# The OpenFOAM Turbomachinery Working Group, and Conclusions from the Turbomachinery Session of the Third OpenFOAM Workshop


Håkan Nilsson

*Chalmers University of Technology, SVC*

**Maryse Page and Martin Beaudoin**

***Hydro-Québec, Research Institute***

Bernhard Gschaider

*ICE Strömungsforschung*

Hrvoje Jasak

*Wikki Ltd.*



*24th IAHR Symposium*

*October 27-31, 2008, Foz do Iguasso*

# Outline

- OpenFOAM CFD toolbox
- OpenFOAM Turbomachinery Working Group
  - Objectives and contributions
  - Third OpenFOAM Workshop in Milano, June 2008:
    - Code
    - Validation: ERCOFTAC conical diffuser
    - Technical presentations
  - Next meeting: Turbomachinery Workgroup day at Fourth OpenFOAM Workshop in Montréal, June 2009

# About the OpenFOAM CFD toolbox

- OpenFOAM: Open Field Operation and Manipulation
  - An OpenSource object oriented C++ tool for solving PDE's
  - http://www.openfoam.org
- OpenSource:
  - We have insight into the code
  - We can make development and tailor-made solvers
  - We can make research implementations available and results reproducable
  - We can use a large number of CPUs without extra cost
- Access to an international community of OpenFOAM users

# About the OpenFOAM CFD toolbox

- **Preprocessing**:
  - Grid generators, converters and manipulators
  - Case setup
- **Postprocessing**:
  - OpenSource Paraview
  - Access to postprocessing tools (Ensight, etc) through conversion
- **Many *specialized CFD* solvers implemented, e.g.**
  - *simpleFoam*: A finite volume steady-state solver for incompressible, turbulent flow of non-Newtonian fluids, using the SIMPLE algorithm
  - *turbFoam*: A finite volume solver for unsteady incompressible, turbulent flow of non-Newtonian fluids, using the PISO algorithm
  - *icoDyMFoam*: Sliding/moving grid
- **Runs in *parallel* using automatic/manual domain decomposition**

# About the OpenFOAM Turbo WG

- Initiated at the Second OpenFOAM Workshop in Zagreb, June 2007
- Steering committee:
  Maryse Page, Martin Beaudoin (Hydro-Québec)
  and Håkan Nilsson (Chalmers)
- Homepage:
  - OpenFOAM Wiki:

    http://openfoamwiki.net/index.php/Sig_Turbomachinery
- Code sharing:
  - A branch of the OpenFOAM-extend project on SourceForge.net
- 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

# Contributions to OpenFOAM Wiki

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

    → Source code on SourceForge.net
- 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

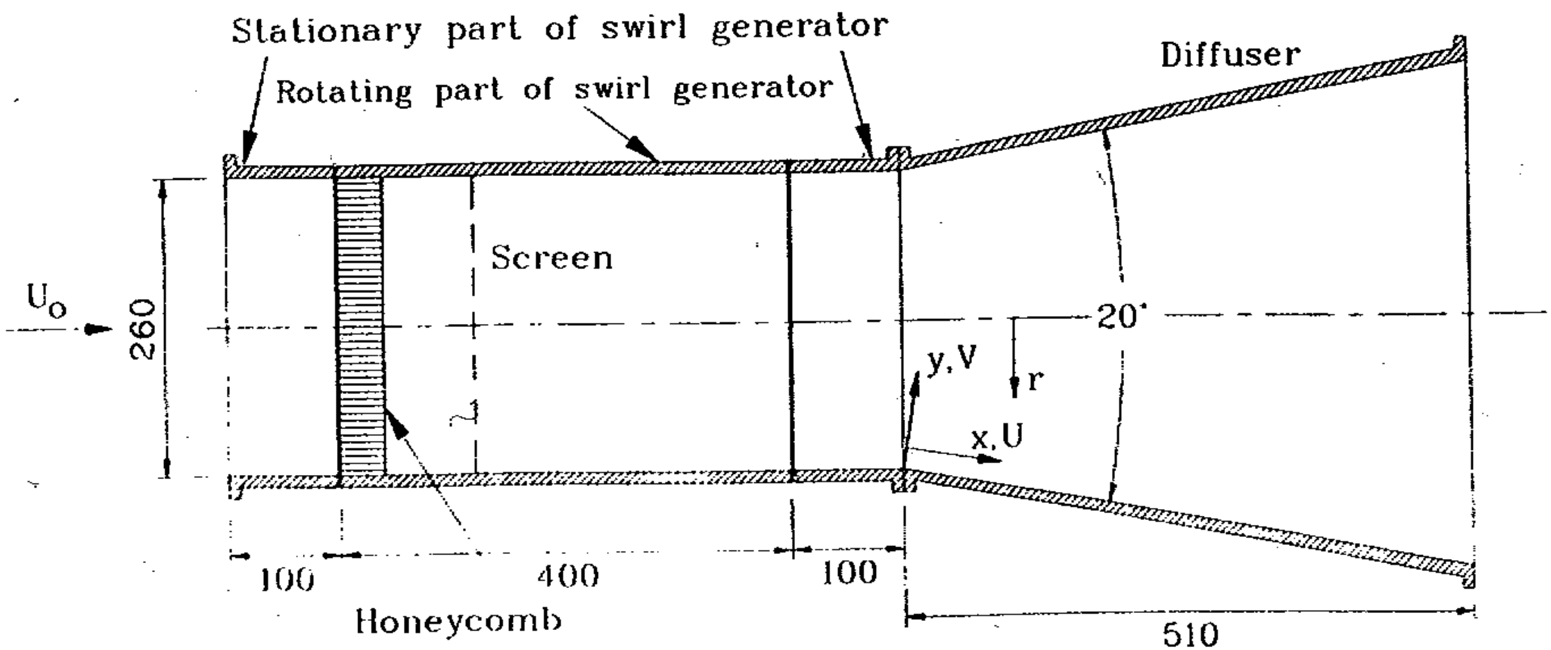
- Contributions to Turbomachinery branch:
  - Source codes:
    - Mesh converters:
      - cgnsToFoam
      - foamToCGNS
    - Pre-processing tools:
      - addSwirlAndRotation
    - Specialized BC:
      - profile1DfixedValue
  - Case-studies/tutorials/validation:
    - ercoftacConicalDiffuser

# Work in progress

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

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

# Workshop Case study: *ERCOFTAC Conical Diffuser*



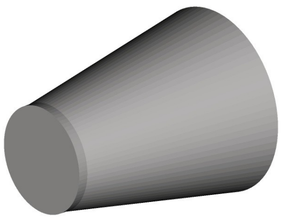

Ref: P.D. Clausen, S.G.Koh and D.H.Wood,
*Measurements of a Swirling Turbulent Boundary Layer Developing in a Conical Diffuser*,
Experimental Thermal and Fluid Science 1993, 6:39-48

# Workshop 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)
- Documentation

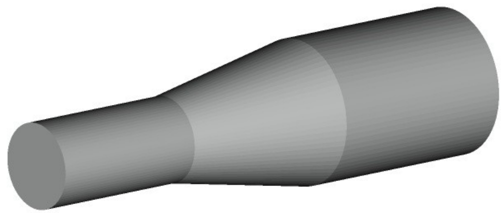
# Cases in OF Wiki and OpenFOAM-extend

Case0



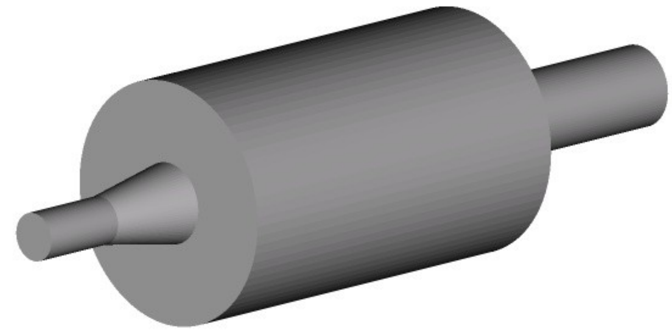
Case0: Base case

Case1



Case1: Extended base case
Case1.1: Radial mesh
Case1.2: MFRSimpleFoam
Case1.3: 2D wedge mesh

Case2



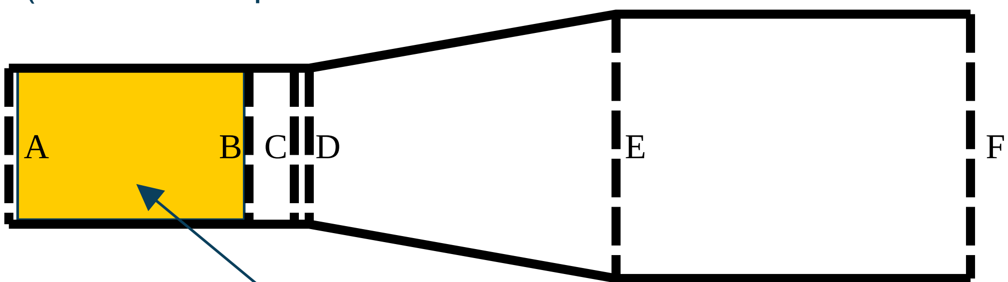
Case2: Case1 with a dump
Case2.1: Inlet velocity profile
Case2.2: Radial mesh

- All cases except Case1.2 use simpleFoam
- k-ε with wall functions, average $y^+$: 25-27
- div(phi,U) Gauss linearUpwind Gauss;
- div(phi,k) Gauss upwind;
- div(phi,epsilon) Gauss upwind;

# Example: Case1.2: Case1 with MRFSimpleFoam

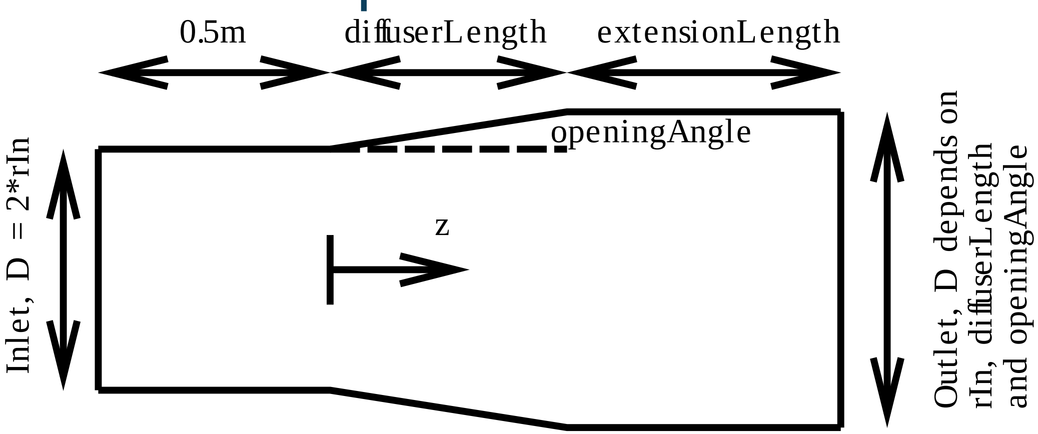
## Cross-sections for mesh control

(there are m4 parameters for axial locations and mesh density)

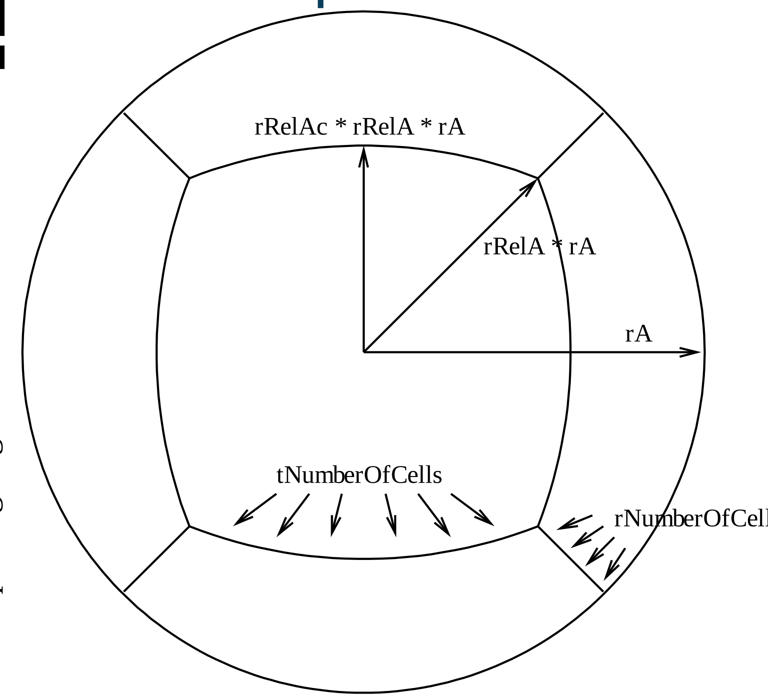



Rotating part (MRF)

## m4 parameters

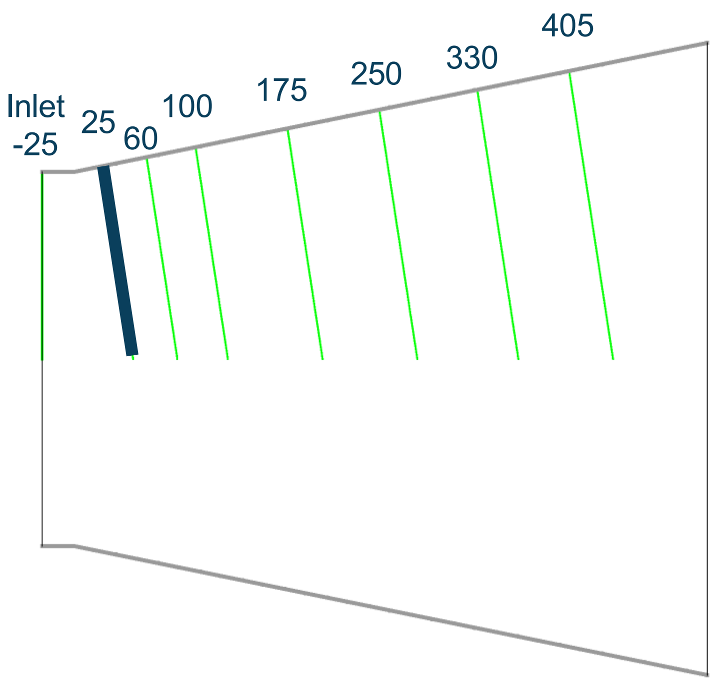


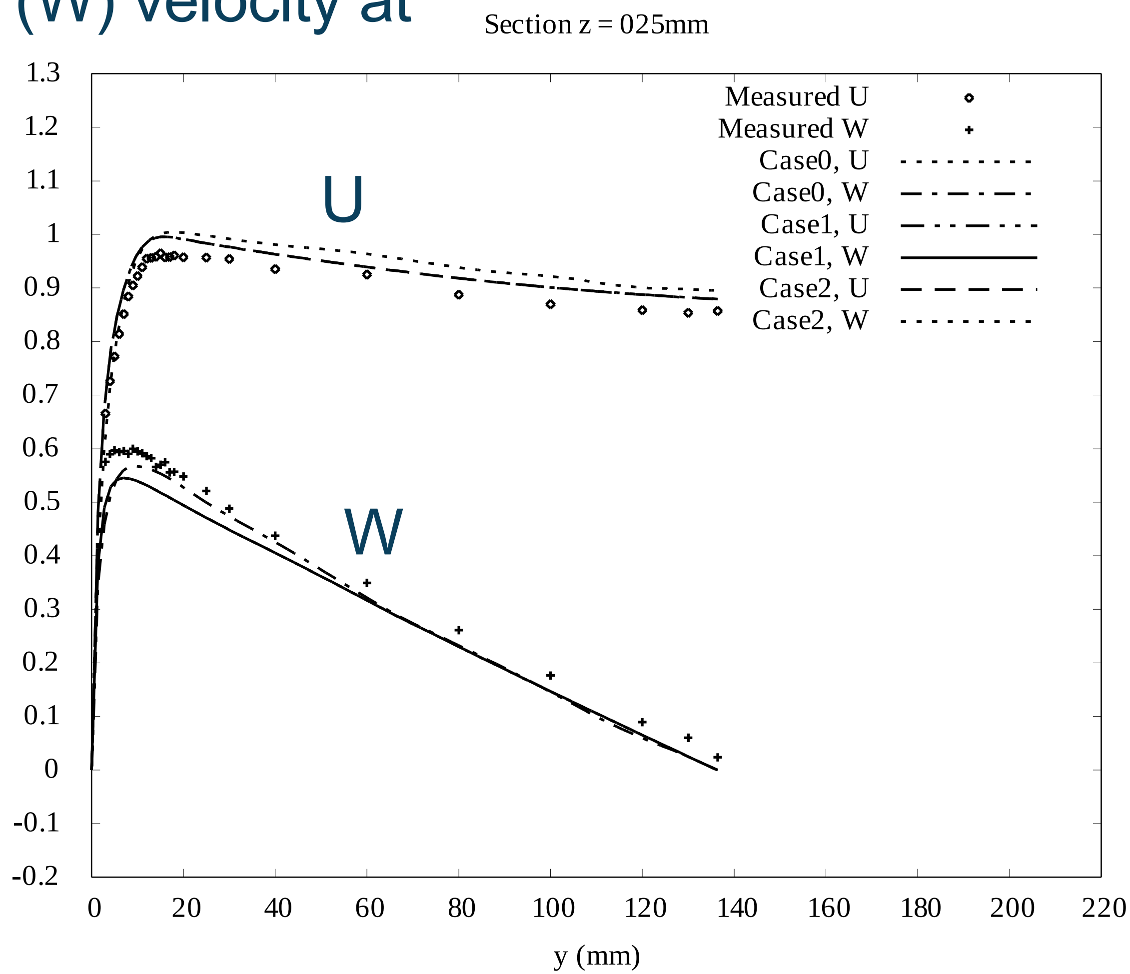

## Cross-section mesh topology and m4 parameters



# Results: ERCOFTAC Conical Diffuser

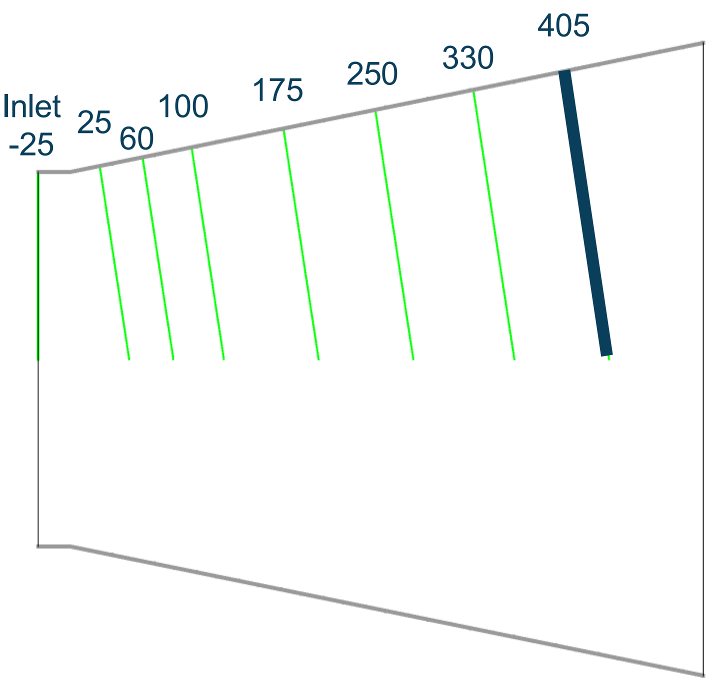
Axial (U) and tangential (W) velocity at Z=25mm, Case0/1/2

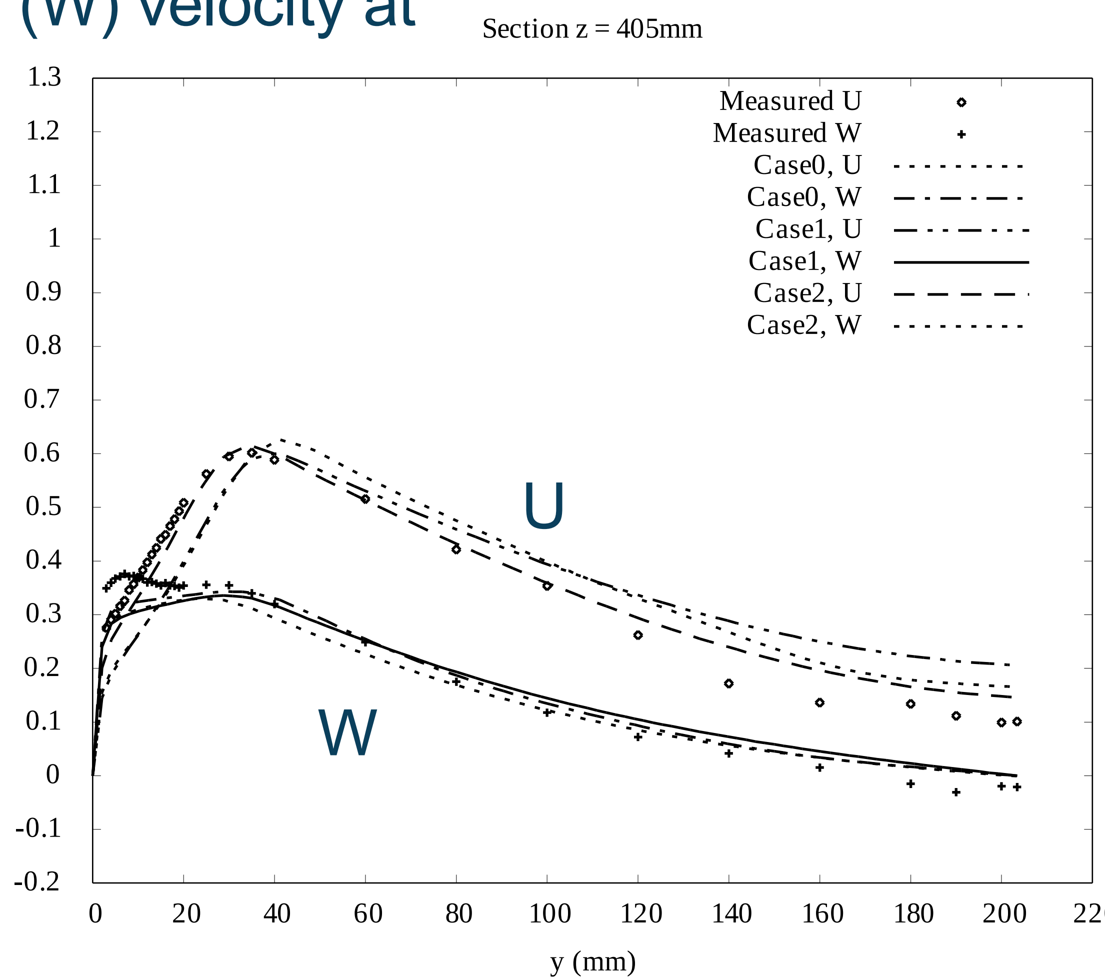


# Results: ERCOFTAC Conical Diffuser

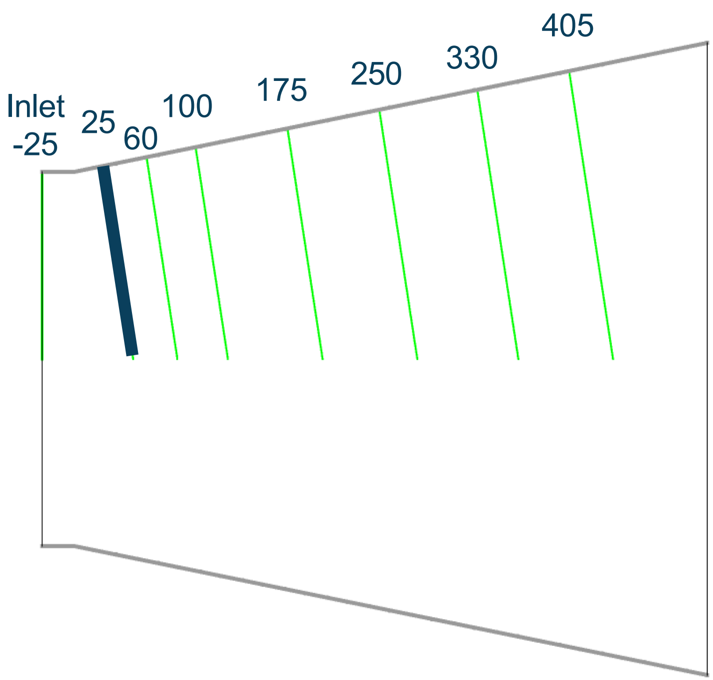
## Axial (U) and tangential (W) velocity at Z=405mm, Case0/1/2

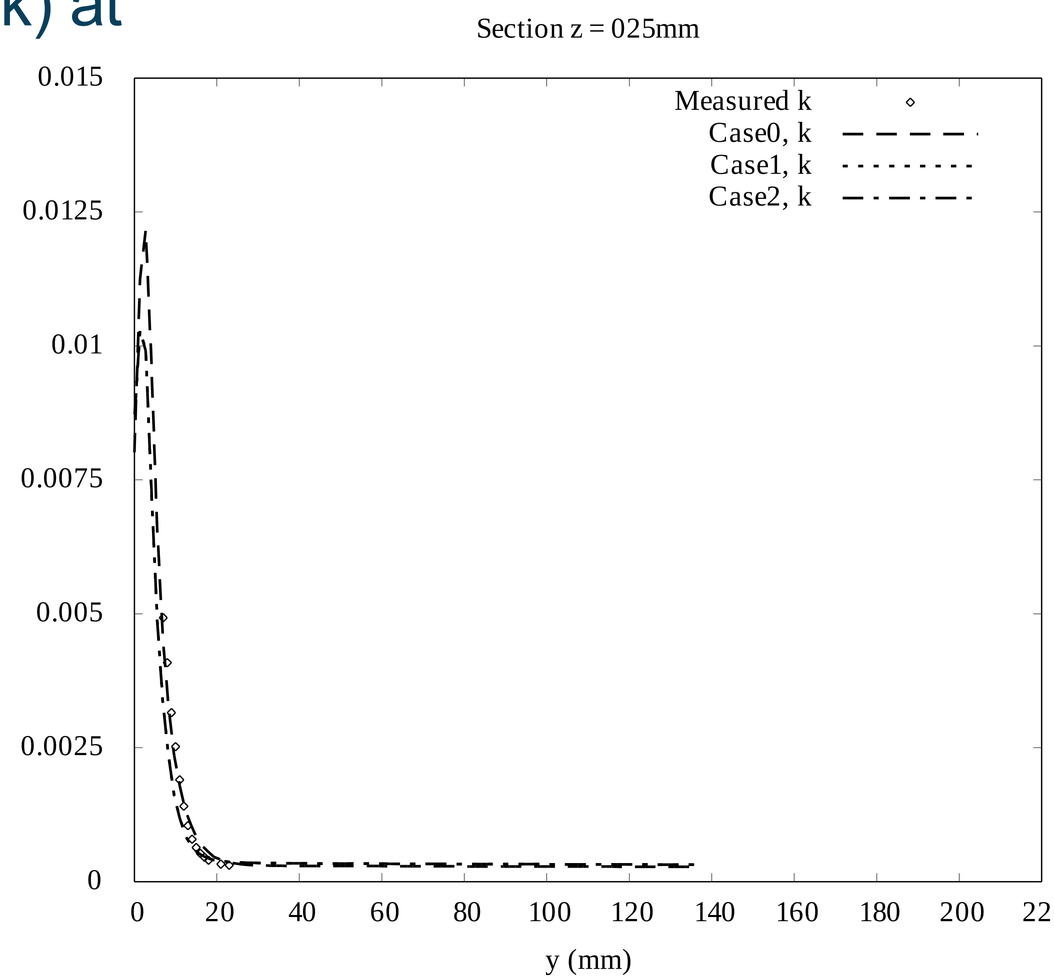


# Results: ERCOFTAC Conical Diffuser

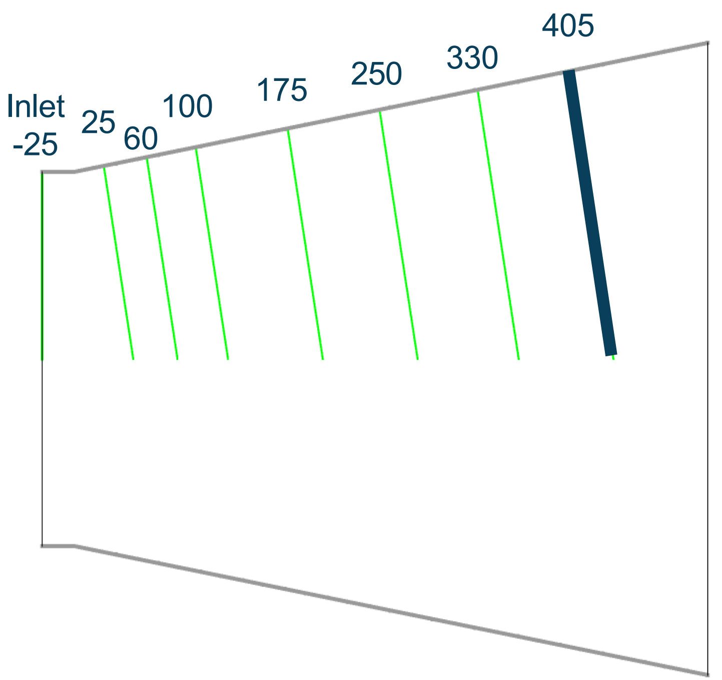
## Turbulent kinetic energy (k) at Z=25mm, Case0/1/2

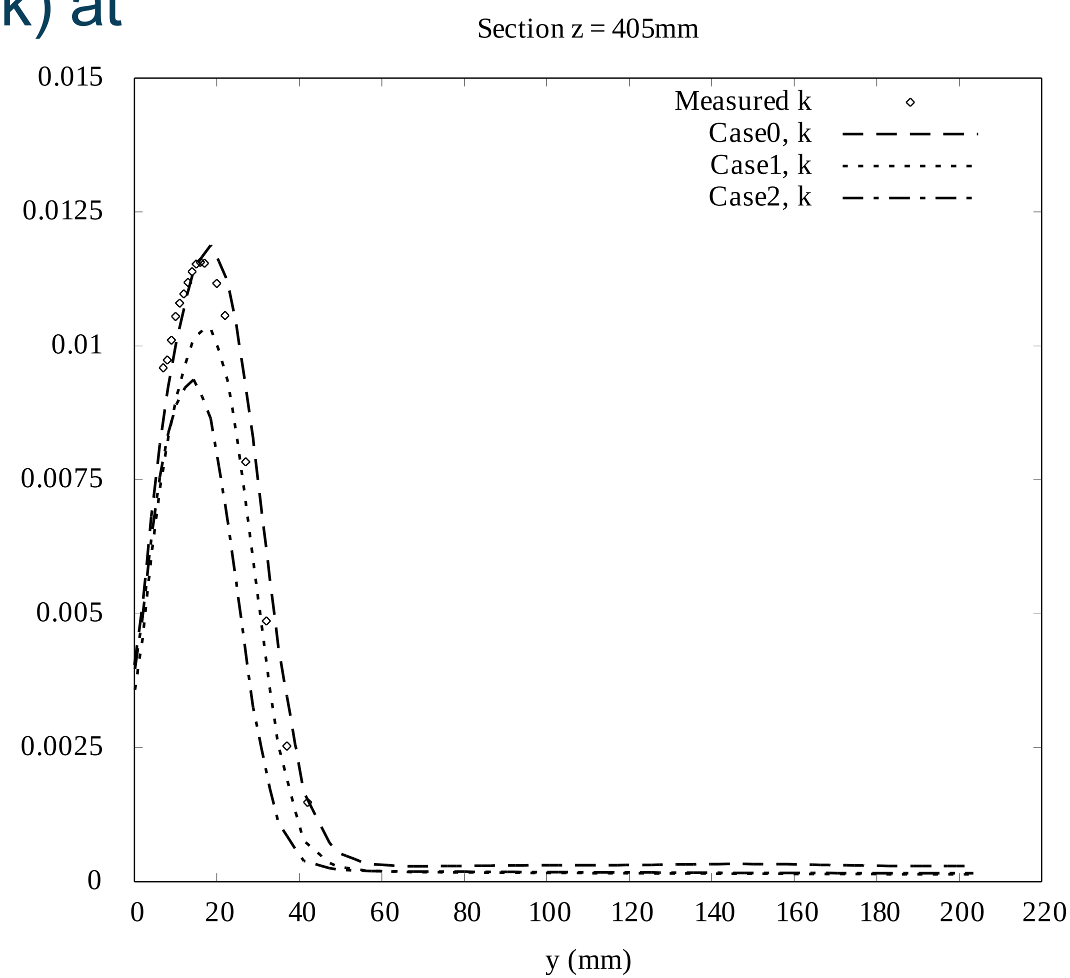


# Results: ERCOFTAC Conical Diffuser

Turbulent kinetic energy (k) at Z=405mm, Case0/1/2



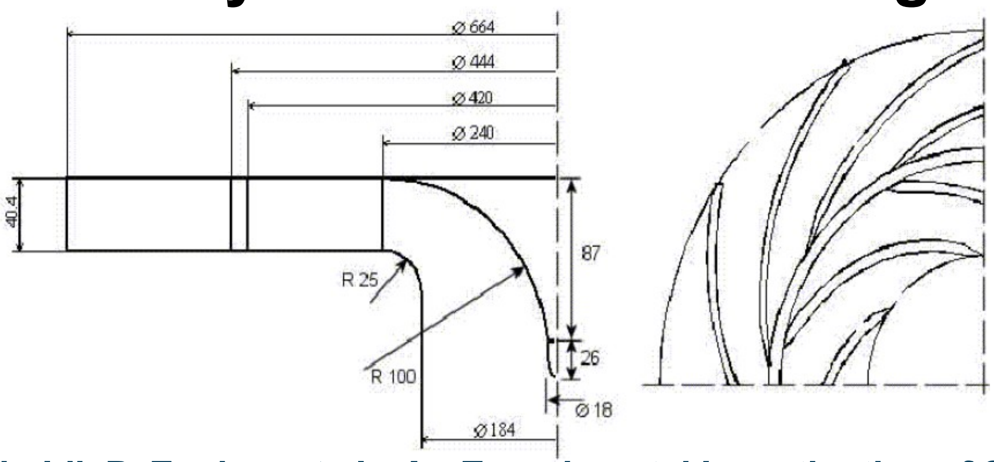



# Tech. Presentations in Turbo Session

- **Third OpenFOAM Workshop in Milano, June 2008:**
  - **Development of a GGI interface** (Beaudoin & Jasak)
  - **LES for swirling flow in pipes** (Duprat & Métais)
  - **Turbo-gas engines: GGI, Steady compressible flow, Conjugate Heat Transfer, Boundary Conditions, Mixing Plane** (Mangani & Bianchini)
  - **Mixing Plane Interface** (Blaim, Borm, Fröbel, Kau)

# Fourth OpenFOAM Workshop

- Montréal, Canada, June 1-4, 2009
  - → More info: http://www.openfoamworkshop.org
- Turbomachinery Session
- OpenFOAM Turbomachinery Workgroup 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.