Adapting OpenFOAM for Turbomachinery Applications

Maryse Page and Martin Beaudoin
IREQ, Hydro-Québec’s Research Institute

Second OpenFOAM Workshop, Zagreb, 7-9 June 2007
Outline of the presentation

➢ Context
➢ OpenFOAM for hydraulic turbines
  ▪ Developments
  ▪ High-performance computing
  ▪ GPL
  ▪ Future development
➢ Validation for hydraulic turbines
Hydro-Québec

- Québec’s electric utility
  - Power generation
  - Transmission
  - Distribution
- 35 GW installed capacity
- 97% power generation based on hydroelectric resources
- 50+ generating stations
- 350 hydroelectric generating units
The MATH technology:
- Analyze the hydraulic behaviour of hydraulic turbine by CFD
MATH technology

- Improve the performance of hydraulic turbine by the design of tailor-made modification

**CFD analysis**

**On-site implementation**

---

Second OpenFOAM Workshop
Zagreb 7-9 June 2007
MATH technology
OpenFOAM: new addition in our CFD toolbox

- Open Source: potential for collaboration
- No runtime licensing fees
- High quality C++ source code
- Sound software architecture
  - Object oriented design and implementation
  - Encapsulation through libraries
  - Already parallelized for high-performance computing
- Strong and active community with experts still actively involved in the development
Objective: Developing basic steady-state capabilities for hydraulic turbines

- Multiple Frames of Reference (MFR)
- Boundary conditions
- Interfaces:
  - General Grid Interface (GGI)
  - Cyclic GGI
  - Mixing plane
- CGNS file converters
**Turbomachinery modeling**

- Unsteadiness from relative motion of rotating and stationary parts
  - Requires 3D flow modeling of all the hydraulic components
- Simplification by introducing:
  - cyclic BC
  - MFR – no relative mesh motion
- For steady-state flow, more simplifications by:
  - neglecting transient interaction
  - averaging (mixing plane) between rotating/stationary parts
## Multiple Frame of Reference (MFR)

<table>
<thead>
<tr>
<th>RANS formulation</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>inertial frame (absolute velocity)</td>
<td>$\text{div}(u_I \otimes \bar{u}_I) = -\text{grad}(p/\rho) + \nu \text{lap}(\bar{u}_I)$</td>
</tr>
<tr>
<td>rotating frame (relative velocity)</td>
<td>$\text{div}(\bar{u}_R \otimes \bar{u}_R) + 2\bar{\Omega} \times \bar{u}_R + \bar{\Omega} \times \bar{\Omega} \times \bar{r} = -\text{grad}(p/\rho) + \nu \text{lap}(\bar{u}_R)$</td>
</tr>
<tr>
<td>rotating frame (absolute velocity)</td>
<td>$\text{div}(\bar{u}_R \otimes \bar{u}_I) + \bar{\Omega} \times \bar{u}_I = -\text{grad}(p/\rho) + \nu \text{lap}(\bar{u}_I)$</td>
</tr>
</tbody>
</table>

- No frame change at interface between stationary and rotating domains
- Application: simpleMFRFoam
**Testcase: ERCOFTAC centrifugal pump**

MFR example: ERCOFTAC centrifugal pump

Second OpenFOAM Workshop
Zagreb 7-9 June 2007
Boundary conditions

Walls:
- Rotating walls for MFR
- Dictionary parameters
- Evaluated at every time iteration

<table>
<thead>
<tr>
<th>Type of wall</th>
<th>Boundary Field dictionary entry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spinning</td>
<td>runnerConeTurbine99</td>
</tr>
<tr>
<td></td>
<td>{</td>
</tr>
<tr>
<td></td>
<td>type turboWallFixedValue;</td>
</tr>
<tr>
<td></td>
<td>frameType &quot;fixed&quot;;</td>
</tr>
<tr>
<td></td>
<td>velocityType &quot;plusOmega&quot;;</td>
</tr>
<tr>
<td></td>
<td>omegaWallRPM -595;</td>
</tr>
<tr>
<td></td>
<td>}</td>
</tr>
<tr>
<td>Counter-rotating</td>
<td>draftTubeCone</td>
</tr>
<tr>
<td></td>
<td>{</td>
</tr>
<tr>
<td></td>
<td>type turboWallFixedValue;</td>
</tr>
<tr>
<td></td>
<td>frameType &quot;rotating&quot;;</td>
</tr>
<tr>
<td></td>
<td>velocityType &quot;minusOmega&quot;;</td>
</tr>
<tr>
<td></td>
<td>omegaWallRPM 0.0;</td>
</tr>
</tbody>
</table>
**Boundary conditions**

- **Inlet / outlet profiles**
  - Radial and meridian profiles
  - File format: CSV files (as CFX-5)
  - Evaluated at every time iteration

<table>
<thead>
<tr>
<th>Type of profile</th>
<th>Boundary Field dictionary entry</th>
</tr>
</thead>
</table>
| Draft tube inlet      | inlet {
|                       |     type cfx5ProfileFixedValue;    |
|                       |     filePath "inlet-operA.csv";   |
|                       |     fileFormat "CSV";             |
|                       |     profileType "Radial";         |
|                       |     fieldName "Velocity";         |
Interfaces

- GGI interface:
  - New type of interface introduced by Hrvoje Jasak in OpenFOAM 1.3_dev
  - Based on PatchToPatchInterpolation
    - Not a sliding interface
    - No remeshing required for the neighbouring cells of the interface
  - Generalization needed:
    - cyclicGGI
    - partialGGI
    - mixingPlaneGGI
  - On-going validation
Interfaces

- Generalization of GGI using simple transformations

<table>
<thead>
<tr>
<th>GGI</th>
<th>cyclicGGI</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Diagram of GGI" /></td>
<td><img src="image2" alt="Diagram of cyclicGGI" /></td>
</tr>
<tr>
<td>- Not 1-to-1 abutting</td>
<td>- Generalization of cyclic for not 1-to-1 abutting</td>
</tr>
<tr>
<td>- Weighted interpolation across interface</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>partialGGI</th>
<th>mixingPlaneGGI</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image3" alt="Diagram of partialGGI" /></td>
<td><img src="image4" alt="Diagram of mixingPlaneGGI" /></td>
</tr>
<tr>
<td>- Not 1-to-1 abutting</td>
<td>- Circumferential averaging across the interface</td>
</tr>
<tr>
<td>- Weighted interpolation</td>
<td></td>
</tr>
<tr>
<td>- Partial overlap with cyclic</td>
<td></td>
</tr>
</tbody>
</table>
Support for the CGNS file format

- AIAA standard for CFD data (www.cgns.org)
- Archival and data exchange between tools
- Importing and exporting:
  - Mesh, solution, BC
  - Interface:
    - Automatic reconstruction of 1-to-1 cyclic
      - Relaxed criteria for matching faces
      - Cyclic patches for non-planar surfaces
    - GGI: on-going activity
  - Converters:
    - cgnsToFoam
    - foamToCGNS
High-performance computing with OF

➢ Recent acquisition of a supercomputer at IREQ
  - IBM 1350 Beowulf cluster
  - 500 AMD Opteron 64-bit CPUs (1000 cores)
  - 250 x 8 GB = 2 TB of distributed RAM
  - 30 TB of fast storage (IBM GPFS parallel file system)
  - Infiniband 4x interconnect (10 Gbps)
  - Water-cooled rear door heat exchangers
  - Running NPACI Rocks 4.2.1 (Centos 4.4)

➢ OpenFOAM will be an important application for this cluster:
  - Optimization loop
  - Unsteady simulations
  - Fluid-structure interactions
  - LES in diffuser
  - Free surface simulations
High-performance computing with OF
Hydro-Québec and the GNU GPL

- The GNU General Public License (GPL) covers the activities of copying, distributing and modifying software source code.
- Users of software covered by the GPL are free or have the rights to:
  - Execute the software, for whatever purpose.
  - Have access to the source code.
  - Redistribute copies of the software, including the source code.
  - Modify the software and distribute the modifications to the public.
- The GPL does not extend to the input data nor to the results generated with GPL covered software.
- The GPL will automatically extend to libraries or source code linked with the original software.
- For more information about the GPL:
Hydro-Québec and the GNU GPL

- Hydro-Québec wants to protect the Intellectual Property coming from its R&D activities, including software source code.
- By default, we cannot “give away” our source code.
- HQ R&D CFD team wants to contribute actively to the development of OpenFOAM.
- We strongly believe in the added value of collaboration through Open Source development.
- We had to explain the ramifications of the GPL to HQ R&D managers and researchers, and we got the approval to share our work related to OpenFOAM with the community.
Future development

- Optimization loop with OpenFOAM as the principal computing engine
- Balancing the allocation of large computing resources between parallel and distributed processing
  - Partitioning parameters
  - Available computing resources
  - Size of problem
- Source code contributions
Validating OpenFOAM for Turbomachinery

- **Testcases:**
  - ERCOFTAC conical diffuser
  - Turbine 99 draft tube
  - Draft tube

- **Computational parameters:**
  - divScheme for momentum equation:
    - limitedLinearV
    - linearUpwind
  - application:
    - simpleFoam
    - transientSimpleFoam
ERCOFTAC conical diffuser
Turbine 99 Workshop 3

http://www.turbine-99.org
**T99-w3, Engineering quantities**

- **Graphs:**
  - The first graph shows the comparison between OpenFOAM and CFX-5 experiments for the elbow and corner regions, with distance in meters on the x-axis and a value on the y-axis.
  - The second graph illustrates the comparison of C_prm values between OpenFOAM and CFX-5 for different regions labeled Ia, Ib, II, III, and IVa.

**Notes:**
- Engineering quantities are highlighted in the text, indicating specific values or parameters relevant to the study.
- The graphs provide a visual representation of the performance of OpenFOAM compared to CFX-5 in different scenarios.

---

**Second OpenFOAM Workshop**
**Zagreb 7-9 June 2007**

---
2 runner configurations:
- original runner
- modified runner

near best efficiency for the hydraulic turbine.
Draft tube

- application: simpleFoam
- divSchemes:
  - div(\(\phi, U\)): limitedLinearV1

original runner | modified runner

residual stagnation
Draft tube - simpleFoam

![Graph showing pressure recovery coefficient vs length for CFX-5 and OpenFOAM with lines for modified and original runner](image-url)

- **CFX-5**
- **OpenFOAM**

- Modified runner
- Original runner
Draft tube

- application: transientSimpleFoam
- divSchemes:
  - div(\phi, \mathbf{U}): limitedLinearV 1
- ddtSchemes: Euler
Draft tube – transientSimpleFoam

unsteady phenomena
quasi-periodic
Draft tube - OpenFOAM
Draft tube - OpenFOAM

![Graph showing pressure recovery coefficient vs length for different runner configurations using OpenFOAM.](image-url)
Draft tube – CFX-5