Niğde Ömer Halisdemir Üniversitesi Mühendislik Bilimleri Dergisi
Niğde Ömer Halisdemir University Journal of Engineering Sciences

Araştırma makalesi / Research article

www.dergipark.org.tr/tr/pub/ngumuh / www.dergipark.org.tr/en/pub/ngumuh

Nondimensionalization of Newtonian flow CFD simulation by manipulating commercially available software

Ticari olarak erişilebilen yazılımı manipüle ederek Newtonsal akış HAD benzeşiminin boyutsuzlaştırılması

Eyüb Canlı[1,*]

[1] *Selçuk Üniversitesi, Makine Mühendisliği Bölümü, 42130, Konya Türkiye*

Abstract

Computational Fluid Dynamics (CFD) is a powerful tool which has a widespread utilization among industry and academia that sometimes leads to superficial evaluation of its means. This work provides an approach and methodology to set up nondimensional CFD analyses through commercially available software packages that are intrinsically dimensional. By the proposed method, one can generate brief and strong results without much post processing thanks to the carefully selecting independent variables that constitute the governing nondimensional numbers. Several aspects of the nondimensionalization scheme are tried via quantitative cases based on the governing nondimensional scale, i.e., the Reynolds number to show their conveniency and disadvantages for investigating different physics. In-house code nondimensionalization is also evaluated based on the present results. Practical experiences on the CFD simulation conduction are shared. It is concluded that the nondimensionalization strategy should be based on the parameters to be investigated to reduce the CFD workload.

Keywords: Axisymmetric flow, Incompressible, Reynolds number, Steady

Öz

Hesaplamalı Akışkanlar Dinamiği (HAD), endüstri ve akademide yaygın olarak kullanılan güçlü bir araçtır; ancak bu yaygın kullanım bazen yöntemin yüzeysel değerlendirilmesine yol açabilmektedir. Bu çalışma, özünde boyutlu olan ticari yazılım paketleri kullanılarak boyutsuz HAD analizlerinin nasıl kurulabileceğine dair bir yaklaşım ve metodoloji sunmaktadır. Önerilen yöntemde yönetici boyutsuz sayıları oluşturan bağımsız değişkenlerin zekice seçilmesi sayesinde analiz sonrası fazla işlem gerekmeksizin öz ve güçlü sonuçlar elde edilebilir. Boyutsuzlaştırma şemasının çeşitli yönleri, belirleyici boyutsuz ölçek (Reynolds sayısı) temel alınarak nicel vakalar üzerinden test edilmiş ve farklı fiziksel olayların incelenmesindeki avantajları ile dezavantajları ortaya konmuştur. Ayrıca, mevcut sonuçlara dayanarak özel (in-house) kod boyutsuzlaştırması da değerlendirilmiştir. HAD simülasyonlarının yürütülmesine dair pratik deneyimler paylaşılmıştır. Sonuç olarak, HAD iş yükünü azaltmak için boyutsuzlaştırma stratejisinin incelenecek parametrelere göre belirlenmesi gerektiği ortaya konmuştur.

Anahtar Kelimeler: Eksenel simetrik akış, Sıkıştırılamaz, Reynolds sayısı, Kararlı

1 Introduction

Computational Fluid Mechanics (CFD) is a common, widespread, powerful tool in engineering and related research. On the other hand, modern day professional world demands and expects too much outcome and skill from real people, due to abundant tools such as data driven software and rapidly changing technology. This partly leads to shallow and superficial utilization of engineering tools, specifically software tools, due to shortened learning and orientation periods for the theory, immense number of sources that harden distinguishing the proper ones, less time for thinking. This is no different for CFD. And it is partly observed in engineering students [1, 2]. The author of the present paper experienced such phenomena. Two preliminary works for a Ph.D. thesis were published after they were discussed in a respected conference [3, 4], however, later on, it was understood that the conference papers contain methodological mistakes, which are understood by means of the Ph.D. thesis that was completed later [5]. One of the significant reasons for superficial and problematic utilization of CFD is that it gives solutions and results even though the setup is not proper for accurate solution of the investigated problem or physics. When validation cannot be done, this may lead to harm and losses. Another phenomenon may be false validation that occurs when the CFD results match the benchmark data while the CFD setup is wrong and does not reflect the actual physics. In that case, the user who thinks that the setup is correct will generate erroneous interpolation or extrapolation results. Another issue is the vast number and volume of CFD data. Dimensional data populates reports and makes evaluation and interpretation difficult whereas nondimensional data represented via fewer variables and parameters ease data handling and evaluation. Last but not least, there is this wrong evaluation and assessment of CFD results issue due to the lack of proper knowledge. As a conclusion, CFD methodology should be discussed more via literal and verbal means. Among important aspects of such discussion, using

* Sorumlu yazar / Corresponding author, e-posta / e-mail: ecanli@selcuk.edu.tr (E. Canlı)
Geliş / Received: 21.08.2025 Kabul / Accepted: 10.10.2025 Yayımlanma / Published: xx.xx.20xx
doi: 10.28948/ngumuh.1763595

similarity and dimensionless analysis retain significant place. Similarity approach that contains dimensionless analysis intrinsically, is sometimes a must where real world scale system analysis via experimentation or computational study is not feasible, possible or available. In the present paper, nondimensionalization, dimensionless analysis, a way to use commercial CFD packages as pseudo-dimensionless, practical aspects and sharing experiences are intended by means of an examination of a solid quantitative case. The added value of the paper is the practical nondimensionalization of the CFD results with less post processing. The rationale is the careful and clever selection of the independent variables that form the governing dimensionless numbers.

There are two significant papers in the literature, where one is about the derivation of dimensionless governing equations from vector form equations for axisymmetric domains [6], and the other one uses a commercial CFD package to conduct a simulation that has a similar domain to the present paper [7]. However, Canli et al. [6] derives the governing equations for in-house coding while Canli et al. [7] reports results only for turbulent flow and create the dimensionless scheme by means of only the dynamic viscosity magnitude. Nevertheless, those two mentioned works are strongly recommended for further reading. Derivation of differential forms of the governing equations shows that the main variable can be reduced only to Reynolds number (Re) in laminar and turbulent cases for Newtonian fluids under specific assumptions. The simulation that shows the developing turbulent flow in the entrance length of the axisymmetric pipe flow is a good example for CFD setup and necessary validation and verification steps, as well as nondimensionalization. Other two good examples of nondimensional CFD analyses of heat transfer and fluid flow phenomena that have axisymmetric cylindrical domains can be viewed by two different conference proceedings in the literature [8, 9]. Canli et al. [8] simulates a simultaneous development in a pipe, i.e., thermal and hydrodynamic development that is being realized simultaneously by means of commercial CFD while taking Re and Prandtl number (Pr) as the main variables. They arrange the dimensional variables in such a way that they obtain the Re and Pr as the desired variable level values. On the other hand, it is seen that generally thermophysical properties have values above and below unity while characteristic length variable is taken as unity. Ceviz et al. [9] on the other hand, simulate natural convection heat transfer in a vertical pipe via commercial CFD. They also use the approach for making the commercial dimensional CFD nondimensional by altering thermophysical properties of the fluid in such a way that the results can be used as non-dimensional without much post processes. However, it should be noted that preparing a commercial CFD setup to have results that can be regarded as nondimensional is relatively harder than obtaining nondimensional results via commercial CFD for Newtonian flow, or forced convection with Newtonian flow, since more thermophysical properties are involved in natural convection and the governing dimensionless numbers and related physics change. According to literature and previous works, nondimensionalization is needed for; benefiting from similarity to reduce experimental and computational hardship; reducing test and trial numbers by only changing the governing nondimensional variables instead of immense number of full factorial of dimensional variables; and scanning robust intervals for more universal results. In in-house coding for nondimensional codes, the formulation is modified to have nondimensional equations and variables. Therefore, the results obtained are nondimensional automatically. In case of commercial CFD, magnitudes and values of dimensional variables are selected to have a certain value of the nondimensional governing variable. The careful and clever selection of dimensional variable values enables one to do less work after dimensional results are obtained to make them nondimensional. For instance, if the density variable is set to unity, and Re is set to the examination value while the density is unity, then the obtained density dependent results will not need to be divided by density to make them nondimensional since dividing a number to unity does not change anything. However, one cannot set each dimensional variable to unity since the governing nondimensional number always becomes unity in such a case. In fact, sometimes, only a few dimensional variables can be set to unity while others have different orders of magnitudes of unity and/or other numbers. So, selecting the variable types becomes an issue for completing CFD examination with less effort, especially with less post processing and data reduction. The present paper focuses on effects of selecting strategies of the unity variables for making analyses with commercial CFD nondimensional while evaluating different aspects of nondimensionalization in respect of CFD accuracy and cost. Before proceeding further, as a speculation raising from vocational intuition, the author of the present paper suspects that commercial codes may also use nondimensional governing equations in the core of their black box code packages, but their user interfaces require dimensional variable inputs for sure.

Literature has been surveyed for similar studies to the current paper regarding its main aims. The most recent and most significant work in literature, in terms of the focus and aim, is found to be from 2010 [10] though effort was paid to find a more recent and significant one. Nevertheless, Dillon et al. [10] lay out a similar task to the present paper by trying to compare a nondimensional analysis and a dimensional one on the same case via using a commercial CFD, i.e., COMSOL. The case is a natural convection heat transfer case in a vertical annulus. The authors have the literature benchmark data available so that they manipulate commercial CFD code by selecting dimensional variables to obtain dimensionless results as in the current paper and then conduct the CFD with a dimensional setup. For the same governing dimensionless number, the Rayleigh number (Ra), the authors claim that the results match each other very well. On the other hand, the authors claim that the strategy for making the commercial CFD code nondimensional by the case setup has effects on code usage feasibility and roundoff errors. However, one cannot see those aspects in the paper via the results that are reported by the authors, though three options for nondimensional setup are presented in a table. As a conclusion, that literature paper differs from the present paper by the differences in the cases, the literature paper has different physics with more complex domain, and it does not deliver all the promises in its abstract and introduction. It does not show the practical implications of the nondimensionalization strategy options. Literature was surveyed for more similar works; however, no more directly related sources were found. Therefore, a few examples of CFD examinations with either in-house codes with nondimensional formulation or results from commercial CFD that are completely nondimensional by data reduction are reviewed in the following. Gamboa et al. [11] uses six independent nondimensional variables to optimize a Tesla valve geometry via commercial CFD. So, we see that both the inputs and the outputs from the commercial CFD are nondimensionalized for examining and evaluating the phenomena. Commercial CFD can also be utilized to realize some specific and certainly noncommon particular custom tasks via User Defined Functions (UDF) [12]. Cheimarios et al. [12] used UDFs to use FLUENT for such a purpose. It is shown that a complex CFD case involving natural and forced convection heat transfer can be processed for nondimensional evaluation of the specific case. Two plots are given as completely dimensionless in Nusselt number (Nu) versus Re scheme while temperature contours have Kelvin unit as a dimension. It is understood that dimensionless formulation can be used via UDFs in Fluent. This means that the dimensional CFD setup should be done carefully and/or there should be a conversion step for the transition of data between FLUENT dimensional solver and UDF dimensionless formulation. The authors also state that they do posteriori data reduction to calculate Nu and Re. Artificial Neural Networks (ANN) are being used also for engineering systems including CFD studies [13]. It is known that ANNs work well with normalized data which is a frequent output of nondimensional CFD. Morshed et al. [13] describes their CFD data by means of dimensionless variables and they also use their dimensionless data in ANN, indicating that there may be side benefits of dimensionless analyses as the data can be easily used in statistical methods such as machine learning. Mason-Jones et al. [14] use nondimensional velocity curves to study tidal stream turbines via CFD. It is a type of similarity approach. The authors derive dimensionless number groups to set up the nondimensional research case. Although all the CFD setup and the case are described by ratios, and results are given nondimensional, it is understood that the input values such as boundary conditions and thermophysical properties have dimensions and their values are realistic as in their real-world states. Yet, it shows the need for dimensionless data for evaluating engineering systems.

The evaluation and review of the literature show that dimensionless analysis and evaluation are key parts of engineering and research studies including CFD. To have this, some studies involve self-coding with dimensionless equations. Others conduct dimensional CFD and then use data reduction to have dimensionless results. Dimensionless results involve dimensionless number groups such as coefficients, ratios, normalized magnitudes and similar elements/features. One solution by commercial CFD that has user interfaces for dimensional analyses is that user inputs such as thermophysical properties and boundary conditions as well as spatial domain dimensions can be selected in such a way that governing dimensionless numbers, coefficients and groups are set to desired variable values and some of the user inputs can be set to unity so that the obtained results do not need additional processing for nondimensionalization and normalization. Literature has a few examples including previous studies of the author of the present paper. Also, as mentioned in the earlier paragraph, Dillon et al. [10] has an explicit methodology paper on the subject. As a conclusion, the literature review encourages more work that discusses the nondimensionalization of commercial CFD by user input selection schemes. This paper may be the first case study where domain length is used to set up the governing nondimensional number and apply the introduced methodology. Since commercial CFD codes are black boxes, possible and probable numerical limitations are being tested systematically. CFD is a tool for research but CFD itself is also being studied as an engineering science. Since rounding errors, truncation errors, black box limitations and arrangements and nonlinearity are involved, this present work has become a numerical experiment.

The present work shows how spatial domain design, selection of length dimensions and their order of magnitudes, selection of thermophysical properties, arranging boundary conditions can be used to manipulate the commercial CFD to have nondimensional results with the least amount of post processing. This is the added value of the work. The rationale is the less data by fewer nondimensional variables and parameters that can represent numerous dimensional data. Since there are options to apply this technique, the present work uses a well-known benchmark case, i.e., axisymmetric pipe hydrodynamic entry length, to show the feasibilities of the options quantitatively. Alongside the prospects of the discussed methodology, dimensionless analyses with nondimensional equations and in-house codes are also discussed considering the implications of the present nondimensionalization strategies. Practical experiences are shared. The following second methodology title describes the commercial CFD setup and the nondimensionalization options. In the results and discussion section, i.e., the third title, some spatial parameter distributions are presented in order to discuss them in respect of the methodological options. Finally, a conclusion summarizes the main findings of the work.

2 Methodology

The method for nondimensionalization in the present work relies on careful and clever selection of the values of the independent dimensional variables so that the nondimensional governing numbers are set to desired parameter levels and less post processing is necessary to make CFD results nondimensional. By this way, one who desires to utilize nondimensionalization for the sake of clarity and generalization spends less effort by using the proposed methodology, as the added value of the present paper.

Demonstration of the present methodology for making commercial CFD nondimensional necessitates a simple yet sufficient case study that can be easily validated. Complex cases might lead to misconceptions while their validation and verification might take up too much space that would be needed for the discussion of the proposed methodology. Therefore, in this work, pipe hydrodynamic entry length is used as case studies for laminar and turbulent flow while axisymmetric conditions apply. This is a well-known case that can be easily validated, and it has relatively low number of independent dimensional variables that reduces into a single dimensionless independent variable, i.e., Re. In addition, the author of the present work has a history of the selected case with similar nondimensionalization approaches. These mentioned facts are the reasons for the selection of the case. By selecting this particular case, mesh independency becomes unnecessary since the results to be obtained are almost known entirely and additional experimentation becomes unnecessary. In the meantime, one should clarify the nuances between the validation and verification terms. Validation is a concept where the analyses and their methodology are compared to a data or case that their accuracy is known, via results. When the analyses' results are sufficiently accurate based on benchmark data, the methodology and the results are deemed validated. Verification is, on the other hand, inspection and examination of methodological steps based on theory. In case of CFD, those steps may be accuracy of boundary conditions, domain setup, values of residuals during iterations, discretization schemes, etc. Sometimes verification and validation are used interchangeably, however, their true implications are different.

A particular problem about methodology descriptions of the CFD studies is the presentation and description of the governing equations. This is especially true for commercial CFD users. Since they do not actually handle the derivation and organization of the equation setting and rather just interact with the software user interface, they tend to give a standard equation set from literature. Sometimes this equation set is in vector form which means the explicit differential set of equations specific to the problem can be anything. However, the methodology section of articles should be like a recipe. In that sense, the governing equations should reflect the assumptions, simplifications, and related physics about the problem. Accordingly, in the present work, the problem assumptions are introduced first, problem domain is explained with a diagram in the following, and the governing equations are presented afterwards.

The case for illustrating the nondimensionalization of the commercial CFD by user inputs assumes;

- The fluid is Newtonian and incompressible
- There is no body forces acting on the fluid and hence the flow, including gravity
- Flow is and accordingly the CFD domain is axisymmetric (hydrodynamic entry length inside a pipe)

The CFD domain is shown in Figure 1. The symbols in Figure 1 that show the primary primitive variables and parameters are explained in the following after the governing equations. The axisymmetric domain has a differential volume difference between the wall and the axis. The differential volume of the wall proximity is larger than the differential volume of the axis proximity. It is like an infinitely thin pizza slice when its cross section that is perpendicular to the flow is viewed. Another good diagram of the domain can be viewed from [8].

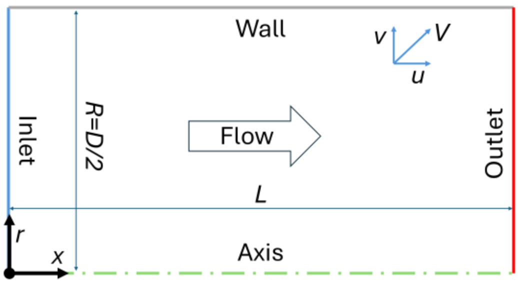


Figure 1. Diagram of the CFD domain

The governing equations for the CFD domain in Figure 1 considering the assumptions are given between Equations (1) – (3).

$$\frac{\partial u'}{\partial x'}+\frac{\partial v'}{\partial r'}+\frac{v'}{r'}=0 \tag{1}$$

$$u'\frac{\partial u'}{\partial x'}+v'\frac{\partial u'}{\partial r'}=-\frac{1}{2}\frac{\partial p'}{\partial x'}+\frac{1}{\mathrm{Re}_{\mathrm{eff}}}\left[\frac{\partial^2 u'}{\partial x'^2}+\frac{1}{r'}\frac{\partial}{\partial r'}\left(r'\frac{\partial u'}{\partial r'}\right)\right] \tag{2}$$

$$u'\frac{\partial v'}{\partial x'}+v'\frac{\partial v'}{\partial r'}=-\frac{1}{2}\frac{\partial p'}{\partial r'}+\frac{1}{\mathrm{Re}_{\mathrm{eff}}}\left[\frac{\partial^2 v'}{\partial x'^2}+\frac{\partial}{\partial r'}\left(\frac{1}{r'}\frac{\partial r'v'}{\partial r'}\right)\right] \tag{3}$$

All the terms in Equations (1) – (3) are dimensionless. The terms are explained in the following. The prime indicates that the primitive variables/parameters are nondimensionalized. Equation (1) is continuity in axisymmetric domain that is defined in cylindrical coordinates. Equation (2) is x momentum and (3) is r momentum, again in the axisymmetric domain defined by cylindrical coordinates and they are also dimensionless. About the symbols, u' is axial velocity while v' is radial velocity components. They are scalar magnitudes. The coordinates x' and r' are axial and radial coordinates, respectively. Dimensionless pressure is shown by p' and $\mathrm{Re}_{\mathrm{eff}}$ is the effective Re. $\mathrm{Re}_{\mathrm{eff}}$ reduces to Re in laminar flow and it contains turbulent viscosity and Re in turbulent flow. So, the governing equations tell us about all the assumptions that are mentioned previously and the axisymmetric domain. The governing equations also tell us more about the handling of turbulence modelling since turbulent viscosity is in fact a turbulence modelling approach. It means that the research involves two equation Reynolds Averaged Navier Stokes (RANS) turbulence modeling depending on calculation of turbulent viscosity. The only term in Equations (1) – (3) is the Re that a user can use as an input as independent variable since turbulent viscosity in Re_{eff} is calculated by CFD. Therefore, Re is defined in Equation (4).

$$\mathrm{Re} = \frac{u_{\mathrm{inlet}} \rho D}{\mu} \tag{4}$$

In Equation (4), axial dimensional inlet velocity component is a constant and actually constitutes the inlet boundary condition. Thermophysical properties of the fluid, ρ and μ are density and absolute/dynamic viscosity respectively. By means of unit homogeneity, Re becomes a dimensionless group or number that represents the ratio of inertial forces over viscous forces. A higher Re indicates that inertial forces dominate viscous forces and create turbulence by destroying lamellae structure in the flow. A lower Re indicates that viscosity governs the flow and leads to laminar flow with orderly flowing fluid. The characteristic length scale in this case is the diameter *D* since the pipe walls create a boundary layer thickness that reaches the pipe axis so that the entire flow domain is actually in the boundary layer for hydrodynamically developed flow. We therefore use the *D* even for the entrance length to measure its length in terms of *D*. The nondimensionalization of either the dimensional terms in the governing equations or the dimensional results can be done via constants that constitute the CFD case. For instance, dimensionless velocity components can be obtained by dividing the dimensional velocity components by the inlet velocity. This is shown by Equation (5).

$$u' = \frac{u}{u_{\mathrm{inlet}}}, v' = \frac{v}{u_{\mathrm{inlet}}} \tag{5}$$

Selecting inlet velocity and hence the inlet boundary condition unity makes the obtained velocity distributions nondimensional. In other words, one can use the quantities of the velocities as if they are nondimensional values since dividing them by unity does not change anything but diminish their dimensions. The coordinates can be nondimensionalized via the characteristic length, i.e., the diameter *D*. This is shown by Equation (6).

$$x' = \frac{x}{D} \text{ or } x' = \frac{x}{D\,\mathrm{Re}}, r' = \frac{r}{D} \tag{6}$$

Dimensionless axial and radial positions, when nondimensionalized by the pipe diameter, show ratios. In case of radial coordinate, this is a normalized value between zero and one since the radial distance cannot exceed the diameter. Since the domain is an axisymmetric one, the radial distance changes between zero and 0.5. On the other hand, dimensionless axial distance is not a normalized one and it is expected to be one or two orders of magnitude bigger than unity since the hydrodynamic entry length is realized at those distances. Equation (7) shows the hydrodynamic entry length in pipes for laminar and turbulent flow, coherent with the assumptions of the present case [8].

$$\begin{aligned} &\mathrm{Re} \le 2300 \Rightarrow L_{\mathrm{entry}} = 0.05\,\mathrm{Re}\,D \\ &\mathrm{Re} > 2300 \Rightarrow L_{\mathrm{entry}} = 1.359\,\mathrm{Re}^{0.25}\,D \end{aligned} \tag{7}$$

It is seen that entry length is at least one order of magnitude bigger than pipe diameter for Re=200 and above in laminar flow. On the other hand, hydrodynamic entry length becomes significantly shorter for turbulent flow. Changing hydrodynamic entry length implies that velocity distributions change as Re change by user inputs. This may lead to repetitive tuning of CFD setup for each Re since mesh dependency may change as well as the convergence. Therefore, one may think to nondimensionalize the axial length not only by the pipe diameter but also by the Re together with the *D*, at least for the laminar interval since the entry length changes linearly with Re. This fixes the dimensionless entry length and creates a normalized like axial distance in the entry length. Therefore, the mesh settings and convergence verification do not change significantly. However, although the domain stays the same for increasing or decreasing Re, especially for the nondimensional in-house codes, physically the domain elongates or shortens, and this can be observed in commercial codes visually when nondimensionalization is done as in the present work. This is shown in the results section. In-house codes involve this effect in the discretization since Re appears in the discretized linearized governing equations, in the coefficients [5]. So, in the in-house codes, the aspect ratio of the mesh elements does not change but numerically, the change rates of the fluxes increase per mesh element. In the commercial codes, when the present nondimensionalization scheme is used, the mesh elements elongate for increasing Re in the laminar flow so that the flux rate stays constant. But this may lead to high aspect ratio mesh elements and should be checked and kept under control.

Another primitive or primary variable in the governing equations is the pressure term. This can be nondimensionalized via Equation (8).

$$p' = \frac{p}{\frac{1}{2}\rho u_{\mathrm{inlet}}^2} = \frac{p}{\frac{1}{2}\rho u_0^2} \tag{8}$$

It is a common way to show inlet velocity as u_0 since it is a constant and reference magnitude. Here in Equation (8), one sees that fluid density is involved. The denominator is in fact dynamic pressure. One may use twice the value. It would only change the average final numerical magnitude of the dimensionless pressure distribution while the physical meaning would not change. Again, if the fluid density is entered as unity, remembering that the inlet velocity is also selected as unity, the spatial pressure distribution of the commercial CFD can be used as nondimensional by only dividing the results to 0.5. If the denominator does not contain 0.5 value in the nondimensionalization description, then the pressure distribution can be directly used as nondimensional.

After primary variables, one may need derivative quantities to review. One of them may be wall shear stress and its nondimensionalization can be done via Equation (9).

$$\tau_w' = \frac{\tau_w}{\mu \frac{u_0}{D}} \qquad (9)$$

In Equation (9), τ_w is wall shear stress and μ is the absolute/dynamic viscosity of the fluid. It can be said that variables that are related to diffusion, for instance, viscosity is related to momentum diffusion, are generally used for nondimensionalization of derivative quantities such as wall shear stress and/or turbulent viscosity. Another nondimensionalization option of the wall shear stress is shown in Equation (10), and it is called skin friction coefficient (C_f).

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho u_0^2} \qquad (10)$$

Coefficients are dimensionless as a principal rule. So, skin friction coefficient is a wall shear stress that is nondimensionalized similar to the pressure term while dimensionless wall shear stress is a proportion based on fluid viscosity. Skin friction factor is also analogue to the Darcy friction factor. Another derivative quantity is strain rate magnitude ($\dot{\gamma}$) and its nondimensionalization can be done via Equation (11).

$$\dot{\gamma}' = \dot{\gamma}\frac{D}{u_0} \qquad (11)$$

The final variable to be nondimensionalize is the turbulent viscosity (ν_{tr}) for turbulent flow modeling. Its nondimensionalization can be done by dividing it by kinematic viscosity (ν) of the fluid as in Equation (12).

$$\nu_{\text{tr}}' = \frac{\nu_{\text{tr}}}{\mu/\rho} = \frac{\nu_{\text{tr}}}{\nu} \qquad (12)$$

It is shown that the only independent variables that are used in the nondimensionalization of spatial primitive and derivative parameters are; inlet velocity, pipe diameter, fluid density and dynamic viscosity, and Re. The first four of them also constitute Re itself. Therefore, in commercial CFD, one can both set the desired Re by only arranging those four independent variables, that are; $u_{\text{inlet}} = u_0$, D, ρ, μ. Giving them the value of 1 make nondimensionalization of the results very easy since dimensional values can be directly used nondimensional. However, one can understand that all four independent variables cannot be set to unity at the same time since Re would have also unity value. So, there are five options which are constituted by changing one of them while the remaining are unity or changing all of them. The last option, changing each independent variable at the same time, does not make any sense since we want to set the independent variables to unity as much as possible. In the preset work, the conveniency evaluation of options for changing a single independent variable is done while others remain unity. Especially, creating a CFD domain that has 500 m radius (corresponding to 1,000 m diameter) to have Re=1,000 and axial length 60,000 m, which is very counterintuitive, is tried. One should enable domain creating tools such as Design Modeler in ANSYS to support those lengths by changing its settings. Working with numerous zeros in both ends, i.e., the decimal side and the integer side, needs to be cautious since black box code may involve unnoticeable quantitative restrictions. The author of the present paper recommends remaining in nine digits at most for either side, i.e., the decimal side and the integer side. One can review Canli et al. [7], Canli et al. [8] for assigning dynamic viscosity to different values while remaining independent variables are unity in turbulent flow for making commercial CFD nondimensional as an option. Other options for more complex and heat transfer involving cases can be viewed from Ceviz et al. [9] and Dillon et al. [10]. However, none of them tried characteristic length to arrange the governing nondimensional numbers.

The first domain was created in the Design Modeler of ANSYS WORKBENCH by drawing a 500 m high and 60,000 m wide rectangle after enabling "Large Model Support". Without enabling this setting, one cannot draw models longer than 1,000 m. A thin surface was created using the rectangle sketch. Then, ANSYS MESH was used to divide the radial length (the vertical side of the rectangle) to 50, which is advised to resolve laminar developed velocity profile sufficiently [8, 9]. The axial length was divided by 6,000 to have 3×10^5 unity aspect ratio mesh elements. Created structured mesh was fed to Fluent solver. Two-dimensional axisymmetric incompressible solver was initiated. Viscous-laminar solver was selected. Fluid viscosity and density were set to unity. Inlet velocity was set to unity as inlet boundary condition. Two different outlet boundary conditions were tried. Outflow boundary condition was tried because developed flow was expected at the outlet. However, pressure outlet boundary condition has more advance settings such as reverse flow restriction and target mass flow rate. Eventually, pressure outlet boundary condition performed better. Nevertheless, pressure results are evaluated in terms of the outlet boundary condition options under the discussion title. Wall and axis edges were assigned to their respected boundary conditions. Reference values in Fluent are used to calculate derivatives, such as skin fiction coefficient. Therefore, independent variable values were also inputted into the reference values section. COUPLED scheme was selected for pressure velocity coupling since the nonlinearity of the problem is relatively low. Spatial discretization options were selected as Green-Gause cell based since the mesh is structured with unity aspect ratio square elements, second order pressure discretization and second order upwind discretization. Other change rate reducer options during iterations such as pseudo transient option were not selected since the nonlinearity of the problem is relatively low as explained before. Momentum and pressure relaxation factors were selected as 0.5 while no other under-relaxation or over-relaxation were applied. Only scaled residuals of continuity, *x*-momentum and *r*-momentum were monitored. Therefore, iterations kept continuing until second dramatic decrease was observed in momentum scaled residuals. The scaled residuals are the total residual of all cells for an iteration divided by the initial iteration total residuals and/or the highest value of the first five iterations. Without residual scaling, number of mesh might increase total residual and make monitoring and comparison harder. Totaling the residuals avoids focusing on a single cell that might restrict one to overview the whole domain. Other cases different from the present article may need monitoring some physical phenomena to decide convergence. However, the present work relies on well known hydrodynamic development for case validation, and therefore, no specific quantities were monitored for finalizing the iterations.

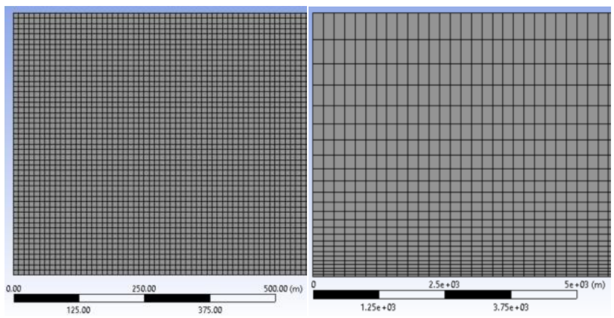
The validation of the data was done by comparing the outlet velocity profile to the hydrodynamically developed velocity profile of laminar flow that obeys the expression in Equation (13) (Hagen–Poiseuille velocity profile). Also, the friction factor or the skin friction coefficient should obey 16/Re expression (Fanning friction factor f_F that is one quarter of Darcy friction factor f).

$$u(r) = u_{max}\left(1-\left(\frac{r}{R}\right)^2\right) \tag{13}$$

Below steps can be regarded as an algorithm of application of the commercial CFD software manipulation for nondimensional results.

1. Review the governing nondimensional numbers of the investigated phenomena (for instance, Re for Newtonian forced flow; Grashof, Pr, and Ra for natural convection heat transfer, etc.)
2. Review the nondimensional parameters to be used in analyzing the problem (for instance, axial velocity component, static pressure distribution, heat transfer coefficient, etc.)
3. Determine the variable levels for the nondimensional governing numbers (for instance, Re=1×10^0, 10^1,10^2, …, 10^n)
4. Try to construct the nondimensional governing numbers to their target levels by assigning independent variables to 1 (unity) as much as possible while prioritizing nondimensional parameters (for instance, take density as 1 for dimensionless pressure and dynamic viscosity as 1 for dimensionless turbulent viscosity)
5. Assign the determined independent variable values in the commercial CFD by using boundary conditions, fluid and solid properties, and drawing the domain.
6. Set up the remaining solver settings as in a dimensional case. Run the solver.
7. After convergence, use the dimensional results directly as nondimensional if their denominators were assigned 1 (unity) value.

Turbulent flow case necessitates a different handling since the hydrodynamic entry length in case of turbulence is much shorter in proportion to the pipe diameter than the laminar counterpart. The axial velocity overshoot at pipe axis, turbulent velocity profile, and the maximum value of the axial velocity are compared to the data in Canli et al. [7] for validation. The pipe diameter was set to 1×10^4 m for 1×10^4 Re, by drawing the vertical edge of the rectangle in the design modeler as 5,000 m. Therefore, horizontal edge of the domain was drawn as 2×10^5 m. The meshing of the domain for turbulent flow was not as straightforward as the laminar case. It is more like an iterative process based on turbulence modeling. In the present work, two equation RANS k-ε turbulence model that involves the turbulent viscosity concept was used. Standard wall function was selected for the near wall treatment. However, standard wall function necessitates a dimensionless wall distance (y^+) value between 15-300. On the other hand, y^+ cannot be predetermined before the simulation. Accordingly, CFD simulation was conducted for a mesh, then y^+ was checked and then mesh was revised to have the desired y^+ distribution along the wall. At the end of the trial-and-error process, vertical edge of the domain was divided to 25 and horizontal edge was divided to 1,000, while mesh structuring enabled rectangular mesh elements. However, to obtain desired y^+ at the neighboring mesh element to the wall, mesh elements were elongated at wall and shortened at the pipe axis. The visual comparison of laminar and turbulent flow case meshes are shown in Figure 2. While laminar flow mesh elements have an aspect ratio of unity for Re=1,000, the case of turbulent flow mesh has different aspect ratio elements from wall towards axis.



a b

Figure 2. Mesh comparison of cases; a. laminar (Re=1,000); b. turbulent (Re=10,000)

The remaining CFD solver settings are similar to the laminar one, for the turbulent one.

The applicability of the presented methodology to different physics, such as heat transfer, temperature dependent thermophysical properties, non-Newtonian flows, phenomena that are governed by multiple dimensionless groups are yet to be explored. The present approach relies on constant values of the thermophysical properties and constant domain sizes. Nevertheless, since nondimensionalization relies on constant reference values, the author thinks that there is a potential to be investigated.

This concludes with the methodology section. In the following results and discussion section, nondimensionalization options as well as the implications of nondimensional in-house codes are discussed in the light of the CFD simulation results of the case study.

3 Results and discussion

It is logical to lay out all the spatial distribution plots for the dimensionless dependent variables of the base case, which is the case for Re=1,000 and the only independent variable different from unity is the pipe diameter as 1,000 m. The plots are shown in Figure 3. The nondimensionalization of the axial length not only by the pipe diameter but also by the Re creates a normalized like axial length, at least for the hydrodynamic entry length for laminar flow (Re=1,000 for Figure 3). Axial distance data should be divided by the pipe diameter and Re as a post process. Since all spatial distributions include length information, this means all the plots need to be post processes at least for the lengths since Figure 3 shows the length option, i.e., the pipe diameter equals to Re for the nondimensionalization of the commercial CFD results. On the other hand, if pipe diameter were taken as 1 m, there would not be any post processing, or there would be just dividing by Re for the normalization. Axial velocity component can be directly used without post processing. If solver graphics such as contour plots are desired to be used directly, the color legend would have a unit label, but an explanation to the reader stating that the results should be viewed as nondimensional would solve the problem. The axial velocity at pipe axis plot shows that dimensionless axial velocity component approaches two times of the inlet velocity through the hydrodynamic entry length. Also, it should be noted that the axial velocity plot at the pipe axis is actually not a line but points. However, the 6,000 points creates a solid line like plot. The second plot in Figure 3 validates the simulation since the simulation results agree with the analytical solution, the axial velocity profile in radial direction. In case of in-house codes, the results are used directly without any post processing for nondimensionalization since their formulation includes the nondimensionalization intrinsically. In the pressure plot, the three trendlines in fact indicate the same physics. The p'_1 trendline is the nondimensionalization that is described by Equation (8) while the outlet boundary condition is set to the pressure outlet. Pressure outlet dictates a pressure value at the outlet and therefore, the pressure potential that drives the fluid and causes the flow to be realized build upon the outlet pressure value. When the nondimensionalization of the pressure does not involve ½ denominator, then the user can directly use the commercial CFD output that is p'_2 trendline. However, if outflow boundary condition is selected, this means that the pressure at the outlet is not specified by the user and therefore the code just calculates the necessary pressure difference at each mesh element to create the velocity vector and add them up towards inlet to approach zero value, which is illustrated by p'_3. The main idea of the outflow boundary condition is that the flow is fully developed at and before the outlet so that the change rates are zero. One can view the p'_3 and decide about the pressure change rate and calculate the pressure difference between inlet and outlet. If the absolute pressure values are needed when there is only p'_3 data exists, then the user should add or subtract the pressure difference from the known inlet or outlet absolute pressure.

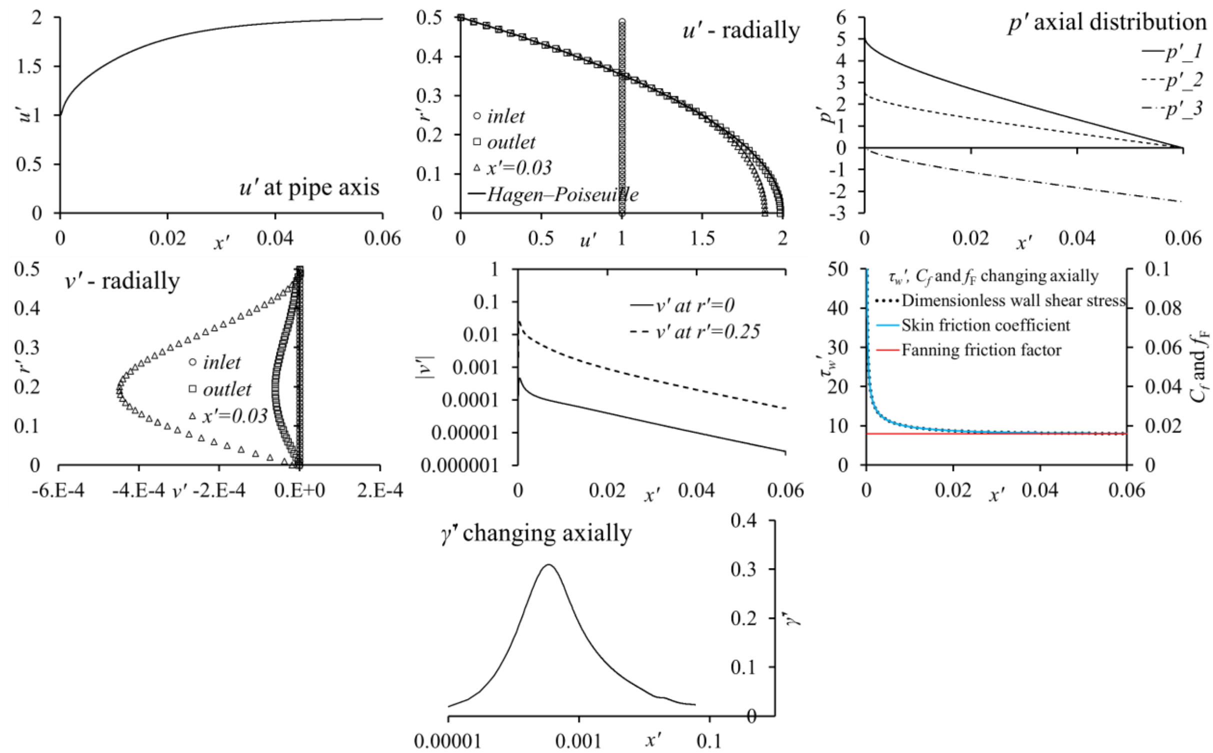


Figure 3. Spatial distribution plots of the primitive primary dependent dimensionless variables for Re=1,000 by the *D*=1,000 m and unity values of remaining independent variables option.

In the pressure plot, all three plot trends show that the pressure change rate is higher at the initial parts of the pipe and then the change rate becomes almost constant. This is another validation that the simulation catches the related physical phenomena since the hydrodynamically developed laminar flow has constant pressure change rate. Radial velocity components can be directly used as nondimensional since the inlet velocity was selected as unity in this nondimensionalization option. At the pipe entrance, boundary layers start to form, and this creates radial velocity component as an attempt by the flow to satisfy the mass conservation. Towards the outlet, radial velocity component diminishes. The flow can be deemed as two dimensional when there are two velocity components and can be regarded as one dimensional when there is only a single velocity component. It should be reminded here that pressure outlet boundary condition somehow performed better than outflow boundary condition since outflow boundary condition led to unrealistic radial velocity component values at the outlet. The more advanced restriction options of the pressure outlet boundary condition such as target mass flow rate and restricting the reverse flow should create this type of better performance. The negative value of the radial velocity component just shows the direction of the vector. The negative direction of the radial velocity component is towards the pipe axis while the positive direction is towards the wall. Here, too small quantities are visible for radial velocity profile. If radial velocity value accuracy is important for the sake of the investigation, then one should check the rounding decimal number settings and/or change the constants of the nondimensionalization scheme so that the nondimensional radial velocity values get bigger quantities. Dimensionless wall shear stress and skin friction coefficient shows how two different nondimensionalization denominators create different quantities yet indicate the same physics. They both show the higher shear at the initial parts of the pipe, and as the boundary layers grow bigger and approach the pipe axis, the shear stress approaches to a constant value. As explained in the methodology section, skin friction coefficient for hydrodynamically developed flow should approach the Fanning friction factor, and it is shown in the respected Figure 3 plot as a validation. The nondimensionalization of the shear stress is done by multiplying the diameter value with the commercial CFD output as a post process, remembering Equation (9) and unity variables. The skin friction coefficient from the software can be directly used. Finally, the fluid element strain has its maximum value at the initial parts of the pipe and diminishes towards the outlet. The strain results from the commercial CFD should be multiplied with the pipe diameter to have the nondimensional strain, recalling from the Equation (11).

The other options for nondimensionalization, which are non-unity values for inlet velocity or fluid density or fluid dynamic viscosity while the pipe diameter would be set to unity would not change the post processing of the results for nondimensional plots too much. In case of those scenarios, the lengths would not need any post processing while velocities should be processed for inlet velocity value other than unity, pressure should be processed for fluid density value other than unity, and wall shear stress should be processed for fluid dynamic viscosity value other than unity. This may imply that there is less post process involved for the options other than the option that makes the nondimensionalization with pipe diameter. However, increasing the Re of the investigated case has different implications to be considered. This is demonstrated in Figure 4 for Re=500 and 1,500 via plots of three different commercial CFD nondimensionalization options. The first plot in Figure 4 compares development of the axial velocity component at the pipe axis for Re=500, 1,000, and 2,000 when the pipe diameter is kept constant at 1,000 m and the inlet velocity, viscosity, and density are changed respectively to arrange the Re value. The second plot in Figure 4 shows the same cases but for different pipe diameters, namely 500 m, 1,000 m, and 2,000 m while the remaining independent variables are unity. The main difference between the first plot and the second plot is that the domain and mesh settings are kept constant for the first plot while second plot necessitates redrawing of the domain. The mesh settings do not change for the second plot, but the aspect ratio of the mesh elements do change since pipe length shortens or elongates for constant mesh element number. The first plot shows that the development of axial velocity at pipe axis changes spatially. This implies that constant domain and mesh settings as well as mesh element dimensions and number may be sensible to the changes and mesh independency may need to be examined for each case. Also, since the domain length was determined based on Re=1,000, Re=2,000 cannot show entry length. Only additional post processing occurs for the inlet velocity nondimensionalization since the plot involves only velocity information. Pressure distribution would need additional post processing for the density variable other than unity and wall shear stress would need additional post processing for the viscosity variable other than unity. The trendlines coincide with each other for the second plot since the nondimensionalization solely done via pipe diameter. Also, the mesh elements become shorter or longer in axial direction based on the Re value, keeping the axial velocity change rate per mesh element and therefore, it is evaluated that additional mesh dependency examination is not necessary. As a conclusion, one may argue that significant changes in the governing nondimensional number may create spatial mesh dependent results via nondimensionalization parameters other than the characteristic length variable. On the other hand, mesh aspect ratio may be checked for too much elongated or shortened mesh elements when there is a big difference in the nondimensional governing number for the nondimensionalization by the characteristic length. In addition, it can be asserted that nondimensionalization is more convenient by thermophysical properties or velocity scale that constitutes the nondimensional governing number when the differences in the nondimensional governing number, Re in the present case, is small, in such a way that the mesh dependency is not expected to change significantly.

The assessment of change in Re or the nondimensional governing number can be done based on order of magnitudes. For instance, Re=100 is one order of magnitude smaller than Re=1,000, hence a new mesh may be generated to mesh independent results when independent variables other than the length scale are changed for nondimensionalization.

After reviewing Figure 4, plot 1 (Figure 4a) and plot 2 (Figure 4b) are re-evaluated in respect of nondimensional in-house codes and the nondimensionalization of the commercial CFD results. When thermophysical properties are modified to have the desired Re in the current commercial CFD nondimensionalization, the axial normalized length becomes longer for Re=500 and becomes shorter for Re=2,000, as a way of indicating the sufficiency of the pipe length for hydrodynamic development. We know that the domain dimensions and mesh setup are fixed for Figure 4a so one can assert that change rate is higher for Re=2,000 per mesh element since the development continues, than the Re=500 case since the axial velocity does not change for almost last half of the pipe. Interpreting this leads us to the conclusion that arranging Re and nondimensionalization of the commercial code via thermophysical properties or the velocity scale while length scale is unity may create spatial mesh dependency changes. Regarding in-house codes, this may not be observed since the whole governing equations are nondimensional including the lengths that may be also divided by Re. Re appears in coefficients of linearized governing equations that are obtained with spatial discretization based on numerical approach. In brief, in-house nondimensional codes create plots that are more like Figure 4b instead of Figure 4a.

Another thing about plot 1 (Figure 4a) is that one should be very careful about the pressure outlet boundary condition if the target mass flow rate option is selected. Since the pipe diameter is 1,000 m, the target mass flow rate calculation is sensitive to the decimals of pi number. In the present case, the author used the calculator pi function instead of three or five decimal approximate pi values. This ensured convergence to the analytical solution. Otherwise, the target mass flow rate may be smaller than it should be leading to lower developed axial velocity value. Nondimensionalization of the commercial CFD via thermophysical properties is attractive because a single drawing and a single mesh setup can be used for several values of the governing nondimensional number. However, one should be careful of spatial mesh dependency. In plot 2 (Figure 4b), the nondimensionalization is done by pipe diameter which means each CFD simulation necessitates redrawing (re-dimensioning actually) the domain. This is the main hardship of length scale nondimensionalization option of the commercial CFD. Then, the mesh setting does not change but mesh element dimensions and aspect ratio change because of the shortened or elongated domain.

The rest of the Fluent solver settings do not change. When the domain length is drawn sufficiently long enough to cover the entry length, the nondimensional normalized axial length and axial velocity profiles at pipe axis coincide onto each other as they are shown in Figure 4b. The changes of mesh elements for the cases in Figure 4b are shown in Figure 5. As the dimensional domain elongates, the mesh elements also elongate. In case of in-house nondimensional codes, the mesh element sizes also stay the same, but the mesh element change in Figure 5 is realized via the spatial discretization scheme of the numerical method. As Re grows bigger the rate of change increases in the nondimensional in-house code. The numerical remedies create a mesh elongation like effect. In Fluent, mesh elements that have an aspect ratio of about 40 and above create numerical problems. In a similar way, too high rate of change in nondimensional in-house codes may create numerical errors so that the user may create a new mesh setting. Therefore, in nondimensionalization commercial CFD by length scale, the user should monitor the aspect ratio of the mesh elements.

The final results to be presented are the spatial distribution of dimensionless variables for the turbulent flow in order to show that pipe diameter as high as 1×10^4 m and the related pipe length can be constructed and solved. They are given in Figure 6. In the meantime, one may view turbulent nondimensional pipe entry length plots from commercial nondimensional CFD that relies on nondimensionalization via thermophysical properties from Canli et al. [7].

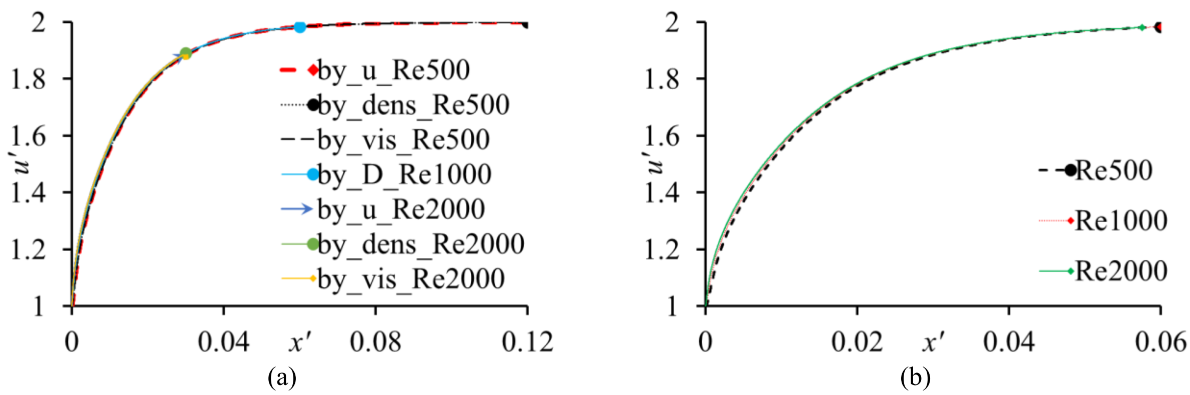


Figure 4. Axial velocity component profile at pipe axis for Re=500, 1,000, and 2,000 (a) nondimensionalization is done via inlet velocity or fluid density or viscosity while pipe diameter is constant (b) pipe diameter changes to set the Re values while remaining independent variables are unity

In Figure 6, the nondimensionalized and normalized axial length or the hydrodynamic entry length indicates that the turbulent entry length is shorter than the laminar one. The y^+ plot verifies the utilization of standard wall functions for the turbulence model where near wall non-isotropic turbulence is calculated via analytical method. The value of y^+ is already nondimensional, and therefore there is no further processing for that value, and Fluent results can be directly used.

Axial velocity profiles in radial direction validates the simulation in comparison with Canli et al. [7]. The dimensionless pressure becomes linear almost throughout the domain. Radial velocity component shows positive and negative signs due to the velocity overshoot phenomenon of the axial velocity. The main reason for the velocity overshoot phenomenon of the turbulent pipe flows hydrodynamic entry length is the initial turbulence intensity assumption, which is assumed 5% in the present work. This assumed value first decreases as the initial parts of the pipe have thin boundary layers that do not merge at the pipe axis yet. As the boundary layers grow bigger towards the pipe axis, turbulence is generated inside the boundary layers. When the boundary layers merge at the pipe axis, generated turbulence also increases the turbulence intensity and axial velocity at the pipe axis reduces slowly. This velocity overshoot phenomenon changes the spatial distribution of the primary variables compared to the laminar case. In general, the proposed methodology for making commercial CFD nondimensional works for the turbulent flow even with the length scale as high as 10,000 m. Below is a tabulated comparison (Table 1) of the introduced nondimensionalization, in-house code nondimensionalization, and conventional dimensional simulation.

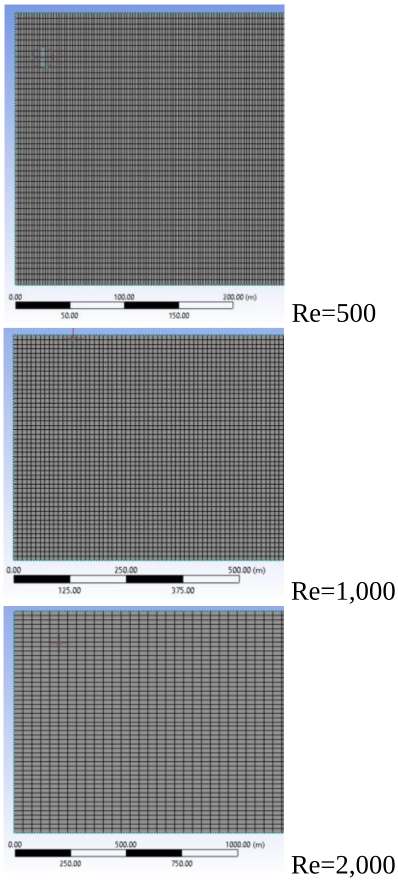


Figure 5. Mesh element aspect ratio changes as Re changes for nondimensionalization by pipe diameter

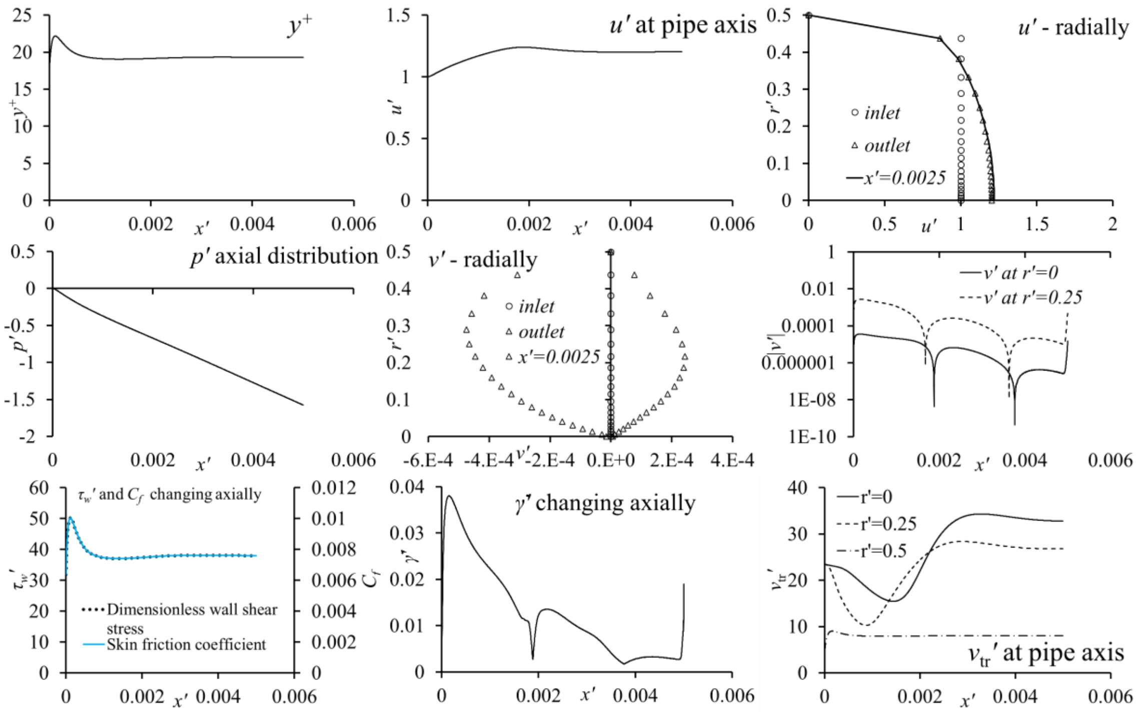


Figure 6. Spatial distribution plots of the primitive primary dependent dimensionless variables for Re=10,000 (turbulent flow) by the D=10,000 m and unity values of remaining independent variables option.

Table 1. Comparison of introduced nondimensionalization, in-house code nondimensionalization, and conventional dimensional simulation.)

Aspect of comparison	Commercial CFD manipulation	Nondimensional in-house code	Commercial CFD with dimensions
Generalization	Moderate	High	Low
Number of parameters and variables	Moderate	Low	High
Preparation hardness	Low	High	Low
Utilization ease	High	Low	High
Reliability	Moderate	High	Moderate
Robustness	Moderate	Low	High

4 Conclusion

Nondimensional variables and parameters enable one to produce generalized yet clearer evaluations and conclusions, replacing and representing the numerous dimensional variables and numbers. The rationale of the present paper is to describe a way of setting up CFD solver so that the post processing necessitates less work. The investigated case and the obtained results proves this added value. Present work reviews a practical methodology to make commercial CFD analyses nondimensional by carefully setting up the whole CFD case starting from the domain design, continuing with meshing, and selecting specific predetermined quantities for independent variables that constitute the governing nondimensional number or numbers of the investigated physics. As a counterintuitive application, the CFD domain characteristic length is changed between 500 to 1×10^4 m to demonstrate the outcomes of the methodology while specific recommendations and practical warnings about settings are asserted. Nondimensional in-house coding is also evaluated in terms of its implications while reviewing the results of the present work. Following remarks can be made about the major outcomes of the article.

- Using length scale as the nondimensionalization independent parameter to set the governing nondimensional number, Re in the present work, while all other independent input variables are unity, and normalization of the distances in spatial plots not only by the length scale but also by the Re create a convenient way of nondimensionalization commercial CFD. Same mesh setting can be used without generating new meshes while mesh element size and aspect ratio changes with changing Re. The only hardship of this approach is drawing the domain for each Re level.
- Small changes in the governing nondimensional number, for instance in the value of Re, one of the thermophysical properties or the inlet velocity can have a value different than unity. In such cases, one should be aware that the solution may become mesh dependent and the spatial distributions may change in distance.
- In terms of reliability of the introduced method, one should be careful about the rounding errors. Since nondimensionalization sometimes grants normalization, i.e., the parameter changes between 0 and 1, too small numbers with significant number of decimals may appear. In case of rounding errors, one should review the nondimensional results and then make necessary arrangement in the numerical solver if too small or too big numbers appear. Another solution may be rearranging the values of the independent variables that base the present nondimensionalization scheme.
- More complex work with several nondimensional groups as physics governing numbers can be set as future goal for the conveniency and feasibility of the reviewed methodology.

The limits of the present method for making commercial CFD software setup nondimensional as a generalization are yet to be explored. In other words, heat transfer involving cases, temperature dependent thermophysical properties, non-Newtonian flows should be tried for the applicability of the method. Those cases may involve several different nondimensional numbers and independent variables may become dependent, creating hardship for applying the methodology in the present work. Nevertheless, this creates a potential for interesting new scientific articles.

Acknowledgement

The author acknowledges the time and efforts of the reviewers to improve the quality of the paper.

Conflict of Interests

The author declares that there is no conflict of interests.

Similarity Rate (iThenticate): %9

References

[1] A. H. Abdulkarim, E. Canli, and A. Ates. CFD Case Study on a Nozzle Flow: Literature Review, Theoretical Framework, Tools and Educational Aspects. International Conference on Engineering Technologies (ICENTE'18), pp. 486-489, Konya, Türkiye, 2018.

[2] A. H. Abdulkarim, A. A. Ghaidan, A. Chelang, and E. Canli. Comprehension, utilization and interpretation problems of engineers and engineering students, associated with commercially available CFD packages: a case study. International Conference On Engineering, Natural And Applied Science 2021 (ICENAS'21), pp. 77-84, Osmaniye, Türkiye, 2021.

[3] E. Canli, A. Ates, and S. Bilir, Comparison of turbulence models and CFD solution options for a plain pipe. EPJ Web of Conferences, 180, 02013, 2018. https://doi.org/10.1051/epjconf/201818002013

[4] E. Canli, A. Ates, and S. Bilir, Conjugate heat transfer for turbulent flow in a thick walled plain pipe. EPJ Web of Conferences, 180, 02014, 2018. https://doi.org/10.1051/epjconf/201818002014

[5] E. Canli, Numerical solution of transient conjugated heat transfer in thick walled pipes with turbulent flow. Ph.D., Selçuk University, Konya, Türkiye, 2020.

[6] E. Canli, A. Ates, and Ş. Bilir, Derivation of dimensionless governing equations for axisymmetric incompressible turbulent flow heat transfer based on standard k-ϵ model. Afyon Kocatepe Üniversitesi Fen Ve Mühendislik Bilimleri Dergisi, 20(6), 1096-1111, 2020. https://doi.org/10.35414/akufemubid.821009
[7] E. Canli, A. Ates, and Ş. Bilir, Developing turbulent flow in pipes and analysis of entrance region. Academic Platform-Journal of Engineering and Science, 9(2), 332-353, 2021. https://doi.org/10.21541/apjes.818717
[8] E. Canli, A. H. Altun, and A. Ates. Hydrodynamic and thermal simultaneous development in pipes for all the three thermal boundary condition types using CFD. 4th International Conference on Life and Engineering Sciences ICOLES 2021, pp. 185-205, İstanbul, Türkiye, 2021.
[9] A. Ceviz, E. Canli, A. Ates, and Ş. Bilir. Numerical scheme for dimensionless natural convection analysis of vertical pipe. 23rd Congress of Thermal Sciences and Technology with International Participation (ULIBTK 2021), pp. 1283-1293, Gaziantep, Türkiye, 2021.
[10] H. E. Dillon, A. Emery, R. Cochran, and A. Mescher. Dimensionless versus dimensional analysis in CFD and heat transfer. Excerpt from Proceedings of the COMSOL Conference 2010 Boston, pp. 1-9, Boston, USA, 2010.
[11] A. R. Gamboa, C. J. Morris, and F. K. Forster, Improvements in fixed-valve micropump performance through shape optimization of valves. Journal of Fluids Engineering, 127(2), 339-346, 2005. https://doi.org/10.1115/1.1891151
[12] N. Cheimarios, E. Koronaki, and A. G. Boudouvis, Enabling a commercial computational fluid dynamics code to perform certain nonlinear analysis tasks. Computers & chemical engineering, 35(12), 2632-2645, 2011. https://doi.org/10.1016/j.compchemeng.2011.03.008
[13] M. A. A. Morshed, N. A. Mini, M. A. Hakim, and M. N. Hasan, Mixed Convective Heat Transfer Characteristic in Vented Cavity Under Dynamic Flow Modulation by CFD and Neural Network Model Approaches. Heat Transfer, 54(5), 3072-3087, 2025. https://doi.org/10.1002/htj.23349
[14] A. Mason-Jones, et al., Non-dimensional scaling of tidal stream turbines. Energy, 44(1), 820-829, 2012. https://doi.org/10.1016/j.energy.2012.05.010