

About the OpenFOAM Turbo WG

- Initiated at the Second OpenFOAM Workshop in Zagreb, June 2007.
- Steering committee:
Maryse Page, Martin Beaudoin (Hydro-Québec)
Håkan Nilsson (Chalmers)
- Homepage:
OpenFOAM Wiki:
http://openfoamwiki.net/index.php/Sig_Turbomachinery
- Code sharing:
OpenFOAM-extend project at SourceForge.net :
<http://sourceforge.net/projects/openfoam-extend/develop>
- Join the e-mail list:
openfoam-extend-turbowg@lists.sourceforge.net

Objectives

- 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:
OpenFOAM Workshop WG day: www.openfoamworkshop.org

Contributions to the OpenFOAM Wiki

- Developments:
Descriptions of contributed solvers, utilities and libraries.
Pointers to shared code and cases in OpenFOAM-extend.
- Tutorials:
How to implement (new application, BC, turbulence model)
Cylindrical coordinate systems
- Howtos:
mergeMeshes, GGI, MRFSimpleFoam, etc.
- Validation test cases (from pre-processing to validation):
ERCOFTAC conical diffuser
ERCOFTAC centrifugal pump
Dellenback Abrupt Expansion (in Turbulence WG wiki)
- List of publications.

OpenFOAM-extend on SourceForge.net

Contributions to Turbomachinery branch:

- Source code:
Mesh converters:
cgnsToFoam
foamToCGNS
Pre-processing tools:
addSwirlAndRotation
Specialized BC:
profile1DfixedValue
- Case-studies / tutorials / validation
ercoftacCentrifugalPump
ercoftacConicalDiffuser

Work in progress

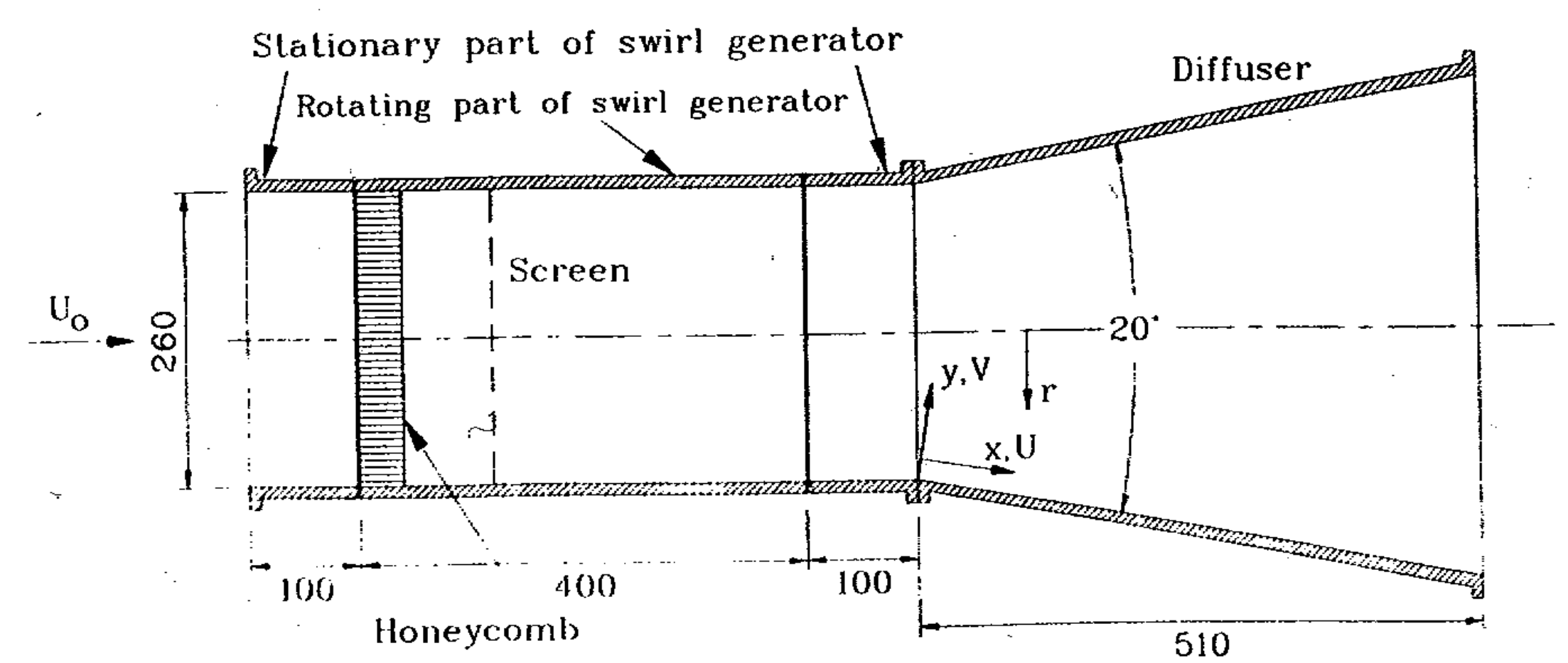
- Development:
Mixing plane interfaces
Automated test-loop
- Validation and testing:
ercoftacCentrifugalPump in 3D
GGI
cyclicGGI
GGI in parallel
- Automatic parallel benchmarking for hydraulic turbine applications on large clusters
- Also, contributions from University of Munich:
Mixing plane implementation
BCs for calculations in rotating reference frame (compr. flow)

Please join and contribute!

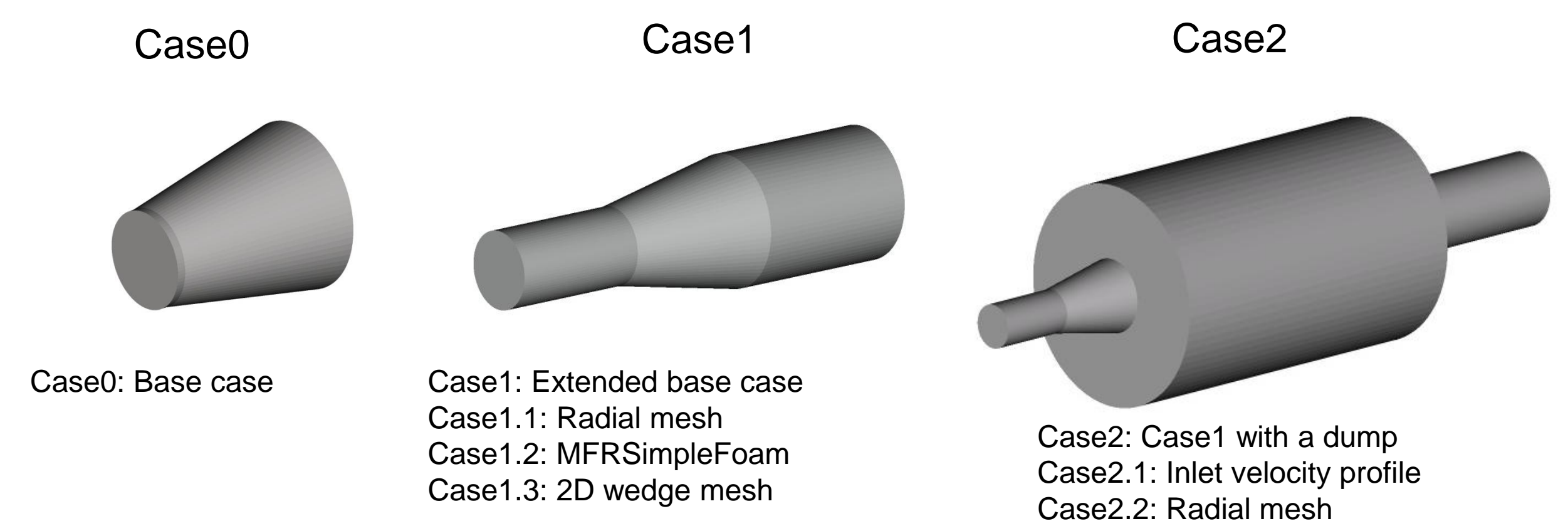
Case-studies

- Instructions and comments in the OpenFOAM Turbo WG Wiki
- All files available on OpenFOAM-extend:
Case set-up
Experimental data
Applications, libraries
- Mesh parametrization for blockMesh using m4 (O-grid, radial grid)
- Automatic post-processing (sample, gnuplot)
- Documentation

ERCOFTAC Conical Diffuser

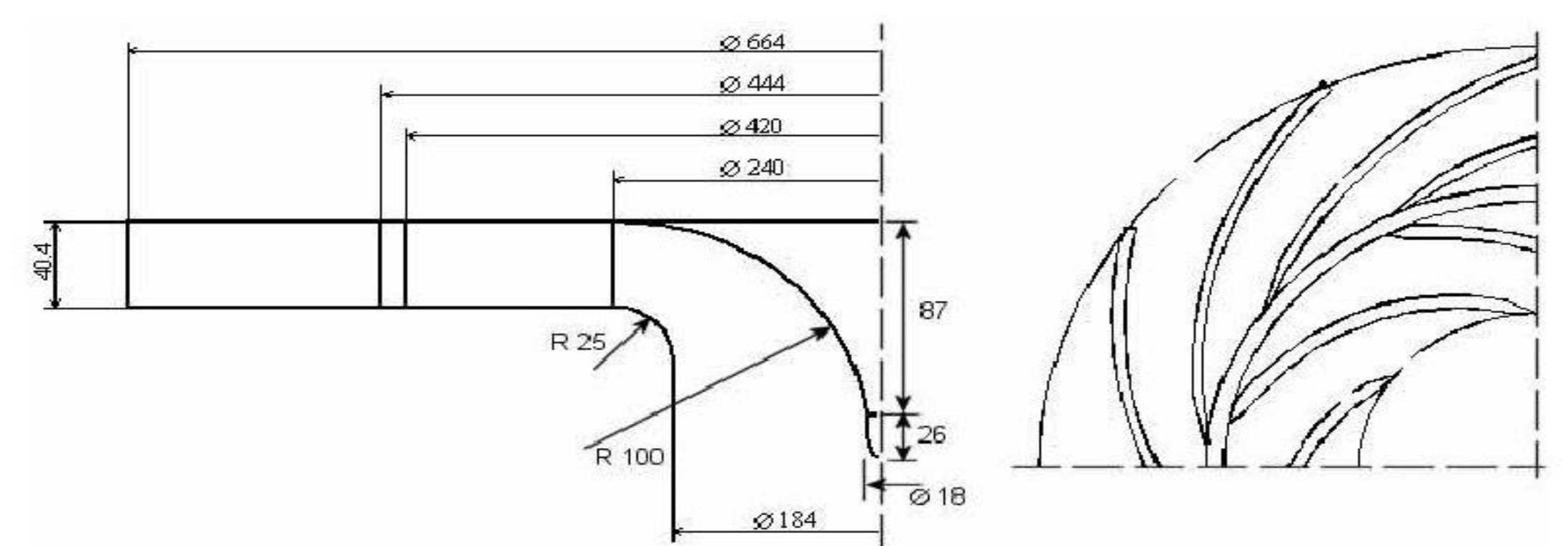


Ref: P.D. Clausen, S.G.Koh and D.H.Wood,
Measurements of a Swirling Turbulent Boundary Layer Developing in a Conical Diffuser,
Experimental Thermal and Fluid Science, 6:39-48, 1993



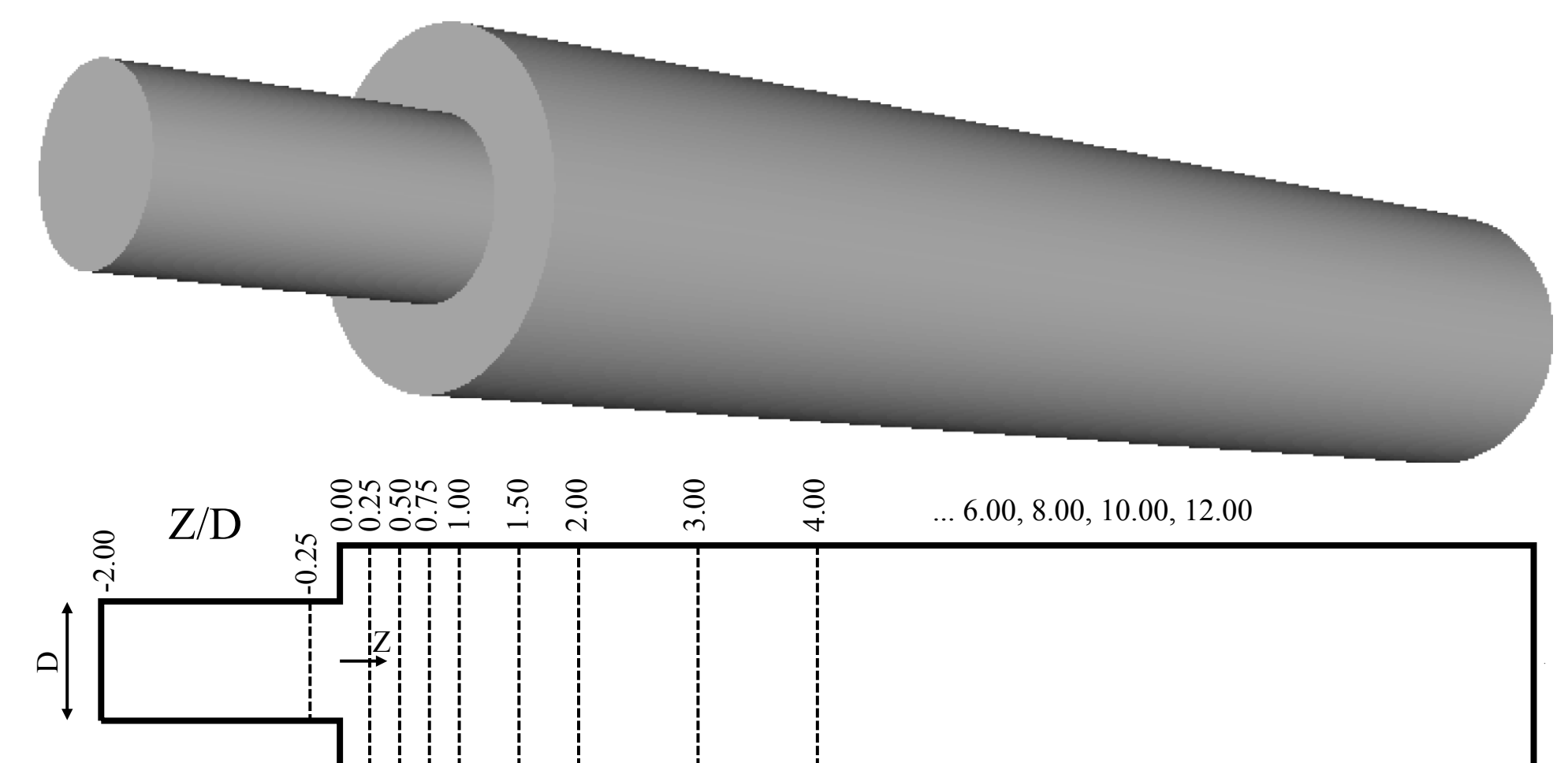
Case0: Base case
Case1: Extended base case
Case1.1: Radial mesh
Case1.2: MRFSimpleFoam
Case1.3: 2D wedge mesh
Case2: Case1 with a dump
Case2.1: Inlet velocity profile
Case2.2: Radial mesh

ERCOFTAC Centrifugal Pump



Ref: M. Ubaldi, P. Zunino, G. Barigozzi and A. Cattanei
An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow,
ASME Journal of Turbomachinery, vol.118, 41-54,1996

Dellenback Abrupt Expansion



Ref: P.A. Dellenback, D.E. Metzger and G.P. Neitzel
Measurements in Turbulent Swirling Flow Through an Abrupt Expansion,
AIAA Journal, 26 (6), 669-681,1987

References

1. H. Nilsson, M. Page, M. Beaudoin, B. Gschaider and H. Jasak, *The OpenFOAM Turbomachinery Working Group, and conclusions from the Turbomachinery Session of the Third OpenFOAM Workshop*, 24th Symposium on Hydraulic Machinery and Systems, Foz do Iguassu, 2008.
2. O. Petit, M. Page, M. Beaudoin and H. Nilsson, *The ERCOFTAC centrifugal pump OpenFOAM case-study*, 3rd IAHR International Meeting of the WorkGroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Brno, Czech Republic, October 14-16, 2009 .