

Numerical simulation and validation of laminar flow through a 2D pipe using computational fluid dynamics

Simulação numérica e validação de escoamento laminar através de um tubo 2D utilizando dinâmica de fluidos computacional

DOI:10.34117/bjdv9n10-084

Recebimento dos originais: 15/09/2023

Aceitação para publicação: 18/10/2023

Mario Cerrogrande Ramos

Doctor in Nuclear Sciences and Techniques

Institution: Universidade Federal de Minas Gerais, Agencia Boliviana de Energia Nuclear

Address: Av. Arica, entre calles 22407 y Punata, Urbanización Parcopata, Distrito 8, El Alto

E-mail: marc5663@gmail.com

Antonella Lombardi Costa

Doctor in Nuclear and Industrial Security

Institution: Universidade Federal de Minas Gerais

Address: Av. Antônio Carlos, 6627, Pampulha, Belo Horizonte – MG, CEP: 31270-901

E-mail: antonella@nuclear.ufmg.br

Vitor Vasconcelos Araújo Silva

Doctor in Nuclear Sciences and Techniques

Institution: Jülich Supercomputing Centre (JSC)

Address: Forschungszentrum Jülich, Postal Code 52428 Jülich, Germany

E-mail: v.silva@fz-juelich.de

Cláudia Pereira Bezerra Lima

Doctor in Environment and Nuclear Safety

Institution: Universidade Federal de Minas Gerais

Address: Av. Antônio Carlos, 6627, Pampulha, Belo Horizonte – MG, CEP: 31270-901

E-mail: claudia@nuclear.ufmg.br

Maria Auxiliadora Fortini Veloso

Doctor of Chemical Engineering

Institution: Universidade Federal de Minas Gerais

Address: Av. Antônio Carlos, 6627, Pampulha, Belo Horizonte – MG, CEP: 31270-901

E-mail: mdora@nuclear.ufmg.br

ABSTRACT

In modern times, there has been a significant increase in the use of software to simulate fluid dynamics. There exist a variety of closed-source software packages that are utilized for both modeling and simulating fluid behavior. These proprietary programs have become increasingly common and are widely utilized in numerous industries that require advanced simulations. This work focuses on the use of a commercial software package designed for computational fluid dynamics (CFD) problems to model laminar flow

through a pipe, with the objective of gaining a deeper understanding of the dynamic structures and interactions that occur within the laminar flow. In this paper, an accurate computation of two-dimensional flow is performed using the Time Volume Element Methodology. A grid sensitivity analysis with Richardson extrapolation is conducted to determine the grid-independent solutions, ensuring that the results obtained are reliable and accurate. The findings of this study suggest that the relationship between various parameters, such as velocity and pressure, is heavily influenced by the position and shape of the pipe through which the flow occurs. The investigation utilized a grid sensitivity analysis with Richardson extrapolation to determine the percentage error between the analytical and grid-independent solutions for different parameters, revealing that the errors are on the order of 10-1 %.

Keywords: ansys fluent, CFD, validation.

RESUMO

Nos tempos modernos, houve um aumento significativo no uso de software para simular a dinâmica de fluidos. Existe uma variedade de pacotes de software de código fechado que são utilizados para modelar e simular o comportamento de fluidos. Esses programas proprietários tornaram-se cada vez mais comuns e são amplamente utilizados em vários setores que exigem simulações avançadas. Este trabalho foca no uso de um pacote de software comercial projetado para problemas de dinâmica de fluidos computacional (CFD) para modelar o escoamento laminar através de uma tubulação, com o objetivo de obter uma compreensão mais profunda das estruturas dinâmicas e interações que ocorrem dentro do escoamento laminar. Neste artigo, um cálculo preciso do fluxo bidimensional é realizado usando a Metodologia de Elementos de Volume de Tempo. Uma análise de sensibilidade da rede com extrapolação de Richardson é conduzida para determinar as soluções independentes da rede, garantindo que os resultados obtidos sejam confiáveis e precisos. As descobertas deste estudo sugerem que a relação entre vários parâmetros, como velocidade e pressão, é fortemente influenciada pela posição e formato do tubo através do qual ocorre o fluxo. A investigação utilizou uma análise de sensibilidade da grade com extrapolação de Richardson para determinar o erro percentual entre as soluções analíticas e independentes da grade para diferentes parâmetros, revelando que os erros são da ordem de 10-1%.

Palavras-chave: ansys fluente, CFD, validação.

1 INTRODUCTION

Researchers have extensively investigated laminar flow in channels and sub-channels of nuclear research reactors, conducting both theoretical and experimental studies. The main focus of these endeavors is to establish fresh correlations that enable the calculation of the convective heat transfer coefficient within the developing regime or laminar flow regime, specifically for Reynolds numbers ranging from 400 to 1700 [1, 2].

While most heat exchangers operate in the turbulent flow regime to achieve high heat transfer coefficients, there are still many heat exchangers that operate in the laminar-turbulent transition regime. By analyzing the fluid dynamics in this transitional regime, researchers can develop new methods to improve the performance of heat exchangers and design more efficient heat transfer system [3].

The analysis of channels flow is very important from the engineering point of view. A lot of engineering problem dealt with it. The analysis of channel flow is of utmost importance due to its rigorous engineering applications and implications. In this regard, it is crucial to consider the viscous effects exhibited by real fluids in laminar flow conditions.

This study focuses on the numerical analysis of laminar flow in the inlet of a pipe for all incompressible fluids. By utilizing a numerical technique that provides a closer approximation to the basic equations of fluid motion than previous investigations, the results show significant differences in both velocity profiles and development lengths compared to previous work.

The aim of this study is to use computational methods to investigate the nature of fully developed laminar flow in a pipe and determine various parameters. Grid refinements were utilized to calculate the parameters, and a grid sensitivity analysis was performed with Richardson extrapolation to compare the results. The accuracy of the simulations was verified by comparing the grid independent solution to analytical values. To conduct the study, the ANSYS-FLUENT code was used, a commercial software package designed for computational fluid dynamics (CFD) problems.

2 THEORETICAL LAMINAR FLOW THROUGH A PIPE

In the literature, the physical principles utilized for the development of mathematical modeling of fluid motion are commonly presented as: conservation of mass, second Newton's law, and conservation of energy [4, 5]. Based on these principles and the continuum hypothesis, the balance of mass, linear momentum and energy are obtained from the so-called equations of continuity (1), conservation of momentum in an inertial (non-accelerating) reference frame (2) and energy (3) respectively in differential form using the Eulerian framework:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla \cdot p - \nabla \cdot \overline{\overline{\tau}} + \rho \vec{g} \quad (2)$$

$$\frac{\partial}{\partial t} \left(\frac{1}{2} \rho v^2 + \rho \hat{U} \right) + \nabla \cdot \left[\left(\frac{1}{2} \rho v^2 + \rho \hat{U} \right) \vec{v} \right] + \nabla \cdot \mathbf{q} + \nabla \cdot (P \vec{v}) + \nabla \cdot (\tau \cdot \vec{v}) - \rho (\vec{v} \cdot \vec{g}) - \dot{G} = 0 \quad (3)$$

Additional equations may be required when additional models are used, for example for the analysis of structural behavior or in the presence of chemical reactions. The governing equations and the boundary conditions are formulated based on following assumptions:

- a) Non-Newtonian fluid, single phase and incompressible flow, the fluid properties, such as dynamic viscosity and density are constant.
- b) Isothermal flow, local changes in temperature are small or nonexistent; this eliminates the need for a differential energy equation.
- c) An internal flow through a (circular) pipe infinitely long in the z-direction, nothing special about any z-location since the pipe is infinite in length—the flow is fully developed.
- d) Axisymmetric about the center line, which makes the problem two dimensional (r,z) instead of three dimensional (r,θ,z).
- e) The effects of gravity is ignored and steady state flow.

With assumptions a) and b) the governing the equations of fluid dynamics of the continuity, momentum and energy are simplified:

$$\rho \nabla \cdot \vec{v} = 0 \quad (4)$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

$$\rho \frac{\partial}{\partial t} (\vec{v}) + \rho \vec{v} \cdot \nabla (\vec{v}) = -\nabla \cdot p + \mu \nabla^2 (\vec{v}) + \rho \vec{g} \quad (5)$$

Both the continuity equation (4) and the Navier-Stokes equation (5) involve the velocity term, which leads to their coupling in the context of fluid dynamics. In addition, pressure appears in all three components of the Navier–Stokes equation, and thus the

velocity and pressure fields are also coupled. These equations in cylindrical coordinates with assumptions c), d) and e) are simplified:

$$\frac{1}{r} \frac{\partial}{\partial r} (rv_r) + \frac{\partial}{\partial z} (v_z) = 0 \tag{6}$$

Radial direction

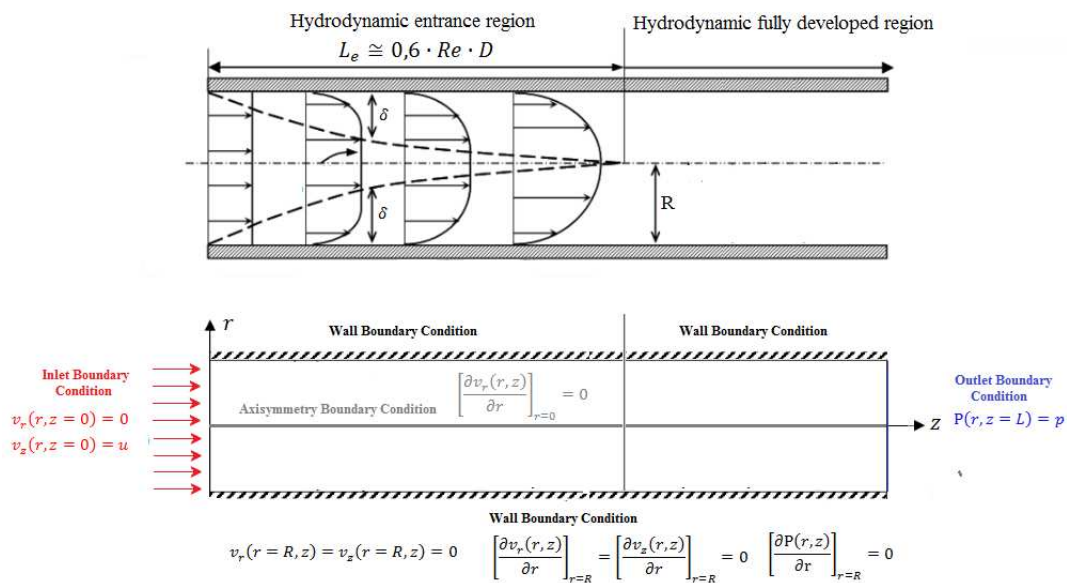
$$\rho \left[v_r \frac{\partial v_r}{\partial r} + v_z \frac{\partial v_r}{\partial z} \right] = -\frac{\partial P}{\partial r} + \mu \left[\frac{\partial}{\partial r} \left(\frac{1}{r} \frac{\partial}{\partial r} (rv_r) \right) + \frac{\partial^2 v_r}{\partial z^2} \right] \tag{7}$$

Axial direction

$$\rho \left[v_r \frac{\partial v_z}{\partial r} + v_z \frac{\partial v_z}{\partial z} \right] = -\frac{\partial P}{\partial z} + \mu \left[\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial v_z}{\partial r} \right) + \frac{\partial^2 v_z}{\partial z^2} \right]$$

In steady, incompressible, laminar flow of a Newtonian fluid in an infinitely long cylindrical pipe of diameter D or radius R = D/2, ignoring the effects of gravity, the boundary condition in pipe is shown in Figure 1:

Figure 1: Scheme of the fluid flow in a pipe and boundary conditions



Source: Adapted from CENGEL, Y.; CIMBALA, J. 2013.

In fully developed laminar flow, fluid particles move at a constant axial velocity along streamlines, while the velocity profile $v_r(r)$ remains constant in the direction of flow. There is no motion in the radial direction, resulting in a velocity component of zero in the direction perpendicular to the flow at all points. The velocity profile $v_r(r)$ is obtained by applying the boundary conditions of symmetry about the centerline and the no-slip condition at the pipe surface. In fully developed laminar flow in a pipe, the velocity profile

is parabolic, with the maximum velocity occurring at the centerline and the minimum velocity (which is zero) occurring at the pipe wall. In addition, the axial velocity $v_r(r)$ is positive for any r , and thus the axial pressure gradient dP/dz must be negative (i.e., pressure must decrease in the flow direction because of viscous effects):

$$v_r(r) = 2V_{ave} \cdot \left(1 - \frac{r^2}{R^2}\right) ; V_{ave} = -\frac{R^2}{8\mu} \cdot \left(\frac{dP}{dz}\right) \quad (8)$$

$$\Delta P = f \frac{L\rho V_{ave}^2}{2D} ; f = \frac{64}{Re} \quad (9)$$

$$\tau_w = f \frac{\rho V_{med}^2}{8} ; f = \frac{64}{Re} \quad (10)$$

The pressure drop is a significant parameter in the analysis of pipe flow as it directly correlates the power demands of the fan or pump needed to sustain the flow. The relation between pressure drop and wall shear stress due to viscous effects for laminar flow in circular pipes is given by equations (9) and (10) respectively.

3 METODOLOGY

The geometry of channel was based in The Gas-Turbine-Modular Helium Reactor (GT-MHR) [6]. The geometry for this 2D analysis consists of half the channel of radius R , which considers a channel wall of length L . The channel is created from four edges forming a rectangle seen in 2D shown in Figure 1. The thermal physical properties of Helium at pressure conditions of 1025 psi (reactor operating pressure) and temperatures of 491 °C (reactor inlet temperature) were extracted from NIST libraries [7], in addition to the parameters to define the laminar study in the fully developed region, such as the hydrodynamic length L_e , the Reynolds number and average velocity, as shown in Table 1.

Table 1: Channel geometry, operative and thermal physical parameters.

Parameter	Value	Reference
Thermal physical properties of Helium		
Pressure, P (psi/MPa)	1025/7.07	GT-MHR
Temperature, T (°C)	491	GT-MHR
Viscosity, μ (Pa s)	$3.83 \cdot 10^{-5}$	NIST
Density, ρ (kg/m ³)	4.4037	NIST
Specific heat, C_p (J/kg K)	5188.7	NIST
Thermal conductivity, k (W/m K)	0.30241	NIST
Channel geometry		
Channel diameter, D (m)	0,015876	GT-MHR

Hydrodynamic length L_e (m)	90D	-
Reynolds number, Re	1500	$L_e \cong 0.6 \cdot Re \cdot D$
Channel length, L (m)	$2 L_e$	-
Average speed, V_{med} (m/s)	0.8211	$Re = \frac{\rho D V_{med}}{\mu}$

Source: Adapted from GA - General Atomic. 1996.

The results obtained in a CFD simulation can be influenced by several parameters, such as the correct description of the physics of the problem, spatial and temporal discretization of the problem (generation of the mesh) and the iterative process until the difference between the solution of the current iteration and that of the previous iteration is less than a certain value. It is necessary that the discretization of the domain be enough to solve the most important physical aspect of the studied flow.

Of all the techniques that currently exist for mesh generation, detailed in [8], the simplest is the creation of a structured mesh, which will be used for laminar analysis. Meshes of this type always have the same number of neighbors and are easier to generate associated with the high intuitiveness of the numerical algorithms used to solve the problems [8].

To properly solve the boundary layer in the region close to the wall, the criteria recommended by the Users' Manual were used [9], where the spacing of the r direction is uniformly distributed and has a recommended value of $R/38$. In the z direction the spacing was also distributed uniformly and adopted different values for the spacings for mesh refinement purposes. Three meshes with different spacing in the z and r direction were modeled; their parameters are presented in Table. 2.

Table 2: Meshes used in modeling.

Mesh	Number of Elements	Mesh type
Mesh 1	19 x 142	Coarse
Mesh 2	38 x 284	Medium
Mesh 3	76 x 568	Fine

Source: own elaboration

The CFD code used for the present study is Fluent version 2019 R3, employing the 2D portion of the code in double precision. The numerical method used in Fluent is the finite volume method. The discretization of the terms is given in the Users' Manual [9]. In this CFD study, the viscous terms were discretized using a 2nd order centered difference scheme, while the pressure term was evaluated using a centered scheme. The convective terms were discretized using the user-specified 2nd order QUICK scheme. The segregated solver was used to solve the discretized equations, with default under-

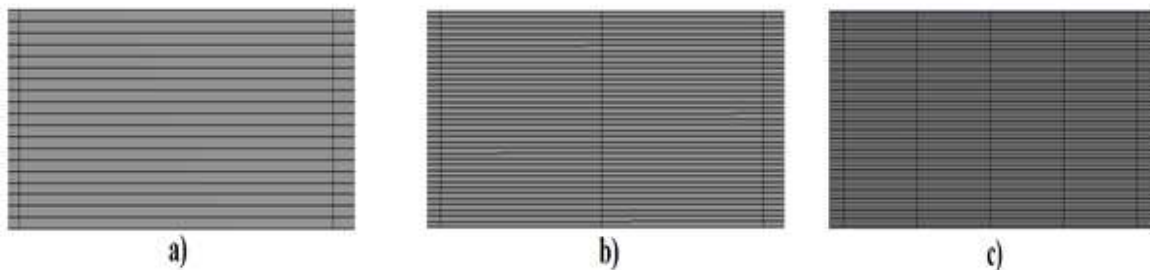
relaxation factors employed. To achieve pressure-velocity coupling, the SIMPLE algorithm was utilized.

The issue of iterative convergence was investigated by comparing Fluent results for fully developed flow in a pipe with the exact analytical solution. This flow is also called Poiseuille flow. Iterative convergence addresses the issue of how low the tolerance levels for equation residuals need to be in order to ensure that the solution is converged. The default value for convergence in Fluent is 1.0×10^{-3} . The solution for a simple flow in a pipe with a length of 90 pipe diameters was compared to the exact solution for convergence tolerances of 1.0×10^{-3} to 1.0×10^{-7} .

4 RESULTS AND DISCUSSION

In the Figure 2 three meshes are shown and have acceptable qualities as recommended in the Users' Manual, although the best quality control parameters are those corresponding to mesh 3 (Fine), mesh 1 (Coarse) and mesh 2 (Medium) are also suitable for the case study and will presumably consume less computational resources.

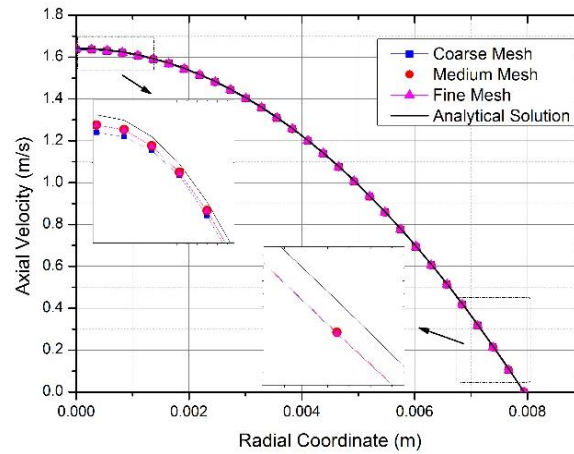
Figure 2: Meshes designed for laminar analysis where a) 19x142 (coarse) b) 38x284 (medium) and c) 76x568 (fine).



Source: own elaboration

The axial velocity profiles in the fully developed laminar region using the meshes projected in the model and for code default tolerances are shown in Figure 3. The modeled velocity profiles have the same behavior as the analytical solution, in the central region of the channel, the maximum velocity on the axis has a theoretical value of 1.6423 m/s, the relative errors were 0.4% for the coarse mesh and 0.2% for fine and medium meshes, as can be seen in the amplified region. The fine and medium meshes present similar results in the central region and in the near wall region.

Figure 3: Axial velocity profiles for meshes 1 (coarse), 2 (medium) and 3 (fine).



Source: own elaboration

In order to perform grid sensitivity analysis, the discretization error will be evaluated using a technique commonly used in the CFD community known as the Grid Convergence Index (GCI), which was developed by Roache [10, 11]. Traditional error convergence analysis relied on the exact (analytical) solution to determine the error and the rate of convergence through graphical representation. However, for most practical problems, the exact solution is unknown.

For the special case, the three meshes are generated with a constant grid refinement ratio. The initial grid used for the simulations had a total of 38x284 cells. It was decided to proceed with coarse grid and one finer mesh. The details of the different meshes are presented in Table 3. The notation ϕ_h is adopted to describe the solution for the finest mesh.

In reference to the finest mesh, all subsequent meshes are denoted accordingly. The subsequent grid size has cell dimensions that are doubled in both directions, leading to the notation ϕ_{2h} .

Table 3: Mesh analysis.

Parameter	ϕ_h (fine)	ϕ_{2h} (medium)	ϕ_{4h} (coarse)
Number of cells in the r Direction	76	38	19
Number of cells in the Z Direction	568	284	142
Smallest cell height (m)	2.089×10^{-4}	4.178×10^{-4}	8.356×10^{-4}
Total number of cells	43168	10792	2698

Source: own elaboration

It can be shown [10] that the discretization error of a grid is approximately (with a constant grid refinement ratio):

$$\varepsilon_h^d \approx \frac{\phi_h - \phi_{2h}}{2^a - 1} \tag{11}$$

where a is the order of the scheme and is given by

$$a = \frac{\log\left(\frac{\phi_{2h} - \phi_{4h}}{\phi_h - \phi_{2h}}\right)}{\log(2)} \tag{12}$$

The notation "2" in both equations signifies the increase in mesh dimensions. Based on Equations (11-12), it is evident that a minimum of three meshes is necessary to evaluate the discretization error. To avoid calculation errors arising from the logarithm of a negative number, it is essential for the three solutions to exhibit monotonic convergence [11, 12]. According to the theory of Richardson Extrapolation, the solution obtained from the finest mesh can be combined with the discretization error derived from Equation (11) to achieve an approximate grid-independent solution. This can be expressed in equation form as follows:

$$\Phi = \phi_h + \varepsilon_h^d \tag{13}$$

The results from the grid sensitivity analysis are shown in Table 4 and plotted in Figure 3. Only maximum axial velocities calculated are part of this comparison.

Table 4: Discretization error and Richardson Extrapolation Results.

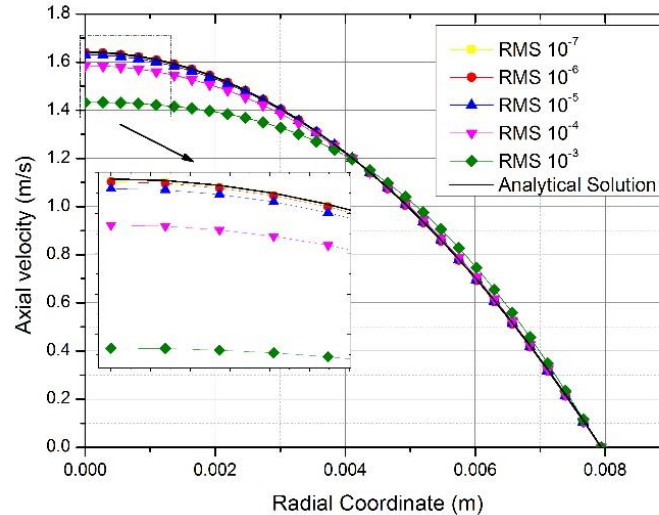
Parameter	Value
Order of the scheme, "a"	1.89
Discretization Error, ε_h^d (m/s)	0.00186
Finest mesh solution, ϕ_h (m/s)	1.63858
Richardson Solution, Φ (m/s)	1.64044
Analytical solution, V_{axial} (m/s)	1.64227

Source: own elaboration

In the case of the iterative convergence study, the medium mesh was used to compare the exact solution with the results in the fully developed region of axial velocity for convergence tolerances from 1.0×10^{-3} to 1.0×10^{-7} , the results obtained can be seen in Figure 4, convergence tolerance between 1.0×10^{-3} to 1.0×10^{-7} with maximum relative errors of 0.2% for 1.0×10^{-7} and 1.0×10^{-6} and 0.7% for 1.0×10^{-5} . The convergence value

that presents a lower computational cost and acceptable results was 1.0×10^{-6} this value will be a reference for the other analysis.

Figure 4: Axial velocity profiles for different convergence tolerances.

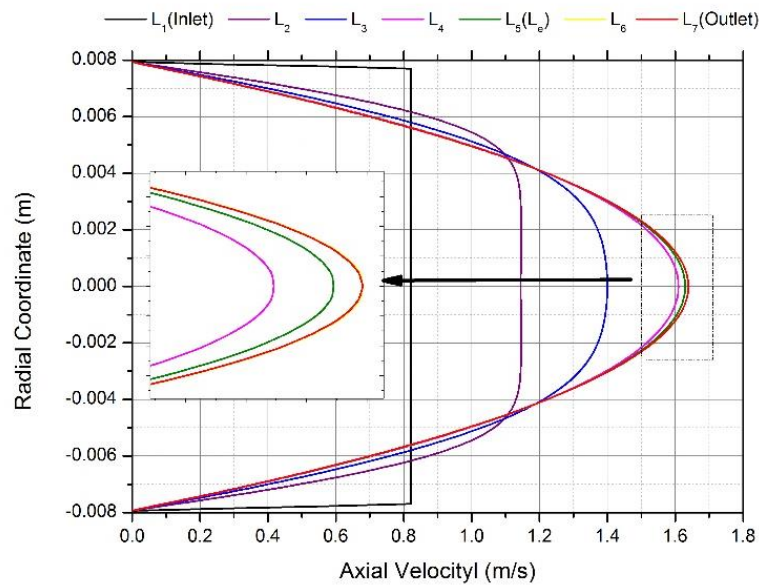


Source: own elaboration

The most suitable mesh and the reference convergence tolerance were the medium mesh and the convergence value of 1.0×10^{-6} . The analysis to characterize the laminar flow in the channel will be modeled, such as velocity distributions, shear stress and pressure drop.

At the entrance of the channel the velocity is uniform, viscous effects become important and a boundary layer develops with increasing channel length. At a certain point in the L_e flow, the boundary layer from the walls grows to such an extent that they all merge on the centerline of the channel (fully developed laminar region). From that point onwards, the viscous effects extend along the entire cross-section of the tube and the velocity profile no longer changes with position as seen in Figure 5.

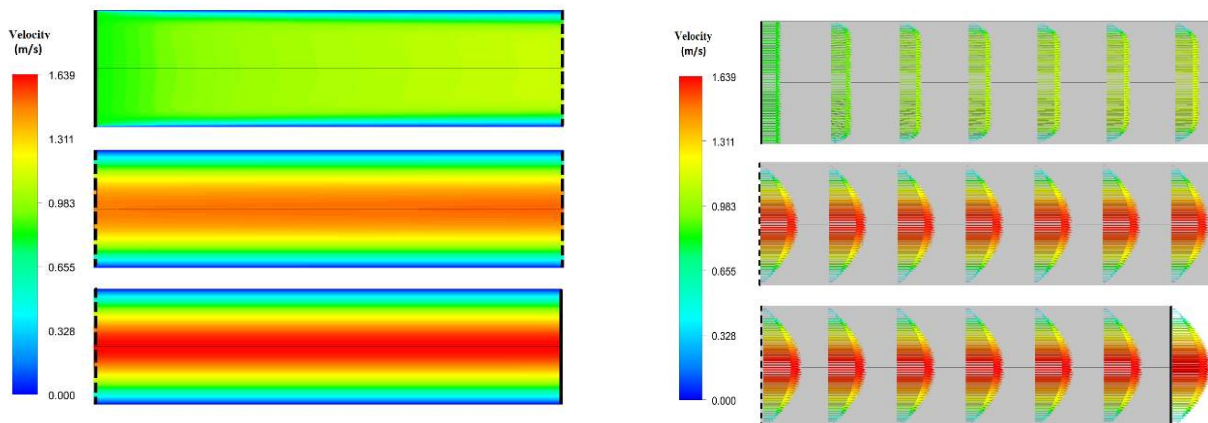
Figure 5: Development of the velocity profiles in a laminar flow.



Source: own elaboration

In Figure 5 and 6, as the laminar flow develops in the channel under frictional forces, the velocity distribution approaches a parabolic-like behavior. In Figure 5, L1 corresponds at the channel entrance where the velocity is uniform; L2, L3 and L4 are located in the intermediate regions where the velocity profiles are flat in the central region. L5 starts the fully developed laminar region (Le) and velocity starts to be parabolic. L6 is located in the fully developed laminar region where in this region the parabolic profile of the velocity no longer changes and finally L7 is the channel output.

Figure 6: Velocity profiles along the laminar flow in three regions of the channel (inlet, middle and outlet).

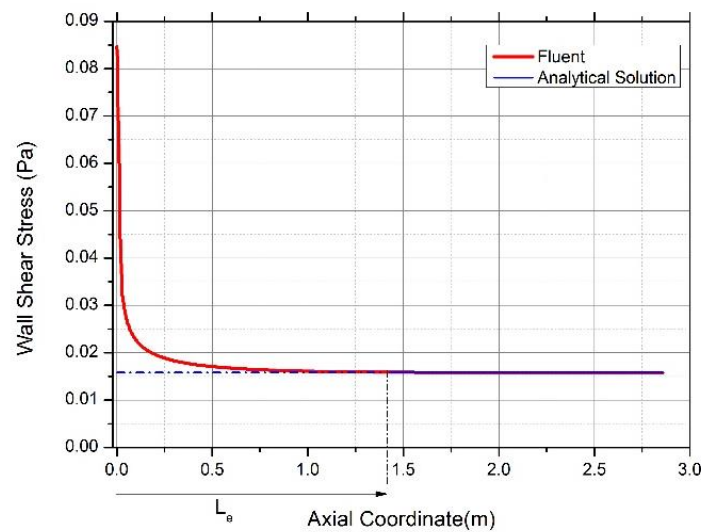


Source: own elaboration

The wall shear stress in the channel is highest at the inlet of the pipe, where the boundary layer thickness is smaller and the pressure drop is higher; in the inlet region of

the tube the effect is always increasing in relation to the average friction factor. The wall shear stress will gradually decrease until the value remains constant along the flow direction in the fully developed region because the boundary layer from the walls grows to such an extent that they all merge on the centerline of the channel and the viscous effects extend along the entire cross section of the tube. The behavior of the shear stress along the entire channel is shown in Figure 7. In the fully developed laminar region, the theoretical wall shear stress (eq. 10) in relation to the simulated one has a relative error of 0.4%.

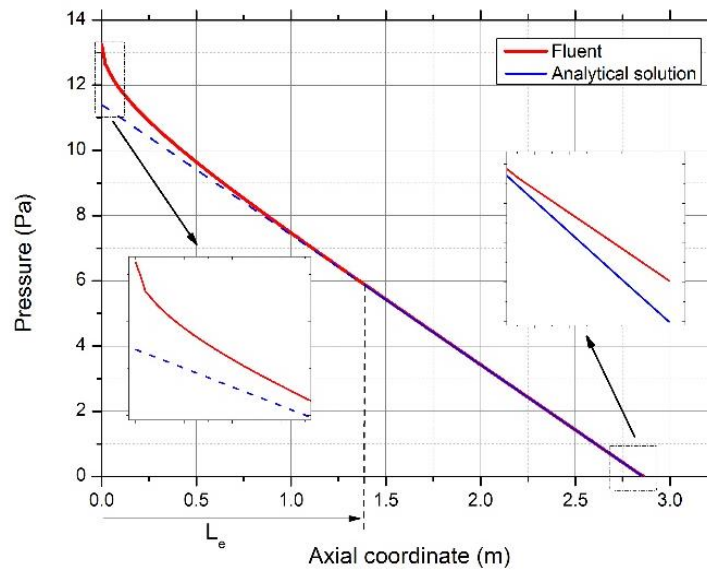
Figure 7: Wall shear stress along the channel, where is constant in the fully developed region.



Source: own elaboration

As the pressure drop depends on the friction factor (in turn, on the wall shear stress) and on the physical and geometric parameters of the flow, and as the friction factor is constant, the pressure drop caused by the friction of the laminar flow does not depend on the roughness of channel. Therefore, the variation of pressure in relation to the variation of the length of the channel $(dP)(dz)$ remains constant along the channel, so that the pressure decreases linearly in the fully developed region. In the laminar region before it becomes fully developed, the pressure drop does not decrease linearly. The comparison of the pressure drop along the simulated and analytically calculated (eq. 9) channel was performed only in the fully developed region, the values show very good agreement as seen in Figure 8 the maximum relative error was 0.2%.

Figure 8: Pressure drop along the channel, where it decreases linearly in the fully developed region.



Source: own elaboration

5 CONCLUSION

To understand the dynamics of laminar flow, a validation exercise was conducted to compare the obtained results with the theoretical basis of flow through a pipe. A proper performance of the CFD code for the studied case was indicated by the good agreement between the CFD results and analytical solution.

The validation of the different parameters of the modeling with the theoretical equations is the point of concern of this laminar analysis in the channel, which will be used for future work in a turbulent analysis of a GT-MHR reactor.

Finally, a grid sensitivity analysis was conducted on the mesh. The discretization error for axial velocity was computed at a specific location in the pipe, and Richardson extrapolation was employed to obtain the grid-independent solution. The resulting values showed good agreement with analytical values from the literature. The percentage error between the analytical and grid-independent solutions for axial velocities is on the order of $10^{-1}\%$.

ACKNOWLEDGMENT

The authors are grateful to CNEN (*Comissão Nacional de Energia Nuclear*), CAPES (*Coordenação de Aperfeiçoamento de Pessoal de Nível Superior*), FAPEMIG (*Fundação de Amparo à Pesquisa do Estado de Minas Gerais*) and CNPq (*Conselho Nacional de Desenvolvimento Científico e Tecnológico*) for the support.

REFERENCES

- [1] MOHANTA, L.; CHEUNG, F. B.; BAJOREK, S. M. ; TIEN, K.; HOXIE, C. L. Experimental study of laminar mixed convection in a rod bundle with mixing vane spacer grids. **Nuclear Engineering and Design**, v. 312, p. 99-105, 2017.
- [2] UMAR, E.; TANDIAN, N. P.; SYURYAVIN, A. C.; RAMADHAN, A. I.; PRAYITNO, J. H. CFD Analysis of Convective Heat Transfer in a Vertical Square Sub-Channel for Laminar Flow Regime. **Fluids**, v. 7, n. 6, p. 207, 2022.
- [3] WANG, Z.; ZHANG, R.; YU, T.; LIU, Z.; HUANG, Y.; WANG, J.; CONG, T. Numerical study on laminar-turbulent transition flow in rectangular channels of a nuclear reactor. **Frontiers in Energy Research**, v. 8, p. 67, 2020.
- [4] CENGEL, Y.; CIMBALA, J. **Fluid Mechanics Fundamentals and Applications**, 4th Edition, McGraw Hill, 2013.
- [5] SWANEY, R. E.; BIRD, R. B. **Transport Phenomena**, 2nd edition, New York, John Wiley & Sons, 2007.
- [6] GA - General Atomic. **Gas Turbine-Modular Helium Reactor (GT-MHR) Conceptual Design Description Report, GA Project No. 7658**, General Atomic, 1996.
- [7] NIST - National Institute of Standards and Technology. Available at: <<https://webbook.nist.gov>>. Last accessed: 08 Aug. 2020.
- [8] Versteeg, H. K.; Malalasekera, W. **An introduction to computational fluid dynamics: the finite volume method**, 2nd edition, Pearson education, 2007.
- [9] ANSYS Fluent. **ANSYS Fluent: Theory Guide**. Release 2019 R3, 2020.
- [10] ROACHE, P. J. Perspective: A Method for Uniform Reporting of Grid Refinement Studies. **ASME Journal of Fluids Engineering**, Vol. 116, September, 1994.
- [11] DE FARIA, F. P., GODINO, D. M., CORZO, S. F., RAMAJO, D. E., COSTA, A. L., & LIMA, C. P. B. Heat transfer in a simulated spent fuel pool for different types of spent fuels. **Brazilian Journal of Development**, 9(1), 5004-5019, 2023.
- [12] FERZIGER, J. H.; PERIC, M.; STREET, R. L. **Computational methods for fluid dynamics**, springer, 2002.