IAHR
24th Symposium on Hydraulic Machinery and Systems
OCTOBER 27-31, FOZ DO IGUASSU

THE OPENFOAM TURBOMACHINERY WORKING GROUP, AND CONCLUSIONS FROM THE TURBOMACHINERY SESSION OF THE THIRD OPENFOAM WORKSHOP

AUTHORS

Håkan Nilsson[1], Maryse Page[2], Martin Beaudoin[2], Bernhard Gschaider[3], Hrvoje Jasak[4]

[1] Chalmers University of Technology, Sweden
[2] Hydro Québec, Canada
[3] ICE Strömungsforschung, GmbH, Austria
[4] Wikki Ltd., Great Britain
openfoam-extend-turbowg@lists.sourceforge.net

ABSTRACT

The OpenFOAM CFD toolbox was released as OpenSource December 10, 2004, and since then the number of users throughout the world has increased dramatically. Many industries concerned with turbomachinery (water turbines) are investing a lot of money each year in commercial CFD solvers. The availability of cheap hardware makes it possible to do simulations on a large number of CPUs, which requires many expensive software licenses. There is thus a need for a high quality CFD tool that is cheap, and OpenFOAM is the first tool to meet those demands. OpenFOAM has many of the features that are available in the commercial CFD codes, and due to the OpenSource distribution under the GPL licence it can be used at no cost. There is however a need to develop and maintain some of the features that are specifically needed for turbomachinery applications, and for that reason a turbomachinery Working Group of OpenFOAM users was formed at the second OpenFOAM workshop in Zagreb 2007. The first task of the working group has been to contribute to the set-up of the OpenFOAM-extend project at SourceForge, which is used as a platform for the collaborative effort within the working group, as well as a way to distribute the work to anyone who need to take part of it. The second task of the working group has been to organize a turbomachinery session at the third OpenFOAM workshop in Milano 2008. At that workshop the ERCOFTAC conical diffuser was studied as a validation testcase for OpenFOAM.

The present paper describes the OpenFOAM CFD toolbox, and the features that are of interest to the water turbine community. It will further present the OpenFOAM Turbomachinery Working Group, the ERCOFTAC conical diffuser validation testcase for OpenFOAM, the conclusions from the turbomachinery session of the third OpenFOAM workshop, and the OpenFOAM-extend project at SourceForge.

KEY WORDS: CFD, OpenFOAM, Turbomachinery, Water Turbine, Diffuser, Subversion

INTRODUCTION

OpenFOAM is an OpenSource library of C++ routines that is mostly used to implement CFD solvers, although it is not limited to those kinds of applications. The code is based on the finite-volume method, but there are also facilities for doing coupled simulations with both the finite volume method and the finite element method. The computational mesh may consist of any kind of polyhedral cells, and the code accepts fully unstructured meshes. Many advanced features can be found in OpenFOAM, such as moving meshes, conjugate heat transfer, and fluid-structure interaction. With regard to basic features, such as turbulence models and discretization schemes, OpenFOAM has a set of those similar to any commercial CFD solver. With OpenFOAM it is however easy to add any modification to any part of the implementation. With commercial CFD solvers one is limited to add implementations through user-defined-functions, which is a strong limitation. With respect to CFD in the field of turbomachinery there are some features lacking in OpenFOAM. There is a need for a set of boundary conditions that makes it easy to set those in a similar way as it can be done in some other CFD solvers. Automatic post-processing specific for turbomachinery results is needed. There is also a need for further development of different aspects of rotating geometries, and interaction between rotating and stationary parts of the geometry. With this in mind, the OpenFOAM Turbomachinery Working Group was initiated at the second OpenFOAM workshop in Zagreb, 2007. The purpose of the group is to develop the features that are needed for turbomachinery simulations, and to help new users learn how to use OpenFOAM to produce high-quality results for turbomachinery applications. The ERCOFTAC conical diffuser testcase was chosen as the first validation testcase of relevance for turbomachinery. The results from the initial studies of this case were presented at the third OpenFOAM workshop in Milano, 2008, and all the files are available at the OpenFOAM-extend project at SourceForge, for anyone who would like to evaluate OpenFOAM, or learn how to use it. Some of those results, and some conclusions from the turbomachinery session of the workshop in Milano are described in the present paper. A description of the OpenFOAM-extend project at SourceForge is also made in order to help people get started to use it.

THE OPENFOAM TURBOMACHINERY WORKING GROUP

The OpenFOAM Turbomachinery Working Group was initiated at the Second OpenFOAM Workshop in Zagreb, June 2007. Responsibles for the group are Maryse Page, Martin Beaudoin (Hydro-Québec) and Håkan Nilsson (Chalmers). The objectives of the group are to:

- 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

The group can be reached by an e-mail to openfoam-extend-turbowg@lists.sourceforge.net. The homepage can be found at http://openfoamwiki.net/index.php/Sig_Turbomachinery, and a Subversion repository can be found at http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Breeder/OSIG/TurboMachinery/.

The group contributes OpenFOAM implementations, descriptions of the OpenFOAM code, tutorials, and validation testcases for turbomachinery applications. Current main contributions are converters back-and-forth between the OpenFOAM format and the CGNS format, a utility for setting swirl and rotation to patches and internal cells, and a boundary condition for axi-symmetric profiles. Current work in progress are the development of a General Grid Interface (GGI), mixing plane interfaces, a filtered k-ω SST turbulence model [1], and a solver for cavitation modeling using the Volume of Fluid (VOF) method and different mass-transfer models [2,3]. Automatic parallel benchmarking for hydraulic turbine applications on large Linux clusters is another topic that is currently being discussed in the group.

An important part of the work in the group is to set up validation testcases. In the next section, the ERCOFTAC conical diffuser testcase will be described in detail. It was chosen as a common testcase for the turbomachinery session of the third OpenFOAM workshop in Milano, July 10-11, 2008. Other testcases that are currently under consideration are the ERCOFTAC centrifugal pump [4], and the Dellenback combustor [5]. The testcases can also be used as tutorials on how to use OpenFOAM for turbomachinery applications.

THE ERCOFTAC CONICAL DIFFUSER TESTCASE

The ERCOFTAC conical diffuser testcase [6] is a swirling boundary layer developing in a conical diffuser (axi-symmetric expansion of a circular pipe). The experimental set-up is such that the inlet swirl prevents boundary layer separation in the diffuser, but is just insufficient to cause recirculation in the core of the flow. The experimental results are available in the ERCOFTAC Classic database (Case 60: Swirling Boundary Layer in Conical Diffuser). The testcase was included in the ERCOFTAC Workshop on Data Bases and Testing of Calculation Methods for Turbulent Flows held in Karlsruhe in 1995 [7]. The testcase is also part of the QNET-CFD Knowledge Base. Figure 1 shows the geometry as it was presented by Clausen et al.. (left), and as the geometry was described for the workshop in Karlsruhe (right). A swirling flow was created by a rotating cylinder with a honeycomb screen at its inlet. The inlet velocity distribution is close to a a plug flow with a solid body swirl. The axial pressure gradient and curvature of the streamlines have been found to be the dominant perturbations imposed to the swirling boundary layer as it exits the cylindrical part and enters the conical diffuser. The swirl is responsible for severe radial gradients near the wall for most of the turbulence quantities. In the experiments it was open air after the end of the expansion.

Measurements were taken in traverses normal to the wall, as shown in Figure 1. The mean velocity profiles were measured using a single wire hot-wire anemometer, with an estimated error of 2%. The Reynolds stresses were measured using an X-wire probe, with an estimated error of 10%. The wall shear stress was estimated using log law of the wall. The static pressure was measured using wall taps.

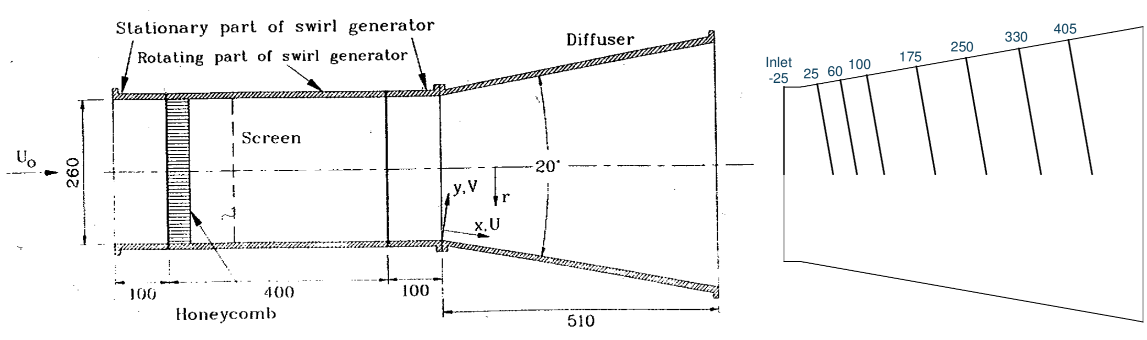

Figure 1 Description of the ERCOFTAC conical diffuser experimental setup, and the measurement traverses.

Several two-dimensional flow computations were submitted to the ERCOFTAC workshop in 1995 [7]. Two-dimensional computations [8] and three-dimensional [9] computations using the experimental data at the inlet of the conical diffuser were also published in 1996 and 1997. More recently, results were presented using OpenFOAM at the Second OpenFOAM Workshop [10].

Computations that were closer resembling the experimental set-up were presented in 2006 [11]. Experimental results related to the same conical diffuser were also presented in 1987 by Clausen and Wood in the conference paper [12]. Prediction of turbulence quantities for this swirling flow were presented by Armfield, Cho and Fletcher in [13]. Bounous [14] used OpenFOAM-1.4.1 for a thorough investigation of the conical diffuser using different choices of geometrical extensions, block topologies, linear solvers, discretization schemes, turbulence models, and inlet velocity profiles.

The ERCOFTAC conical diffuser was used as a case-study for the third OpenFOAM workshop in Milano, July 10-11 2008. 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 different m4-preprocessor parametrizations files of the geometry and the grid to generate the input files for the blockMesh mesh generator (distributed within OpenFOAM), complete OpenFOAM cases that solves the flow in the domain, automatic post-processing of the results and comparison with measurements, and xfig files for generation of the figures in this section. The cases that are currently available are listed here, and the geometries are shown in Figure 2.

- Case0: Base case
- Case1: Extended base case
- Case1.1: Radial mesh
- Case1.2: MFRSimpleFoam
- Case1.3: 2D wedge mesh
- Case2: Case1 with a dump
- Case2.1: Inlet velocity profile
- Case2.2: Radial mesh

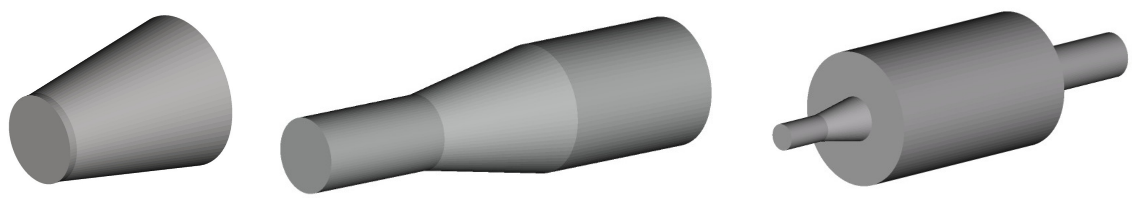

Figure 2 Geometries of Case0, Case1 and Case2

By default, the cases are run with the simpleFoam solver, which is a steady state solver with RANS turbulence models. In Case1.2 the MRFSimpleFoam solver is used, which is a similar solver, but for multiple reference frames. Turbulence is by default modeled using the standard k-ε model with wall functions. The first-node wall distance is approximately $y^+=26$. A second-order upwind scheme is used for the velocity, and a first-order upwind scheme is used for the turbulence quantities. The inlet boundary condition is set either by a utility that adds swirl and rotation to patches, or by a boundary condition that interpolates the values from a 1D profile. For Case0 the inlet velocity and turbulent kinetic energy is interpolated from the experimental results, and the eddy dissipation is computed to yield a ratio between the turbulent viscosity and the laminar viscosity as $\mu_T/\mu=14.5$. For the other cases turbulence is specified using a turbulent length scale of 0.0032m and a turbulent intensity of 10% ($\mu t/\mu=27.3$). A Neumann boundary condition is used for all other boundary conditions, except for the outlet pressure which is set to zero, and no-slip at walls. There are examples attached with the cases, showing how to use other kinds of outlet boundary conditions for the pressure.

Figure 3 shows a view of the axi-symmetric computational domain of Case1, which includes extensions before and after the diffuser. The cross-sections refer to A: The beginning of the honeycomb and the inlet of the computational domain, B: The edge of a rotating part of the wall, C: A measurement plane, D: The inlet of the expansion, E: The outlet of the expansion, F: The outlet of the artificial extension and the outlet of the computational domain. At the original ERCOFTAC workshop, it was recommended to compute only the part between cross-sections C-E, and to use the experimental values at the inlet. The case named Case0 follows these recommendations. Case1 extends the computational domain to include the swirl generator, and to move the outlet boundary condition further away from the region of interest. The reasons for this are to reduce the need for experimental data for the inlet boundary, and to better resemble the outlet flow into open air. An inlet plug-flow with a solid body rotation is used to resemble the flow after the honeycomb, and the walls between cross-sections A and B are rotating with the honeycomb. The parametrization of the cases allows the user to modify both the geometry and the computational mesh. Some of the parameters are shown in Figure 3.

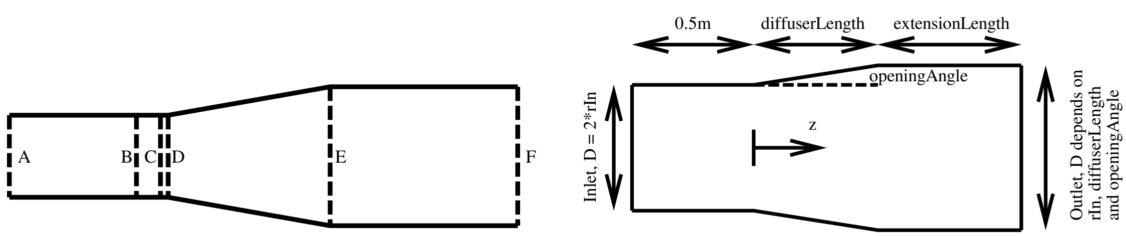

Figure 3 Parametrization of Case1. Cross-sections A-F are used to define the mesh topology at those locations. The geometry may easily be modified using the parameters shown in the right figure.

Figure 4 shows the three different cross-section mesh topologies that are currently available for Case1: an O-grid, a radial mesh and a 2D wedge mesh. The parameters allow the user to modify the mesh distribution easily at each cross-section plane (A-F).

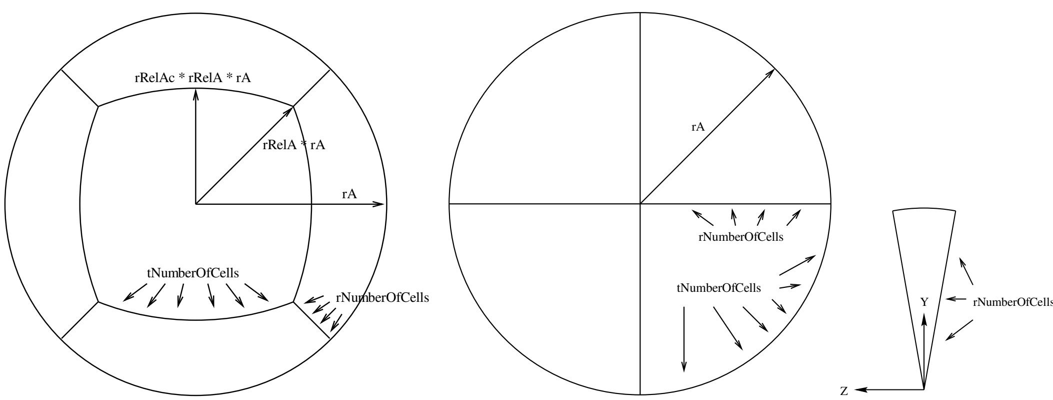

Figure 4 Cross-section mesh topology parametrization for Case1. Left: O-grid topology, center: radial mesh, right: 2D wedge mesh.

Figure 5 shows the geometry and parametrization for Case2, which is an extension of Case1 to further resemble the real flow, with an outlet to open air. At the outlet of the diffuser there is a large dump, where the swirling flow may undergo a vortex breakdown. An outlet pipe is added to accelerate the flow in order to have a well-defined outlet flow direction. The rest of Case2 is similar to Case1, and there are both O-grid and radial mesh topologies available. The geometry and O-grid mesh of Case2 is similar to that used by Gyllenram and Nilsson [11].

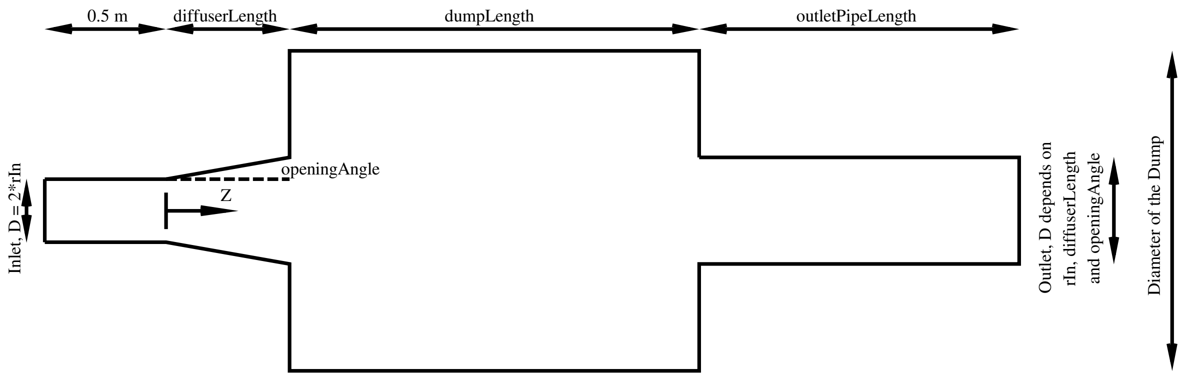

Figure 5 Geometry parametrization for Case2. Available cross-section topologies are O-grid, and radial.

The testcases are accompanied with automatic post-processing routines for validation against measurements, and view of the residuals.

Figure 6 shows an example of a validation of the computational results for standard cases Case0, Case1 and Case2, at cross-sections 25mm and 405mm downstream the entrance of the diffuser. The results correspond well with the measurements. A detailed study of these cases, and variations of the cases, was done by Bounous [14]. These studies concluded that the solution is independent from the linear solver, that the solution had a dependence on the order of accuracy of the discretization scheme (1st order upwind insufficient), that the k – ε and k – ω SST turbulence models give the same velocity results but different turbulent kinetic energy, and that the difference in the results of different cross-section grid topologies is negligible

A study of the effects due to the rotation in the multiple-frame-of-reference solver was presented at the workshop by Olivier Petit. For rotational speeds up to 100rpm those simulations yield the same results as for Case1. However, for a rotational speed of 1000rpm the results deteriorate severely, and the convergence is highly affected.

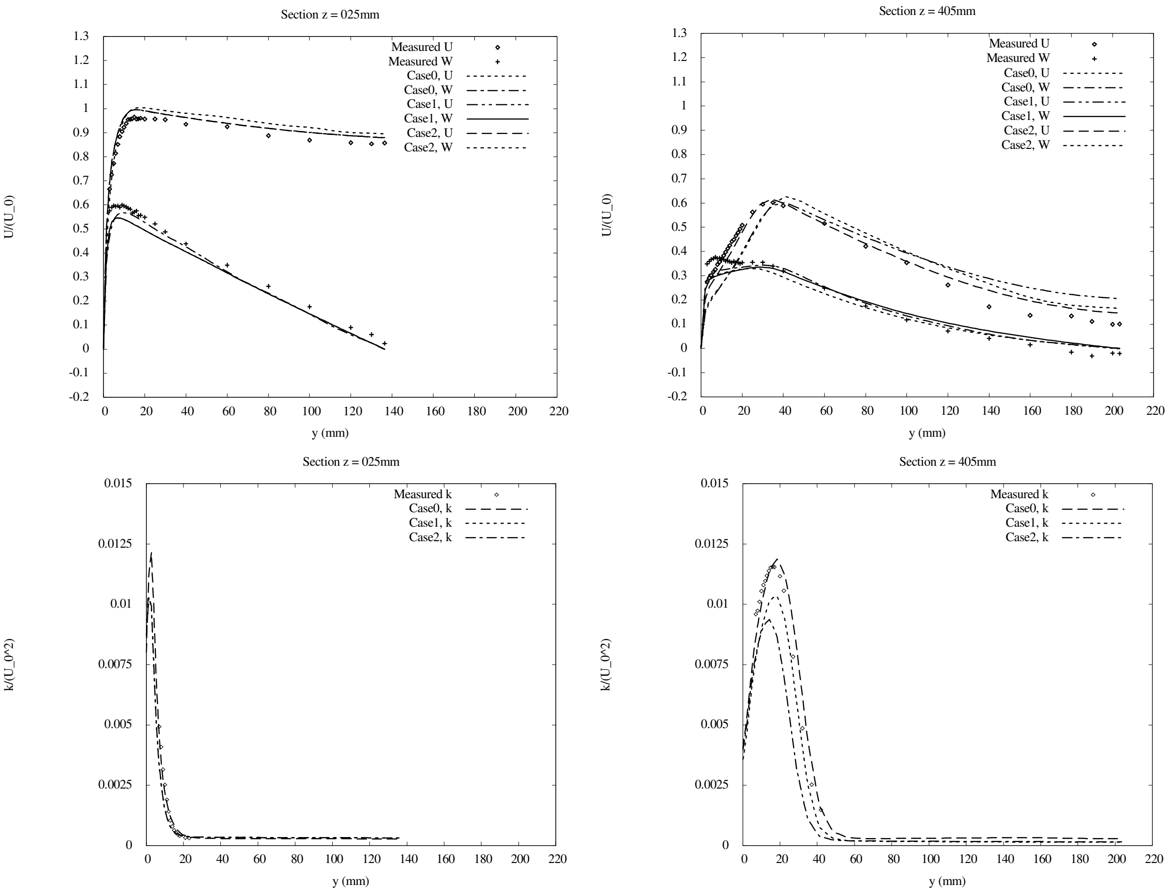

Figure 6 Examples of validation of the computational results for standard cases Case0, Case1 and Case2. Coordinate y=0 at the diffuser wall. At section z=025, Case1 and Case2 yield the same results.

THE TURBOMACHINERY SESSION OF THE THIRD OPENFOAM WORKSHOP

In addition to the results for the ERCOFTAC conical diffuser, some other interesting aspects of the use of OpenFOAM for turbomachinery applications were presented in the turbomachinery session of the third OpenFOAM workshop. Beaudoin and Jasak, Hydro Québec, Canada, and Wikki Ltd, Great Britain, presented a development of a General Grid Interface (GGI), which may be used for connecting non-conformal grids, cyclic interfaces, frozen rotor, and stage interfaces [15]. Duprat and Métais, LEGI/MoST, Grenoble, presented LES simulations for swirling flow in pipes, and in the ERCOFTAC conical diffuser. They proposed an implementation of an alternative near-wall modeling, and a method for generation of swirling turbulent flow at the inlet boundary. Mangani and Bianchini, University of Florence, Italy, presented work in the field of turbo-gas engines. They presented an alternative approach to the implementation of a GGI, improvements for steady-state compressible flow simulations, developments of an implicit method for conjugate heat transfer, boundary conditions specific for simulations of turbo-gas engines, and a mixing plane implementation. Blaim, Borm, Fröbel and Kau, Technische Universität, München, presented the use of virtual patches for the implementation of a mixing plane interface in OpenFOAM.

The turbomachinery working group has now started the preparations for the fourth OpenFOAM workshop, and further contributions to the OpenFOAM turbomachinery community.

THE OPENFOAM-EXTEND PROJECT AT SOURCEFORGE.NET

The OpenFOAM Turbomachinery Working Group share files through the OpenFOAM-extend project [16,17] at sourceforge.net (http://sourceforge.net/projects/openfoam-extend). The goal of the

OpenFOAM-extend project is to open the OpenFOAM CFD toolbox to community contributed extensions. The project consists mainly of a Subversion repository for sharing source files, and a Wiki that describes how to use it. Figure 7 shows a graphical view of the structure of the OpenFOAM-extend Subversion repository. The Subversion repository is arranged according to the recommendations of such repositories (http://subversion.tigris.org). The main activites are currently taking place in trunk/Core, trunk/Breeder, tags/Core, and branches/OpenCFD_Release. In trunk/Core, the cutting-edge development version by Professor Hrvoje Jasak can be found. The development version is an extension of the OpenCFD version, and includes many advanced features that are not available in the OpenCFD version. In trunk/Breeder there are user-contributed libraries and applications that can easily be added to your installed OpenCFD version. In tags/Core there are a number of stable versions of the development version. In branches/OpenCFD_Release, the release by OpenCFD can be found, with applied up-to-date bug-fixes. The OpenFOAM Turbomachinery working group is using the trunk/Breeder/OSIG/Turbomachinery directory to add applications, libraries, validation testcases, and tutorials. There is an Allwmake script that compiles all the contributed features automatically. The current code contributions from the OpenFOAM Turbomachinery Working Group are converters between the OpenFOAM and the CGNS formats, a boundary condition for axi-symmetric profiles, and a utility for adding swirl and rotation to patches and interiour cells. The main work in the Turbomachinery Working Group until now has been to add the ercoftacConicalDiffuser testcase.

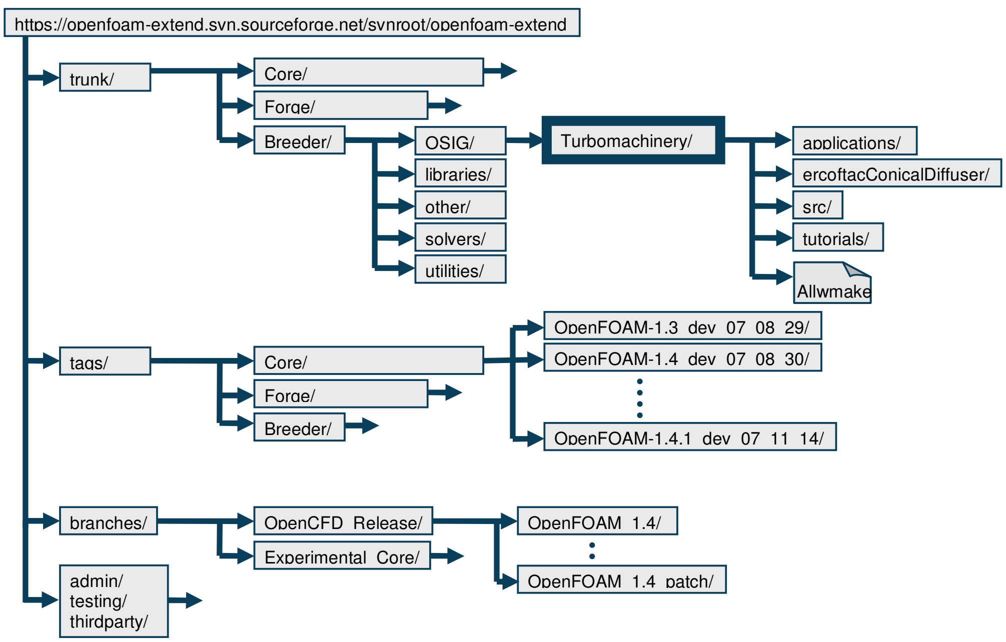

Figure 7 Structure of the OpenFOAM-extend Subversion repository, and the OSIG/Turbomachinery repository

Some of the advantages of using a version control system (not only for the source code, but especially for the comparison cases) are that

- The information can easily be shared and modified by people working far apart, without the risk of losing any intermediate version. In the present work all the code and testcase distribution has been through the Subversion system at SourceForge, and the people have been located in Sweden, Canada, Austria and Great Britain.
- Every modification that has been done can be traced back to its originator, and it is possible to make a short comment of each modification.

CONCLUSIONS

The OpenFOAM Turbomachinery Working Group has been active for one year, and has already contributed both implementations that are useful for turbomachinery simulations, and many variants of a testcase for validation of OpenFOAM as well as for learning how to use OpenFOAM for turbomachinery applications. The validations show that OpenFOAM gives results that are close to the experimental results. The testcase is accompanied with complete setups for the cases, which can be viewed as Best-Practice-Guidelines for those kinds of simulations. OpenSource routines for mesh generation and automatic post-processing are included.

ACKNOWLEDGEMENTS

We would like to express our greatest acknowledgements to Phil Clausen and David Wood for making the experimental results of the ERCOFTAC conical diffuser available. We would also like to acknowledge the OpenFOAM community, SourceForge Inc., and MediaWiki.org. Håkan Nilsson is 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. Some of the simulations presented in this work has been made on clusters financed by SNIC, the Swedish National Infrastructure for Computing, whom we greatfully acknowledge.

BIBLIOGRAPHICAL REFERENCES

[1] W. GYLLENRAM AND H. NILSSON, *Design and Validation of a Scale-Adaptive Filtering Technique for LRN Turbulence Modeling of Unsteady Flow*, May 2008, J. Fluids Eng., Volume 130, Issue 5, 051401 (10 pages), [DOI: 10.1115/1.2911685]
[2] N.WIKSTRÖM, *Approaching Large Eddy Simulation of Cavitating Flows for Marine Applications,* 2006, Doctoral thesis, Chalmers University of Technology, ISBN/ISSN: 91-7291-791-X
[3] T. HUUVA, *Large eddy simulation of cavitating and non-cavitating flow*, 2008, Doctoral thesis, Chalmers University of Technology, ISBN/ISSN: 978-91-7385-063-6
[4] M. UBALDI, P. ZUNINO, ET AL., 1996, *An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow*, ASME Journal of Turbomachinery.
[5] P.A. DELLENBACK, D.E. METZGER, AND G.P. NEITZEL, 1987, *Measurements in Turbulent Swirling Flow Through an Abrupt Expansion*, AIAA J., 26(6), pp. 669–681.
[6] P.D. CLAUSEN, S.G. KOH AND D.H. WOOD, 1993. *Measurements of a swirling turbulent boundary layer developing in a conical diffuser.* Experimental Thermal and Fluid Science, 6:39-48
[7] W.RODI, J.-C. BONNIN AND T.BUCHAL (organizers), April 3-7,1995, *ERCOFTAC Workshop on Data Bases and Testing of Calculation Methods for Turbulent Flows*. Karlsruhe, Germany
[8] M.PAGE, A.-M.GIROUX AND B.MASSÉ, June 2-6, 1996, *Turbulent swirling flow computation in a conical diffuser with two commercial codes*. CFD'96, Fourth Annual Conference of the CFD Society of Canada, Ottawa, Canada
[9] M.PAGE, B.MASSÉ AND A.-M.GIROUX, June 17-19, 1997, *Turbulent swirling flow computations in a conical diffuser*. FIDAP User's Meeting, Burlington, USA
[10] M.PAGE AND M.BEAUDOIN, June 7-9, 2007, *Adapting OpenFOAM for Turbomachinery Applications.* Second OpenFOAM Workshop, Zagreb, Croatia
[11] W. GYLLENRAM AND H. NILSSON, 2006, *Very Large Eddy Simulation of Draft Tube Flow*. In Proceedings of 23rd IAHR Symposium, Yokohama
[12] P.D. CLAUSEN AND D.H. WOOD, Sept 7-9, 1987, *Some measurements of swirling flow through an axisymmetric diffuser*. Proceedings of Sixth Symposium on Turbulent Shear Flows, Paul Sabatier University, Toulouse, France
[13] S.W. ARMFIELD, N.-H. CHO AND C.A.J. FLETCHER, 1990, *Prediction of Turbulence Quantities for Swirling Flow in Conical Diffusers.* AIAA J., Vol.28, No.3, pp..453-460
[14] O. BOUNOUS, 2008, *Studies of the ERCOFTAC conical diffuser using OpenFOAM*, Report, Chalmers University of Technology, Department of Applied Mechanics. Presented at the 3rd OpenFOAM Workshop, Milan, Italy
[15] M. BEAUDOIN AND H. JASAK, July 10-11, 2008, *Adaptation of the Generalized Grid Interface (GGI) for Turbomachinery Simulations with OpenFOAM*. Presented at the 3rd OpenFOAM Workshop, Milano, Italy.
[16] M. BEAUDOIN, B. GSCHAIDER, H. JASAK AND H. NILSSON, June 7-9, 2007, *Open Subversion Reposity Access For OpenFOAM*. Presented at the 2nd OpenFOAM Workshop, Zagreb, Croatia.
[17] M. BEAUDOIN AND B. GSCHAIDER, July 10-11, 2008, *The OpenFOAM-extend project on SourceForge: current status*. Presented at the 3rd OpenFOAM Workshop, Milano, Italy.