INVESTIGATION OF A FLOW FIELD GENERATED BY A FRACTAL GRID BASED ON EXPERIMENTAL DATA AND CFD SIMULATIONS

A. Fuchs[1], W. Medjroubi [1], N. Reinke [1], G. Guelker [1] & J. Peinke [1]
[1] *ForWind, Institute of Physics, University of Oldenburg, Oldenburg, Germany*

Abstract

Fractal grids generate turbulence by directly exciting many length-scales of different sizes simultaneously, rather than using the nonlinear cascade mechanism to obtain multiscale excitation, as it is the case for classical grids. These scales influence each other and show very different properties compared to all previously documented turbulent flows [1, 2, 3, 5]. In this work we present experimental wind tunnel and computer fluid dynamics (CFD) studies of the turbulent flow generated by a fractal grid under the same conditions. We did an extensive statistical study and a direct comparison between the experimentally and numerically acquired time series in order to investigate and compare one-point- and two-point-statistics. In addition we present an application of a stochastic method, so-called Langevin approach, to the experimentally and numerically acquired velocity increment time series to examine three-point-statistics in terms of Kramers-Moyal coefficients.

INTRODUCTION

The grid used in the present work is shown in figure 1, it is placed at the inlet of the test section. In general fractal grids are constructed from a multiscale collection of obstacles which are based on a single pattern which is repeated in increasingly numerous copies with different scales. Our fractal grid is based on a square shape with $N = 3$ fractal iterations. The fractal iterations parameter is the number the square shape is repeated with different scales. At each iteration $j = 0, ..., N - 1$ the number of squares is four times higher than in the iteration $j - 1$. Each scale iteration j is defined by a length L_j and a thickness t_j of the squares bars constituting the grid.

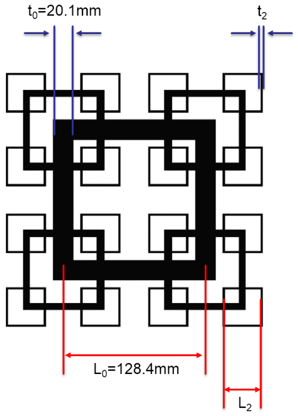


Figure 1: Illustration of the $N = 3$ space-filling square fractal grid (used for the present work).

EXPERIMENTAL SETUP

The experiments were conducted in a closed loop wind tunnel with test section dimensions of 200 cm x 25 cm x 25 cm (length x width x height) at the University of Oldenburg. The wind tunnel has a background turbulence intensity of approximately 2% for $U_\infty \leq 10$ m/s. The inlet velocity was set to 10 m/s, which corresponds to a Reynolds number related to the biggest bar length $L_0 = 128.4$ mm of about $Re_{L_0} = U_\infty L_0/\nu = 83800$, where ν is the kinematic viscosity. Constant temperature anemometry measurements of the velocity were performed using (*Dantec 55P01* platinum-plated tungsten wire) single-hot-wire with a wire sensing length of about $l_w = 2.0 \pm 0.1$ mm and a diameter of $d_w = 5$ μm which corresponds to a length-to-diameter ratio of $l_w/d_w \approx 400$. A *StreamLine* measurement system by *Dantec* in combination with CTA Modules 90C10 was used for the measurements. The hot-wire was calibrated with *Dantec Dynamics Hot-Wire Calibrator*. The overheat ratio was set to 0.8. In the streamwise direction, measurements were performed on the centerline in the range between 5 cm $\leq x \leq$ 176 cm distance to the grid. The data was sampled with $f_s = 60$ kHz with a *NI PXI1042* AD-converter and 3.6 million samples were collected per measurement point, representing 60 seconds of measurements data. To satisfy the Nyquist condition, the data were low-pass filtered at frequency $f_l = 30$ kHz.

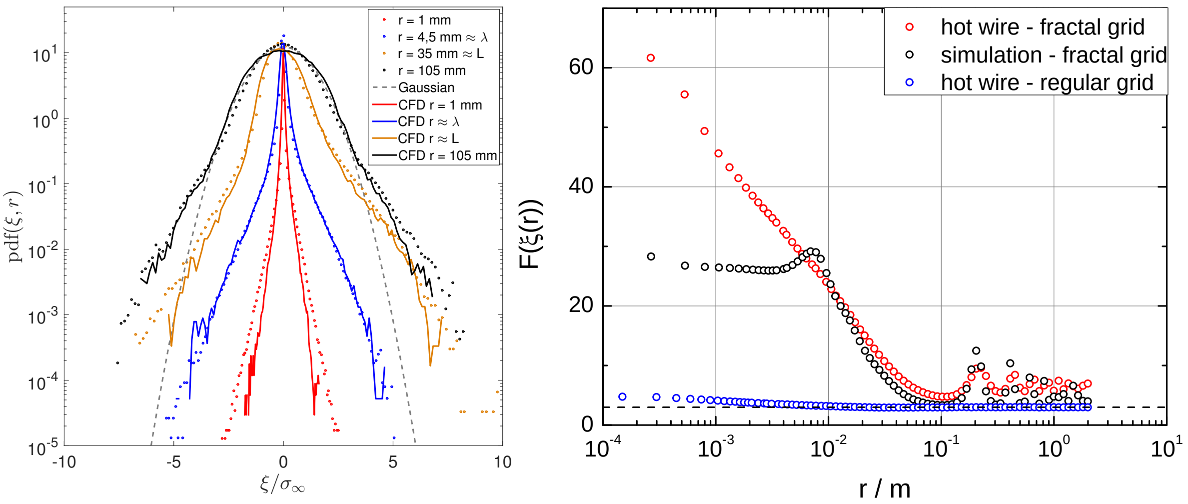
COMPUTATIONAL SETUP

The numerical simulation was set up analogous to the experiments in order to compare the obtained results with the experiments in a consistent manner. The simulations were conducted for equal Reynolds number, numerical data was sampled with the same sampling frequency of 60 khz at same probes locations and the computational domain dimension is in accordance to the dimension of the test section in the wind tunnel. A Delayed Detached Eddy Simulation (DDES) with a Spalart-Allmaras background turbulence model [6] was performed using the open source code OpenFOAM [4] to solve the three dimensional, incompressible Navier-Stokes equations. DDES is a hybrid method stemming from the Detached Eddy Simulation method (DES) [7], which involves the use of Reynolds Averaged Navier-Stokes Simulation (RANS) at the wall and Large Eddy Simulation (LES) in the free flow. The numerical mesh was generated using the built-in OpenFOAM meshing tools blockMesh and snappyHexMesh [4]. As a result, an unstructured mesh of 24 million cells is obtained, where regions of interest in the wake are refined.

RESULTS AND DISCUSSION

The results are presented and compared for the experimental and computational studies in terms of one-point- and two-point-statistics on the centerline as a function of the distance to the grid. A very good qualitative and quantitative agreement was found between the results from the experiment and the simulation regarding the investigated one point statistics such as exponent of turbulence decay power law, higher-order statistics (skewness and flatness) and probability density functions of turbulent velocity fluctuations. The experimentally and numerically acquired time series are further compared using two-point-statistics such as analysis of energy spectra and structure functions. In figure 2a the probability density functions (pdfs) of the velocity increments $\xi(r)$ normalized to σ_∞ is given at four different spatial scales r. We use normalized velocity increments so that normalized increment pdfs can be compared to a normal distribution of standard deviation 1. At least for the qualitative behavior both techniques show that the turbulence statistics are clearly non-Gaussian distributed at a distance of 20 cm to the grid. Especially for small spatial scales r heavy tails presented in the pdfs so therefore the turbulence is highly intermittent. This intermittent behavior of the fractal turbulence can be quantified through the flatness of the increment pdfs as a function of the spatial scale r which is presented in figure 2b. In accordance to the investigation of increment pdfs, it is obvious that the smaller the spatial scale r the higher the flatness of increment pdfs. In addition, this plot shows the effect of RANS turbulence modelling for spatial scales smaller than approximately 10 mm. The results from the experiment and the simulation are completely off in this region. This is also shown in the differences of the pdfs at a spatial scales $r = 1mm$ (figure 2a). To compare the results with ordinarily turbulence generation we also experimentally investigated a classical grid with the same blockage ration (36 %), the same effective mesh size and the same edge length of the grid.

The analysis will be extended to include the Langevin approach to the experimentally and numerically acquired velocity increment time series. These analysis will provide a better insight on the turbulence generated by fractal grids and the suitability of CFD models (DDES) to characterize this turbulence.



(a)

(b)

Figure 2: (a) Probability density functions of the velocity increments $\xi(r)$ normalized to $\sigma_\infty = \lim_{r\to\infty} \langle \xi(r) \rangle$ at four different spatial scales r obtained for the experimentally (dots) and numerically (solid lines) acquired time series at a distance of 20 cm to the grid and (b) flatness of the increment pdfs as a function of the spatial scale r at a distance of 20 cm to the grid. The increment pdfs are shifted in the vertical direction for clarity of presentation.

The presentation will show comparisons between the experimental and computational studies in terms of one-point-, two-point and three-point-statistics.

References

[1] D. Hurst and J. C. Vassilicos., (2007), Scalings and decay of fractal-generated turbulence. *Physics of Fluids,* ***19****(3):035103,*

[2] S. Laizet and J.C. Vassilicos, (2011), DNS of fractal-generated turbulence. *Flow, Turbulence and Combustion,* ***87****(4):637âĂŞ705,*

[3] B. Mazzi and J. C. Vassilicos., Fractal-generated turbulence. *Journal of Fluid Mechanics,* ***502****:65âĂŞ87*

[4] OpenFOAM, (2013), The open source computational fluid dynamics toolbox. *http://www.openfoam.com/*

[5] R. Stresing, J. Peinke, R.E. Seoud, and J.C. Vassilicos,(2010), Defining a new class of turbulence. *Physical Review Letters,* ***104****:194501âĂŞ4,*

[6] P. R. Spalart, S. R. Allmara, (1994), A One-Equation Turbulence Model for Aerodynamic Flows. *La Recherche Aerospatiale 1,****5-21***

[7] P. R. Spalart, M. Strelets, S. R. Allmara, (1997) Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. *Advances in DES/LES 1,* ***137-147***