IMPRESS WP2 D2.2.2 – CFD modelling of flow in stacks

Jan Gersl (jgersl@cmi.cz), Czech Metrology Institute

Motivation

Measurement of flow rate in stacks is important component for determining of amount of emissions released to the atmosphere. Current regulation documents (e.g. EU Industrial Emissions Directive) are introducing lower emission limit values bringing new requirements for accuracy of the emission measurement methods. The most common method for flow rate measurement in stacks is using Prandtl tubes for determining gas speed in a net of points inside a stack (e.g. ISO 10780, EN ISO 16911). Another method is using ultrasonic flow meters. Both of them are sensitive to orientation of the meter with respect to the gas flow direction and therefore also to a swirl which often appears in the stacks. The swirl was identified by stakeholders as a factor influencing the flowrate measurement accuracy in stacks which needs a deeper investigation.

What we want to model

The aim of the research within the IMPRESS project is to investigate the flow in certain typical stack configurations, to determine swirl velocities for various parameters like shape of the supplying pipe or Reynolds number of the gas flow and to estimate related errors in flow rate measurements using the two above mentioned methods. The stack configurations considered consist of a vertical pipe with T-shape connection of a supplying pipe. The supplying pipe is either straight or with 90° elbow or with double out-of-plane 90° elbow. The range of Reynolds numbers typical for the stack flow is 10^5 to 10^7. The determination of errors of the flow rate measurement with Prandtl tubes is based on known experimental data on indication of S-type Prandtl tubes exposed to air streams of various directions [1]. The influence of flow direction to ultrasonic flow meters is derived from their physical principle.

The software and hardware used

The method used for this investigation is CFD (computational fluid dynamics) modelling using OpenFOAM software validated by experimental data from air velocity profile measurements in T-shape geometry (e.g. [2]). The OpenFOAM is free opensource software (www.openfoam.org) which is widely used by both academic and commercial sector. It contains tools for mesh generation, computing and postprocessing. The range of physical problems which can be modelled with OpenFOAM covers single phase and multiphase flows of various media with variety of options for turbulence modelling, heat transfer, combustion and others.

The computations are performed at CMI's cluster with 544 processor cores and around 6 TB RAM.

Where we are now

Several structured meshes with various cell densities have been created for the stack geometries with straight and single elbow supplying pipes (Fig. 1-2) and testing of the mesh convergence and turbulence models is running now. After the sufficient mesh density will be determined and the suitable turbulence model giving predictions corresponding to the experimental data will be selected the swirl modelling in stacks can be started. Some preliminary results for the flow field in a stack with single-elbow supplying pipe with 1.5 m diameter and inlet air flow velocity of 10 m/s are shown in the figures 3 - 5.

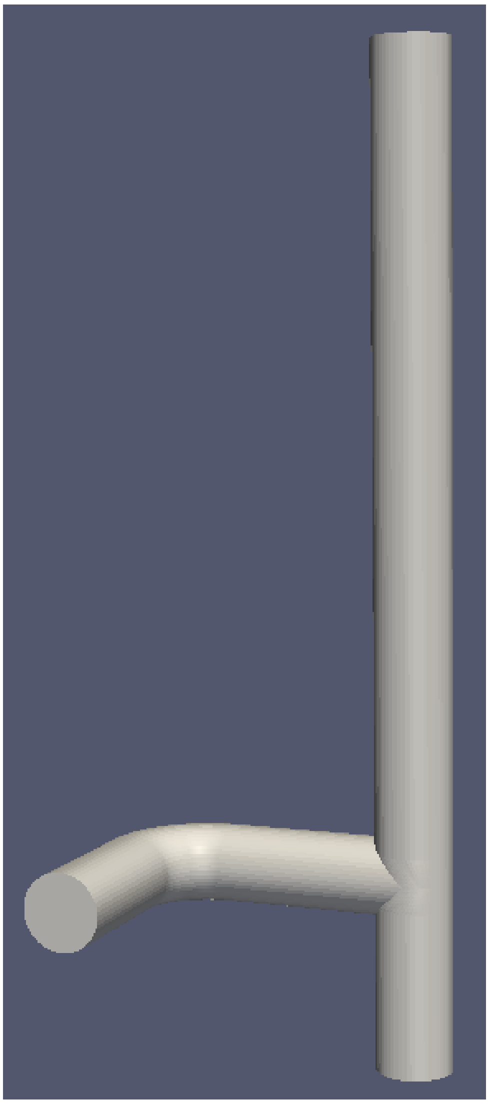

Fig. 1 Stack geometry with a single elbow supplying pipe.

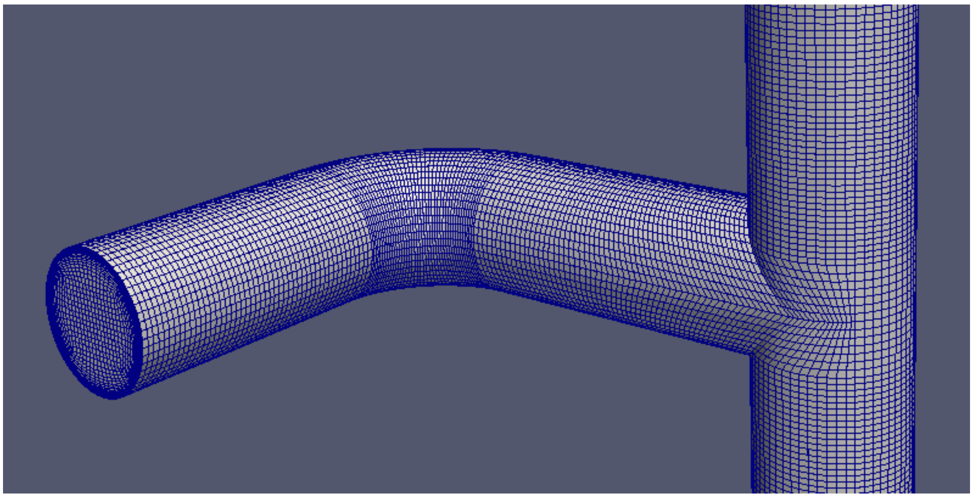

Fig. 2 Example of a mesh for the stack geometry.

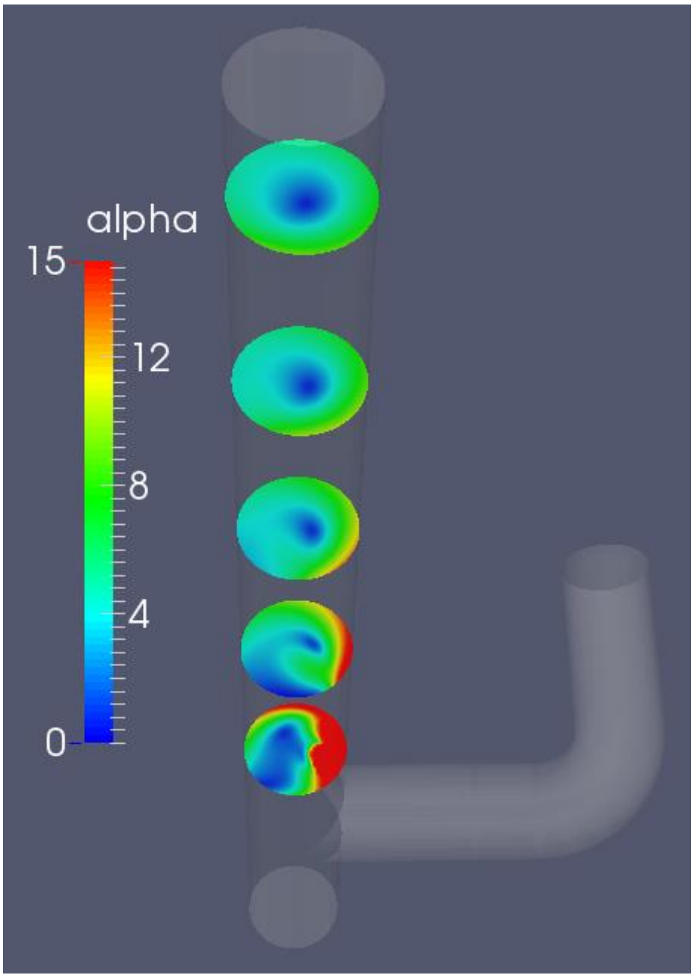

Fig. 3 Angles of flow velocity vector from the stack axis in various cuts. The colour scale is in degrees.

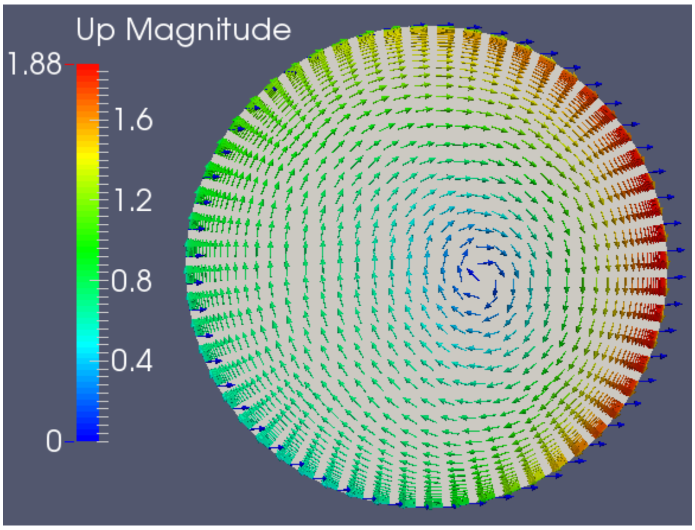

Fig. 4 Swirl in a cut through the stack. The colour scale is in m/s and Up is the velocity vector projection into the cut.

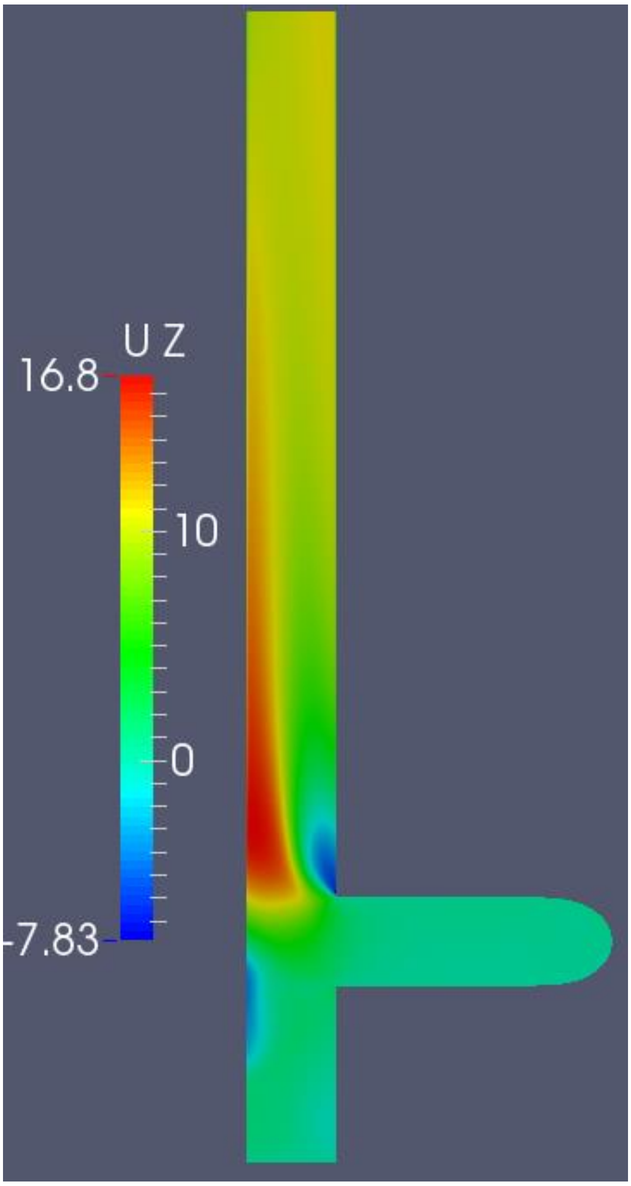

Fig. 5 Vertical velocity component. The colour scale is in m/s.

References

[1] I. I. Shinder et al., *NIST's New 3D Airspeed Calibration Rig Addresses Turbulent Flow Measurement Challenges*, Proceedings of ISFFM 2015

[2] J. Szmyd et al., *Experimental and numerical analysis of the air flow in T-shape channel flow*, Arch. Min. Sci., Vol. 58 (2013), No 2, p. 333–348