Simulation with different turbulence models in an Annex 20 benchmark test using Star-CCM+

DCE Technical Report No. xxx

AALBORG UNIVERSITY
Department of Civil Engineering

Aalborg University
Department of Civil Engineering
Indoor Environmental Engineering Research Group

DCE Technical Report No. xxx

Simulation with different turbulence models in an Annex 20 benchmark test using Star-CCM+

by

Jérôme Le Dréau
Per Heiselberg
Peter V. Nielsen

January 2013

Scientific Publications at the Department of Civil Engineering

Technical Reports are published for timely dissemination of research results and scientific work carried out at the Department of Civil Engineering (DCE) at Aalborg University. This medium allows publication of more detailed explanations and results than typically allowed in scientific journals.

Technical Memoranda are produced to enable the preliminary dissemination of scientific work by the personnel of the DCE where such release is deemed to be appropriate. Documents of this kind may be incomplete or temporary versions of papers—or part of continuing work. This should be kept in mind when references are given to publications of this kind.

Contract Reports are produced to report scientific work carried out under contract. Publications of this kind contain confidential matter and are reserved for the sponsors and the DCE. Therefore, Contract Reports are generally not available for public circulation.

Lecture Notes contain material produced by the lecturers at the DCE for educational purposes. This may be scientific notes, lecture books, example problems or manuals for laboratory work, or computer programs developed at the DCE.

Thesis are monograms or collections of papers published to report the scientific work carried out at the DCE to obtain a degree as either PhD or Doctor of Technology. The thesis is publicly available after the defence of the degree.

Latest News is published to enable rapid communication of information about scientific work carried out at the DCE. This includes the status of research projects, developments in the laboratories, information about collaborative work and recent research results.

Published 2013 by
Aalborg University
Department of Civil Engineering
Sohngaardsholmsvej 57,
DK-9000 Aalborg, Denmark

Printed in Denmark at Aalborg University

ISSN xxxx-xxxX
DCE Technical Report No. xxx

Contents

1. INTRODUCTION

The purpose of this investigation is to compare the different flow patterns obtained for the 2D isothermal test case defined in Annex 20 (1990) using different turbulence models. The different results are compared with the existing experimental data.
Similar study has already been performed by Rong et al. (2008) using Ansys CFX 11.0. In this report, the software Star-CCM+ has been used.

2. PRESENTATION OF THE ISOTHERMAL CASE

The simulations are performed with the two dimensional isothermal Annex 20 room benchmark test described by Nielsen (1990). The sizes of the annex 20 room are specified as:

L = 9m, H = 3m, h_1 =0.168m, h_2 =0.48m

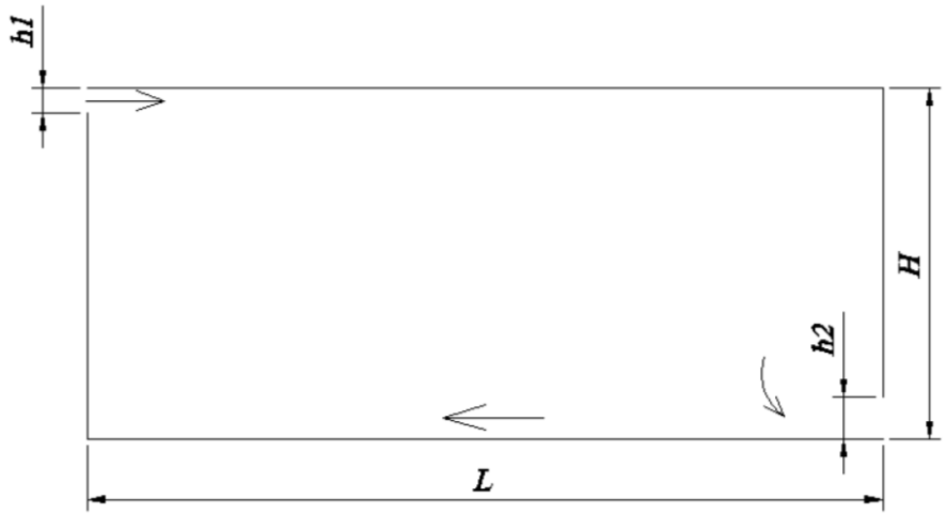


Figure 1: Sketch of the benchmark test

The air is supplied from left top with velocity in 0.455 m/s and exhausted from right bottom. The inlet conditions are listed in the following:

$$u_0 = 0.455\, m/s$$

$$k_0 = 1.5 * (0.04\, u_0)^2 = 4.97\,.\,10^{-4}\, J/kg$$

$$\varepsilon_0 = \frac{{k_0}^{1.5}}{l_0} = 6.59\,.\,10^{-4}\, J/kg.s \qquad \text{with} \qquad l_0 = \frac{h_1}{10}$$

According to Nielsen (1990), the inlet conditions correspond to a turbulence intensity of 4%.

3. PRESENTATION OF THE SIMULATIONS

It has been shown by Olmedo et al. (2010) that the studied case can be considered as a steady two dimensional case (steady or unsteady) in the major part of the domain. Therefore the simulations have been performed with a two dimensional geometry, and assuming a steady state. The CFD models are solved using the segregated flow model and air is considered as an ideal gas.

In this report the results obtained with different meshes are presented, and different k-ε and k-ω models are tested.

Figure 2 shows four line locations where the simulations results can be compared with measurements.

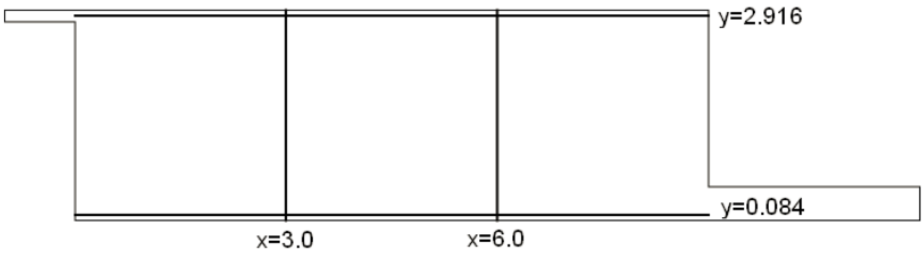


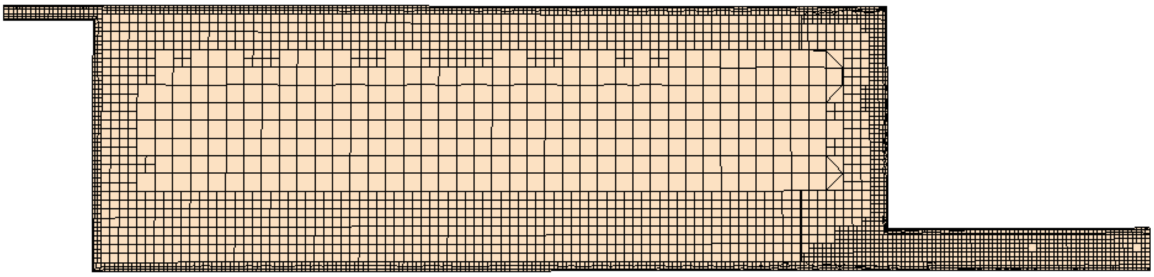
Figure 2: Sketch of the vertical and horizontal lines where the velocity profiles have been measured

4. DIFFERENT COMPUTATIONNAL GRIDS

The number of cells in the computational grid has been chosen similar to Voigt (2000). Different grids are used in combination with the k-ε and k-ω models.

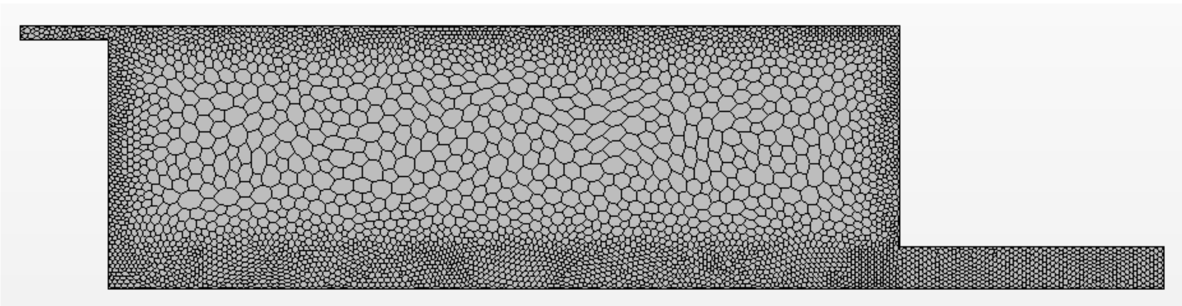
- Mesh 1: coarse & structured

Characteristics	Surfaces mesh 3D	Volume mesh 3D	Surface mesh 2D
Structured Base size: 0.2m Custom size at the border: 30% (0.06m)	38 930 faces (surface remesher)	185 125 faces (trimmer with prism layer mesher)	4 068 cells



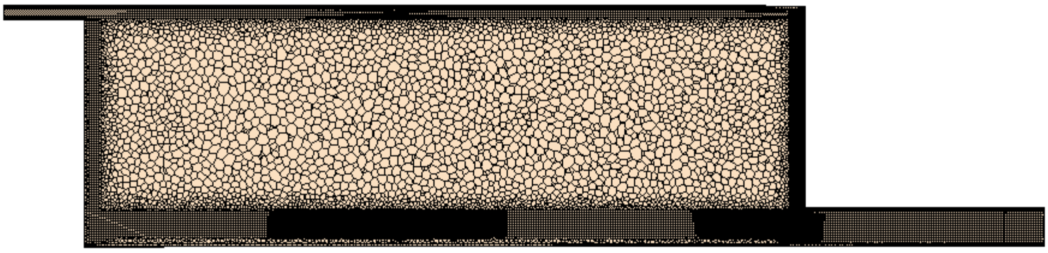
- Mesh 2: coarse & unstructured

Characteristics	Surfaces mesh 3D	Volume mesh 3D	Surface mesh 2D
Unstructured Base size: 0.2m Custom size at the border: 30% (0.06m)	38 930 faces (surface remesher)	117 567 faces (polyhedral with prism layer mesher)	4 793 cells



- Mesh 3: fine & unstructured

Characteristics	Surfaces mesh 3D	Volume mesh 3D	Surface mesh 2D
Unstructured Base size: 0.1m Custom size at the border: 30% (0.03m)	140 762 faces (surface remesher)	4 840 630 faces (polyhedral with prism layer mesher)	16 658 cells



4.1 LIST OF SIMULATIONS

Nbr	Type of turb. model	Wall function	Others...	Mesh
2	k- ε	High y + Wall treatment	Steady state – 2D Ideal gas Segregated flow	1- Structured Coarse (4,1 k-cells)
1				2- Unstructured Coarse (4,8 k-cells)
3				3- Unstructured Fine (16,7 k-cells)

4.2 Convergence

The figures below show the convergence with different grids. Convergence is longer to reach with a finer mesh.

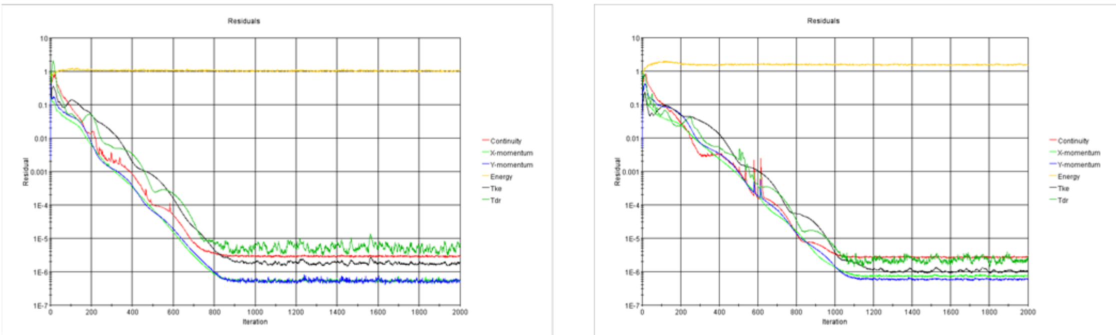


Figure 3: Structured grid (left) vs. Unstructured grid (right) – Coarse mesh

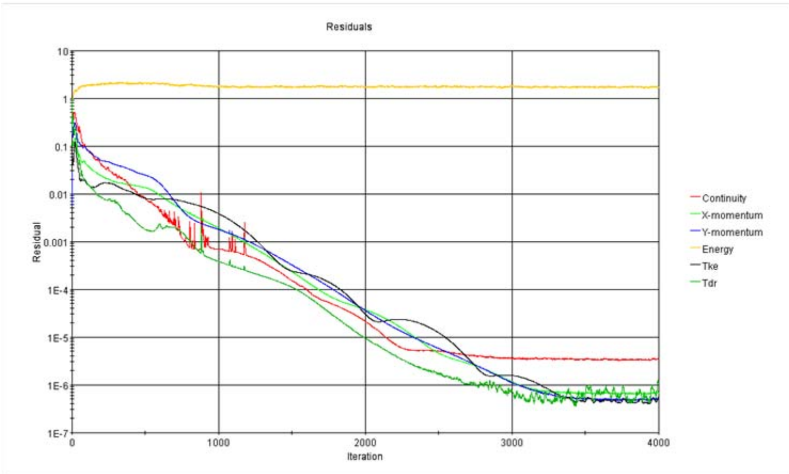
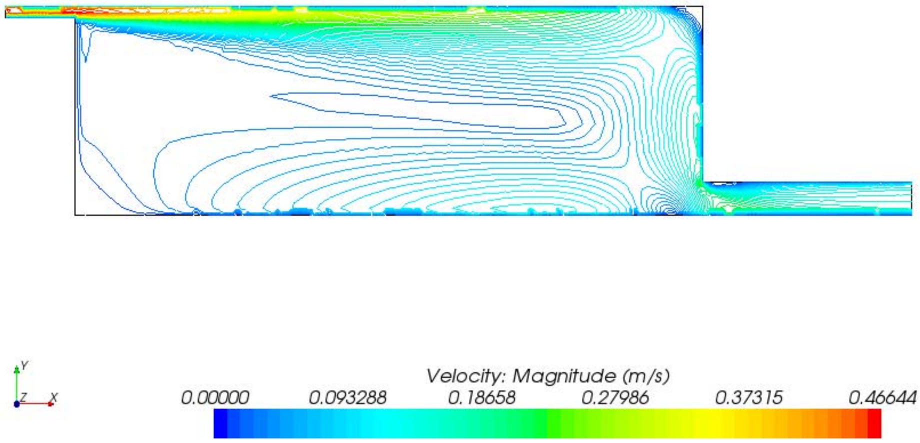


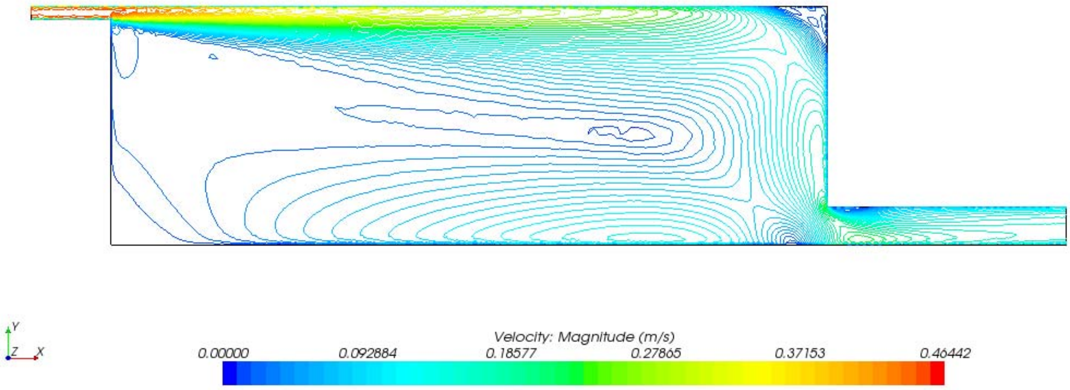
Figure 4: Fine and unstructured mesh

4.3 STREAMLINES

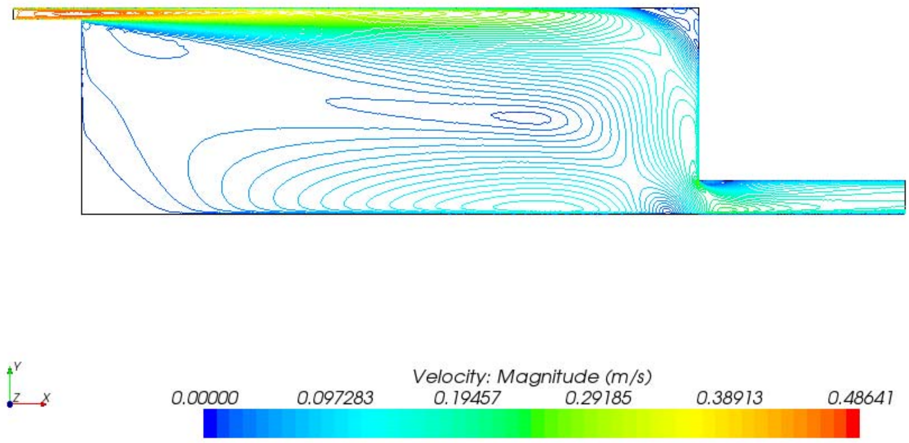
Simulation 2: Structured & coarse



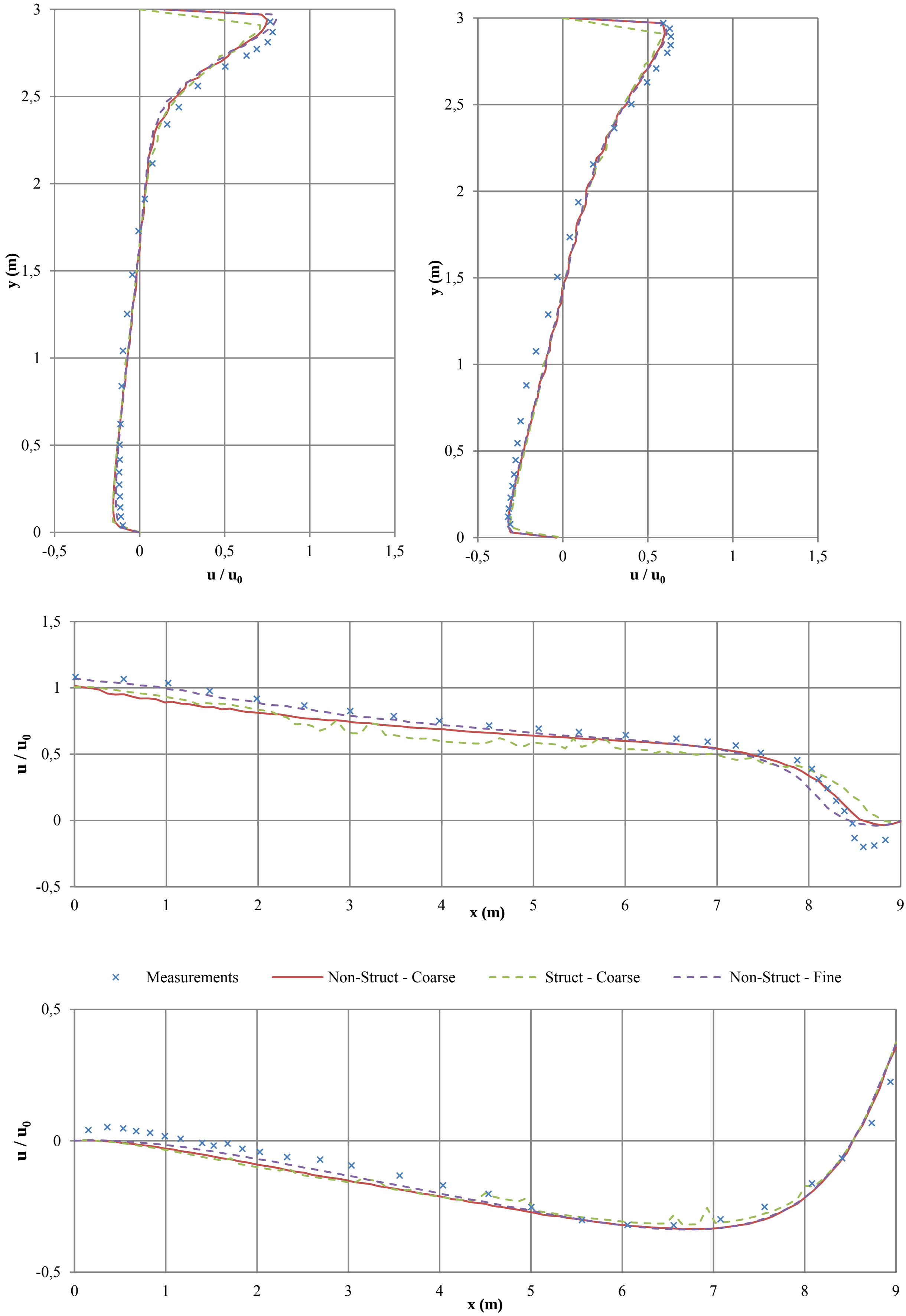
Simulation 1: Non-structured & coarse



Simulation 3: Non-structured & fine



4.4 VELOCITY PROFILES



⇒ The structured mesh showed the less accurate results. This could be due to the slightly lower number of cells used, or due to the layout, which might be less efficient. A finer mesh is required. With an unstructured grid, a finer mesh gives better results compared to experiment, but still doesn't fit exactly with the results.
Nevertheless it has to be noticed that this analysis has been performed only with the standard k-ε model. Conclusions might change with another turbulence model.

5. DIFFERENT K-Ε MODELS

5.1 LIST OF SIMULATIONS

Nbr	Type of turb. model	Wall function	Others…	Mesh
3	Standard k- ε	High y + Wall treatment	Steady state – 2D Ideal gas Segregated flow	3- Unstructured Fine (16,7 k-cells)
4	Standard k-ε Low-Re	All y + Wall treatment		
5	AKN k-ε Low-Re	All y + Wall treatment		
6	Realizable k-ε	High y + Wall treatment		

The inlet conditions are defined with:

$$k_0 = 1.5 * (0.04\, u_0)^2 = 4.97\,.\,10^{-4}\, J/kg$$

$$\varepsilon_0 = \frac{{k_0}^{1.5}}{l_0} = 6.59\,.\,10^{-4}\, J/kg.s$$

When using the Standard k- ε model, the inlet turbulent viscosity is therefore equal to (CD Adapco - 2011):

$$\mu_t = \rho\ C_\mu\ k\ max\left(\frac{k}{\varepsilon}\ ;\ 0.6\sqrt{\frac{v}{\varepsilon}}\right)\ \approx\ 1.225\ \cdot 0.09 \cdot \frac{k^2}{\varepsilon}\ \approx\ 4.13\,.\,10^{-5}\, Pa.s$$

And the inlet turbulent viscosity ratio:

$$\frac{\mu_t}{\mu} \approx \frac{4.13\,.10^{-5}}{1.855\,.10^{-5}} \approx 2.22$$

5.2 CONVERGENCE

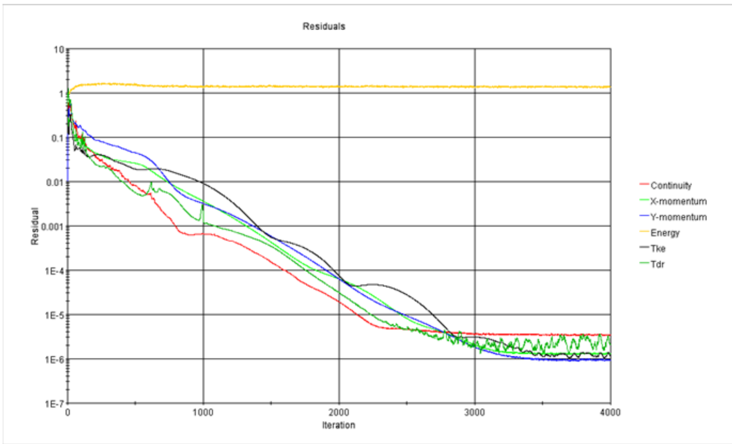


Figure 5: Simulation 4

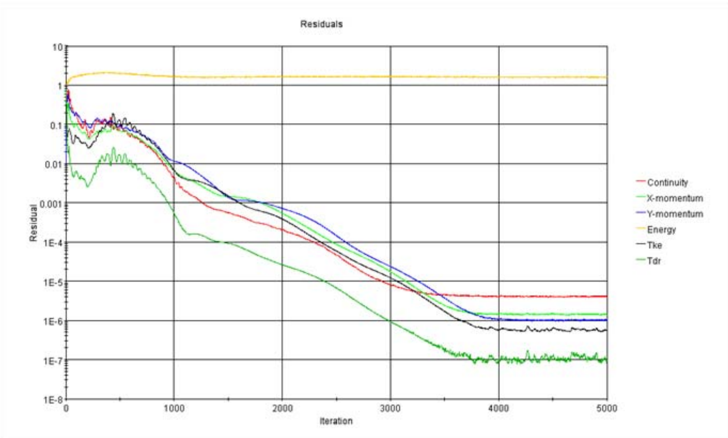


Figure 6: Simulation 5

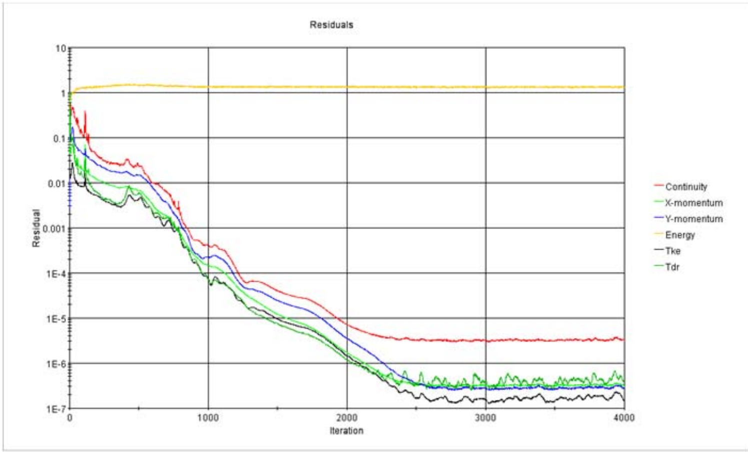
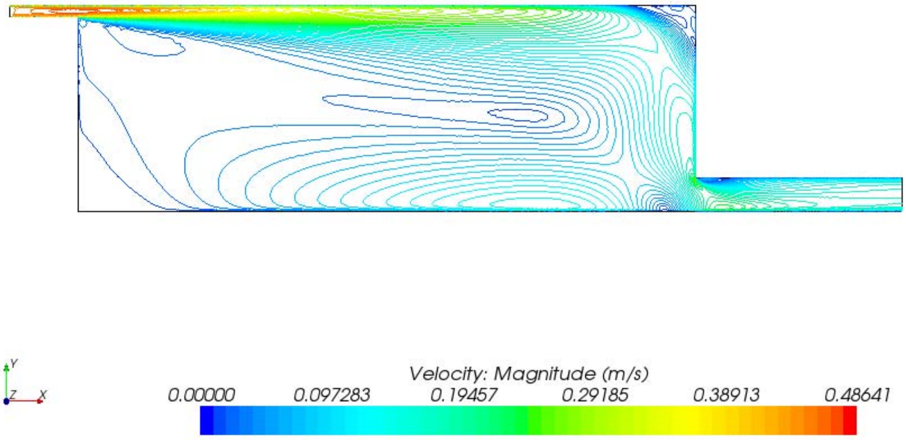


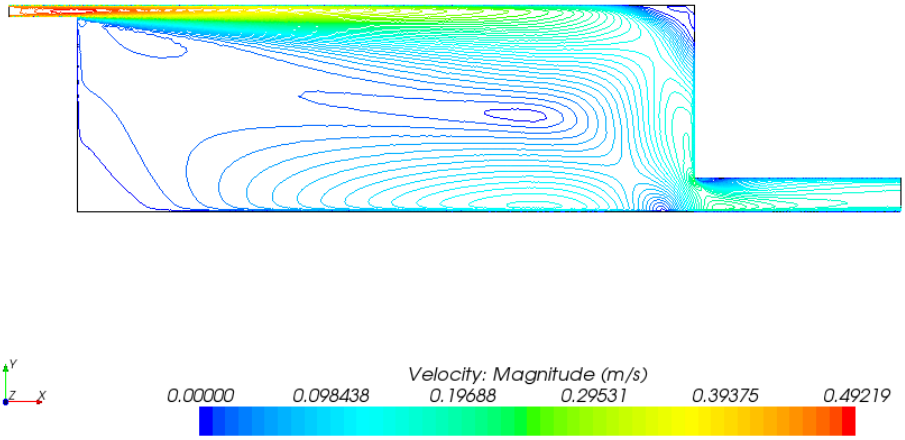
Figure 7: Simulation 6

5.3 STREAMLINES

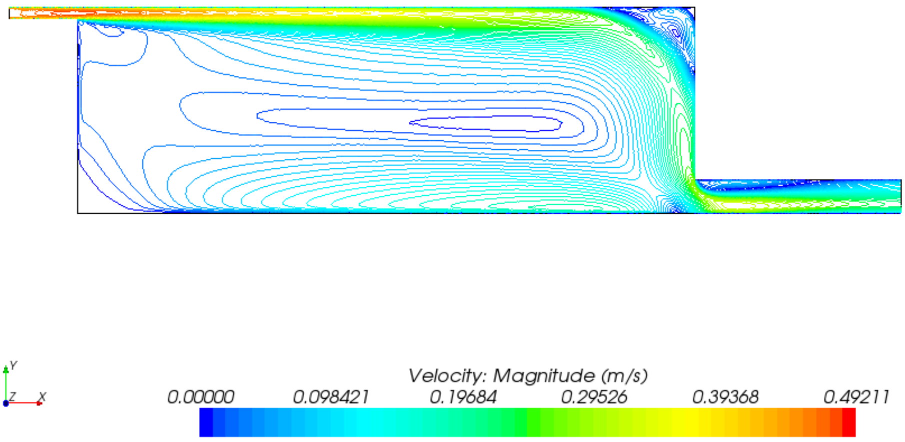
Simulation 3: Standard k- ε



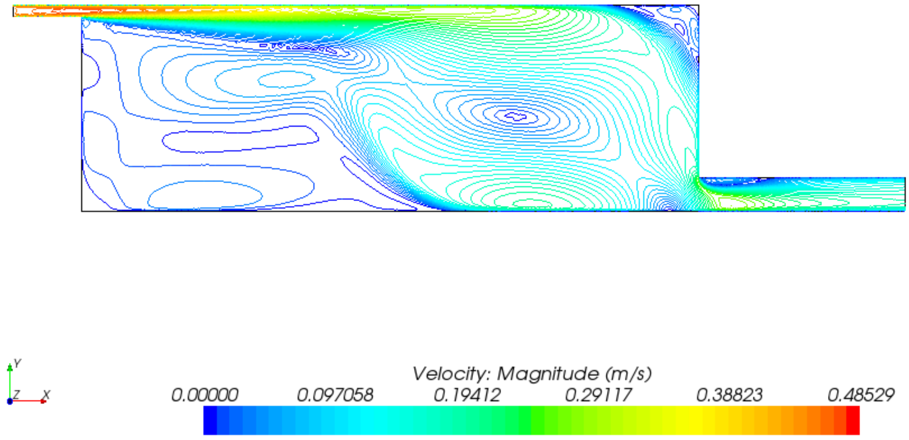
Simulation 4: Standard k- ε Low-Re



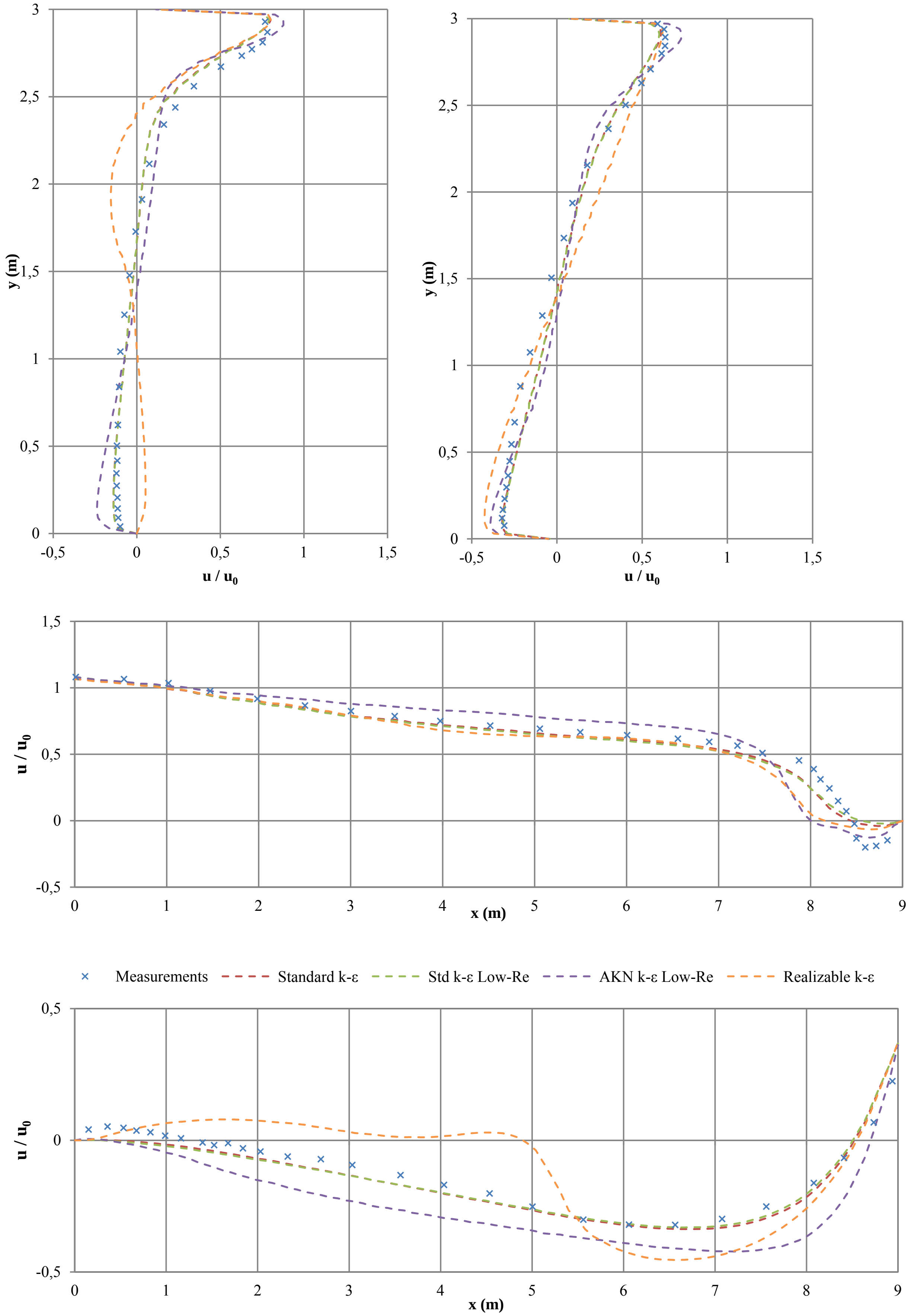
Simulation 5: AKN k- ε Low-Re



Simulation 6: Realizable k- ε



5.4 VELOCITY PROFILES



⇒ The performance of the standard k-ε and standard k-ε Low-Re are comparable and quite close to the measurements. Nevertheless the wall bounded flow is difficult to predict with all the models.
When applying the model AKN k-ε Low-Re, a recirculation zone appears in the right-upper corner. A less accurate correspondence with the results can be observed.
The results obtained with the turbulence model Realizable k-ε diverge the most with the experimental results. This is due to the presence of a complex zone of recirculation on the left part of the room.

5.5 About the Low-Reynolds number model

It has been seen that the results of the standard k-ε and standard k-ε Low-Re are almost the same. This might be due to the fact that the Low-Reynolds model is almost never activated during the simulation, and the standard k-ε model dominates. In fact, a standard k-ε model can be considered as a special version of a Low-Re number model, without any damping function at low local turbulence Reynolds number R_t .
Nielsen (1995) suggested that the limit for R_t was 400: above this value, a Low-Re model corresponds to a standard k-ε model. Even if the Low-Re model used in Star-CCM+ is slightly different (model from Lien, and not from Launder and Sharma), this limit has been kept, as the basis of the two models are the same.
This means that if the turbulent viscosity ratio is above around 40, the damping functions will disable the Low-Reynolds number model.

The turbulent viscosity ratio obtained from the simulation can be seen on the figure below. It can be seen that this ratio is in most of the domain above 40, therefore deactivating the Low-Re model. This explains why the results of the standard k-ε and standard k-ε Low-Re are similar.

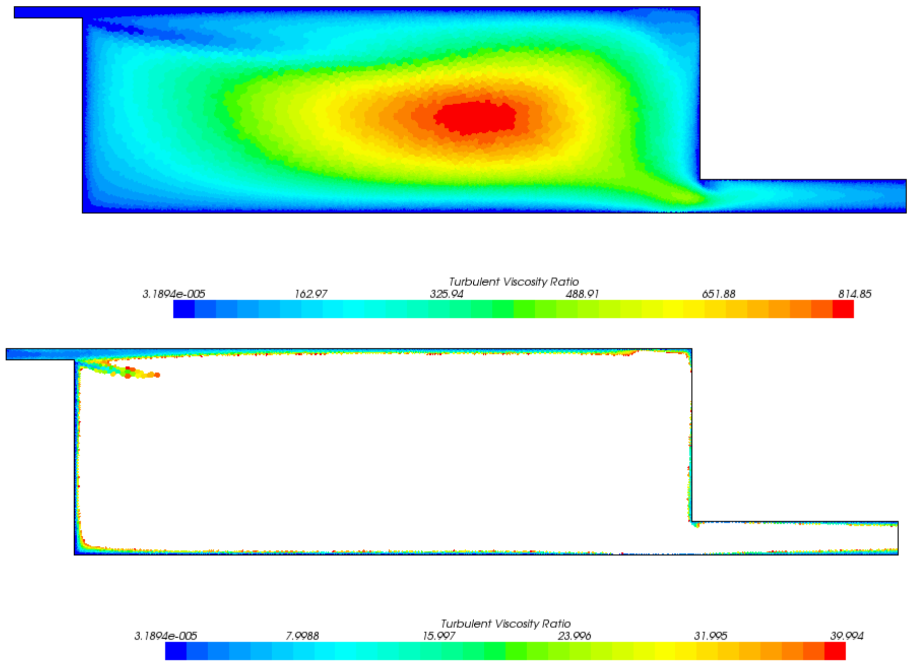


Figure 8: Turbulent viscosity ratio - Standard Low-Re turbulence model
Below are only represented the cells where the turbulent viscosity ratio is below 40.

When having a closer look to the inlet, it can be observed that the turbulence viscosity ratio is relatively small, close to 2. This can be explained by the values of k_0 and ε_0 that have been set at the inlet: it will lead to a turbulence viscosity ratio around 2.2 for the standard k-ε model (cf. page 10).

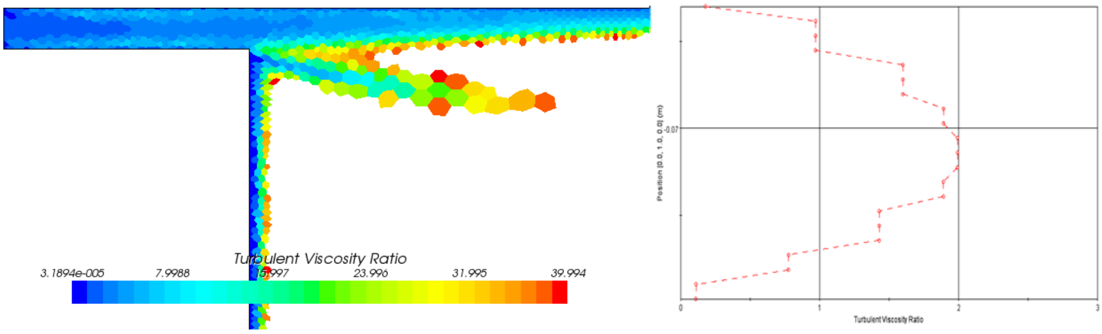


Figure 9: Turbulence viscosity ratio at the inlet

6. DIFFERENT K-Ω MODELS

6.1 LIST OF SIMULATIONS

Nbr	Type of turb. model	Wall function	Others...	Mesh
7	Standard k-ω (Wilcox)	All y + Wall treatment	Steady state – 2D Ideal gas Segregated flow	3- Unstructured Coarse (4,8 k-cells)
8	SST k-ω (Menter)	All y + Wall treatment		

The inlet conditions are defined with:

$$turbulence\ intensity = 0.04$$

$$l_0 = \frac{h_1}{10} = 0.0168\ m$$

6.2 CONVERGENCE

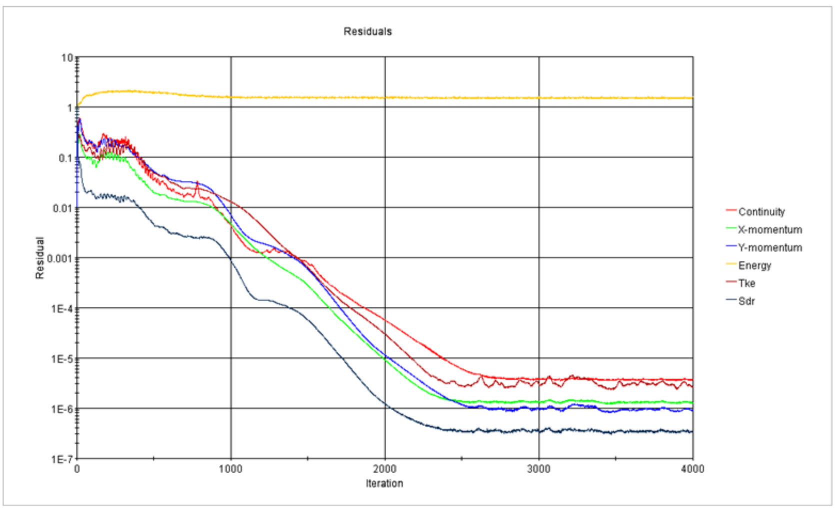


Figure 10: Simulation 7

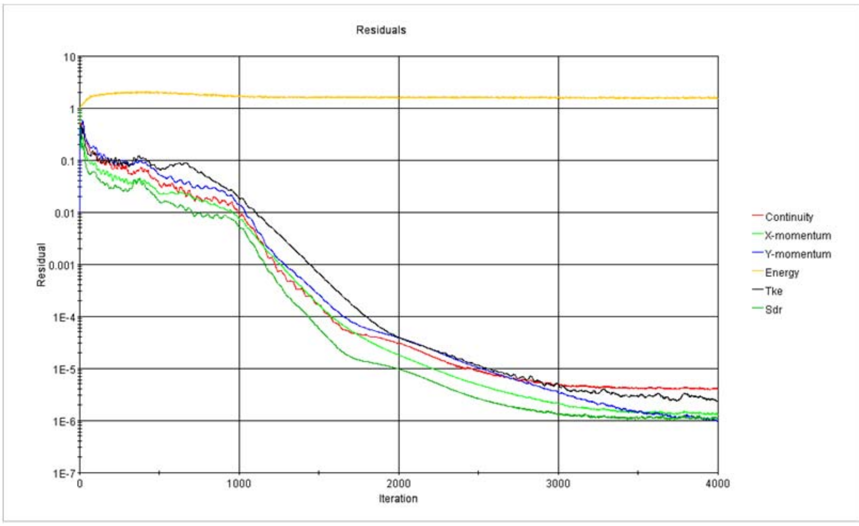
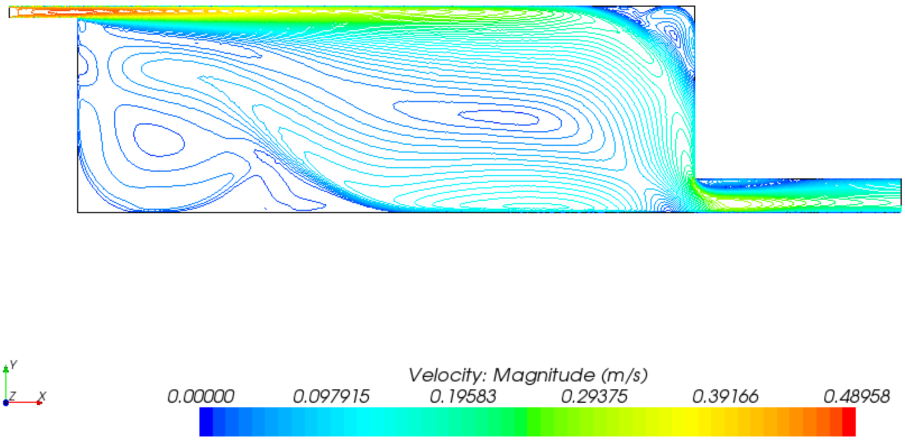


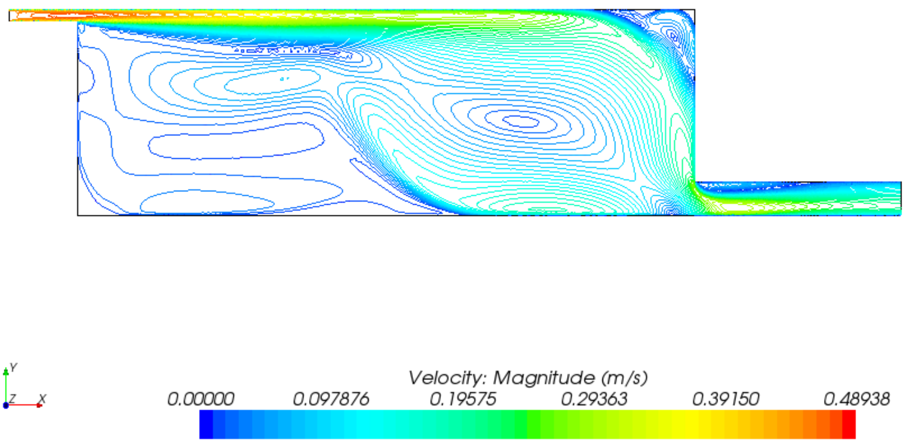
Figure 11: Simulation 8

6.3 Streamlines

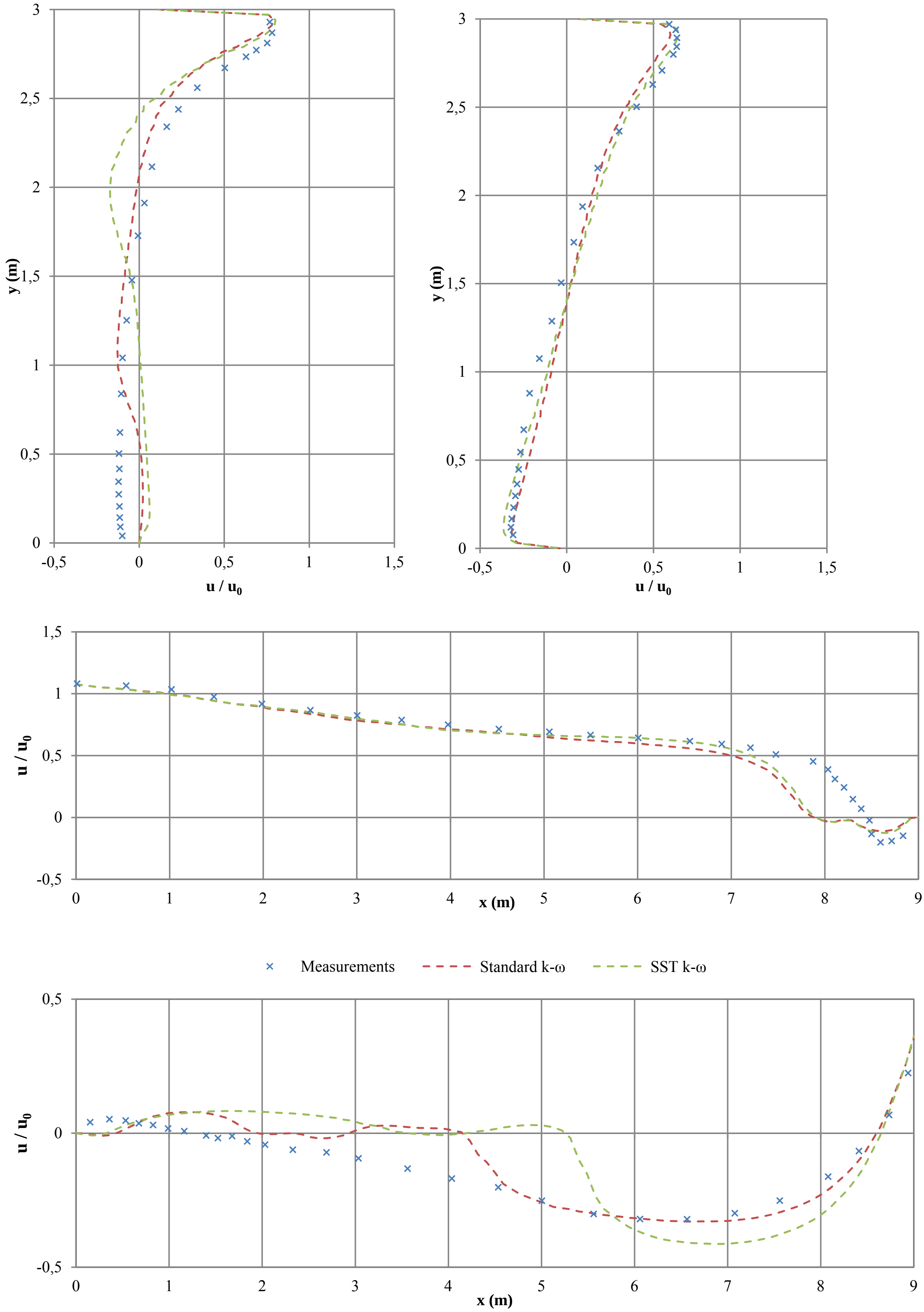
<u>Simulation 7:</u> Standard k-ω



<u>Simulation 8:</u> SST k-ω



6.4 Comparison profiles



⇒ The results obtained with a k-ω model are not in accordance with the experimental results, especially away from the main stream. In these two cases, a complex recirculation zone can be observed on the left part of the room.
But the eddy recirculation in the right upper corner is better predicted with these models than with the k-ε model.

7. Conclusion

In this report, simulations with the Annex 20 benchmark test have been performed using Star-CCM+. The goal was to study how different turbulence models predict the velocity distribution in a ventilated room.

In the main stream away from the wall (upper part and right part of the test room), all turbulence models give relatively accurate results. Differences can be observed in the recirculation zone on the upper right corner and on the left part. The standard k-ε or the standard k-ε low-Re models predict well the recirculation zone on the left part, but no eddy recirculation is predicted in the right upper corner.
The other models (k-ω, k-ε AKN and Realizable) predict this eddy recirculation, but underpredict its intensity. Nevertheless the velocity profile in the left part of the room is not well-predicted with these models.

These simulations pointed out the fact that numerical simulations should always be compared with measurements since different turbulence models will give different results in a specific case.

References

CD Adapco. *Star-CCM+ (v. 6.04.014)*. 2011.

Lars Koellgaard Voigt. *Comparison of Turbulence Models for Numerical Calculation of Airflow in an Annex 20 Room*. International Center for Indoor Environment and Energy Department of Energy Engineering. Technical University of Denmark. ISBN 87-7475-225-1.

Peter V. Nielsen. *Specification of a Two-dimensional Test Case*. Department of Building Technology and Structure Engineering. Aalborg University, 1990. ISBN 0902-7531 R9040.

Peter V. Nielsen. *Flow in Air Conditioned Rooms: Model experiments and numerical solution of the flow equations*. PhD thesis, Technical University of Denmark, 1974.

Peter V. Nielsen. *Airflow in a world exposition pavilion studied by scale-model experiments and computational fluid dynamics*. ASHRAE Transactions,101(2), 1118-1126, 1995.

Inés Olmedo, Peter V. Nielsen. Analysis of the IEA 2D test. 2D, 3D, steady or unsteady airflow? Aalborg University, 2010. ISSN 1901-726X.

Li Rong, Peter V. Nielsen. *Simulation with different turbulence models in an annex 20 room benchmark test using Ansys CFX 11.0*. Aalborg University, 2008. ISSN 1901-726X.