STUDIA UBB PHYSICA, Vol. 68 (LXVIII), 1-2, 2023, pp. 25-38
(RECOMMENDED CITATION)
doi:10.24193/subbphys.2023.03

A COMPARISON OF COMPUTATIONAL FLUID DYNAMICS TURBULENCE MODELS FOR DIFFERENT REYNOLDS NUMBERS FLOWS

K. GERGŐ[1,2], V. REDNIC[1], R.V.F. TURCU[1,3*]

ABSTRACT. The present research was made to improve the engineering perspective concerning the computational fluid dynamics method. The main purpose is to obtain the limits of different turbulence models and to establish which of them is the most suitable for different Reynolds number flows. The flow configuration was a typical one, named backward facing step, which is widely used to investigate flows experimentally and computationally. The study involves multiple stages, from creating the geometry and meshing the experimental model to simulating all the conditions to obtain sufficient data to compare the turbulence fluid flow models with experimental results.

Keywords: *backward facing step**,** turbulence models, atomization, CFD, fluid flow, Reynolds-number*

INTRODUCTION

Computational fluid dynamics (CFD) is a very powerful tool for technological development engineering since it makes easier, cheaper and more time efficient the process of taking measurements. The method is based on a computational,

[1] *National Institute for Research and Development of Isotopic and Molecular Technologies, Center of Advanced Research and Technologies for Alternative Energies (CETATEA), Donat 67-103, Cluj-Napoca, Romania.*
[2] *Technical University of Cluj-Napoca, Faculty of Electronics, Telecommunications and Information Technology, 26-28 George Barițiu str., 400027 Cluj-Napoca, Romania.*
[3] *Babeş-Bolyai University, Faculty of Physics, 1 Kogălniceanu str., 400084 Cluj-Napoca, Romania.*
[*] *Corresponding author: flaviu.turcu@ubbcluj.ro*

iterative process. The main point is to approximate the propagation of different physical phenomena in a pre-defined system.

CFD was introduced in technological development in the 1970s, when the first commercial software packages were developed.[1] These early programs were limited by the available computing power, and could only handle relatively simple two-dimensional problems. As computing power increased in the following decades, so did the capabilities of CFD. Three-dimensional models became common, and more complex problems could be analyzed. This led to the adoption of CFD in a variety of industries, including aerospace, automotive, and energy. For example, CFD simulations are used extensively in car racing for designing the aerodynamics of the car, optimizing the airflow over certain geometrically designed shapes, and improving the cars performance. CFD models are also used to predict the car's behavior under different driving conditions and to evaluate new designs before they are physically tested.

Preparing a CFD analysis involves multiple stages of pre-processing and post-processing. These stages, presented in the Results and Disscussions section.

An important quantity in CFD simulations which is involed in the present study as well is the **Reynolds number**. The Reynolds number is a dimensionless parameter used in fluid mechanics to characterize the behavior of a fluid flow. The Reynolds number (Re) is defined as the ratio of inertial forces to viscous forces within a fluid flow. It can be expressed mathematically as:

$$Re = \rho \frac{vL}{\mu}$$

where ρ is the density of the fluid, v is the velocity of the flow, L is a characteristic length scale (such as the diameter of a pipe or the chord length of an airfoil), and μ is the dynamic viscosity of the fluid. The Reynolds number has the same value for all fluid flows that have the same geometric shape and the same fluid properties. The value of the Reynolds number determines the type of flow regime that exists within the fluid flow. At low Reynolds numbers (typically less than 2000), the flow is laminar, meaning that the fluid flows in smooth, parallel layers without turbulence. At high Reynolds numbers (typically greater than 6000), the flow becomes turbulent, meaning that the fluid flows in a chaotic, unpredictable with eddies and vortices forming within the flow. Between 2000-6000 **Reynolds number (Re)** values, there is a transitional regime, in which vortices form and dissipate quickly. [2]

The Reynolds number is an important parameter in many areas of fluid mechanics, including aerodynamics, hydrodynamics, and heat transfer. It is used to predict the onset of turbulence, to design and optimize fluid flow systems, and to compare different fluid flow configurations. It is a key parameter in the design of many engineering systems, such as pipelines, pumps, and aircraft.

Fluid flow models

There are a lot of choices for fluid flow models. A CFD engineer should know the advantages and disadvantages of each in order to create a proper model for a certain phenomena. The following chapter will contain a brief description of the most widely used fluid flow models:

Inviscid flow is a type of fluid flow where the fluid is assumed to have zero viscosity, meaning that it has no internal friction. Inviscid flow is an idealized model that is used to simplify the analysis of fluid dynamics problems, especially in aerodynamics and assumes that the fluid is homogeneous and the flow is non-rotational, meaning that the fluid particles move in straight lines and do not rotate around their own axis. It is often used to analyze the flow of fluids around solid objects, such as wings, propellers, and airfoils.

Laminar flow, on the other hand, is a type of fluid flow where the fluid particles move in parallel layers or streams, with minimal mixing between them. Laminar flow occurs when the fluid is moving at low velocities, and the flow is characterized by smooth, steady motion. Laminar flow is often observed in small channels and pipes, and it is important in the design of fluid handling systems. Laminar flow is often modeled using the Navier-Stokes equations, which describe the behavior of viscous fluids. There are other models that reflect much better the behaviour of certain flows based on the Reynolds Averaged Navier-Stokes (RANS) equations, which are used to solve for the time-averaged flow field.

The **k-epsilon model** is a two-equation model, which solves for the turbulent kinetic energy (k) and the dissipation rate of turbulent kinetic energy (epsilon). The k-epsilon model assumes that the turbulence is isotropic and that the eddies in the flow are small and rapidly decaying. This model is suitable for moderate Reynolds number flows where the turbulence is not very intense.

The k-epsilon model is computationally less expensive than the k-omega model. The model is relatively easier to set up and calibrate and it is more suitable for low-to-moderate Reynolds number flows where the turbulence is not very intense. The k-epsilon model assumes isotropic turbulence, which may not be appropriate for all flow situations. The model is less accurate in predicting separated flows, flow separation, and swirling flows. It can also produce unphysical results in cases where the turbulence is anisotropic, such as in flows with strong curvature or rotation.

The **k-omega model** is also a two-equation model that solves for the turbulent kinetic energy (k) and the specific dissipation rate (omega). The k-omega model assumes that the turbulence is anisotropic and that the eddies in the flow are larger and less rapidly decaying than in the k-epsilon model. This model is suitable for high Reynolds number flows where the turbulence is intense.

The k-omega model is more accurate than the k-epsilon model in predicting turbulent flows with intense anisotropy, such as those involving swirling flows or strong curvature. The model is also better suited for high Reynolds number flows where the turbulence is more intense and it can better capture the effect of pressure gradients on the turbulence, which is important in many engineering applications.

The k-omega model is more computationally expensive than the k-epsilon model, it is more difficult to set up and requires more calibration. The model may be less accurate in predicting laminar-turbulent transition and wall-bounded flows. The main difference between the two models is in their treatment of the dissipation rate of turbulent kinetic energy. In the k-epsilon model, the dissipation rate is calculated using an empirical formula that assumes a constant ratio between the length scale of turbulence and the dissipation rate. In the k-omega model, the specific dissipation rate is directly related to the turbulent viscosity, which is determined by solving a transport equation for omega.

Overall, both models have their own advantages and limitations, and the choice of model depends on the specific flow conditions and the desired level of accuracy.

The aim of this paper is to present the general path of building a CFD model for a physical phenomena and to compare the results obtained by using turbulence fluid flow models available in today's CFD technologies with experimental data.

THEORETICAL DETAILES

Until this point we presented various turbulence simulation models for a certain phenomena. One of the main steps of setting up a proper model in CFD technology is to select the proper fluid-flow model that fits best the parameters of the flow in cause. For example, there are models that go well with high-turbulence flows but are expensive comupationally. If you have a simple non-viscous fluid that has low flow velocity, the best reccomendation is to use a laminar model that goes well with this kind of phenomena and converges relatively fast. The setup (geometry) used for this present research is named backward facing step. It's a simple geometry but it's also suitable to study turbulent flows in different conditions.

A **backward facing step** is a flow configuration in which a fluid flows in a channel or duct and encounters a sudden expansion, which creates a step-like geometry with a vertical wall facing upstream and a horizontal wall downstream.

The fluid flow is said to be backward facing because the direction of the flow is opposite to the direction of the step.

When the fluid flows through the narrow channel before the step, it accelerates and develops a boundary layer near the walls. As the flow encounters the step, the sudden expansion causes the flow to slow down and the boundary layer to thicken. This results in the formation of a recirculation region downstream of the step, where the fluid flows back towards the step before turning downstream again. The backward facing step is a common flow configuration used in experimental and numerical studies to investigate fluid dynamics phenomena such as turbulence, boundary layer separation, and heat transfer. It is also relevant to many engineering applications, such as in the design of heat exchangers and combustion systems.

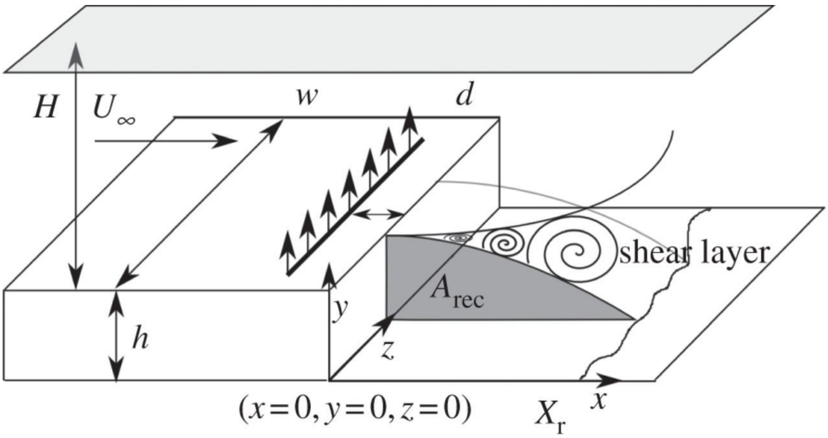

Fig. 1: Backward facing step geometry and parameters [3]

As the duct thickens and the fluid passes through the „step”, an inverse flow domain forms right beside the step. This study will compare the X_r (**Fig. 1**) values obtained from simulations with real experimental data. The source for experimental data is [4], (**Fig**. **2)**. The x-axis value is the Reynolds number of the flow mesured right before the flow passes the step,vertically in the middle of the duct. X_r is the lenght of inverse flow domain, but it's represented in dimensionless units X_r/H, H being the height of the step. In this case H=1.5 cm.

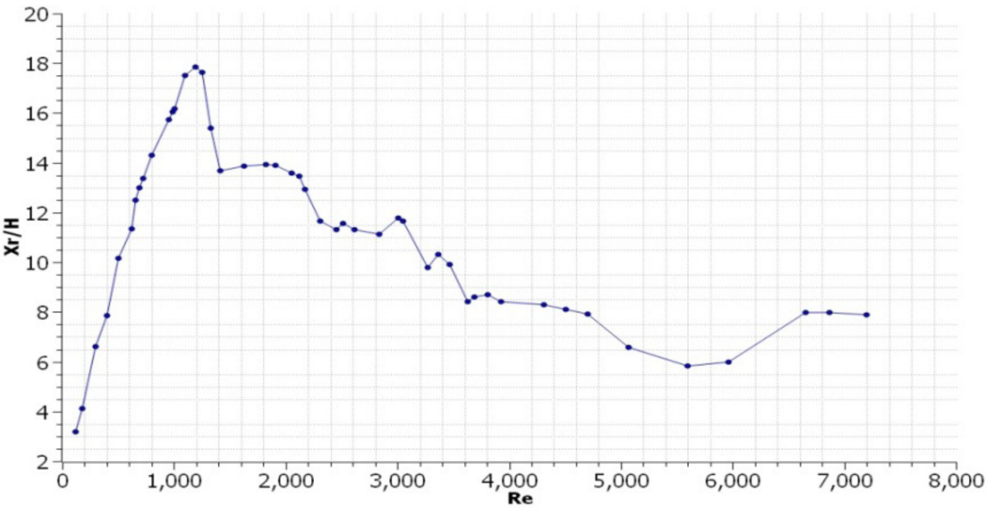

Fig. 2: The length of the inverse flow domain as a function of Reynolds number of the flow before the step [4]

RESULTS AND DISCUSSION

In the following pharagraph, we will present the steps of building a CFD model, illustrating each step with the model built for this research.

The first step is to build a **geometry** using a CAD (computer-aided design) software. Making a geometry assumes only designing the appearance of the studied object without defining the physical properties of the material. In this case, we will build the simple 2D geometry of the backward facing step presented previously.

The next step is to **mesh** the geometry as a preceding operation of the calculations. Meshing is the process of dividing a physical domain, such as an object or fluid volume, into a finite number of small elements known as “meshes” or “elements.” The meshes are usually simple geometric shapes, such as triangles, quadrilaterals, tetrahedra, or hexahedra, that collectively form a complex representation of the original physical domain.

The mesh has two undimensional quantities that caracterizes its quality: skewness and ortogonality. Skewness refers to the distortion or non-uniformity of the mesh elements. Ideally, mesh elements should be as regular as possible, with uniform angles between their faces. However, in practice, mesh elements can become distorted, resulting in non-uniform angles between their faces. Skewness can negatively impact the accuracy and stability of the simulation results, as it can cause numerical errors and convergence problems. Therefore, it is desirable to keep the skewness of mesh elements as low as possible. Orthogonality, on the other hand, refers to the degree to which mesh elements are perpendicular to each other. In an ideal mesh, all elements should be perfectly orthogonal to their neighboring elements. However, in practice, orthogonality can be compromised due to the geometry of the physical domain or the meshing algorithm. Non-orthogonal elements can lead to inaccurate flow predictions and numerical instabilities in CFD simulations, especially in regions of high flow gradients. Therefore, it is important to maintain a high degree of orthogonality in the mesh, especially in regions where accurate flow predictions are crucial.

The mesh size and shape are determined by several factors, including the geometry of the physical domain, the desired level of accuracy, the available computational resources, and the type of analysis being performed. A fine mesh with small elements can provide more accurate results, but at the cost of increased computational resources and longer simulation times. A coarser mesh with larger elements can provide faster results, but with lower accuracy. The process of meshing involves generating the mesh, refining or coarsening it as needed, and then exporting it to the appropriate simulation software. Meshing can be done manually, but for complex geometries, automated meshing tools are typically used.

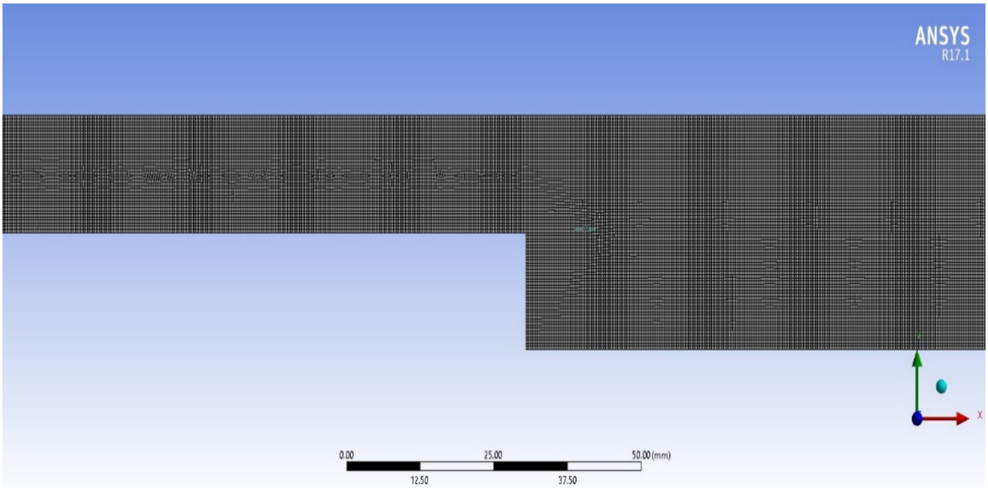

Fig. 3: The meshed form of the backward facing step geometry

The meshing for this simple geometry is represented in **Fig. 3** completed with mesh quality parameters. Orthogonality is a value near to 1 and skewness to 0. It indicates that we realized a decent mesh to run the simulation on.

Pre-processing is the initial stage of a simulation process, which involves setting up the problem, preparing the input data, and creating a numerical model to represent the physical system to be analyzed. It has two main stages:

1. Boundary conditions: Once the mesh is created, boundary conditions must be defined for the simulation. This includes specifying the inflow and outflow boundaries, the wall boundaries, the type of fluid being simulated, and any other relevant physical properties. The boundary conditions are essential for defining the initial conditions of the simulation.
2. Solver Settings: The solver settings must be defined before running the simulation. This includes the choice of solver, the numerical method used to solve the governing equations, and the convergence criteria. The solver settings must be selected to ensure that the simulation results are both accurate and efficient.

Generally, in CFD the **calculations** executed by the solver are differential equations, but solved numerically with a certain error value. These errors, called residuals, should be minimized to meet the convergence criteria. A graphical representation of this process is shown in **Fig. 4**.

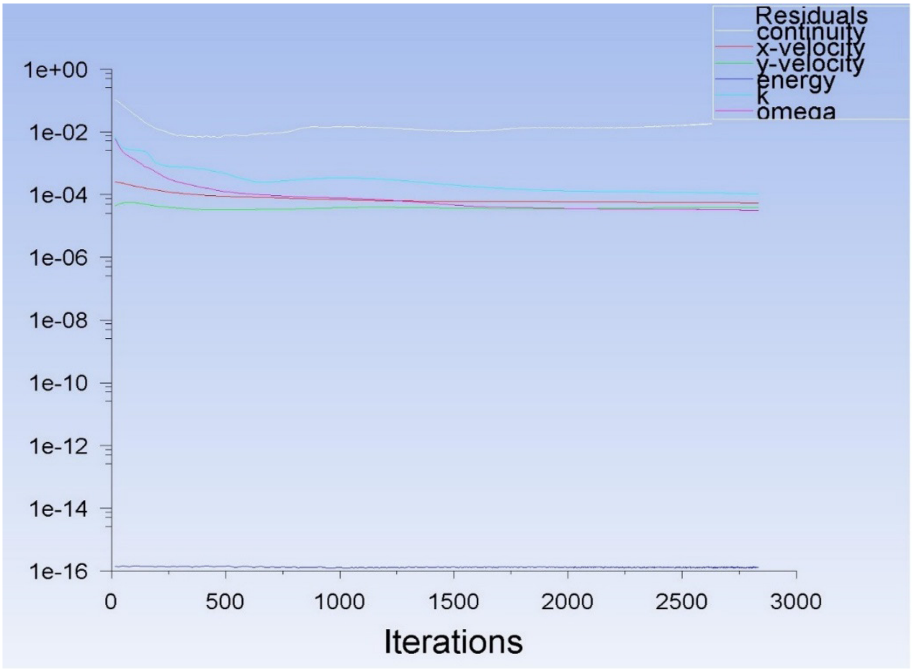

Fig. 4: Evolution of residual values for backward facing step model calculations

The convergence is an essential part of a model. A simulation works with repetitive calculations which assumes minimizing the residual values throughout the iterations. Reaching the convergence criteria means making the residual values smaller than a certain value. Residuals are a measure of the error in the numerical solution of the governing equations. The rate of decrease of the residual values is a measure of the convergence rate of the simulation. In general, the goal of a CFD simulation is to obtain a numerical solution that is both accurate and time efficient. The accuracy of the solution depends on the convergence criteria used, while the efficiency of the solution depends on the rate of convergence. A good convergence criterion should ensure that the solution is accurate enough to meet the desired level of precision while still allowing the simulation to converge in a reasonable amount of time.

In fluid dynamics simulations the most typical and wide applicable set of differential equations is the Navier-Stokes system. The Navier-Stokes equations are a set of partial differential equations that describe the motion of fluids, including liquids and gases [5]. The Navier-Stokes equations describe the fundamental laws of fluid mechanics, including the conservation of mass, momentum, and energy. They take into account the effects of viscosity, pressure, and gravity on the fluid motion.

The general form of the Navier-Stokes equations can be written as:

$$\frac{\partial u}{\partial t} + u \cdot \nabla u = -\frac{\nabla p}{\rho} + \nu \cdot \nabla^2 u + g$$

$\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{t}}$ represents the acceleration of the fluid, $\boldsymbol{u} \cdot \boldsymbol{\nabla u}$ represents the convective acceleration due to the motion of the fluid. $-\frac{\boldsymbol{\nabla p}}{\boldsymbol{\rho}}$ represents the pressure forces, while g represents the viscous forces. The last term represents the effect of gravity on the fluid.

The Navier-Stokes equations are a system of coupled, nonlinear partial differential equations that are difficult to solve analytically. Instead, numerical methods are used to solve them in practice [6].

Post-processing typically involves the following steps:

The first step in post-processing is to extract the data generated by the simulation. This data may include the velocity, pressure, temperature, and other variables at various locations within the simulated domain. The data can be extracted from the output files generated during the simulation and stored in a format that can be easily processed and analyzed. Once the data is extracted, it must be analyzed to identify any patterns or trends that may be relevant to the problem being studied. This can be done using various statistical and data analysis techniques. For example, contour plots, velocity vectors, streamlines, and other visualizations can be used to understand the flow patterns and other features of the system. Post-processing also involves validation and verification of the simulation results. Validation is the process of comparing the simulation results to experimental data to ensure that the simulation accurately represents the physical system being studied. Verification is the process of ensuring that the simulation is solving the correct equations and is providing a reliable solution.

In this particular case, the post-processing consists of gathering data about the lenght of the backflow domain named X_r previously. The data extraction took place by representing the shear force (**Fig. 6**) of the swirl (**Fig. 5**) acting upon the lower wall and reading the coordinate when the horizontal shear force becomes positive again.

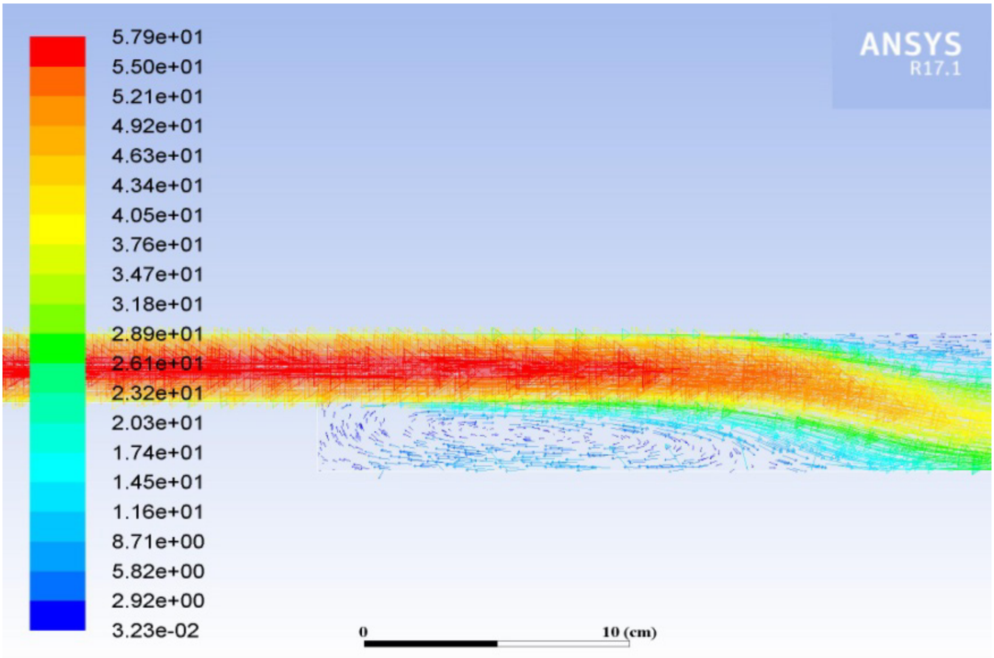

Fig. 5: Velocity vectors of the flow in the pipe in the proximity of the step

The measurements were taken this way, simulating the flow for every Reynolds number and reading the data point when the shear becomes again positive (red dot on **Fig 6**). In this case negative shear means back flow and positive shear forward flow.

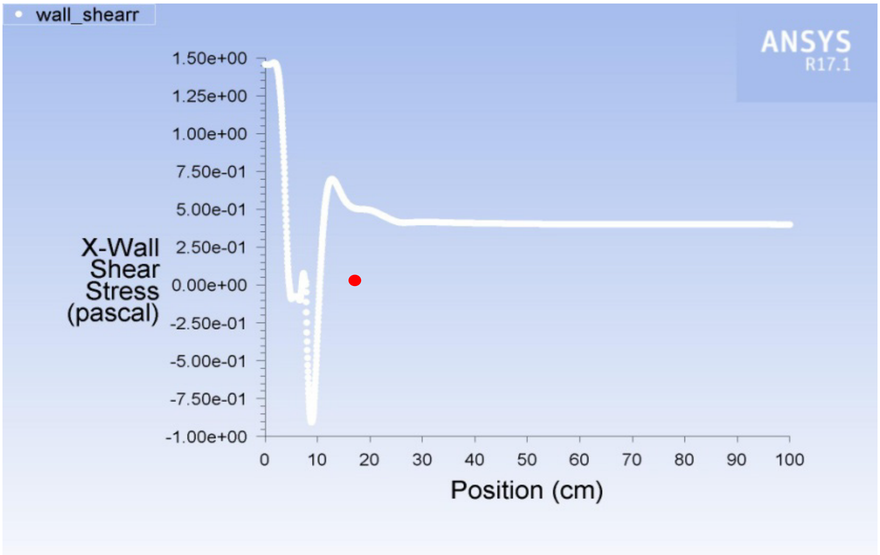

Fig. 6: The horizontal component of the shear-force on the lower wall of the geometry

The results of the CFD simulations in the 20-8000 Re range, including laminar, transitional and turbulent regimes, are presented in **Fig. 7** for both k-epsilon and k-omega models. The CFD results are compared with the experimental ones.

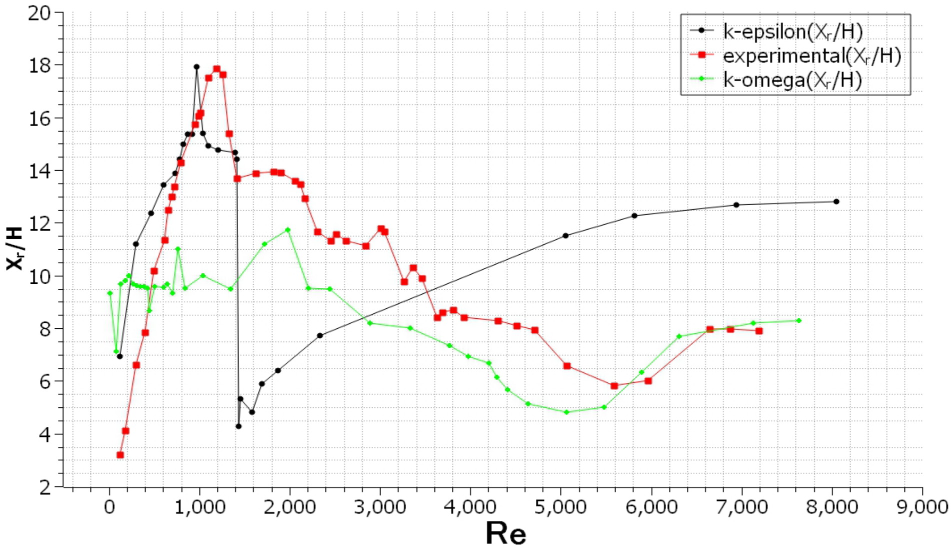

Fig. 7: Comparison between the computational and experimental [4] results

Experimentally, the behaviour of the curves varies in respect with the regime of the flow. For Re<1000, the air flow can be considered laminar. In this interval the lenght of the backflow domain grows linearly with the Reynolds number. In transitional flow regime (Re 2000-6000), the lenght X_r tends to decrease until it reaches a minimum value. Afterwards it has a tendency to grow and to stabilize at a constant value. Reaching the interval when the size of the backflow domain doesn't change anymore with the Reynolds number, means that the flow became turbulent. This behavior takes place for Re>6000. Taking into account the concordance of the curves, we can notice that the k-epsilon model works well with laminar regime but doesn't describe so well transitional and turbulent regimes. As well, we can notice that the k-omega model works well with transitional and turbulent regimes but fails to describe the laminar regime.

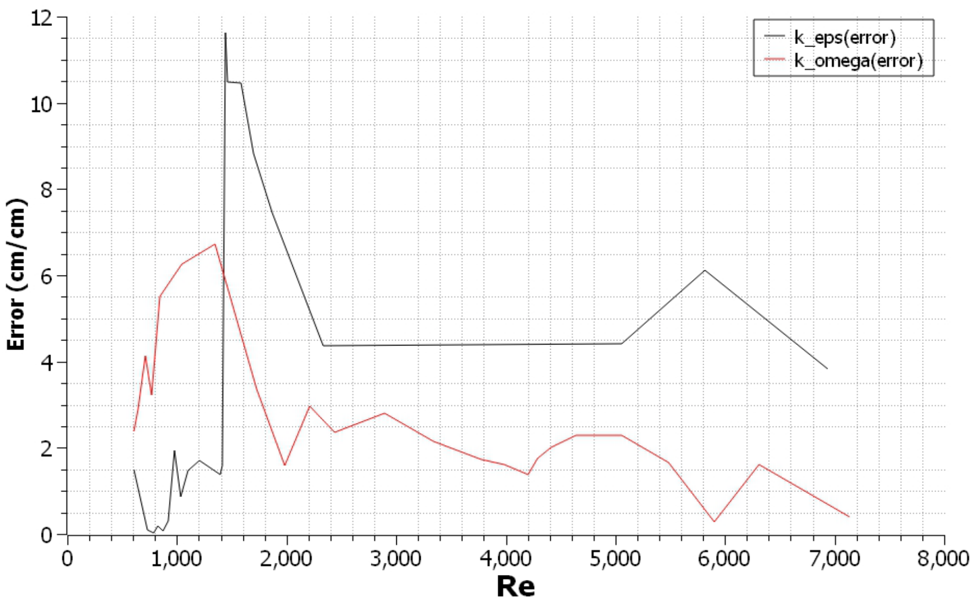

Fig. 8: Absolute errors in respect to experimental curve for both models

On **Fig. 8** is represented the error for both models compared with experimental data. As it can be noticed, the k-epsilon model has smaller errors in the laminar regime (Re<1000) and the k-omega has smaller errors in the transitional and turbulent regimes (Re>1000).

CONCLUSIONS

In conclusion, based on the obtained data can be confirmed that creating a suitable model for a physical phenomenon is strongly influenced by the proper selection of fluid flow model. For laminar and low transitional regime (Re < 1500) k-epsilon model performs very well but afterwards, not even the tendency of evolution of X_r matches the experimental results. Besides, on transitional regime k-omega tends to overperform k-epsilon, as it reflects the decreasing tendency of X_r. On turbulent regime the best choice is obviously k-omega because it predicts correctly the constant value at which the X_r stabilizes. K-epsilon also predicts a constant value for this regime, but with significantly higher error.

This research gives evidence of the fact that using two different fluid flow models can give very different results at the end. Of course, these results don't give a one hundred percent precise guideline of turbulence model selection for every kind of flow but can significantly help the CFD engineers upon taking this decision.

ACKNOWLEDGMENTS

This project is funded by the Ministry of Research, Innovation and Digitalization through Programme 1 - Development of the National Research and Development System, Subprogramme 1.2 - Institutional Performance - Funding Projects for Excellence in RDI, Contract No.37PFE/30.12.2021 and the "Nucleu" Programe within the National Plan for Research, Development and Innovation 2022-2027, project PN 23 24 02 01.

REFERENCES

1. Yufeng Wei, "The development and application of CFD technology in mechanical engineering", **2017**, IOP Conf. Ser.: Mater. Sci. Eng. 274 012012
2. Vaclav Uruba, "Reynolds number in laminar flows and in turbulence", **2019**, AIP Conference Proceedings 2118(1):020003, 38th meeting of departments of fluid mechanics and thermodynamics
3. Gautier N. and Aider J.-L. Control of the separated flow downstream of a backward-facing step using visual feedback Proc. R. Soc. A.46920130404, **2013**
4. F. Armalyt, F. Dursts, J. C. F. Pereira and B. Schonung: "Experimental and theoretical investigation of backward-facing step flow", **1983**, J. Fluid Mech., vol. 127, pp. 473496

5. Giovanni P. Galdi, “An Introduction to the Navier-Stokes Initial-Boundary Value Problem”, **2000**, Part of the Advances in Mathematical Fluid Mechanics book series (AMFM)
6. M. Ahammad, M. A. Rahman, J. Alam, S. Butt, “A computational fluid dynamics investigation of the flow behavior near a wellbore using three-dimensional Navier–Stokes equations”, **2019**, Advances in Mechanical Engineering 11(9)