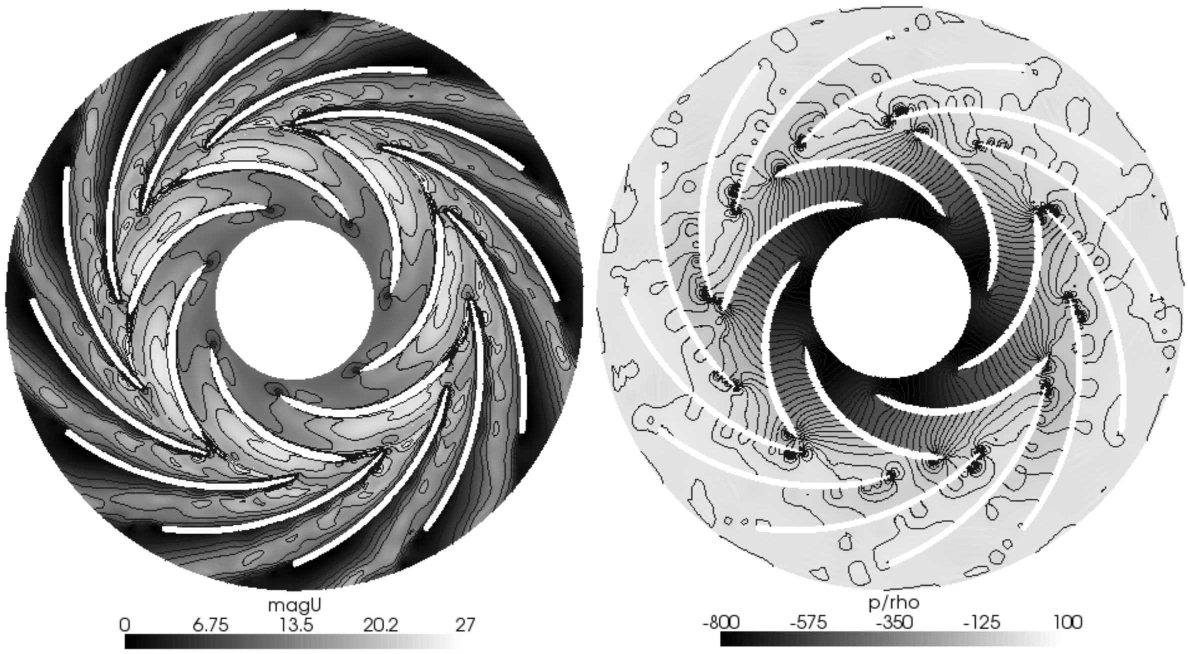

Fig. 5: Velocity magnitude and static pressure for the ERCOFTAC centrifugal pump, for the unsteady simulation

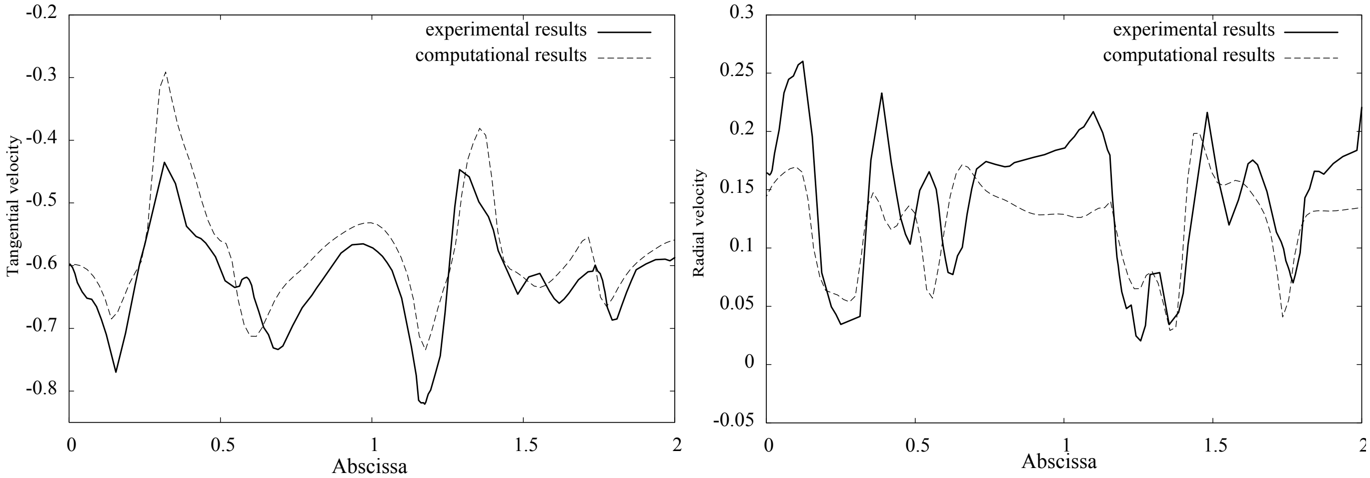

Fig. 6: Distributions of tangential and relative velocity for the unsteady simulation, compared to the experimental data.

4 IMPLEMENTATION DETAILS

4.1 Steady-state solver for Multiple Reference Frames

The solver used for the frozen rotor simulations, based on the Multiple Reference Frame (MRF) approach is named MRFSimpleFoam. It is a steady state solver for incompressible turbulent flow, using the SIMPLE algorithm for pressure-velocity coupling. The MRF approach implies that there is no relative mesh motion of the rotating and stationary parts. In a rotating reference frame, where the relative velocity is computed, the momentum equations must be modified with Coriolis and centrifugal terms. In the MRF approach, the momentum equations uses a mix of inertial and relative velocities, and only one extra term appears in the equations. Tab. 3 summarizes the formulation of the Reynolds Averaged Navier-Stokes equations in the inertial and the rotating frames, and the third alternative is what is implemented in MRFSimpleFoam.

Frame	Convected velocity	Steady incompressible Navier-Stokes equations
Inertial	Absolute velocity	$\begin{cases} \nabla\cdot(\vec{u}_I\otimes\vec{u}_I) = -\nabla(p/\rho)+\nu\nabla\cdot\nabla(\vec{u}_I) \\ \nabla\cdot\vec{u}_I = 0 \end{cases}$
Rotating	Relative velocity	$\begin{cases} \nabla\cdot(\vec{u}_R\otimes\vec{u}_R)+2\vec{\Omega}\times\vec{u}_R+\vec{\Omega}\times\vec{\Omega}\times\vec{r} = -\nabla(p/\rho)+\nu\nabla\cdot\nabla(\vec{u}_R) \\ \nabla\cdot\vec{u}_R = 0 \end{cases}$
Rotating	Absolute velocity	$\begin{cases} \nabla\cdot(\vec{u}_R\otimes\vec{u}_I)+\vec{\Omega}\times\vec{u}_I = -\nabla(p/\rho)+\nu\nabla\cdot\nabla(\vec{u}_I) \\ \nabla\cdot\vec{u}_I = 0 \end{cases}$

Tab. 3: Summary of the Navier-Stokes equations for steady flow in multiple reference frames.

4.1.1 Frozen rotor

In the frozen rotor formulation, the rotating and stationary parts are considered to be at a fixed position relative to each other. The coupling between the rotor and stator domains are still resolved 360˚, but fixed in time. Since the rotor and the stator parts have been meshed separately, a connection must be made between these meshes. Two methods can be used for that in OpenFOAM. The first one is based on the topological changes technology (see section 4.3). The second method is the GGI (see section 4.4).

4.1.2 Mixing plane

Another solution to model the interface between the rotating and stationary parts is the mixing plane. At the interface the flow properties are circumferentially averaged. This will of course remove all transient rotor-stator interactions, but it still gives fairly representative results. A mixing-plane simulation only requires one rotor blade and one stator blade per stage, which significantly accelerates the solution procedure.
The mixing plane is under implementation in OpenFOAM at the moment, but there should be an implementation available in the coming year.

4.2 Unsteady solver for sliding grid

The solver used for the sliding grid simulations in this work is named turbDyMFoam. This is a solver for incompressible RANS simulations using the PISO algorithm for pressure-velocity coupling. The solver also uses libraries for mesh motion and deformation of polyhedral meshes [7], of which one of them handles partly rotating meshes. In the simulations the physical motion of the mesh is directly addressed. The coupling between the rotating and non-rotating parts of the mesh can be accomplished by either topological changes (see section 4.3) or GGI (see section 4.4).

4.3 Topological changes

Two parts of a mesh with coinciding faces can be attached to each other by connecting the faces of both parts. If the resulting mesh would not be conformal, OpenFOAM allows polyhedral cells, and may thus cut the faces to make the mesh conformal with hanging nodes. The sliding interface with topological changes in OpenFOAM uses this functionality at each time step, and deals with the topological changes associated to it. The topological modifier attach-detach is then used, and the rotating part of the mesh is detached from the static part. The rotation then occurs, and the mesh is attached again, as explained in Fig. 7. The black points show the vertices of a 2D cell, and it is shown that a non-conformal connection gives control volumes with hanging nodes.

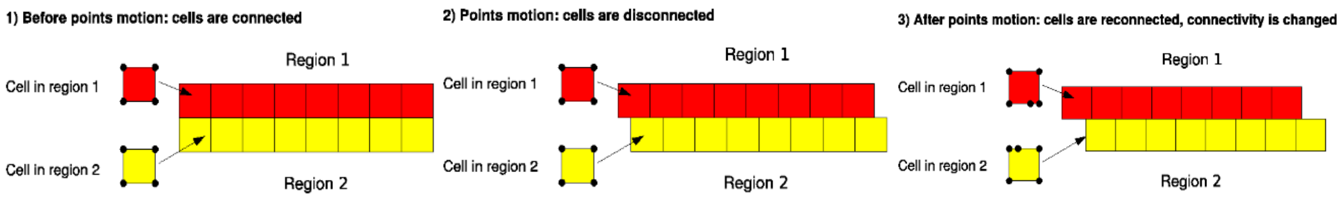

Fig. 7: Operation of a sliding interface, pictures from [7].

This operation can be quite time-consuming, as it needs to re-organize the topology and the internal numbering of faces and cells at each time step. However, the method is fully conservative.

4.4 General Grid Interface (GGI)

The GGI can be used to speed up the operation of the coupling between the rotor and stator domains. With the GGI, the modus operandi is quite similar as for the topological change method. The neighbours still need to be evaluated, but the attach-detach topological modifier doesn't need to be called anymore. This is a gain of time at each time step that is very noticeable on the whole simulation.
Implicit couplings are present in OpenFOAM in order to join multiple mesh regions into a single contiguous domain. But most of them are built to join conformal mesh regions, where

the patches nodes on each side of the interface are matching one to one. The GGI, developed by M. Beaudoin and H. Jasak [8] is a coupling interface used to join multiple non-conformal regions where the patches nodes on each side of the interface do not match.
A GGI is commonly used in turbomachinery, where the flow is simulated through a succession of various complex geometries. The requirement to fit all meshes with conformal matching interface is often very hard to reach. Using the GGI, non-conformal meshes can be designed separately, and joined together using one of many GGI alternatives.
The basic GGI is using weighted interpolation to evaluate and transmit flow values across a pair of conformal or non-conformal coupled patches. It is similar to the static sliding interface, although much simpler in the sense that no re-meshing is required for the neighbouring cells of the interface. The GGI weighting factors are basically the percentage of surface intersection between two overlapping faces.
The GGI implementation is using the Sutherland-Hodgman algorithm to compute the master and shadow face intersection surface area [9]. This algorithm is simple, fast and robust; being re-entrant, it allows for a very compact source code implementation. The Sutherland-Hodgman algorithm is also generic enough to handle any convex n-sided polygons.

5 Conclusions

The rotor-stator interaction features of OpenFOAM have been investigated and compared with experimental data of the ERCOFTAC centrifugal pump with a vaned diffuser. Both steady state simulations using Multiple Reference Frames and unsteady simulations using a transient solver with sliding mesh were performed. Good agreements were found with the experiments, but improvements can be made. The unsteady simulation showed a good behavior of the wakes being advected through the diffuser. It can be concluded that all the functionality is available in OpenFOAM for accurate rotor-stator analysis.
In order to improve the numerical results the same simulations will be performed in 3D, and alternative boundary conditions will be investigated. The coming mixing plane implementation will also be evaluated in the same case, and a tutorial on how to use the mixing plane will be released to the OpenFOAM community.

6 ACKNOWLEDGEMENTS

We would like to express our greatest acknowledgements to M. Ubaldi who made the experimental results of the ERCOFTAC centrifugal pump available, to share with the OpenFOAM community. We would also like to acknowledge OpenCFD Ltd, as well as Hrovje Jasak from Wikki Ltd, who distribute OpenFOAM, and helped us a lot to create this case study. We are very grateful to Hydro-Québec for its financial support and its participation in this project. Olivier Petit and Håkan Nilsson are partly financed by SVC (the Swedish Water Power Center, www.svc.nu). SVC has been established by the Swedish Energy Agency, ELFORSK and Svenska Kraftnät together with Chalmers University of Technology, Luleå University of Technology, Uppsala University and the Royal Institute of Technology.

7 REFERENCES

[1] Weller H.G, Tabor G, Jasak H, Fureby C., *“A tensorial approach to computational continuum mechanics using object-oriented techniques”*, Computers in Physics, Vol.12, No.6, 1998.

[2] Nilsson H., Page M., Beaudoin M., Gschaider B. and Jasak H., *"The OpenFOAM Turbomachinery Working Group, and Conclusions from the Turbomachinery Session of the Third OpenFOAM Workshop"*, 24th IAHR Symposium on Hydraulic Machinery and Systems, October 27-31, 2008, Foz Do Iguassu, Brazil.

[3] Combès, J.F., *"Test Case U3: Centrifugal Pump with a Vaned Diffuser"*, ERCOFTAC Seminar and Workshop on Turbomachinery Flow Prediction VII, Aussois, jan 4-7, 1999.

[4] Ubaldi M., Zunino P., Barigozzi G. and Cattanei A., *"An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow",* Journal of Turbomachinery, vol.118, 41-54, 1996.

[5] Combès, J.F., Bert, P.F. and Kueny, J.L., *"Numerical Investigation of the Rotor-Stator Interaction in a Centrifugal Pump Using a Finite Element Method"*, Proceedings of the 1997 ASME Fluids Engineering Division Summer Meeting, FEDSM97-3454, 1997.

[6] Page, M., Théroux, E. and Trépanier, J.-Y., *"Unsteady rotor-stator analysis of a Francis turbine"*, 22nd IAHR Symposium on Hydraulic Machinery and Systems, June 29 – July 2, 2004, Stockholm, Sweden.

[7] Jasak, H., *”Dynamic Mesh Handling in OpenFOAM”*, 47th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition, 5-8 January, Orlando, Florida, 2008 (AIAA 2009-341).

[8] Beaudoin M. and Jasak H., *"Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM"*, Open Source CFD International Conference 2008.

[9] Sutherland I.E. and Hodgman G.W., *"Reentrant polygon clipping*”, Communication of the ACM, vol.17, Number 1, 32-42, 1974.