The ERCOFTAC centrifugal pump OpenFOAM case-study

Olivier Petit and Håkan Nilsson

Chalmers University of Technology, SVC

Maryse Page and Martin Beaudoin

Hydro-Québec, Research Institute

4th OpenFOAM workshop

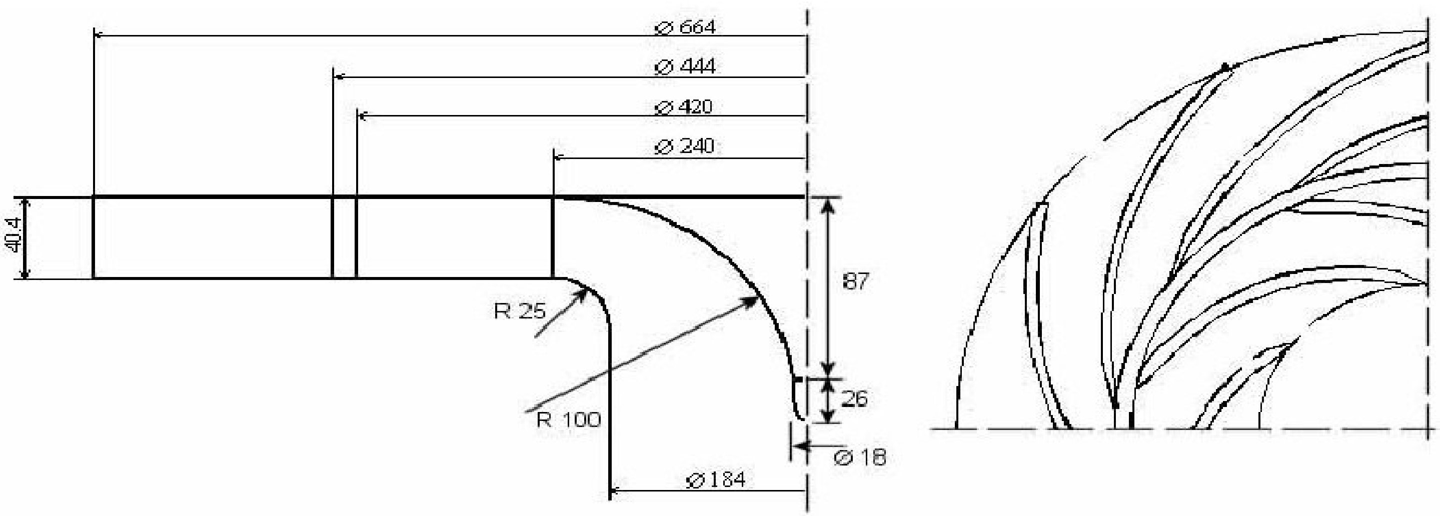
Montréal 1-4 June 2009

Outline

- Description of the ERCOFTAC centrifugal pump case study.
- Tutorial for a steady simulation using the GGI interface and the MRFSimpleFoam solver.
- Tutorial for an unsteady simulation with mesh motion using the GGI interface and the turbDyMFoam solver.
- Conclusions
- Future work

Test case description

- The centrifugal pump has 7 impeller blades, 12 diffuser vanes and 6% vaneless radial gap. The pump operates on air, and at constant rotational speed of 2000 rpm.



Ref: 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

How to get the cases and descriptions

- Need OpenFOAM-1.5-dev + developments available in Breeder branch on OpenFOAM-extend
- http://openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_centrifugal_pump_with_a_vaned_diffuser
- How to do a full check-out of all the ERCOFTAC centrifugal pump files:

```
svn checkout
http://openfoam-extend.svn.sourceforge.net/svnroot/
openfoam-extend/trunk/Breeder_1.5/OSIG/
TurboMachinery/ercoftacCentrifugalPump
```

- How to update: svn update
- How to commit: svn commit

Please discuss with us before you commit.

Structure of the case study

In the svn, under ercoftacCentrifugalPump:

```
|-- cases
    |-- MRFSimpleFoam
        |-- ECPGgi2D
        |-- ECPStitchMesh2D
        |-- ECPMixingPlane2D
    |-- turbDyMFoam
        |-- ECPMixerGgiFvMesh2D
        |-- ECPMixerFvMesh2D
    |-- meshes
|-- measurements
```

The grey cases are not yet on svn

Basic procedure to run the cases

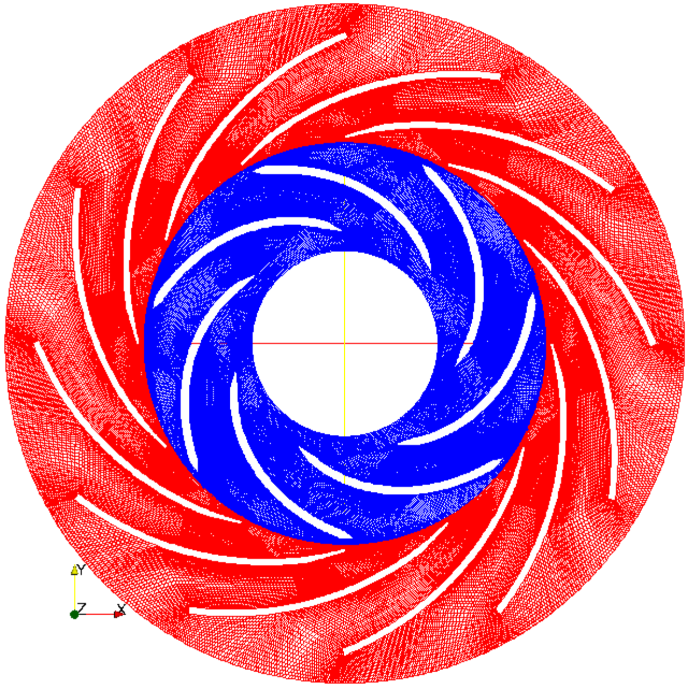
- All cases are coming with an Allrun script that builds the case and starts the simulation using the appropriate solver.

- To start over, an Allclean is provided, deleting the previous simulations.

- Post-process using sample, foamLog, and gnuplot scripts. Here as well, Allrun and Allclean scripts are provided.

Mesh

- The mesh was generated with ICEM-HEXA and exported in the Fluent mesh format. The rotor and stator were meshed separately.
- 93886 hexa cells, Average Y+ value is 35.
- 2D model for tutorial purposes



The MRFSimpleFoam solver

- Steady-state solver for incompressible turbulent flow with Multiple Reference Frames regions
- OpenFOAM/tutorials/MRFSimpleFoam
- Need to define one or more rotating zones in constant/MRFZones

```
1
(
  rotor
  {
    patches                          (BLADE_ROT);
    origin     [0 1 0 0 0 0 0]      (0 0 0);
    axis       [0 0 0 0 0 0 0]      (0 0 -1);
    omega      [0 0 -1 0 0 0 0]    209;       //2000 RPM
  }
)
```

GGI interface, basic setup

- Generalized Grid Interface to couple non-conformal regions (Beaudoin & Jasak, OpenSource CFD Int. Conf., Berlin, 2008)

constant/polyMesh/boundary

```
GGI_INT
{
  type            ggi;
  nFaces          707;
  startFace       374119;
  shadowPatch     GGI_EXT;
  bridgeOverlap   false;
  zone            GGI_INT_ZONE;
}
```

```
GGI_EXT
{
  type            ggi;
  nFaces          756;
  startFace       374826;
  shadowPatch     GGI_INT;
  bridgeOverlap   false;
  zone            GGI_EXT_ZONE;
}
```

0/[U p k epsilon] boundaryField

```
GGI_INT
{
  type ggi;
}
GGI_EXT
{
  type ggi;
}
```

- Additional step for serial/parallel computing:

```
setBatch file: faceSet GGI_INT_ZONE new patchToFace GGI_INT
               faceSet GGI_EXT_ZONE new patchToFace GGI_EXT
               quit
setSet –batch setBatch
setsToZones -noFlipMap
```

- Useful utility, ggiCheck, to monitor the GGI characteristics

ECPGgi2D case study

- Use of MRFSimpleFoam solver and GGI interface

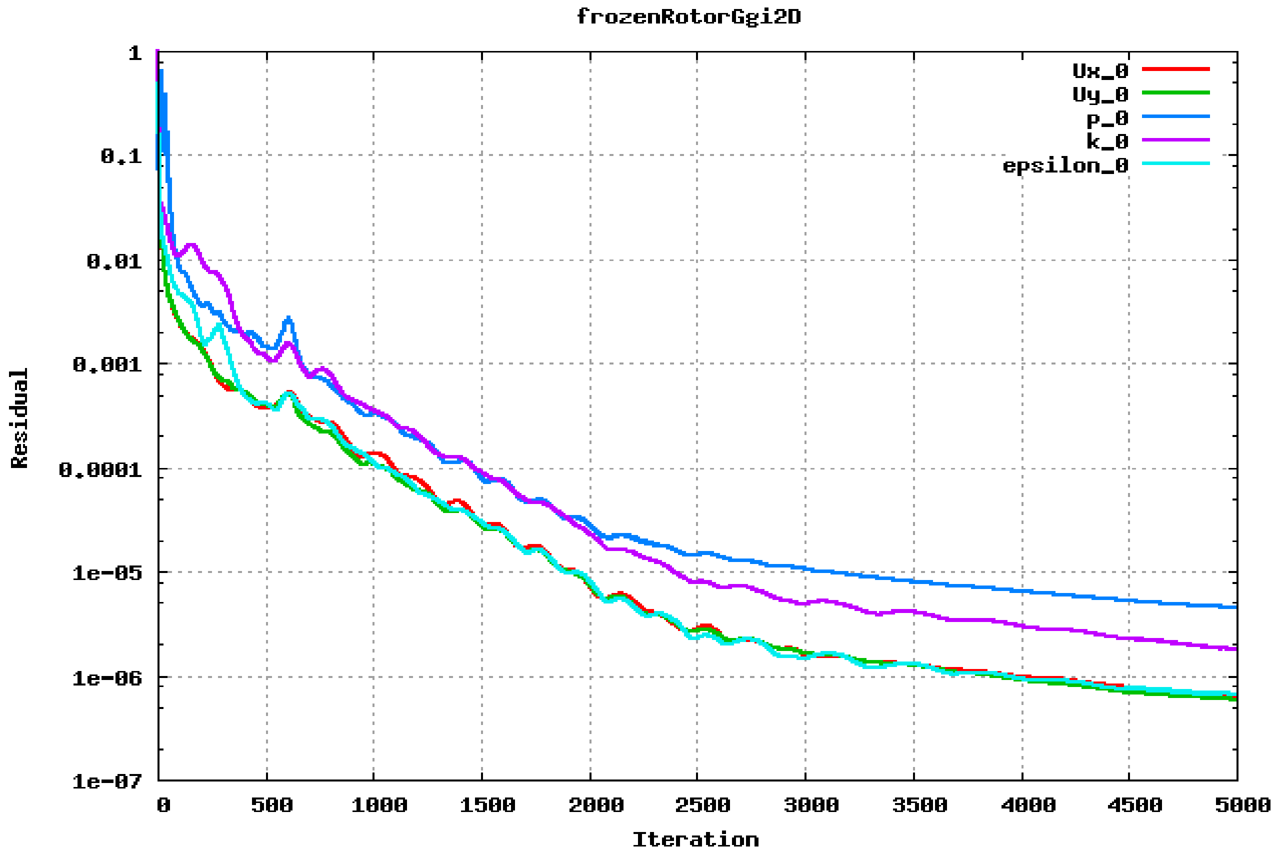
OpenFOAM version	1.5-dev, svn revision 1240		
Solver	MRFSimpleFoam		
Div Schemes	U	linearUpwind	
	k,ε	Upwind	
Solvers	p	GAMG	
	U,k,ε	SmoothSolver	
Turbulence model	kEpsilon		
Boundary conditions	Inlet	U,k,ε	profile1DFixedValue
		p	zeroGradient
	Outlet	k,ε	zeroGradient
		p	fixedMeanValue 0

Profile1DFixedValue: in rotor2d_abs.csv

- U_r=11.4 m/s
- k=0.48735 m^2/s^2
- ε=138.342 m^2/s^3

Results

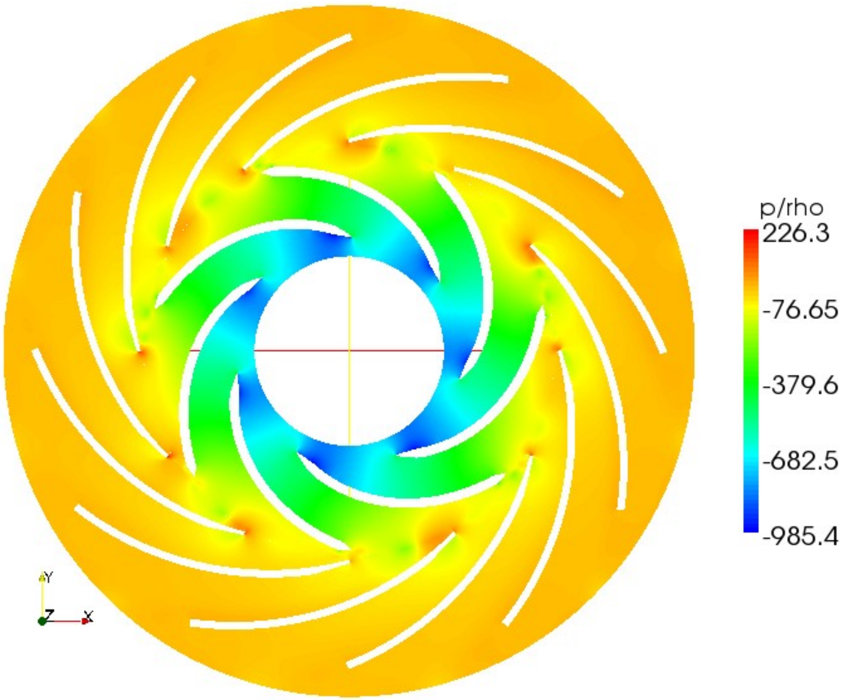
- The simulation is converged, and stopped at 5000 iterations.

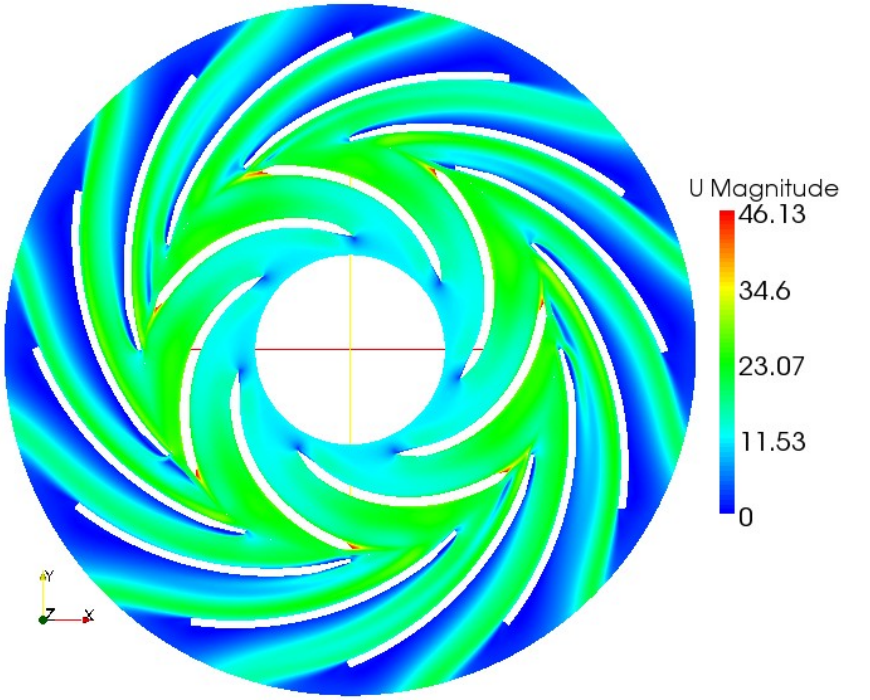


CHALMERS SVC

Hydro Québec Institut de recherche

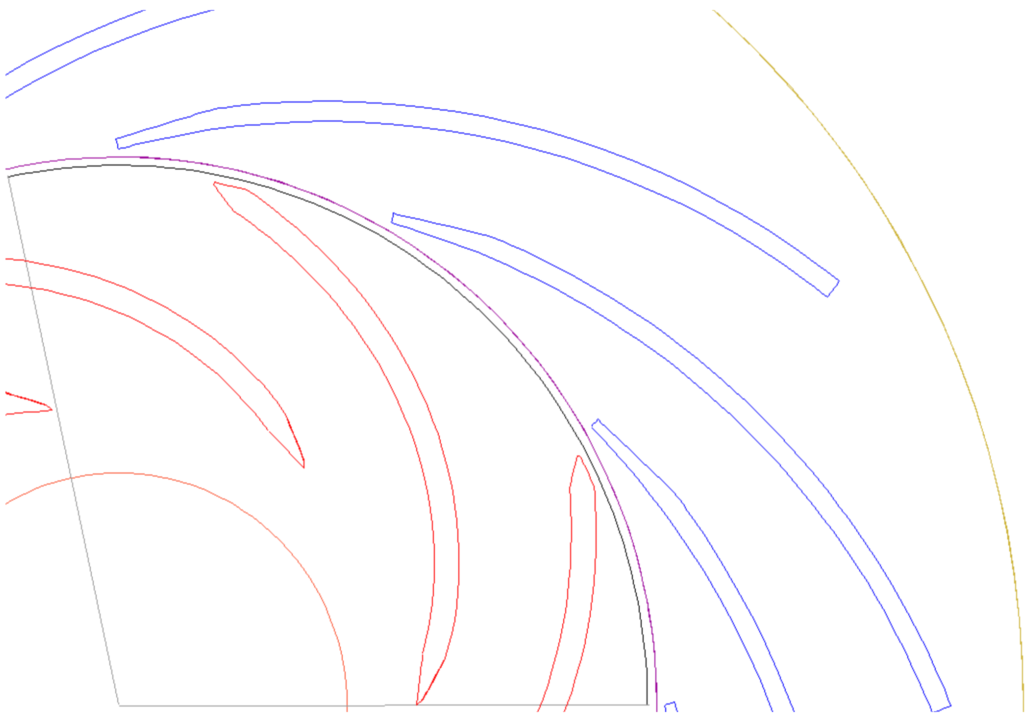
Results


p/rho
226.3
-76.65
-379.6
-682.5
-985.4


U Magnitude
46.13
34.6
23.07
11.53
0

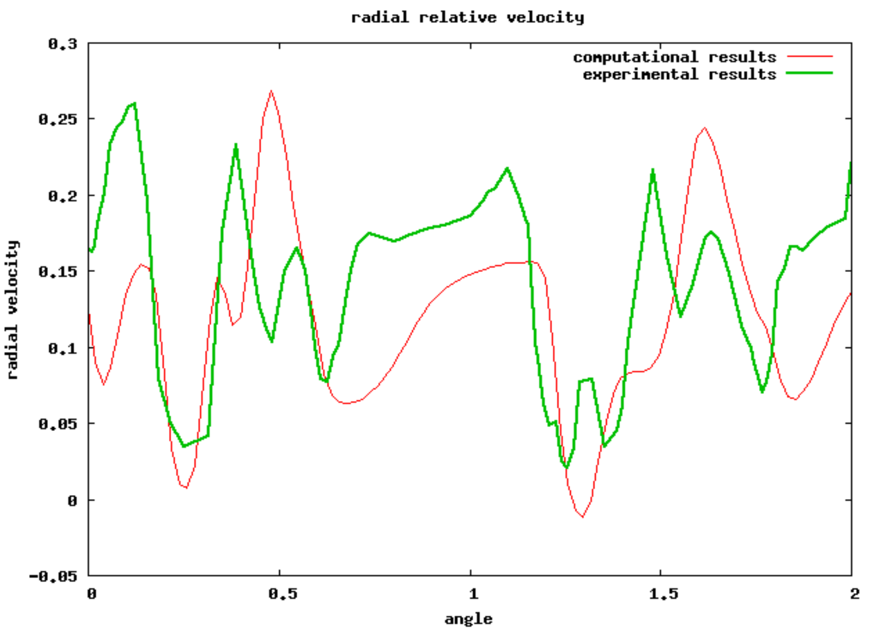
Comparison with the experimental data

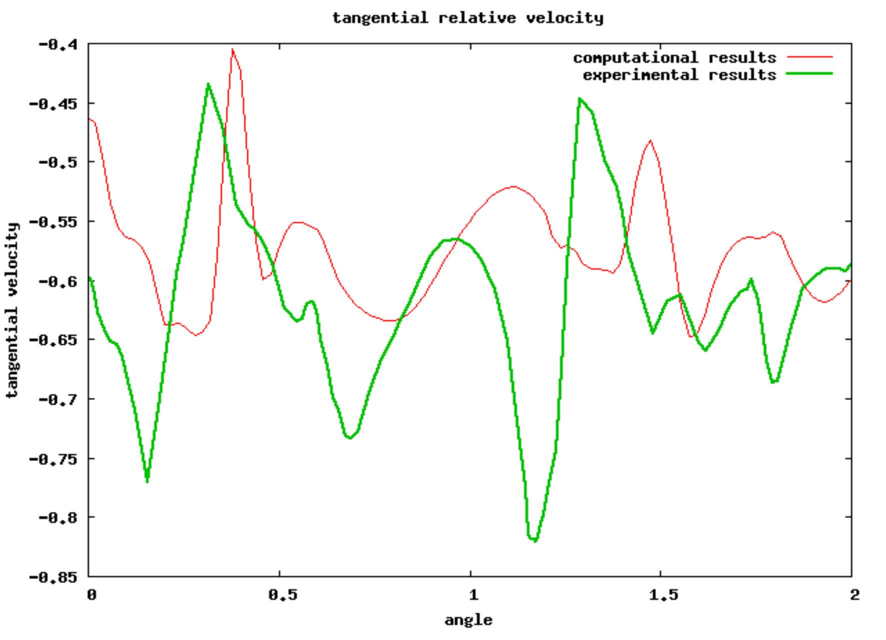
- Experimental results obtained at mid-span, for R/R2=1.02, R2 being the outlet diameter of the impeller
- Measured on 2 blade passages



CHALMERS
SWC
Hydro Québec
Institut de recherche

Comparison with experimental data





Discussion about ECPGgi2D

- The results using the GGI interface are identical to those when using stitchMesh (ECPStitchMesh2D case study).

- Phase shift in the wake between the experimental and simulated results, probably due to the frozenRotor formulation.

The turbDyMFoam solver

- Transient solver for incompressible turbulent using the PISO solver.
- The mesh motion is set-up in constant/dynamicMeshDict:

```
dynamicFvMeshLib   "libtopoChangerFvMesh.so";
dynamicFvMesh      mixerGgiFvMesh;

mixerGgiFvMeshCoeffs
{
    coordinateSystem
    {
        type                cylindrical;
        origin              (0 0 0);
        axis                (0 0 1);
        direction           (1 0 0);
    }

    rpm                         -2000;

    slider
    {
        moving       ( GGI_INT );
        static        ( GGI_EXT );
    }
}
```

ECPMixerGgiFvMesh2D case study

- Use of turbDyMFoam solver, GGI interface and mixerGgiFvMesh
- Physical rigid motion of the mesh.

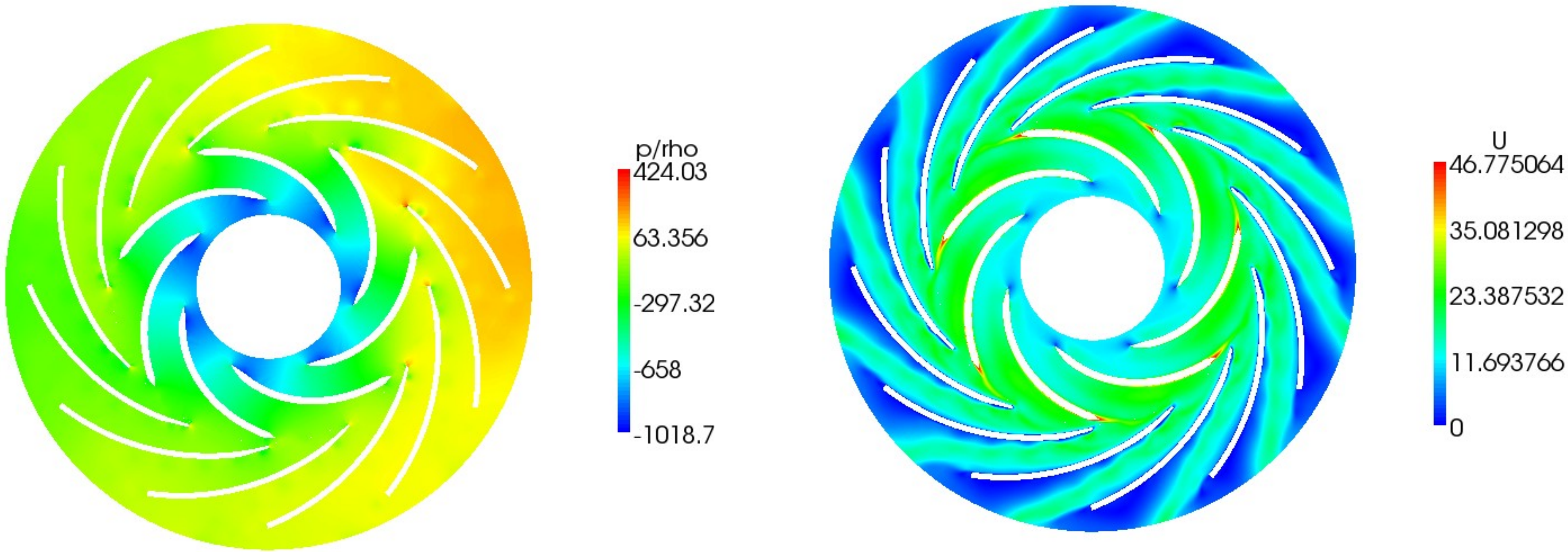
OpenFOAM version	1.5-dev, svn revision 1240		
Solver	turbDyMFoam		
Div Schemes	U	linearUpwind	
	k,ε	Upwind	
Solvers	p	BiCGStab	
	U,k,ε	BiCGStab	
Turbulence model	kEpsilon		
Boundary conditions	Inlet	U,k,ε	profile1DFixedValue
		p	zeroGradient
	Outlet	k,ε	zeroGradient
		p	fixedMeanValue 0

Profile1DFixedValue:
in rotor2d_abs.csv

- U_r=11.4 m/s
- k=0.48735 m^2/s^2
- ε=138.342 m^2/s^3

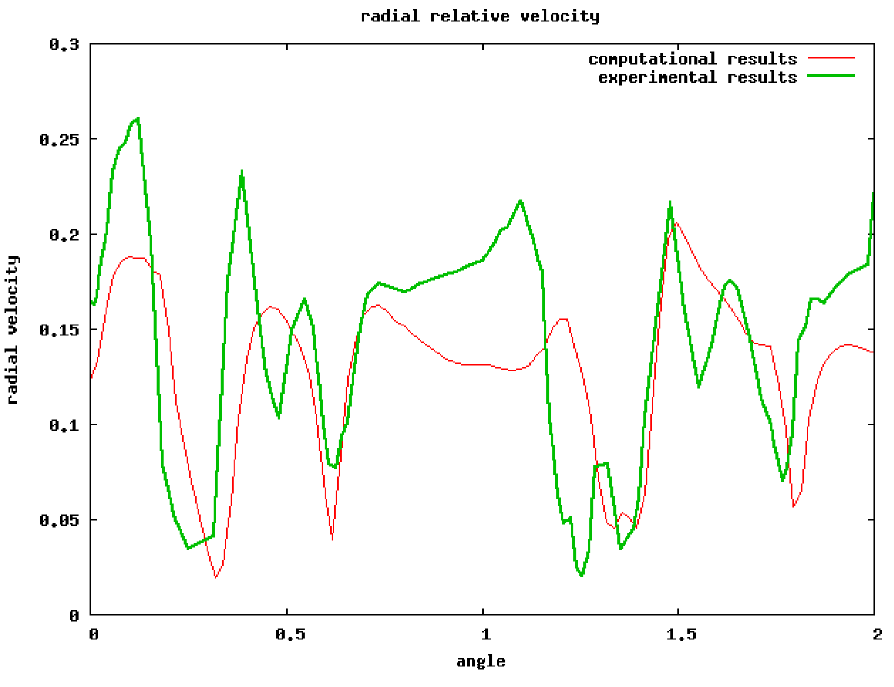
CHALMERS SVC Hydro Québec Institut de recherche

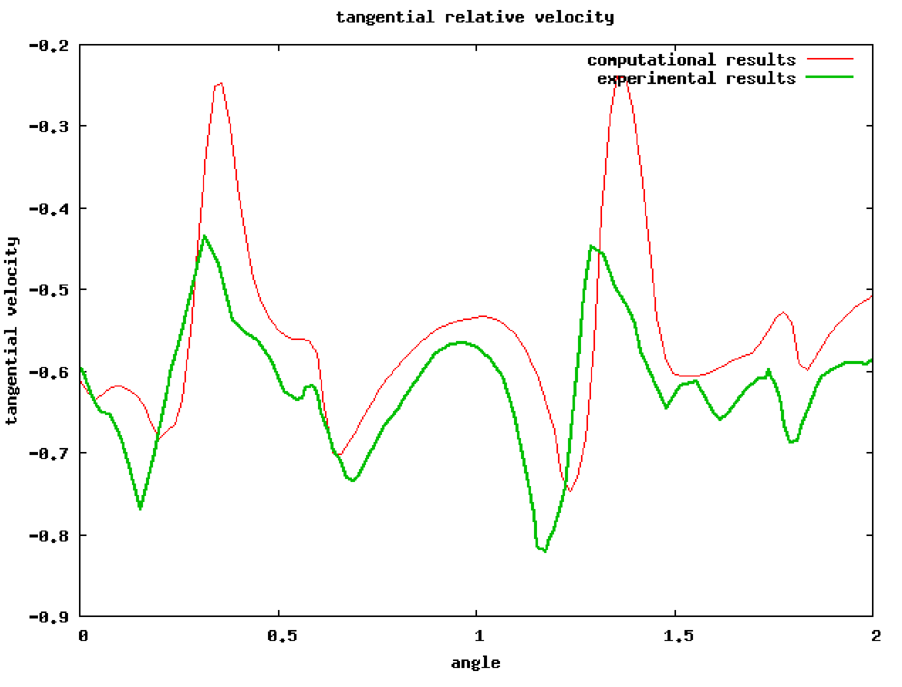
Results



- The unsteady test case shows a more accurate simulation of the wakes.
- The pressure field shows a large-scale oscillation for the current settings, and further investigations of this are being made

Comparison with experimental data





General conclusions

- The case study covers various objectives of the Turbo Working Group:
  - Validation of some useful utilities for turbomachinery simulation (such as GGI interface)
  - Various tutorials for different solvers
  - Collaboration
  - Easy diffusion

- More test cases can be added to this case study, we invite more people to participate.

- A lot more information can be found at http://openfoamwiki.net/index.php/Sig_Turbomachinery

Future work

- A more detailed study of the unsteady simulations will be presented at the IAHR conference in Brno, Czech Republic, October 14-16, 2009 (http://khzs.fme.vutbr.cz/iahrwg2009/)
- A tutorial for the mixing plane should be added as well, validating this new interface.
- A tutorial for simpleTurboMFRFoam, another implementation of a steady-state solver for incompressible turbulent flow for MFR regions.
- A tutorial for transientSimpleDyMFoam, another implementation of a transient solver for incompressible turbulent flow with rigid mesh motion.

Thank you very much !

All information and tutorials can be found at:
http://openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_centrifugal_pump_with_a_vaned_diffuser

The ERCOFTAC conical diffuser case-study, developed for the 3rd OpenFOAM Workshop, can be found at:
http://openfoamwiki.net/index.php/Sig_Turbomachinery_/_ERCOFTAC_conical_diffuser

The Turbomachinery working group website is located at:
http://openfoamwiki.net/index.php/Sig_Turbomachinery