XXXVI International School of Hydraulics

Jachranka, Poland. May 25th 2017

Comparison of Two Numerical Turbulent Flow Models of a Sharp-Groyne Field

Oscar Herrera-Granados

Faculty of Civil Engineering. Chair of Geotechnics, Hydrotechnics, Underground Construction and Hydro-engineering.

HR EXCELLENCE IN RESEARCH

Wrocław University of Science and Technology

Outline

Outline

1. **Introduction**
2. Experimental Works
3. Numerical Modelling
4. RANS vs LES
5. Conclusions

Application of Computational Fluid Dynamics for engineering purposes in Poland

Computational Fluid Dynamics (CFD) is a field of fluid mechanics that it is becoming very popular among water specialists to analyse hydro-engineering projects, e.g.:

- ✓ Water-structure interaction
- ✓ Powerhouses and turbines
- ✓ Fish passes and fish ways
- ✓ Spillways and flood alleviation structures
- ✓ Groynes and river training infrastructure
- ✓ Phenomena related with the motion of sediments
- ✓ Scientific research

River training refers to the measures which are taken to improve the hydraulic behavior of a river and its banks.

Photo source: http://www.life-enns.at/ziele_en.php

Groynes are river training devices built at an angle to the flow that can fulfill multiple objectives (McCoy, et-al. 2008).

The most usual are:

1. Maintaining channel navigability by keeping the flow away from the banks and increasing the mean velocity in the center of the channel as well as the efficiency of sediment transport;
2. Protecting against flooding by increasing the ability of the river reach to pass a relatively large amount of flood flow;
3. Minimizing bank erosion especially in river reaches with incised banks;
4. Restoring fish habitat to degraded streams and
5. Enhancing the diversity of the river.

River groynes along the River Odra in Poland: a traditional river training solution

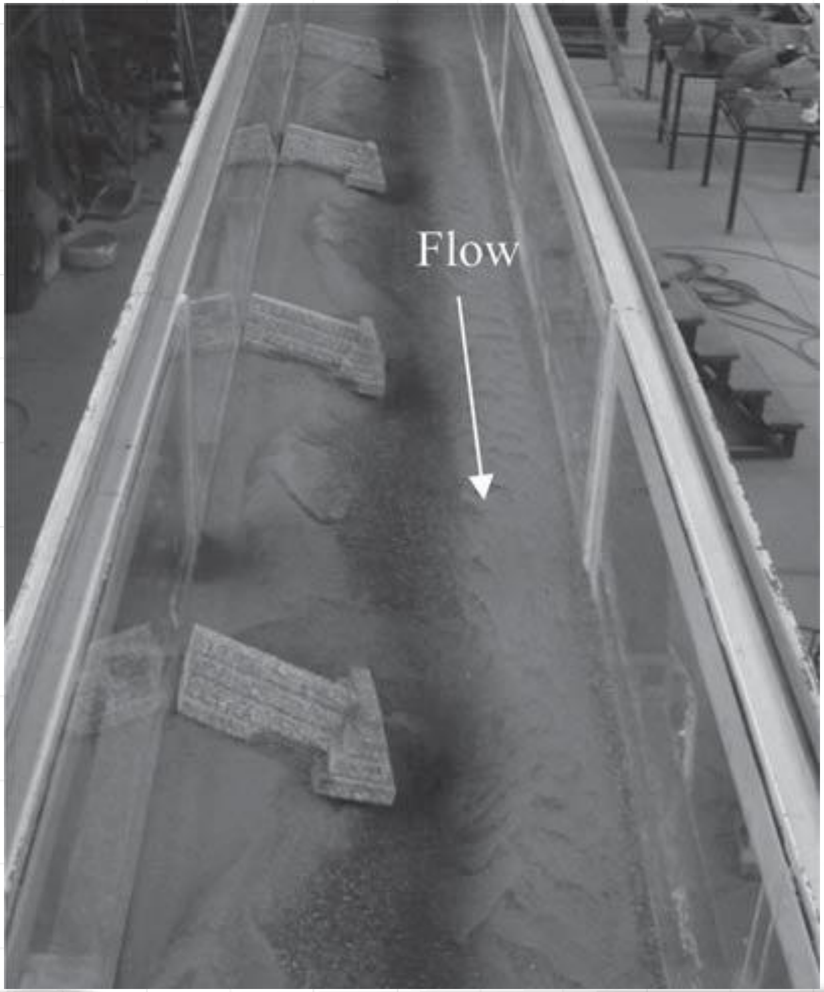
The adequate maintenance of these hydraulic structures plays a vital role in order to fulfil the previously mentioned objectives.

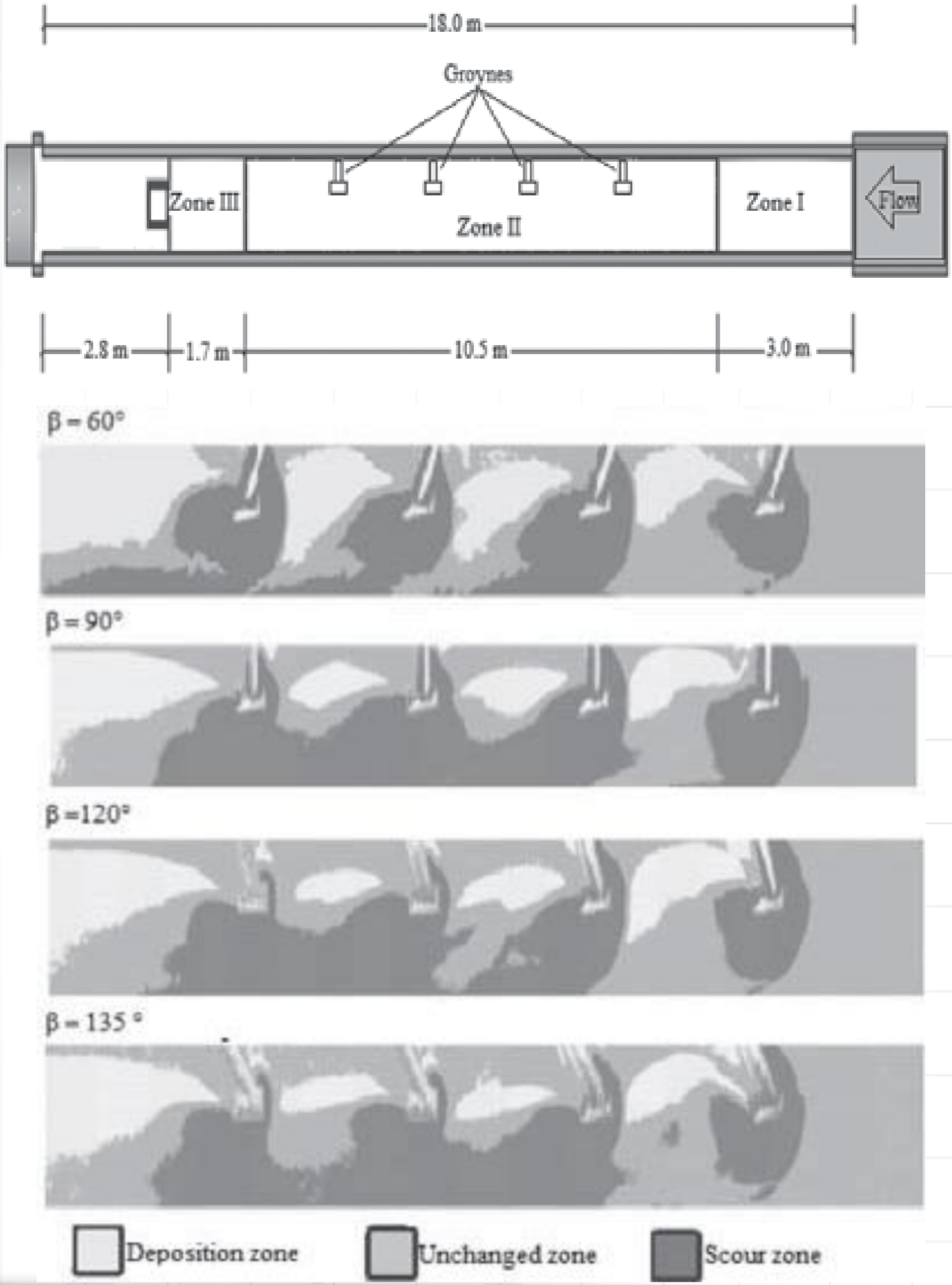
The mismanagement of our natural resources can derive on the malfunctioning of the river training infrastructure. In contradiction, a proper management can add an extra value to the training structures and their objectives.

Fig. above: Groyne in the river *San* in Poland after the flood wave of 2013.

Fig. below: Groyne in the Dutch *Waal*. Groynes can acquire additional recreational and architectural purposes.

The LHUMSS in Bolivia is analysing the efficiency of different arrangements of gabion groynes (Romero et al, 2014) with physical modelling. We are collaborating with them but using numerical modelling.





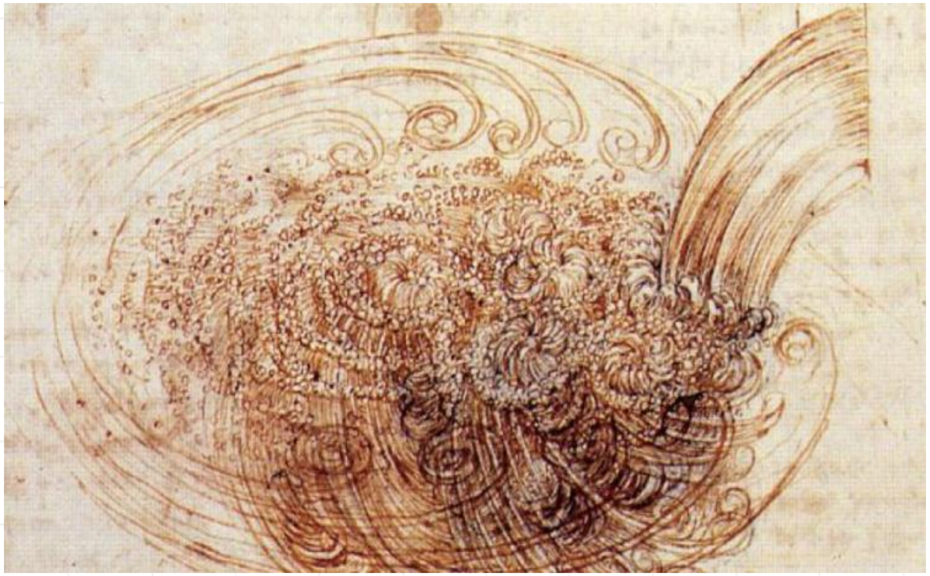
Media1.avi

Outline

1. Introduction
2. **Experimental Works**
3. Numerical Modelling
4. RANS vs LES
5. Conclusions

This uncomfortable Turbulence

Langley Research Center of the United States. *National Aeronautics and Space Administration (NASA)*. Photo ID: EL-1996-00130.



Monaghan, JJ & Kajtar JB. *Leonardo da Vinci's turbulent tank in two dimensions*. European Journal of Mechanics B/Fluids, 44 (2014): 1-9.

Turbulence in air makes itself visible because of this tireless spinning dust that shines under the sunlight.

Nonetheless, it does not happen with turbulence in water. As the eye can not perceive it, one needs to discover it with laboratory instrumentation.

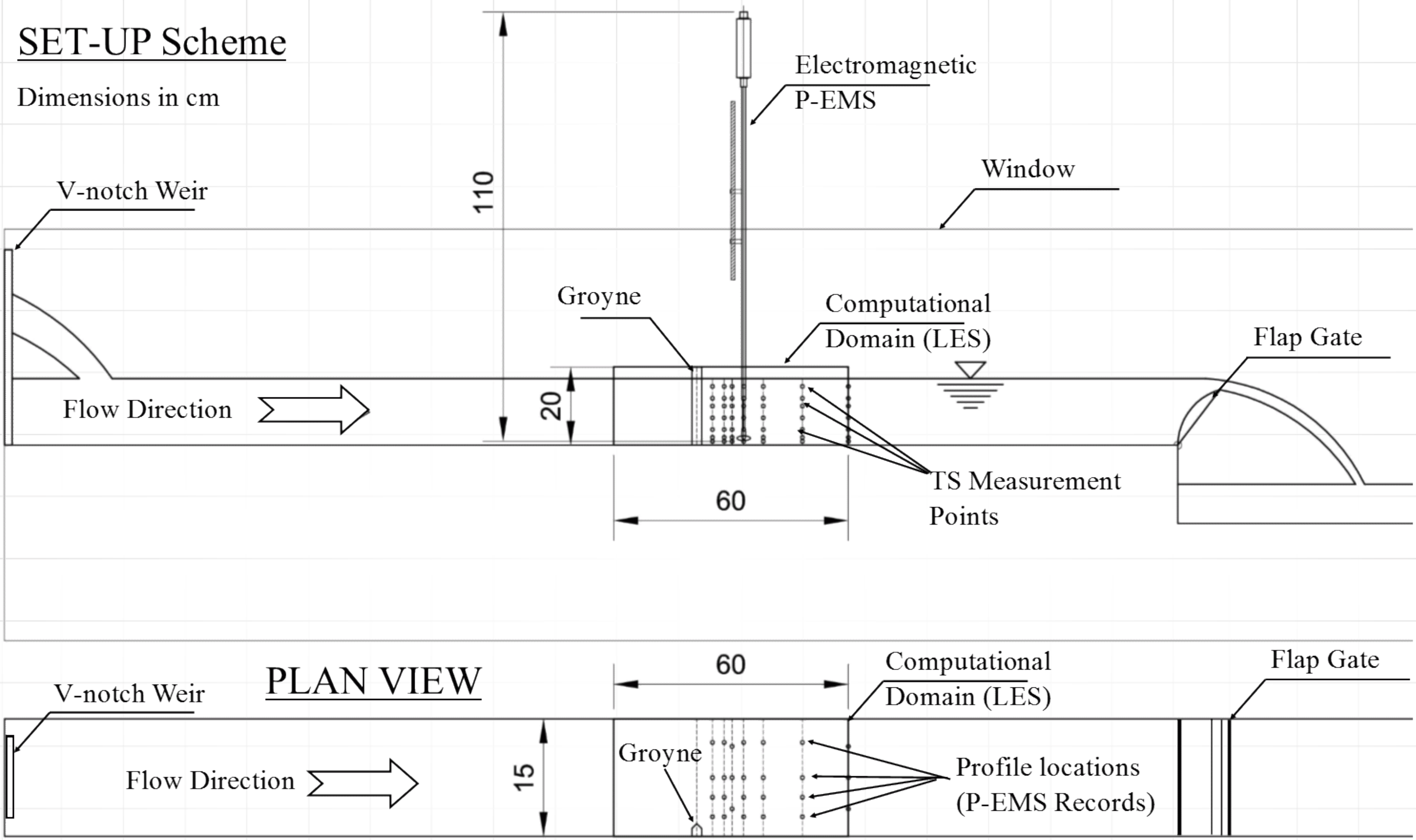
Enzo Levi

Water according to Science

Physical modelling allows specialists to visualize the dynamics of water in well-established scaled constructions at the laboratory. These models are a feasible and reliable choice to analyze the interaction between hydraulic structures and water bodies.

If we have difficult questions, lets solve them with simple cases in the laboratory!!

Some 2D velocity time series were recorded. The interaction between water in motion through a rectangular flume and one small groyne was analyzed and compared using the results obtained with the Flow-3D LES and RNG k-ε models.



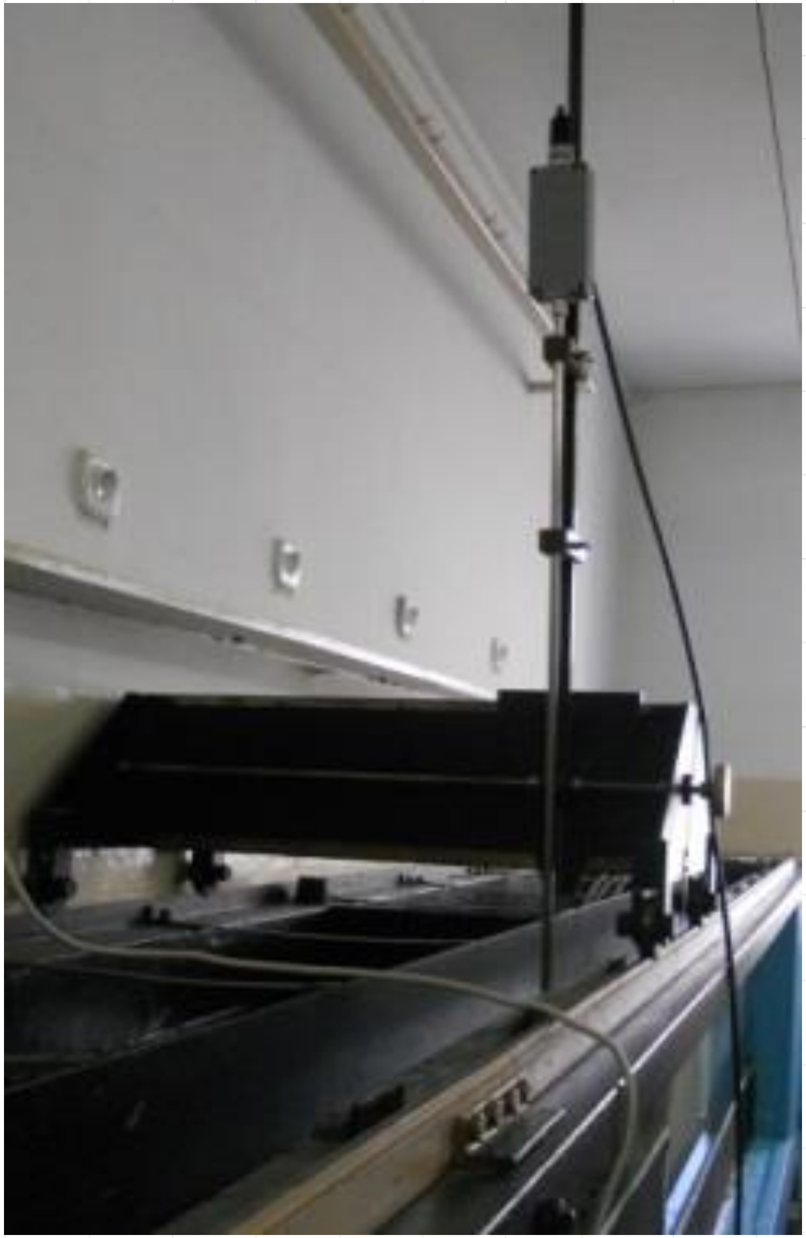
The flow conditions in the laboratory were controlled with a Thomson Weir (upstream) and with a hinged crested gate (downstream). Dimension in cm.

Experimental set-up

Several series of turbulent flow measurements were recorded with a programmable Electro-Magnetic Liquid Velocity-Meter (P-EMS).

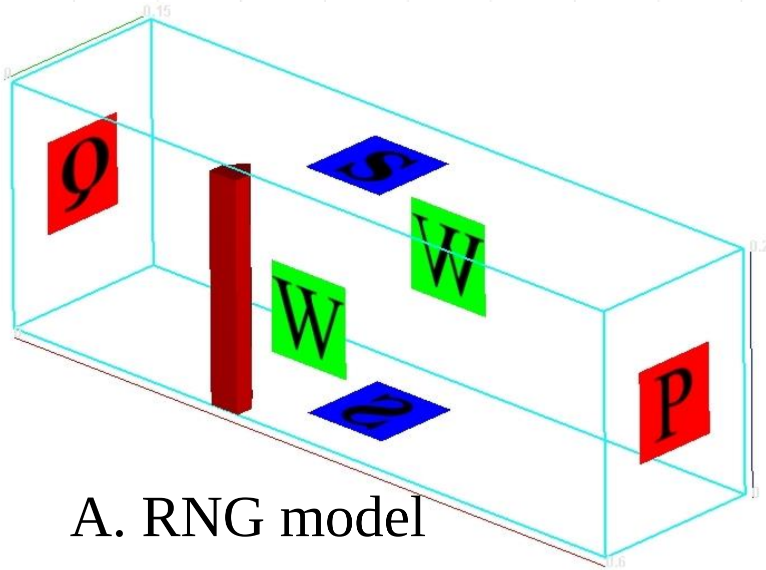
The flume's width is 15cm and the length of the experimental zone is around 6 meters (60 cm for CFD).

Three profiles along five cross sections were measured at seven water elevations. Three flow rates were used for the experiments. Nonetheless, for this presentation, the results of the flow rate $Q = 5\ dm^3s^{-1}$ are presented.

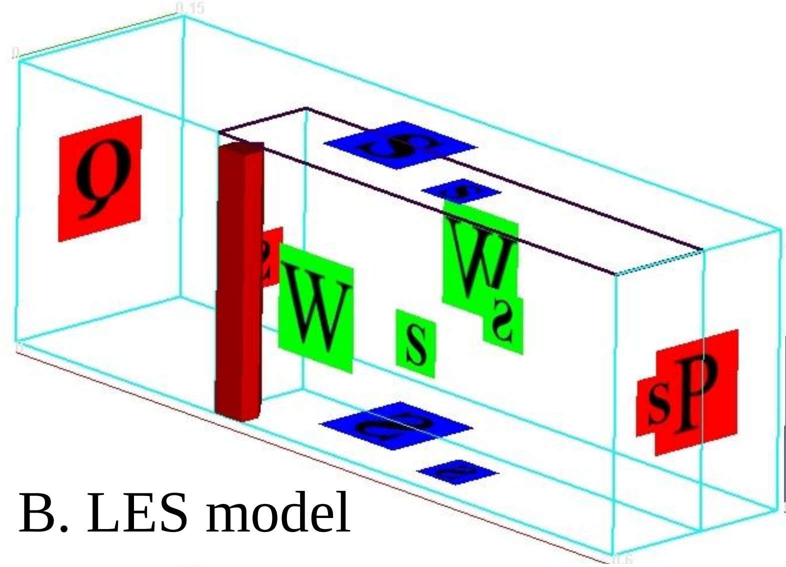


Outline

1. Introduction
2. Experimental Works
3. **Numerical Modelling**
4. RANS vs LES
5. Conclusions



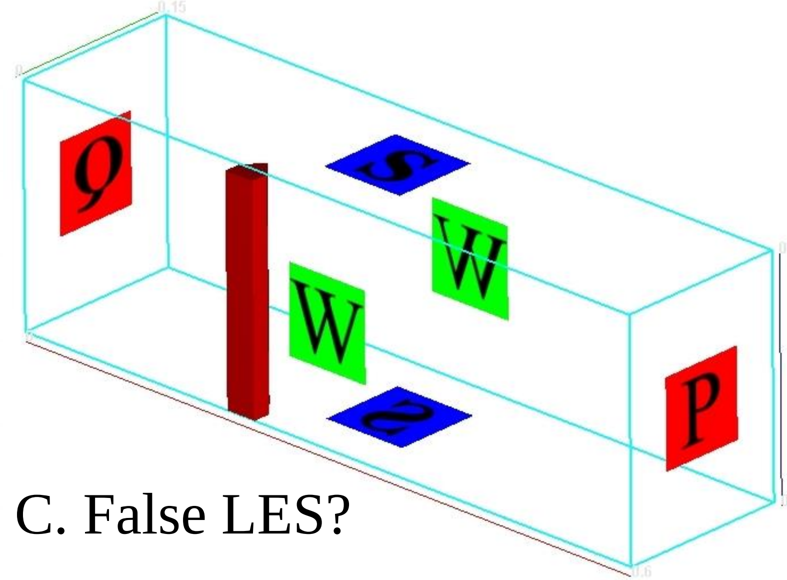
A. RNG model



B. LES model

For this research, three scenarios were modeled:

A. One RANS (RNG) model
B. One LES model
C. One FALSE LES?



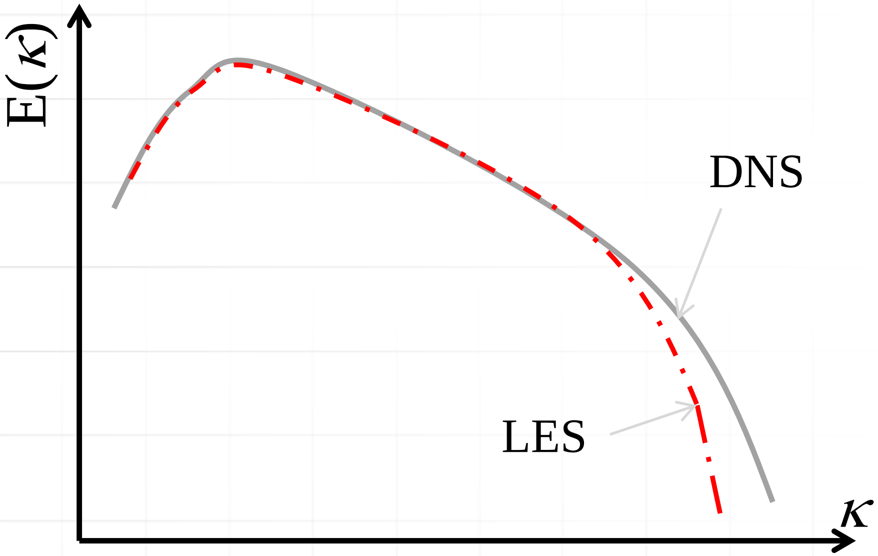
C. False LES?

The philosophy of LES

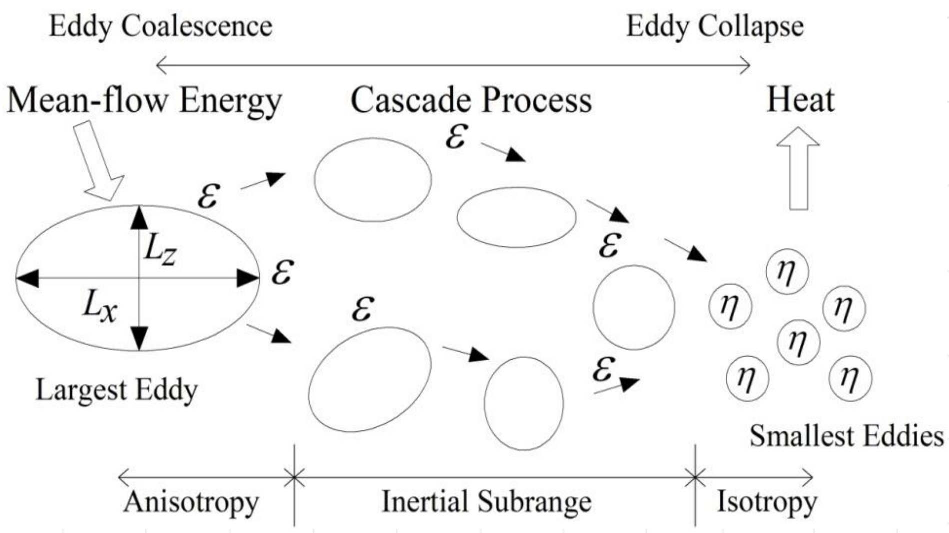
The idea of Large Eddy Simulations is to calculate explicitly the large scales by solving the 3D unsteady equations and to model the motion of the small scales (Rodi et-al, 2013).

Big whorls have little whorls,
Which feed on their velocity;
And little whorls have lesser whorls
An so on to viscosity.

Richardson (1922)



Approximating the real turbulence spectra to the modeled spectra is one of the goals of a LES run.



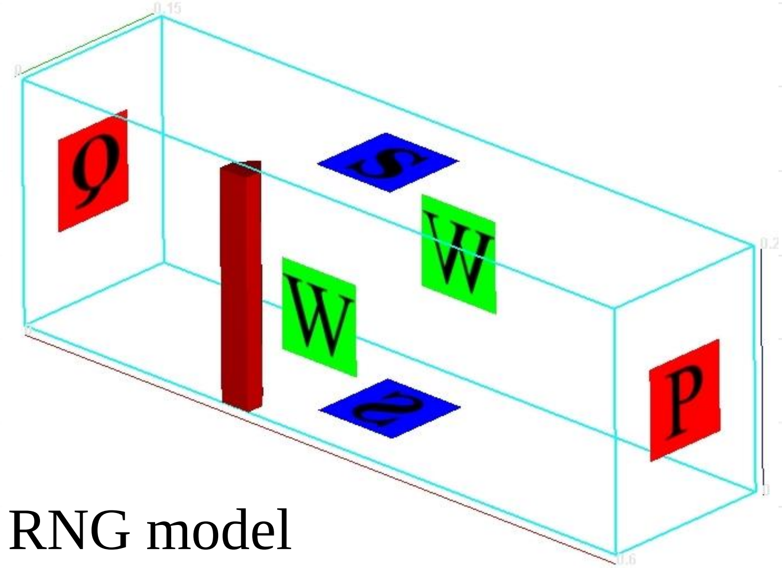
Scenario 1. RNG Model

The boundary conditions were defined using the conditions reprodced at the lab.

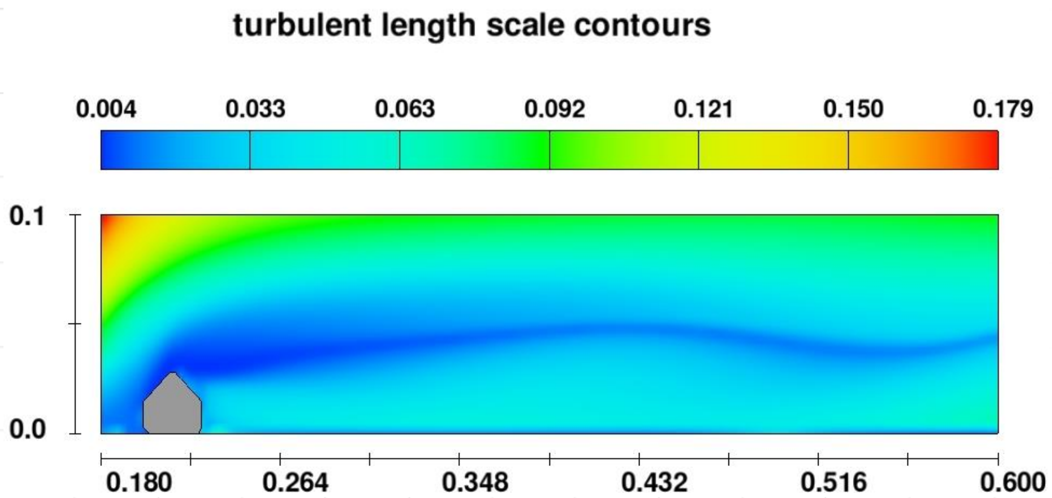
Scenario 2. LES

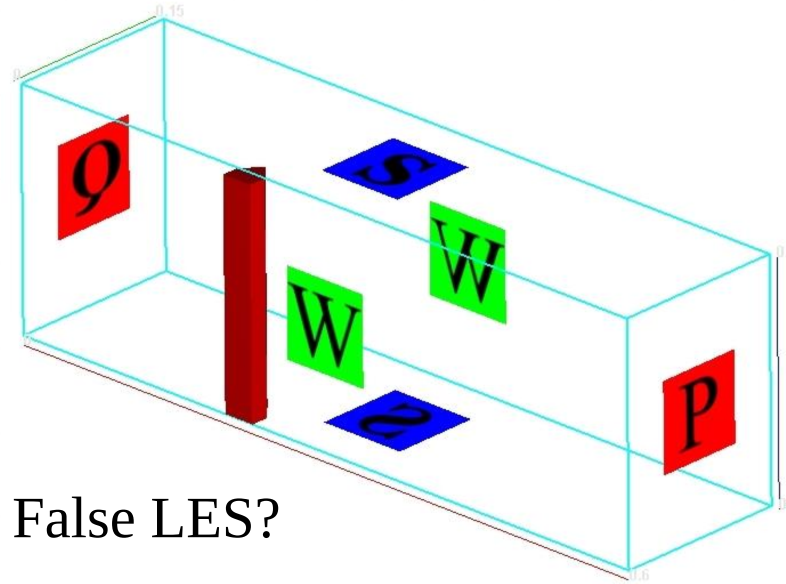
Scenario 3. False LES?

Is the size of the mesh proper in order to carry out a LES?

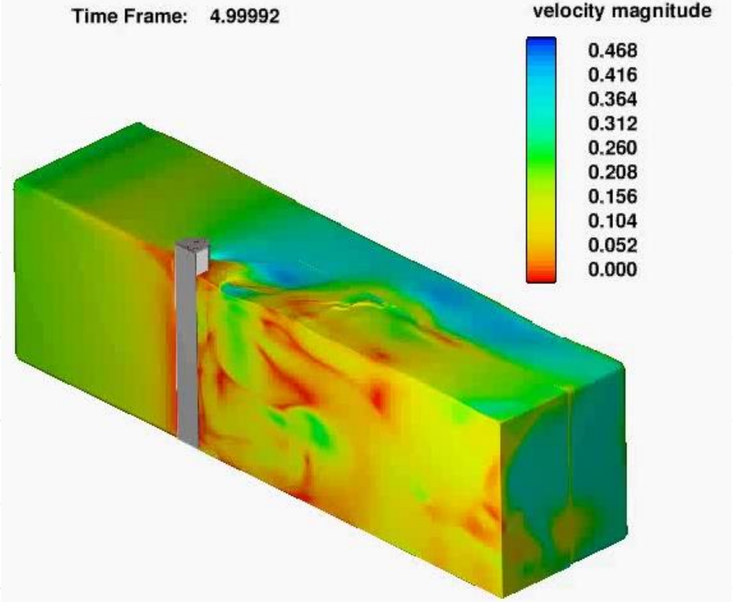


RNG model



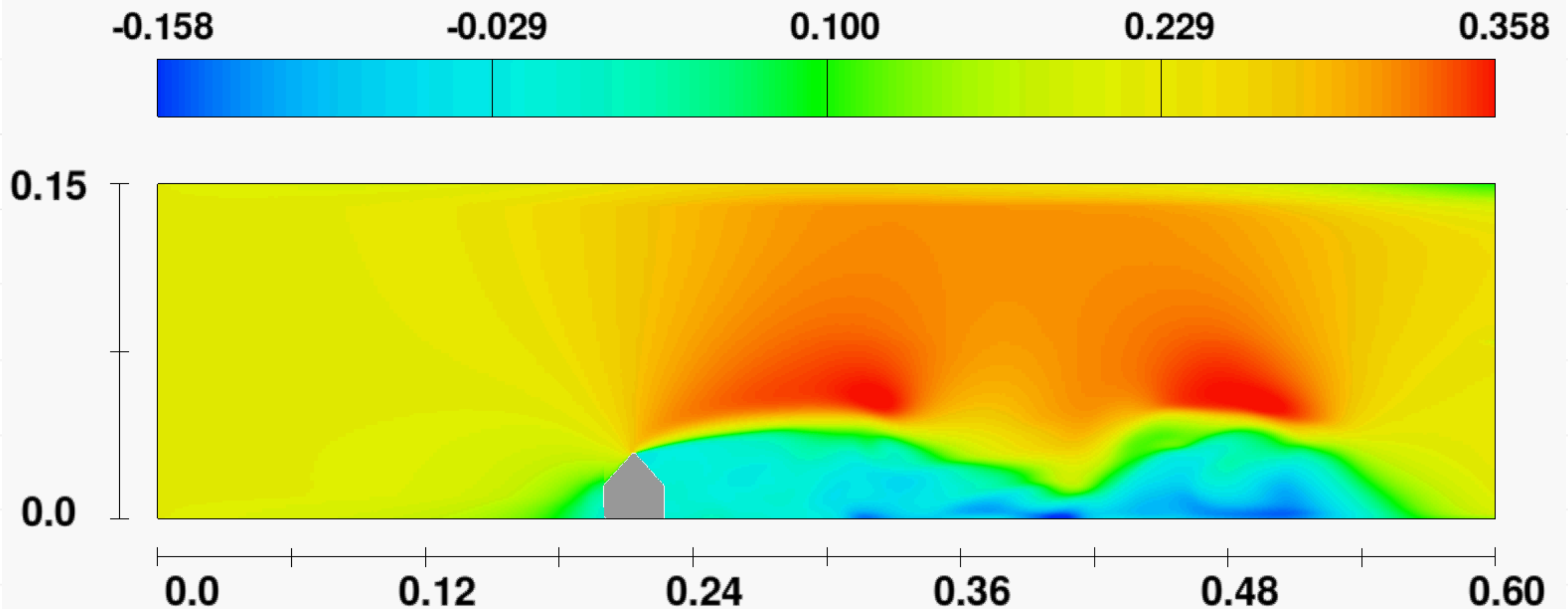


False LES?

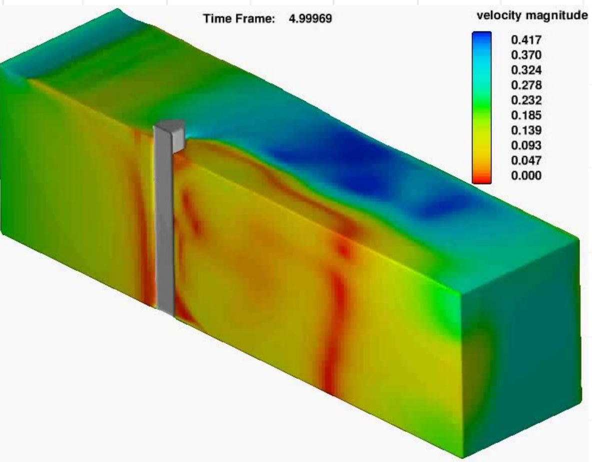


Scenario 2. LES Model

The output of the second scenario is depicted in the left. The velocity magnitude is displayed using colored contours. In Fig. below, the velocity component, in downstream direction, at a depth of 8 cm is depicted. It is possible to observe the dead zones and secondary currents provoked by flow turbulence.



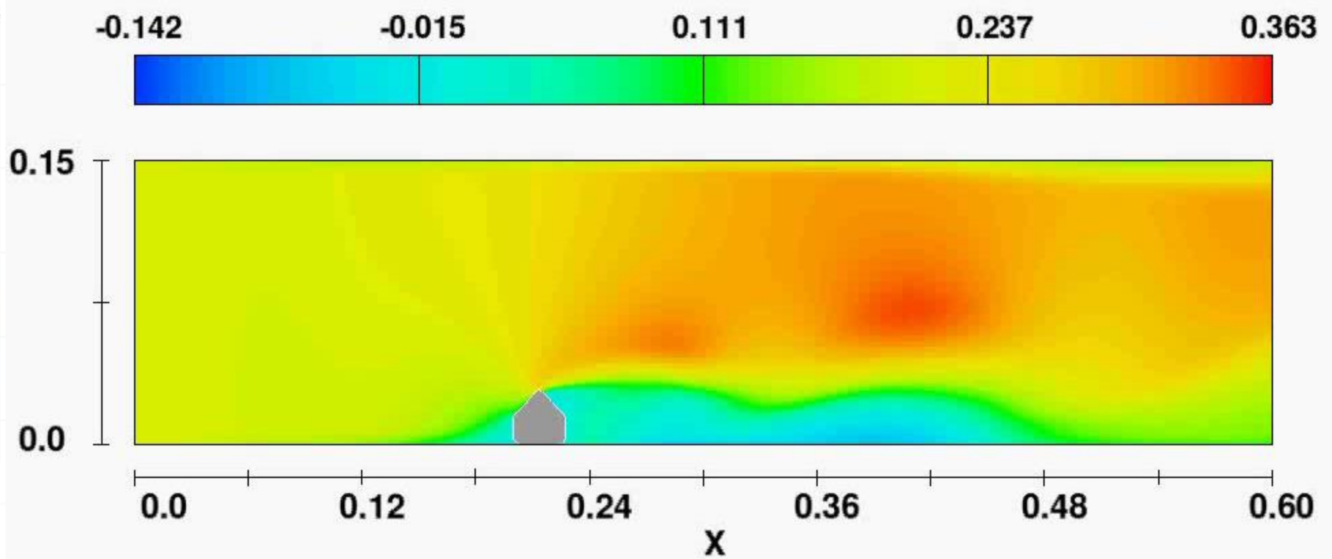
Wrocław University of Science and Technology



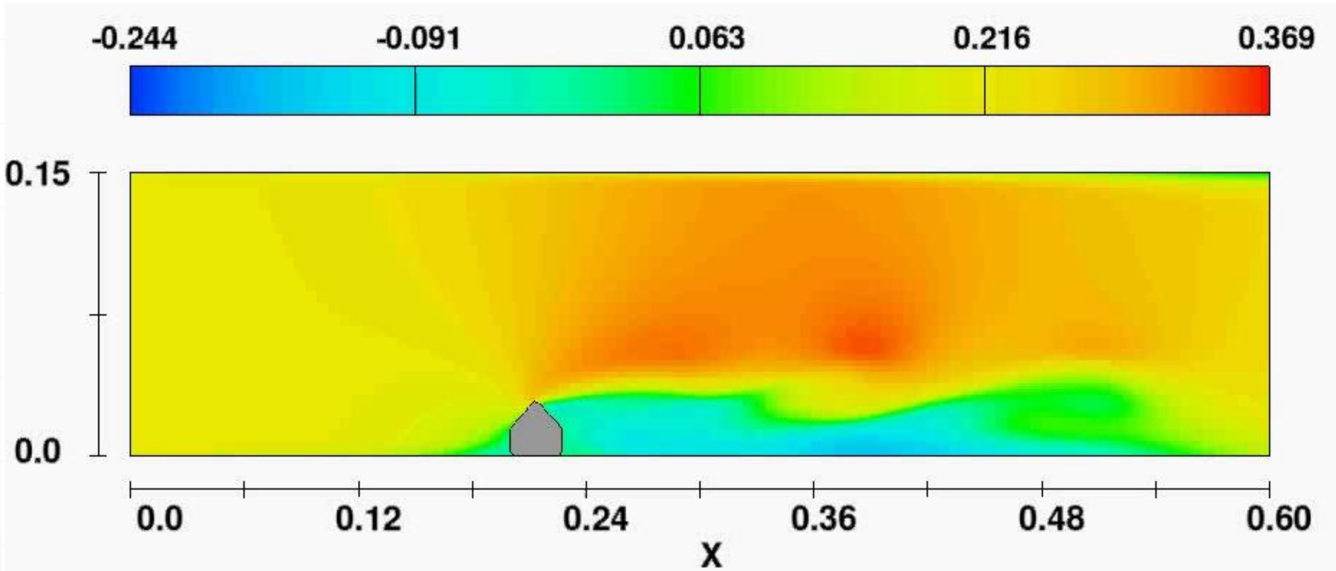
Scenario 1. RANS Model

The output of the first scenario is depicted in above (left and rigth).

The velocity magnitude is displayed. No concrete turbulence structures can be accurately identified.



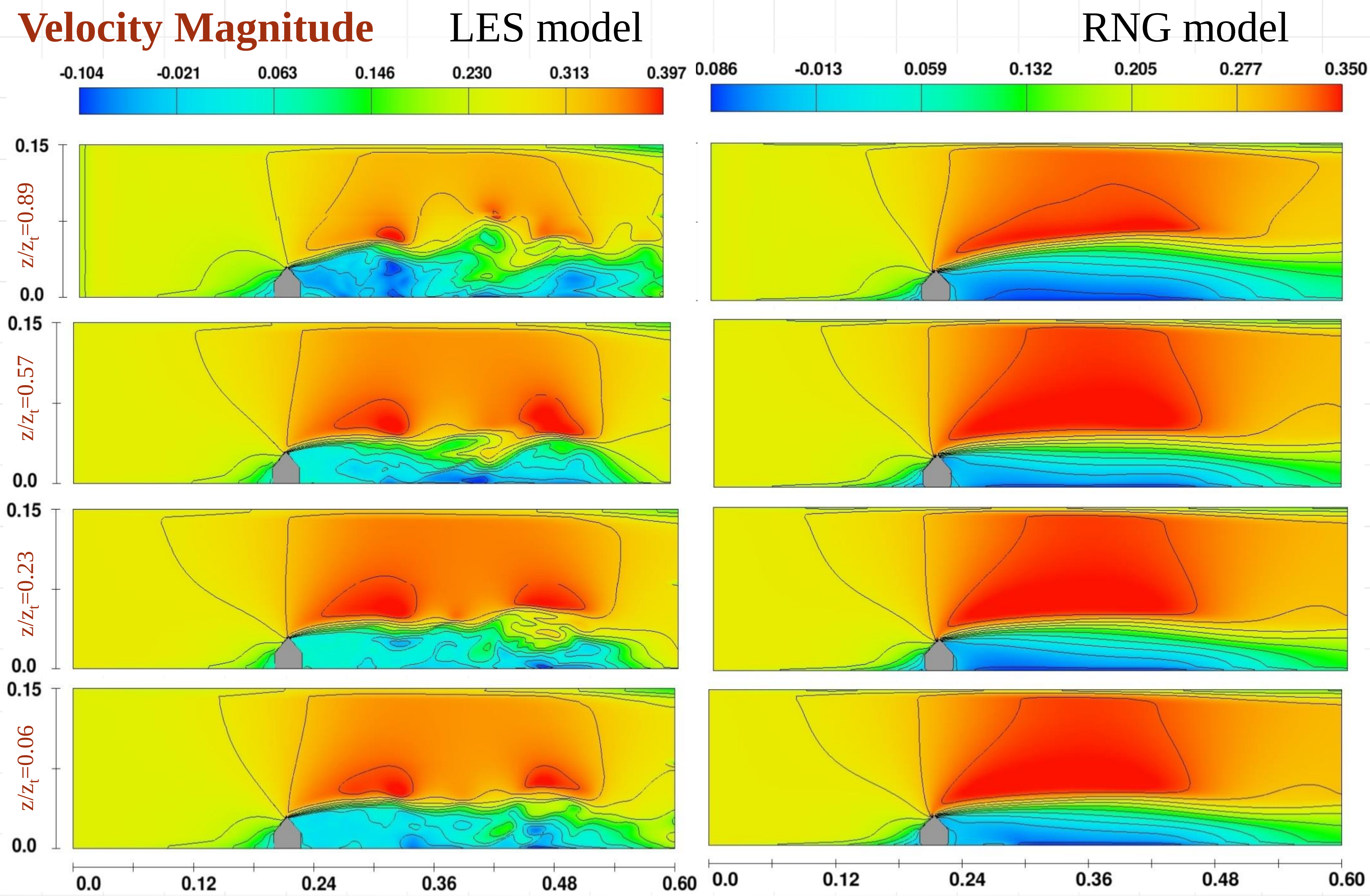
Scenario 3. False LES: The output of the third scenario is depicted below. The velocity field is displayed.



Outline

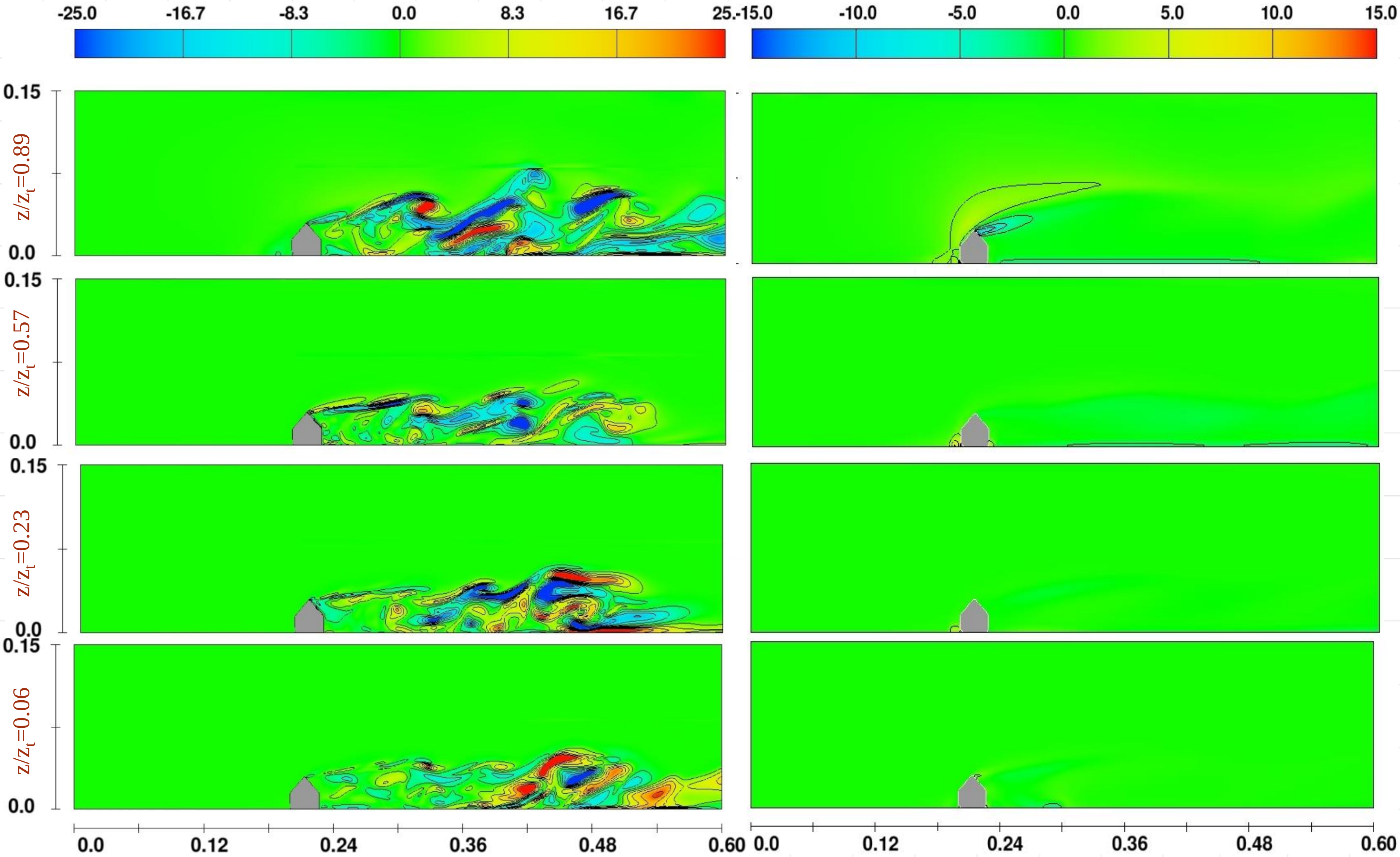
1. Introduction
2. Experimental Works
3. Numerical Modelling
4. **RANS vs LES**
5. Conclusions

Velocity Magnitude

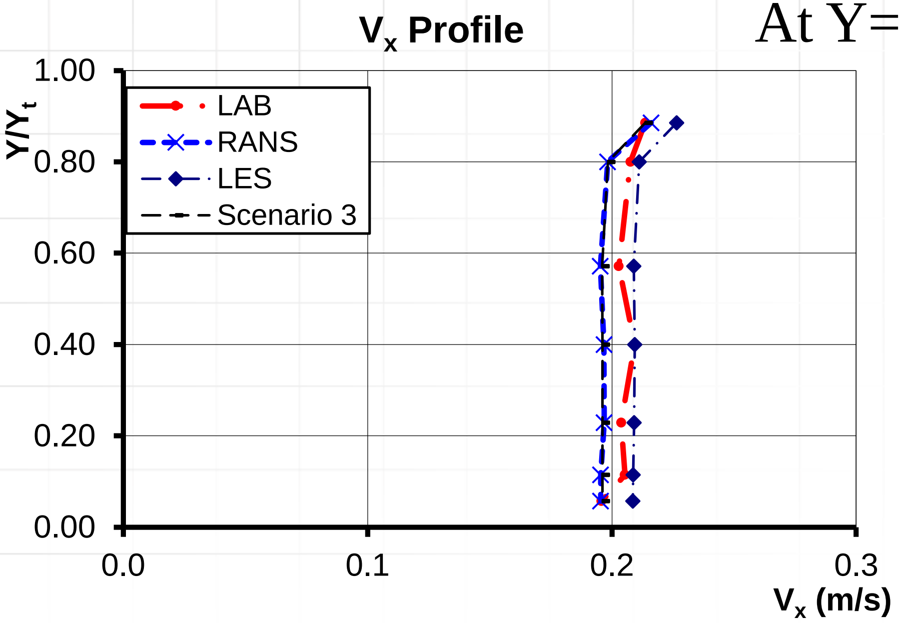


Vorticity

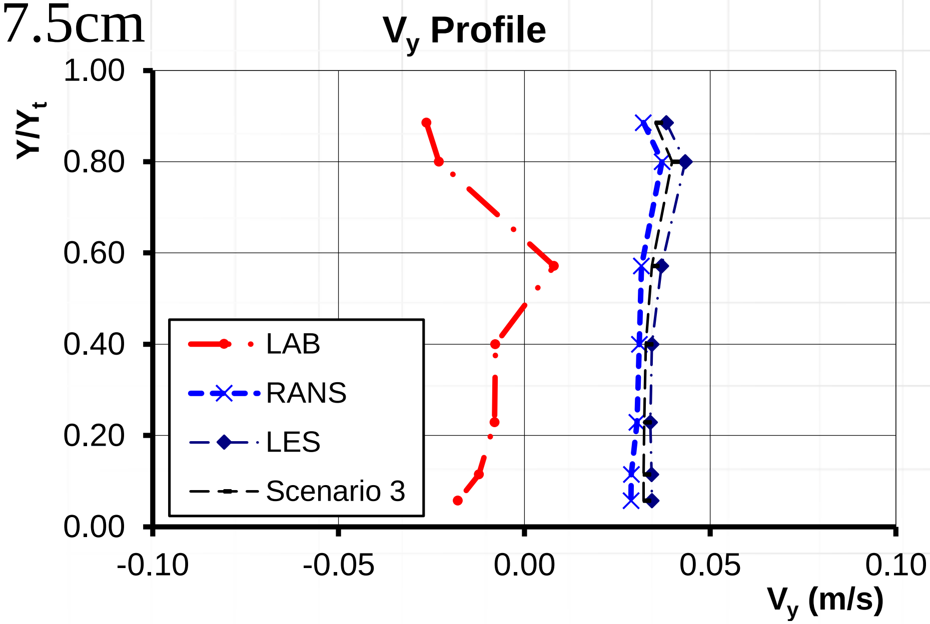
LES model | RNG model



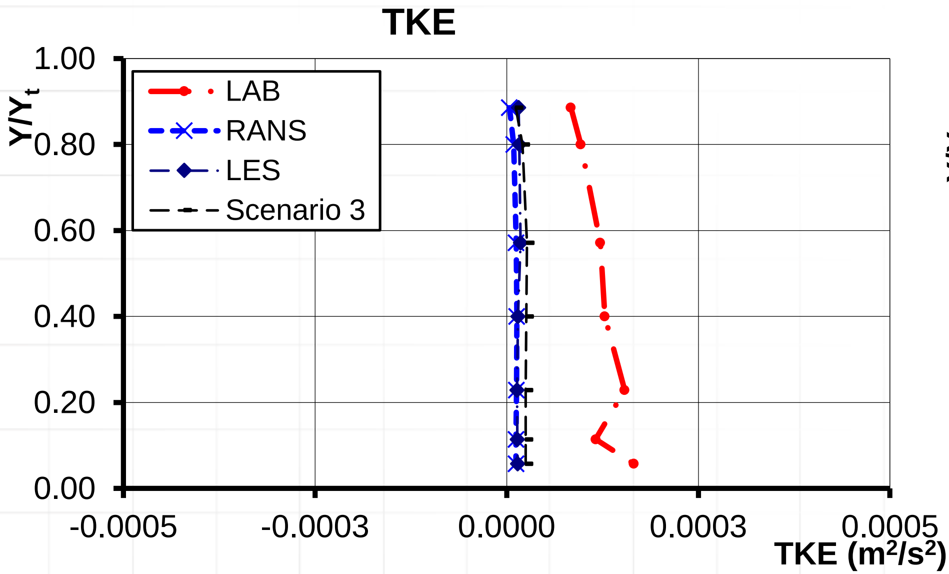
At Y=7.5cm



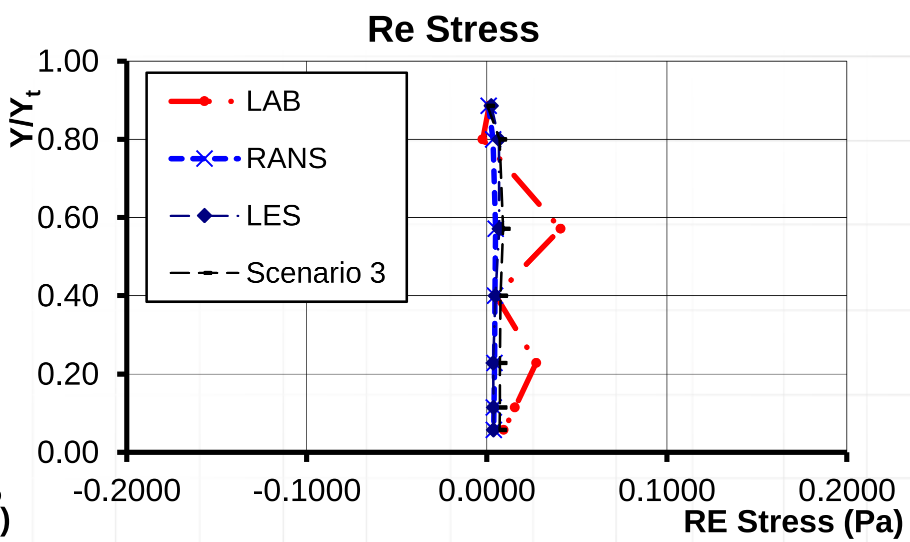
V_x Profile
Y/Y_t
1.00
0.80
0.60
0.40
0.20
0.00
0.0
0.1
0.2
0.3
V_x (m/s)
LAB
RANS
LES
Scenario 3



V_y Profile
Y/Y_t
1.00
0.80
0.60
0.40
0.20
0.00
-0.10
-0.05
0.00
0.05
0.10
V_y (m/s)
LAB
RANS
LES
Scenario 3

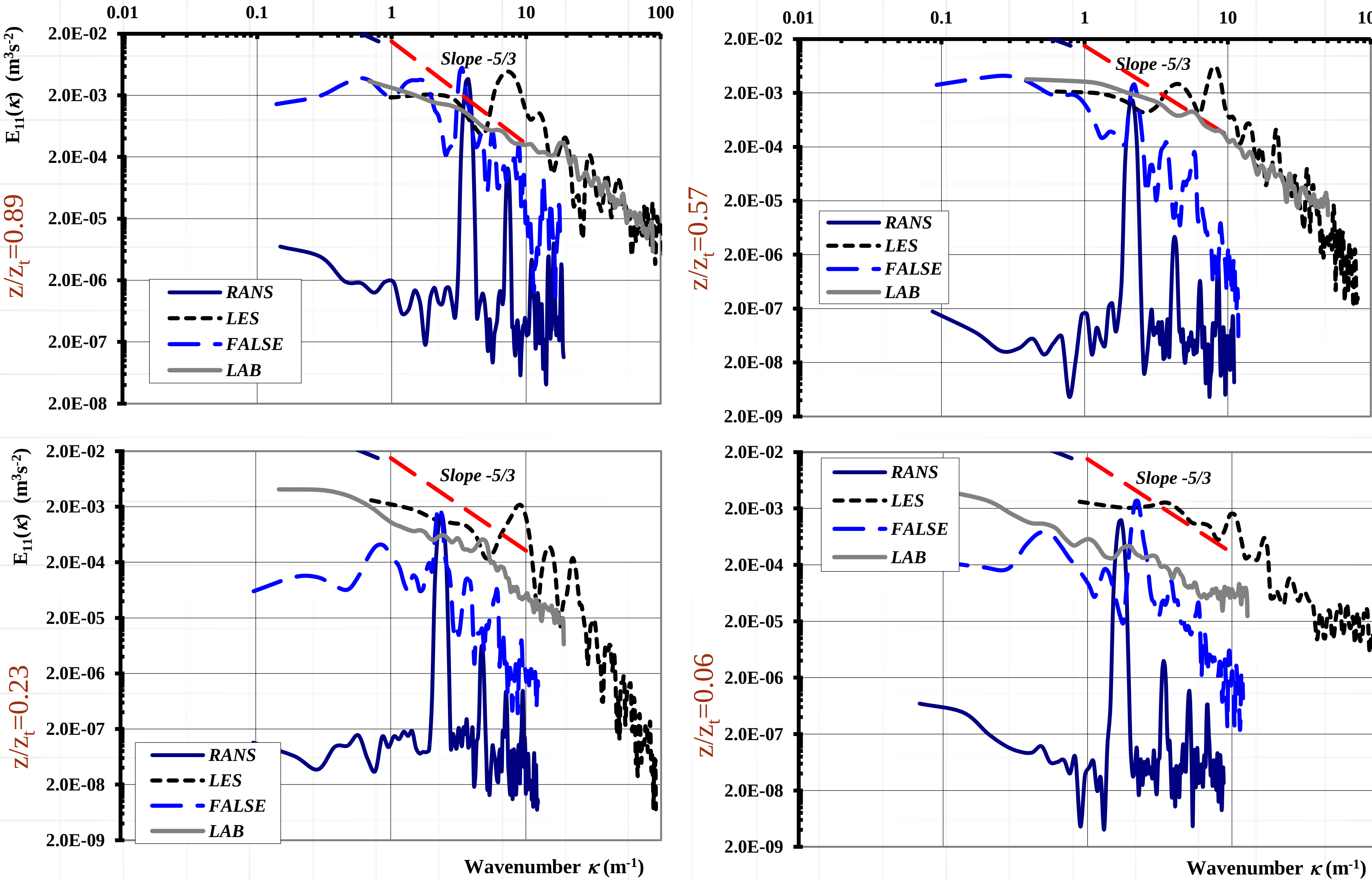


TKE
Y/Y_t
1.00
0.80
0.60
0.40
0.20
0.00
-0.0005
-0.0003
0.0000
0.0003
0.0005
TKE (m^2/s^2)
LAB
RANS
LES
Scenario 3



Re Stress
Y/Y_t
1.00
0.80
0.60
0.40
0.20
0.00
-0.2000
-0.1000
0.0000
0.1000
0.2000
RE Stress (Pa)
LAB
RANS
LES
Scenario 3

At Y=3.5cm



Wrocław University of Science and Technology

Parameter	RANS	LES	Scenario 3
No. grid elements	285 000	6 685 000	285 000
Time step (mean):	1.36E-03s	6.80E-04	1.75E-03
Comp. time (per sec. of real time)	around 12 min	around 305 min	around 9.5 min
Max. pressure residual	1.0821	0.3428	1.8667
Turbulence structures	Non depicted	Depicted	Depicted?
Turbulence Power Spectrum	No spectrum	Theoretically correct	Theoretically correct?
Velocity profile	Correct	Correct	Correct

Outline

1. Introduction
2. Experimental Works
3. Numerical Modelling
4. RANS vs LES
5. **Conclusions**

- The fluid-structure interaction between a groyne and water in motion was analyzed at the laboratory and using CFD.
- One of the purposes of this research was to choose the most proper computational tool to analyze turbulence dynamics.
- Three simulations were carried out: a RANS model, a LES model and a second LES using a coarse mesh. The results of the simulations were compared with the results of the experiments.
- The results of the simulations and the laboratorial works present acceptable agreement.
- Choosing a correct mesh size plays a vital role to carry out a proper analysis of turbulence. Only the LES model (Scenario 2) was able to estimate the most important turbulence characteristics.
- LES demonstrated that this is the most proper tool to analyze the behavior of water in the dead zones downstream the groyne. Thus, other related phenomena should be analyzed using LES.

Is LES the right approach?

The first point to be made in response to this question is that, given the broad range of turbulent flow problems, it is valuable to have a broad range of approaches that can be applied to study them. There is not one 'right' approach. As discussed more fully elsewhere, while the use of LES in engineering applications will certainly increase in the future, the use of simpler RANS models will be prevalent for some time to come. It is valuable, therefore, to continue to seek improvements to the full range of useful turbulence modelling approaches (Pope, 2004).

Why and when to do LES?

1. To understand turbulence structure;
2. To acquire more accurate estimates of turbulence statistics;
3. More valuable as a CFD tool for scientific purposes;
4. To analyze problems at a considerable small scale.

Thank you very much for your attention

Dziękuję bardzo za uwagę

Oscar Herrera-Granados