Physica A 474 (2017) 260–266

Contents lists available at ScienceDirect

Physica A

journal homepage: www.elsevier.com/locate/physa

Comparative analysis on turbulent regime: A self-affinity study in fluid flow by using OpenFoam CFD

A.S. Nascimento Filho [a,*], J.W.G. de Souza [a], A.R.B. Pereira [a], A.A.B. Santos [a], I.C. da Cunha Lima [a,b], A.T. da Cunha Lima [c], M.A. Moret [a,d]

[a] *SENAI CIMATEC Salvador, Bahia, Brazil*
[b] *Pursuelife - Consultancy on Applied Science - Salvador, BA, Brazil*
[c] *Universidade Federal da Bahia, Salvador, BA, Brazil*
[d] *Universidade do Estado da Bahia, Salvador, BA, Brazil*

HIGHLIGHTS

- We studied self-affinity process in fluid velocities simulation.
- We compare sub-diffusive process between two different systems.
- We observed chaotic behavior on both process.
- We discuss aspects of the similarity measures.

ARTICLE INFO

Article history:
Received 3 September 2016
Received in revised form 23 December 2016
Available online 18 January 2017

Keywords:
Detrended fluctuation analysis
Self-affinity process
Turbulence

ABSTRACT

We investigate the self-affinity of a fluid velocity time series in turbulent regime. We use the CFD OpenFoam library to simulate the flow of fluid in a duct with two obstacles injecting vorticity downstream. We calculated the fluid velocity in three sections in of the structure. The simulation presented more turbulences in sections further away from obstacles. We assess the self-affinity of properties of those sections by using the Detrended Fluctuation Analysis method. This scaling method presented two behaviors. First, a persistent behavior after passing through obstacles. Second, a sub-diffusive behavior in turbulent sections, presenting a long-range correlation α exponent ranging between 1.0 and 1.5. The findings suggest that chaotic states tend to follow a sub-diffusive process.

1. Introduction

During the last years the study of processes involving Fluid Dynamics gained great importance in many fields of science, from Astrophysics to Medicine [1–3]. A great part of the success of the use of Fluid Dynamics is due to the advances on observation techniques and developments on high performance computation. The dynamics of a classical fluid is governed by two conservation laws, the mass conservation, and the momentum conservation. Once the Stokes hypothesis [4] is assumed, the momentum conservation law reduces to the Navier–Stokes equations, a second order, non-linear, partial differential equation on the velocity vector field. The relative importance of two terms in that equation determine the flow regime. These terms are the one due to the viscosity, responsible for the dissipation process, and the so called inertia term, the

* Corresponding author.
E-mail address: aloisio.nascimento@gmail.com (A.S. Nascimento Filho).

http://dx.doi.org/10.1016/j.physa.2017.01.074
0378-4371/

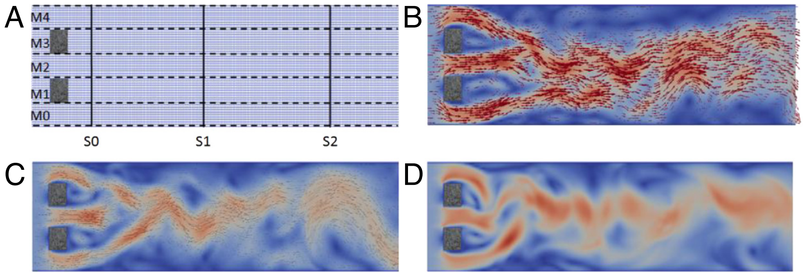

Fig. 1. (A) Mesh of points where the simulation is performed, showing two obstacles generating turbulence, and the sections (S0, S1 and S2), together with the points at the axes M0 to M4; (B) Snapshot of the turbulent flow at with display of arrows in size 5.5; (C) snapshot of the turbulent flow at with display of arrows in size 3.5; (D) snapshot of the turbulent in high resolution (on solid mode).

non-linear one. Whenever the viscous term dominates the flow is laminar. On the other side we have a turbulent flow, where the velocity field corresponds to a stochastic process [5]. The ratio between the non-linear and the viscous term is known as the Reynolds number [6]. The transition from a laminar into a turbulent flow is associated to a certain value of the Reynolds number, which depends on the system under consideration. Physically, near the transition a minimal perturbation leads the system to respond strongly, reflecting the importance of the non-linearity of the Navier–Stokes equation. Although the equation of motion is well defined, rarely an exact solution exists without a severe approximation on the model representing the real system. In general, this is possible in cases of fluids with high viscosity. On the other hand, in cases of very small viscosity, we face a problem of singular perturbation [7]. In such cases, we can imagine a solution in which a laminar flow exists, except for a very thin layer near a fluid–solid interface, a region called boundary layer [4]. It is worthwhile to stress that flows in nature are rarely laminar.

In this work we are interested in comparing two completely different kind of flows, being treated by totally different methods. One is the internal turbulent flow of water in a channel. The other system is the thermal radiation originating from the combustion of acetylene gas enriched with oxygen and natural gas, at different concentrations, during the combustion process. The turbulent regime of flow in the latter case is treated experimentally, while that in the former is treated by numerical simulation. In both cases we are interested in obtaining correlation functions of the velocity field determining some characteristics of the stochastic process. The study of the turbulence is performed by analyzing the self-affinity time series of the velocity field, using the stochastically Detrended Fluctuation Analysis (DFA) [8]. We show that the results confirm remarkable similar characteristics, from the statistical point of view.

In the case of the turbulent flow in a channel, we simulate the flow in a domain as described in (Fig. 1(A)). Water is injected into the channel from the left side. Two obstructions are placed in order to produce turbulence. They work as vorticity injector into the flow. Water is treated as an incompressible, Newtonian fluid. The Navier–Stokes equation, as decried below, are solved numerically using the OpenFOAM library [9], an open source package written in C++ language, using the finite volume method.

In the case of the thermal radiation, an experiment was performed using an apparatus consisting of a combustion chamber coupled with instruments to analyze gases and soot and to measure thermal radiation. The experimental device has a burner composed of two concentric tubes where a flame is generated in the chamber. According to the schema of the experimental device, the radiation was measured by a photo-detector (orifices) located outside the chamber. For more details about that apparatus see Ref. [10].

2. Turbulent regime simulation

The motion of a fluid can be considered either as a flow of a continuous medium or as the motion of a fluid particle. Since the fluid particle moves solidary to the continuous medium, its speed at a given position $\mathbf{r}$ and time t, is the same as the velocity field of the continuous media at that position and time. Together with the mass conservation of the fluid and the Stokes hypothesis, the momentum conservation determine the flow dynamics to be governed by the Navier–Stokes equations:

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \eta \nabla^2 \mathbf{u} - \rho g \hat{k}, \qquad (1)$$

where the material derivative is defined by:

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (\mathbf{u} \cdot \nabla). \qquad (2)$$

In Eq. (1) ρ is the specific density of the fluid, the operator ∇^2 is the Laplacian, η is the viscosity of the fluid, p is the pressure and $\hat{k}$ is a unitary vector in the direction of gravity. As mentioned in Section 1, the second term of the material derivative when operating over the velocity field gives rise to the non-linear term of the Navier–Stokes equation. Fluids with weak viscosity,