OpenFOAM Turbomachinery Working Group:

Progress report

Maryse Page

Hydro-Québec, Research Institute

Håkan Nilsson

Chalmers University of Technology

Third OpenFOAM Workshop
Milano, 10-11 July 2008

About the OpenFOAM Turbo WG

- **Initiated at the Second OpenFOAM Workshop in Zagreb, June 2007**
- **Responsibles:**
  Maryse Page, Martin Beaudoin (Hydro-Québec) and Håkan Nilsson (Chalmers)
- **Homepage:**
  http://openfoamwiki.net/index.php/Sig_Turbomachinery
- **Subversion repository:**
  http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Breeder/OSIG/TurboMachinery/
- **Contact:**
  openfoam-extend-turbowg@lists.sourceforge.net

Objectives of the Working Group

- Identify common interests with OpenFOAM for turbomachinery, and plan joint activities
- Develop OpenFOAM for turbomachinery applications, including pre-processing, solution methods, and post-processing.
- Provide tutorials on how to produce accurate results using OpenFOAM in turbomachines.
- Distribute relevant validation test cases and corresponding OpenFOAM applications.
- Use OpenFOAM to develop Best Practice Guidelines for CFD in turbomachines.
- Connect people with the same interest: OpenFOAM and turbomachinery.
- Organize meetings, workshops and collaborations

Contribution to OpenFOAM Wiki

- **Developments**
  - **Descriptions of contributed solvers, utilities and libraries.**

    → Source code in the svn.
- **Tutorials**
  - **How to implement (new application, boundary condition, turbulence model)**
  - **Cylindrical coordinate systems**
- **Validation test cases**
  - **ERCOFTAC conical diffuser**
  - **Link to Turbine-99 draft tube**
- **List of publications**

OpenFOAM-extend on SourceForge.net

- **Contributions to trunk/Breeder/OSIG/Turbomachinery:**
  - **applications:**
    - **mesh/conversion/cgnsToFoam**
    - **postProcessing/dataConversion/foamToCGNS**
    - **preProcessing/addSwirlAndRotation**
  - **src:**
    - **OpenFoamTurbo/finiteVolume/fields/fvPatchFields/derived/profile1DfixedValue**
  - **Case-studies/tutorials/validation:**
    - **ercoftacConicalDiffuser**

Work in progress

- **Development and testing:**
  - **General Grid Interfaces (GGI)**
  - **Mixing plane interfaces**
  - **Filtered k-ω SST turbulence model**
  - **cavInterFoam – interFoam with mass-transfer**

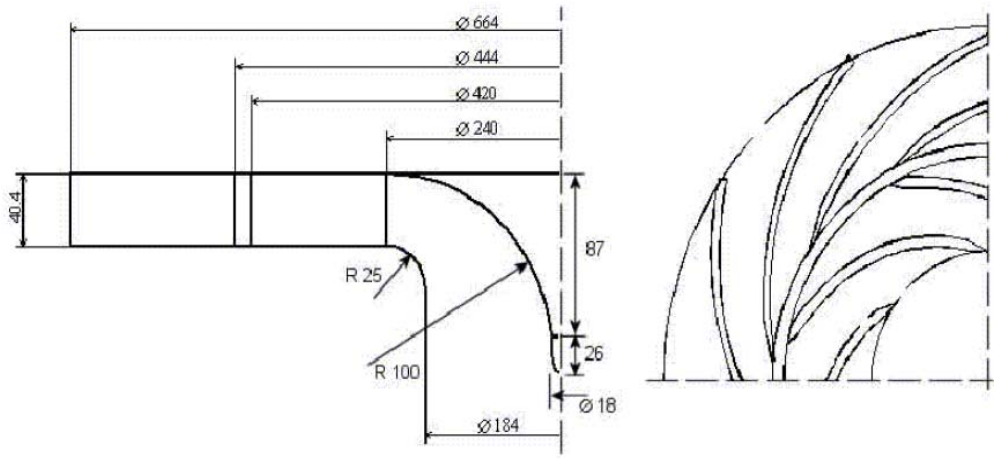
- **Automatic parallel benchmarking for hydraulic turbines applications on large clusters**

Case study: ERCOFTAC Conical Diffuser

- Instructions and comments on OpenFOAM Wiki
- All files available on OpenFOAM-extend
  - cases set-up
  - experimental data
  - applications, libraries
- Mesh parametrization for blockMesh using m4 (O-grid, radial grid)
- Automatic post-processing (sample, gnuplot)
- Space for documentation

Next workshop

- Turbomachinery session
- OF Turbo lecture? Extra OF Turbo day?
- Next case study:
  - ERCOFTAC centrifugal pump



M. Ubaldi, P. Zunino, et al., *An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow*,
ASME Journal of Turbomachinery, 1996.

Next steps for the working group

- **Selection/suggestion of relevant validation test cases:**
  - **Cavitation modelling in Turbo (test case?)**
  - **Dellenback swirl combustor**
  - **Other suggestion ??**

- **Looking for active participation/involvement from others !!**