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 ERCOFAC 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.

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

```plaintext
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 patchToFracce GGI_INT
faceSet GGI_EXT_ZONE new patchToFracce GGI_EXT
quit
setSet -batch setBatch
setsToZones -noFlipMap

Useful utility, ggiCheck, to monitor the GGI characteristics
Use of MRFSimpleFoam solver and GGI interface

<table>
<thead>
<tr>
<th>OpenFOAM version</th>
<th>1.5-dev, svn revision 1240</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>MRFSimpleFoam</td>
</tr>
<tr>
<td>Div Schemes</td>
<td></td>
</tr>
<tr>
<td>U</td>
<td>linearUpwind</td>
</tr>
<tr>
<td>k,ε</td>
<td>Upwind</td>
</tr>
<tr>
<td>Solvers</td>
<td></td>
</tr>
<tr>
<td>p</td>
<td>GAMG</td>
</tr>
<tr>
<td>U,k,ε</td>
<td>SmoothSolver</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>kEpsilon</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td></td>
</tr>
<tr>
<td>Inlet</td>
<td>U,k,ε</td>
</tr>
<tr>
<td>p</td>
<td>zeroGradient</td>
</tr>
<tr>
<td>Outlet</td>
<td>k,ε</td>
</tr>
<tr>
<td>p</td>
<td>fixedMeanValue 0</td>
</tr>
</tbody>
</table>

Profile1DFixedValue: in rotor2d_abs.csv

- Ur=11.4 m/s
- k=0.48735 m²/s²
- ε=138.342 m²/s³
Results

- The simulation is converged, and stopped at 5000 iterations.
Results
Comparison with the experimental data

- Experimental results obtained at mid-span, for \( R/R_2 = 1.02 \), \( R_2 \) being the outlet diameter of the impeller
- Measured on 2 blade passages
Comparison with experimental data

Comparison with experimental data

![Graphs showing radial and tangential relative velocity comparisons between computational and experimental results.](image-url)
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:

```plaintext
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.

<table>
<thead>
<tr>
<th>OpenFOAM version</th>
<th>1.5-dev, svn revision 1240</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>turbDyMFoam</td>
</tr>
<tr>
<td>Div Schemes</td>
<td></td>
</tr>
<tr>
<td>U</td>
<td>linearUpwind</td>
</tr>
<tr>
<td>k, ε</td>
<td>Upwind</td>
</tr>
<tr>
<td>Solvers</td>
<td></td>
</tr>
<tr>
<td>p</td>
<td>BiCGStab</td>
</tr>
<tr>
<td>U, k, ε</td>
<td>BiCGStab</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>kEpsilon</td>
</tr>
<tr>
<td>Boundary conditions</td>
<td></td>
</tr>
<tr>
<td>Inlet</td>
<td>U, k, ε</td>
</tr>
<tr>
<td></td>
<td>profile1DFixedValue</td>
</tr>
<tr>
<td></td>
<td>p</td>
</tr>
<tr>
<td></td>
<td>zeroGradient</td>
</tr>
<tr>
<td>Outlet</td>
<td>k, ε</td>
</tr>
<tr>
<td></td>
<td>zeroGradient</td>
</tr>
<tr>
<td></td>
<td>p</td>
</tr>
<tr>
<td></td>
<td>fixedMeanValue 0</td>
</tr>
</tbody>
</table>

Profile1DFixedValue: in rotor2d_abs.csv

- Ur=11.4 m/s
- k=0.48735 m²/s²
- ε=138.342 m²/s³
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

Radial relative velocity

Tangential relative velocity
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_conifugal_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