Petroleum & Coal
ISSN 1337-7027

Available online at www.vurup.sk/petroleum-coal
Petroleum & Coal 55 (1) 20-25, 2013

A COMPARATIVE NUMERICAL INVESTIGATION OF FLOW THROUGH CHANNEL EXPANSION SYSTEM, USING FINITE VOLUME AND FINITE ELEMENT METHODS

Iman Abrishamchi, Ahmad Okhovat, Seyed Mostafa Nowee

Department of Chemical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran
Corresponding author: Seyed Mostafa Nowee, E-mail: Nowee@um.ac.ir

Received September 17, 2012, Accepted January 30, 2013

Abstract

Behavior of the fluid in relation to the geometry of the system is of great interest in piping systems due to the erosion-corrosion and heat transfer phenomena. This paper deals with computation of flow through a two dimensional, symmetric constriction of an incompressible Newtonian fluid in turbulent state using two commercial finite element and finite volume codes, namely COMSOL MULTIPHYSICS and FLUENT respectively. Computation has been performed in order to determine axial and radial velocities at different position with both of these simulators in order to compare finite element and finite volume results in a practical system. In order to assess the accuracy of the findings of the present study, the results have been compared with the experimental data. The results of the present study reveal that the standard k-ε model could be applied safely in such geometries for predicting the flow behavior. Also, results shows that using finite element methods gives more compatible results with experimental data.

Keywords: Turbulent flow; Finite element; Finite volume; k-ε model; sudden expansion.

1. Introduction

For too many applications of pipelines in industries such as fluid transport pipelines, coolers, air conditioning systems and etc., dynamic of fluid flow has been considered widely in engineering. Fluid flow pattern is changed while it passes contracting-expanding geometry, so this may influence over some phenomena such as heat transfer, corrosion, erosion rate, efficiency.

Several studies have been done on the flow patterns in pipes without sudden expansion and contraction which have been done by T. Mullin and J. Peixinho [8], M. Sahu et.al. [12], Michael A. Traetow [14], A.P. Willis et.al. [15]. But few researchers such as M. Founti and A. Klipfel [4], and Cian Davis and Patrick Frawley [2] , because of its complication, have studied on fluid flow in contracting-expanding systems.

In general, the overall pattern of flow lines depends on point to point changes in axial and radial velocity. In the present investigation, simulation of turbulent flow in pipe lines with an abrupt expansion have been done by COMSOL MULTIPHYSICS v3.5a, which uses finite element method, and FLUENT v6.3, which uses finite volume method, software's and finally the results have been compared with experimental data.

2. Theory

In industrial scale, fluid flow patterns are often turbulent and for the prediction of the process, mathematical modeling is needed. Turbulent flow motion has a significant effect on chemical and physical phenomena such as heat transfer rate.

Mathematical modeling based on known physical and chemical principles express the behavior of engineering systems. Computational fluid dynamics (CFD) can simulate many processes such as turbulent combustion and radiation by using mathematical modeling. Simulation of this phenomenon can be useful and reliable if the infrastructure of models is sufficiently accurate. Numerical method is the basis of computational fluid dynamics. Computational fluid dynamics methods based on numerical mass, energy and momentum continuity equations. Solving a problem in computational fluid dynamics method consists of two stages: First, total fluid space is divided into small components, then the continuity differential equations for each of these components are resolved. As a result, a large number of simultaneous equations based on numerical algorithms must be solved.

Finite element and finite volume methods are numerical methods for solving differential equations that describe many engineering problems. One of the reasons for popularity of this methods is that this methods results in computer programs versatile in nature that can solve many practical problems with a small amount of training [9]. Obviously, it is very important to use accurate methods. On this basis, a practical investigation was done to compare the results of finite element and finite volume methods with experimental data.

In this study, the k-ε model was used to model flow in studied geometry. The k-ε model was first proposed by Launder and Spalding [7] and has been subject of various improvements and modifications since. The k-ε model is one of the most used turbulence models for industrial applications. Standard k-ε model is very suitable for completely disturbanced flow.

The k-ε turbulence model relies on several assumptions, the most important of which are that the Reynolds number is high enough and that the turbulence is in equilibrium in boundary layers, which means that production equal dissipation. The assumptions limit the accuracy of the model; since this is not always true [1].

3. Experimental description

Schematic geometry was studied in this research is given in Fig. 1. This geometry which is presented by Founti and A. Klipfel [4] is chosen to make comparison of simulation results with the experimental ones possible. Experimental investigations of turbulent flow through a sudden expansion have been conducted using laser Doppler anemometry (LDA) by numerous authors [3, 6, 13]. Difficulties in accurately measuring close to the wall were often reported [2].

Founti and Klipfel conducted their experiments in a closed loop system with a vertically mounted working section. Upstream, the pipe has an internal diameter, d, of 25.5 mm and downstream, the diameter, d, was twice as large at 51 mm. The working section incorporated an expansion with a ratio of 2:1 and fully developed flow in the direction of gravity. A schematic diagram and indication of notation is given in Fig. 2.

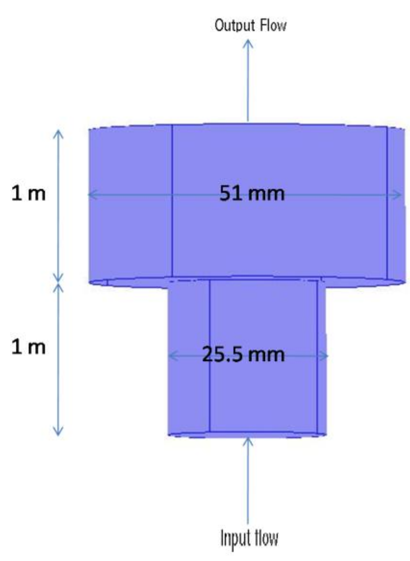

Fig. 1. The computational domain and geometry specifications

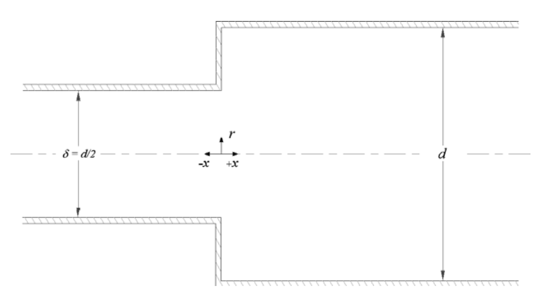

Fig. 2. Schematic diagram of experimental setup [7]

3.1. Fluid properties

A mix of diesel oils was used in order that the refractive index of the working fluid matched the walls of the test section. Experimental parameters are given in Table 1. Measurements were taken using LDA and the error in results was given as 1% and 2% for the mean and turbulent measurements respectively.

Table 1. Experimental properties and operating conditions [7]

Density	830 kg m^{-3}
Viscosity	0.0043201 kg m^{-1} s^{-1}
Mass flow rate	1.7304 kg s^{-1}
Upstream Reynolds number	20,000
Downstream Reynolds number	10,004
Bulk velocity (upstream of expansion)	4.082 m s^{-1}
Bulk velocity (downstream of expansion)	1.021 m s^{-1}

4. Numerical investigation of studies system

4.1. Finite volume analysis with FLUENT simulator

Using FLUENT simulator, simulation was done in two-dimensional symmetric, isothermal and steady state conditions. Since the flow did not include a swirl component, the flow was modeled as 2D axisymmetric. k-ε turbulent model is applied. Characteristics of the simulation in this case are summarized in Table 2.

Table 2. Mesh characteristics for finite volume analyzing

Number of elements	39588
Triangular	39588
Quadrilateral	0

Geometry of the problem and type of meshing, which is used in software FLUENT, is illustrated in Fig. 3.

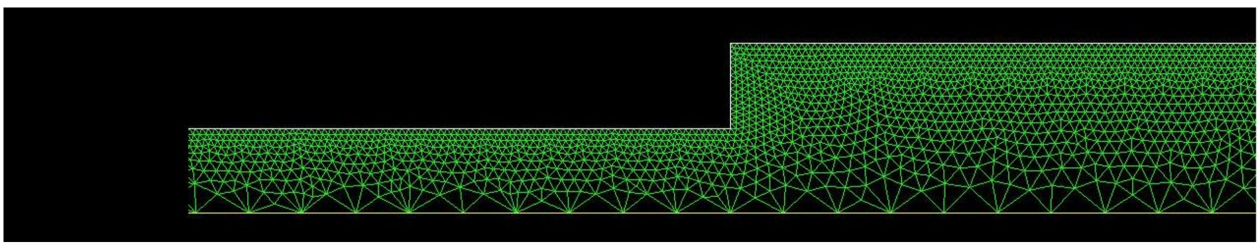

Fig. 3. Constructed mesh for finite volume analysis of studied geometry

4.2. Finite element analysis with COMSOL MULTIPHYSICS simulator

Using COMSOL MULTIPHYSICS, simulation was done in two-dimensional symmetric, isothermal and steady state conditions. Since the flow did not include a swirl component, the flow was modeled as 2D axisymmetric. k-ε turbulent model is applied. Introducing the geometry to the software, two networks with characteristics that given in table 2 was constructed. First, in order to make a reasonable comparing with finite volume methods results, triangular meshing was used. As we see in Fig. 4, triangular cells in this software couldn't cover boundary layers completely. As we will see in the results, this meshing isn’t the best choice for the studied geometry. So, boundary layers’ meshing was used to cover all system geometry, in particular the boundary layers. Characteristics of this two meshing are given in Table 3. Constructed meshes for finite element analysis of the studied system are illustrated in figures 4 and 5.

Table 3 Mesh characteristics for finite element analyzing

Meshing Mesh Characteristics	Triangular	Boundary Layers Meshing
Number of degrees of freedom	12999	14436
Number of mesh points	851	896
Number of elements	1336	964
Triangular	1336	334
Quadrilateral	630	0
Number of boundary elements	196	364
Number of vertex elements	6	6
Minimum element quality	0.0257	0.7027
Element area ratio	0.0068	0.0751

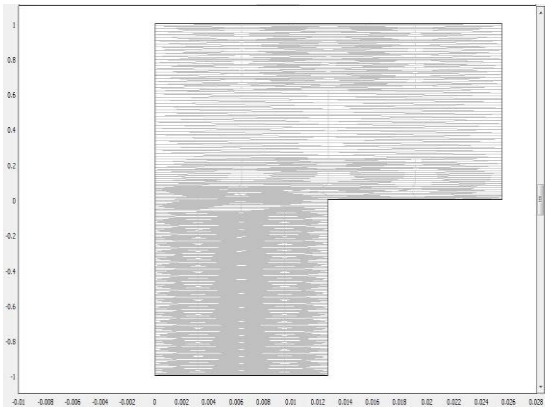

Fig. 4. Triangular grid for finite element analysis

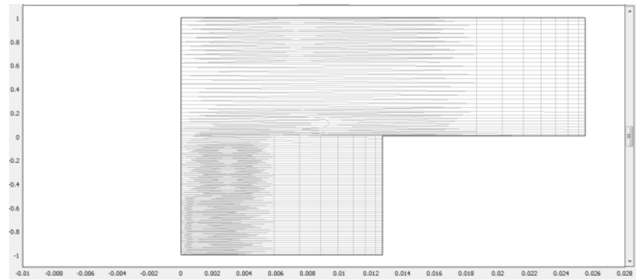

Fig. 5. Boundary layer grid for finite element analysis

In each case, in order to verify that the mesh used for verification was grid independent, a concentrated mesh with more cells was constructed and the governing equations solved. The maximum axial velocity differed by only 2.37%. On this basis, 3 mesh with characteristics that given in table 2 and 3 were regarded as grid independent.

5. Results and Discussion

Solving the governing equations and applying boundary conditions simultaneously, distribution pattern can be obtained. Both finite element and finite volume method results are presented in figures 6-13. According to the figures, with a sudden expansion in pipe, flow lines get far away from each other and thus, the velocity decreased. Comparisons of axial velocities in simulation results using COMSOL and FLUENT with experimental data are shown in figures 6-9. These comparisons for radial velocities are illustrated in figures 10-13.

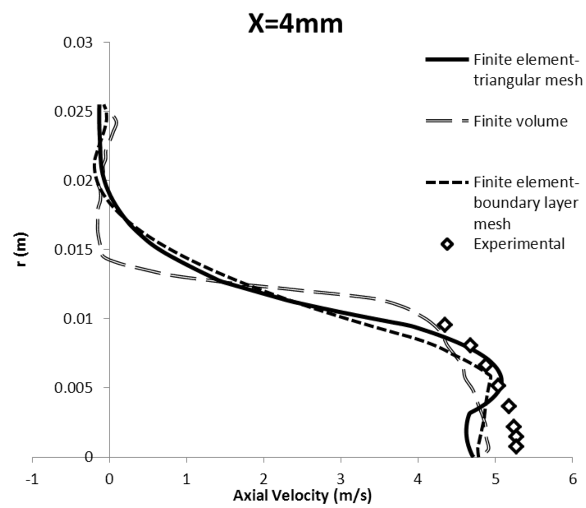

Fig. 6. Variation in the axial velocity at X=4mm

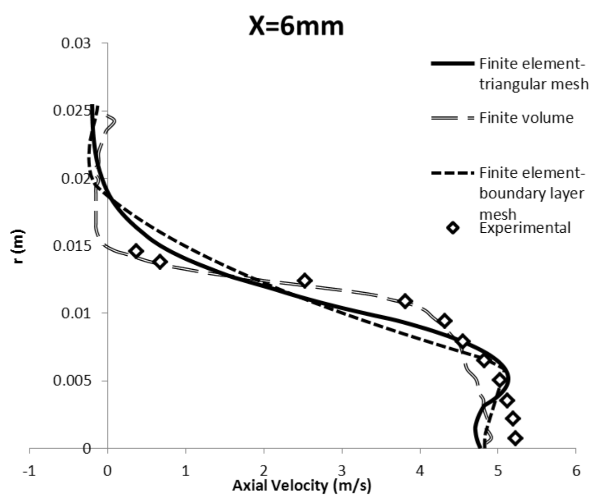

Fig. 7. Variation in the axial velocity at X=6mm

As illustrated in figures, there is fine agreement in axial velocities of the simulation results with COMSOL and FLUENT with experimental data. In COMSOL MULTIPHYSICS simulation, first, meshes with triangular elements were applied, and as we see, this meshing could not predict accurate velocities in boundary layers of walls. So then boundary layer meshes were applied and the results are considerably more compatible with Experimental results rather than triangular meshing. It was found that as the ratio of r/0.025 becomes larger than 0.6, slight over prediction of axial velocity will happen. For the radial velocities, simulation results of both finite element and finite volume methods and experimental data do not fit well with each others, which has two reasons; first, estimation accuracy for radial velocities is lower than axial ones; And, second, flow is extremely turbulent in radial direction and the Reynolds number in this direction is higher than the computational average Reynolds number. So, radial velocities don’t have a significant effect on velocity patterns of the fluid. These results are in agreement with some previous researches [2, 5, 10, 11].

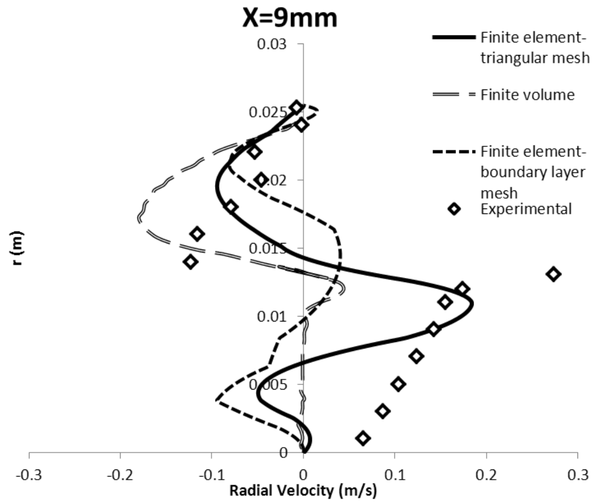

Fig. 8. Variation in the axial velocity at X=9mm

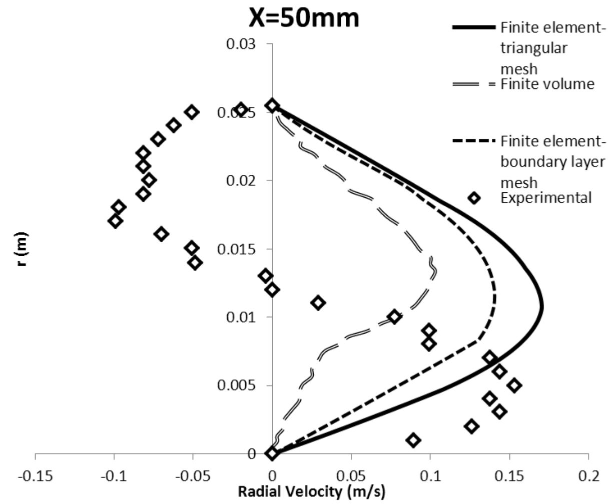

Fig. 9. Variation in the axial velocity at X=50mm

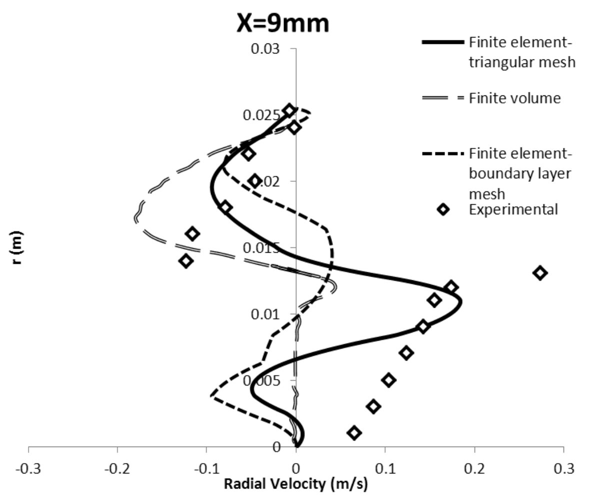

Fig. 10. Variation in the radial velocity at X=9mm

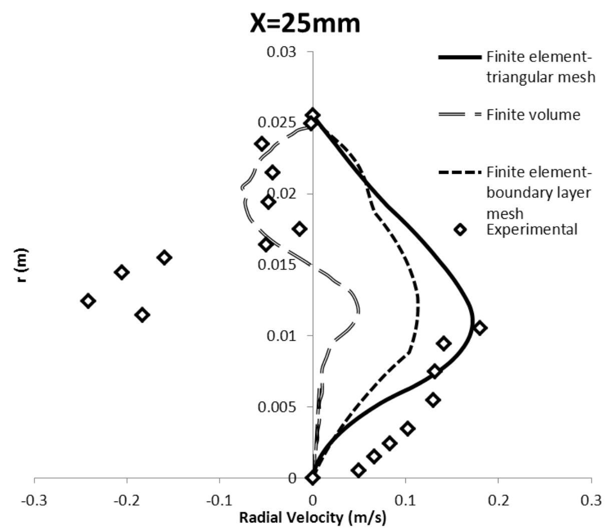

Fig. 11. Variation in the radial velocity at X=25mm

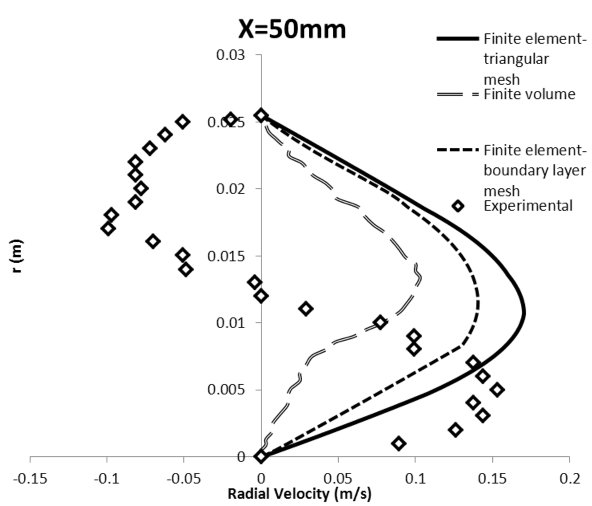

Fig. 13. Variation in the radial velocity at X=50mm

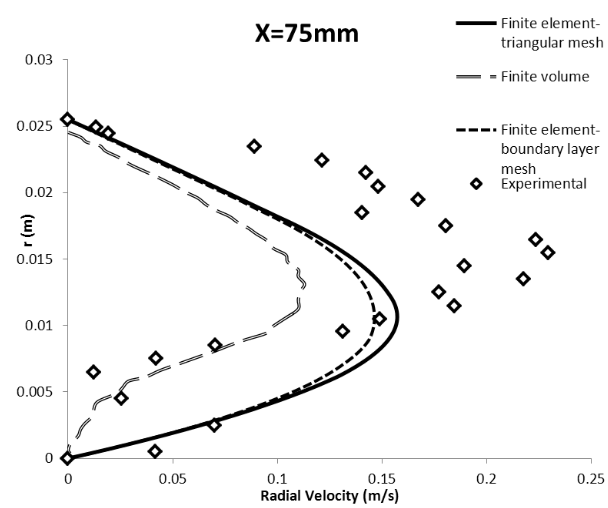

Fig. 13. Variation in the radial velocity at X=75mm

6. Conclusion

In many industries such as oil and chemical ones, it is essential to investigate the flow patterns in the pipelines due to fluid transmission in expansion-contraction geometry. Changing in flow pattern of fluids will effects on a bunch of phenomenon such as heat transfer and corrosion, erosion. The objective of this investigation was to compare the results of finite element and finite volume numerical methods in a practical system. Also, prediction of the flow pattern and axial-radial velocities distribution in pipelines with an abrupt expansion was investigated. The result reveals that there is better agreement between experimental data and the numerical results by using finite element method rather than using finite volume method. Overall, very good matching of experimental and computational data is obtained. However, there is poor agreement between experimental and computational radial velocity results. But, Due to the small magnitude of the radial component relative to the axial component, the deficiencies in modeling the radial velocity do not have a significant effect on the overall fluid solution. The present investigation provides valuable insight into the dynamics of flow in fluid transport pipelines. Further investigation in the dynamics of the flow and velocity distribution must be carried out in more details, which could influence the heat transfer, corrosion and erosion rate, especially for three-dimensional flow.

References

[1] Chemical Engineering Module User's Guide, COMSOLAB, 2007.
[2] Davis, C., Frawley, P.: 2009. Modelling of erosion–corrosion in practical geometries, J. Corrosion Science, 5, 769-775.
[3] Durret, R.P., Stevenson, W.H., Thompson, H.D.: 1988. Radial and axial turbulent flow measurements with an ldv in an axisymmetric sudden expansion air flow, ASME Journal of Fluids and Engineering, 110, 367–372.
[4] Founti, M., Klipfel, A.: 1998. Experimental and computational investigations of nearly dense two-phase sudden expansion fows, J. Experimental Thermal and Fluid Science, 17, 27-36.
[5] Keating, A.: 1999. A Model for the Investigation of Two-phase Erosion–Corrosion in Complex Geometries, Msc., University of Queensland.
[6] Khezzar, L., Whitelaw, J.H., Yiannekis, M.: 1985. An experimental study of round sudden expansion flows, in: Proc. 5th Symp. on Turbulent Shear Flows, Cornell University, Ithaca, NY, 5–25.
[7] Launder, B.E.: Spalding, D.B., 1972. Lectures in Mathematical Models of Turbulence, Academic Press, London, England.
[8] Mullin T., Peixinho, J.: 2006. Transition to Turbulence in Pipe Flow, Journal of Low Temperature Physics, 145, 75-88.
[9] Nam-Hokim, V.Sankar, B., 2009. Introduction to finite element analysis and design, 1st ed. WILEY, United States of America.
[10] Nesic, S., Postlethwaite, J.: 1991. Hydrodynamics of disturbed flow and erosion–corrosion, Part i – single-phase flow study, The Canadian Journal of Chemical Engineering, 69, 704–710.
[11] O'Mahony, A.P., 2006. Numerical Simulation of Slurry Erosion using Lagrangian and Eulerian Techniques, Ph.D., University of Limerick.
[12] Sahu, M., Khatua, K.K., Patra, K.C., Naik, T.: 2009. Developed laminar flow in pipe using computational fluid dynamics, 7th International R&D Conference on Development and Management of Water and Energy Resources, Bhubaneswar, India.
[13] Stieglmeier, M., Tropea, C., Weiser, N., Nitsche, W.: 1989. Experimental investigation of the flow through axisymmetric expansions, ASME Journal of Fluids and Engineering, 111, 464–471.
[14] Traetow, M.A.: 1999. Simulation of Turbulent Pipe Flow, http://www.engineering.uiowa.edu/fluidslab/pdfs/57-020/cfdlab1.pdf.
[15] Willis, A.P., Peixinho, J., Kerswell, R.R., Mullin, T.: 2008. Experimental and theoretical progress in pipe flow transition, *Phil. Trans. Roy. Soc. A* 366, 2671-2684.