International Journal of Multidisciplinary Research and Growth Evaluation.

Result Validation through code Verification and Mesh Sensitivity Test

Alabodite Meipre George [1*], Biralatei Fawei [2]
[1] Department of Mathematics, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria
[2] Department of Computer Science, Niger Delta University, Wilberforce Island, Bayelsa State, Nigeria

* Corresponding Author: **Alabodite Meipre George**

Article Info

ISSN (online): 2582-7138
Volume: 05
Issue: 06
November-December 2024
Received: 04-10-2024
Accepted: 09-11-2024
Page No: 901-909

Abstract
An investigation on the validation of the results from George and Kay, (2017) and the reproduction of results from Srynarayana *et al*. (2008) was conducted with the aid of COMSOL MultiPhysics through Code Verification and Mesh Sensitivity Test for code validation and correctness of the results. Mesh dependence test was conducted to examined temperature distribution and velocity field at the point (0,2) at the core of the plume and plotted as a function of time for different mesh sizes. The results confirmed that both velocity field and temperature distribution at the specified point converged as mesh size decreases. For code verification the results showed that values from the individual terms in the continuity equation vary slightly with a maximum variation of $\approx \pm 0.010$ when taken their sum. Interpolated values showed that the Y-Momentum equation were resolved properly and positive on both sides of the equation: though, with a maximum variation of $\approx \pm 0.003$. Fluid flow in the upward direction is very key for a rising plume because water at the plume's core at that point is expected to be undiluted and positively buoyant. Results from the X-Momentum equation showed both positive and negative values with a maximum variation of $\approx \pm 0.011$. The values from the temperature equation showed both negative and positive values but approximately zero in all. In reproducing the results from Srinarayana *et al*. (2008) with COMSOL MultiPhysics. The overall behaviour showed that results appear similar. A flapping behaviour was also observed which seems to have occurred earlier within the simulation time. However, the Comsol Multiphysics used is reliable as this could reproduce results from other CFD code (Gerris) while the validated results from George and Kay, (2016) are also reliable and independent of mesh size ≤ 0.05.

DOI: **https://doi.org/10.54660/.IJMRGE.2024.5.6.901-909**

Keywords: Code Verification, Plume, Validation, Comsol Multiphysics

1. Introduction

It is becoming interesting over the decades with numerical simulations, as software developers have stepped up to the challenges in computational fluid dynamics (CFD) (Gupta *et al*. 2020) [6]. This in turn has encour- aged researchers to carryout complex investigations that has to do with real life situations using some of these mathematical tools, Navier-Stokes Equation and Reynolds-averaged Navier-Stokes Equation (RANS equations) for a 2-dimensional and 3-dimensional time-depended flows respectively (Jauregui and Silva, 2011; Bistafa, 2018; Argyropoulos and Markatos, 2015) [13, 7, 9]. Other sophisticated approaches such as Large- eddy simulation, Direct numerical simulation, etc., have also been considered which are relevant to many field of studies and have also aided reduced computational cost but then increase the computational speed (Engelmann, 2023; Argyropoulos and Markatos, 2015; Lintermann, 2020) [9, 14]. However, before the use of any CFD tool, there has been a means with which the ability of these codes are being verified for correctness. This we can achieve by reducing or eliminating the identified errors through some exactness measures, the use of some procedures to establish mesh or grid convergence, etc., during the developmental stage (Jauregui and Silva, 2011; Altaie *et al*. 2020; Benay *et al*. 2003; Lintermann, 2020) [13, 14, 2, 1].

After the successful design and development of these CFD tools (codes) that solves the specified equation set, then the concern becomes not only to secure the solution of the given problem but also verifying that the said solution is valid. Thus, it is very important to sometimes cross-check if the quantified difference between the numerical solution and reality is minimal or not. But then, it is a known fact that numerical methods are mostly employed in cases where the proposed mathematical models (especially coupled system of equation that are non-linear) are very difficult to solve analytically. Thus, the method in such a scenario is to make use of these numerical means so as to produce results that are as close as possible to depict the real world scenario in numbers, and in turn displayed in the form of pictures, graphs (Gupta *et al*. 2020; Bistafa, 2018) [6, 7]. For us to be able to know if the solution from the numerical scheme is consistent or corresponds to the real world scenario; it then implies that the proposed model must capture all the possible areas that has to do with the real world scenario, and that the numerical scheme must also solve the proposed equations correctly and exactly. This seems to be the beginning of ensuring that we move towards obtaining a good result (Gupta *et al*. 2020; Lintermann, 2020) [6, 14]. After being successful in all these (by solving the said equations), the next is to see if approximated results agree or are very similar to real life scenarios being simulated for by using a method of validation (Gupta *et al*. 2020) [6]. There are different methods with which numerical solutions can be validated and some of these include: validation by using experimental results, analytical solutions, other numerical solutions, convergence method, and so on (Gupta *et al*. 2020) [6]. Note that all the possible means of validation follows a common pattern, which is by comparison (i.e., comparison of a pattern with previous results or a reference model with the model used in the current investigation) (Gupta *et al*. 2020) [6].

Another area of importance to be considered in terms of result exactness is meshing, and this has also become a special area of research and have also gained attention because of its peculiarity. It is known that before mathematical equations are solved or simulated using a CFD tool, there is the need for the domain of computation to be sub-divided into smaller separate mesh cells or grids; where the formulated equations are initiated and solved (Vanova and Kvocak, 2021; Patil, and Jeyakarthikeyan, 2018; Linter- mann, 2020) [5, 10, 14]. The results obtained during this process are computed by employing the mesh as a base, where this procedure may derive spatial discretization errors. And if there are any, conducting a mesh sensitivity test will enable us to better fathom the quantity of spatial discretization errors present. This in turn will enable us to know the level of effect the mesh may have on the solution. However, it is also known that as mesh size decreases to zero, so the spatial discretization error also approaches zero, and this in turn leads to a reliable solution. Thus, it is evident that getting quality mesh is crucial which a link to achieving the desired reliable solutions is. (Patil, and Jeyakarthikeyan, 2018) [10]. Note that mesh sensitivity or independence test requires carrying out the same simulation or computation using mesh with different sizes in order to know how the solution varies with the various meshes (Jauregui and Silva, 2011; Patil, and Jeyakarthikeyan, 2018; Vanova and Kvocak, 2021) [13, 5, 10].

We have also come to notice that because of the importance of meshing, most CFD developers have made provision for an in-built meshing system or mesh generating system in their commercial codes or tools. This might also be to the fact that it is usually a very difficult task to generate mesh especially for beginners (ie., making the right choice on the type of mesh to be used and their arrangement, the pattern in which nodes are connected in the domain of computation, etc) (Patil, and Jeyakarthikeyan, 2018) [10]. Thus, this present work is aimed at validating some results by George & Kay in their work entitled "Warm discharges in cold fresh water: 2. Numerical simulation of laminar line plumes", with the aid of a CFD tool (COMSOL MultiPhysics) for a given set of fluid flow equations through Code Verification and Mesh Sensitivity Test. And as well attempting to reproduce some results by (Srynarayana *et al*. 2008) in their work entitled "Height and stability of laminar plane fountains in a homogeneous fluid" with the same COMSOL MultiPhysics for code validation and correctness of the results. Particularly those in Fig. 3 and Fig. 5 with Froude number Fr = 2&8, Prandtl number Pr = 7, Reynolds number Re = 100 with source temperature $\varphi = -1$ within the time range $10 \leq \tau \leq 240$.

2. Methodology

Real life problems are mainly space and time-dependent, and as such they are mostly described mathemat- ically in terms of partial differential equations. In fact, seeking an analytical solution of these equations is always a difficult task for a significant number of geometries, which may be as a result of the coupled and complex nature of the equations that describes real scenario (fluid flow and heat transfer) as the case may be. Instead, we rather make use of an approximation method that are built on different types of descritiza- tion, which in turn approximates these partial differential equations with numerical model equations and are solved using CFD approach. Here in this study, we have considered COMSOL Multiphysics software as the simulation package. This CFD tool is a commercial multi-purpose software that uses higher numerical methods for modelling and simulation of real life problems. Though, it primarily uses the Finite Element Method (FEM) to solve single and multiphase simulations. As a matter of fact, it is also known that the Navier Stokes equation forms the foundation for most of the CFD tools that model problems together with other equations such as turbulent model equations that takes care of other aspect of real life problems. But then, it is worth noting that the finite element discretisation method in COMSOL Multiphysics is the Galerkin's method. Meanwhile, the Backward Differentiation Formula is the time stepping scheme used by the time dependent solver for fluid flow (more important details can also be found in the COMSOL Multiphysics Cyclopedia). With COMSOL, we can easily choose from the physics interface the type of fluid flow that is suitable for the desired study. In the investigation carried-out these authors opted for a Single-Phase Flow; where the CFD Model allows the computation of various variations of Navier-Stokes equations to model flows in all velocity regimes such as: modelling of laminar flows, turbulent flow, etc. Though, the case of a turbulent flow is beyond the scope of what we intend to investigate. From the physics interface, we can also choose Non-isothermal Flow; where thermally induced buoyancy forces are studied for laminar flow cases when coupled with heat transfer. This seemed suitable based on the coupled nature of the equations to be solved and this will capture the real dynamics of the problem as temperature changes.

Once FEM is to be used, it is a known fact that the correctness of the results is also linked to the mesh size used. As we have highlighted earlier, when the mesh size decreases towards zero, the more our solution tends to accuracy. COMSOL has an in-built mesh generating system. Thus, from the pre-defined meshing options, we can choose a suitable mesh from the Physics-controlled meshes for our work. But then, there is a provision also for User-Controlled meshing, where we have opted for 0.05 mesh size for the entire domain and the element type used is triangular and it is of the first order. One of the main aims while carrying out simulations, is to try as much as possible to reduce the error between the exact and the numerical solution. Even if the exact solution may not be known, it is important to ensure that the error if it exist is below some acceptable tolerance level. Now, in the absence of the exact solution, we may estimate the relative error in the approximated solution. It is also worth stating that relative tolerance is the convergence criteria value in COMSOL, and this they have set to be 10−4 in their work. COMSOL Multiphysics automatically uses stabilisation methods to prevent spurious oscillations, knowing that seeking a numerical solution for convection dominated transport problems, sometimes leads to some numerical uncertainty, such as spurious oscillations in the solution. Thus, we have also applied stabilisation method to prevent such unwanted instabilities (consider the following COMSOL Multiphyscs (COMSOL Multiphysics Cyclopedia and COMSOL Multiphysics Reference Manual) for more details).

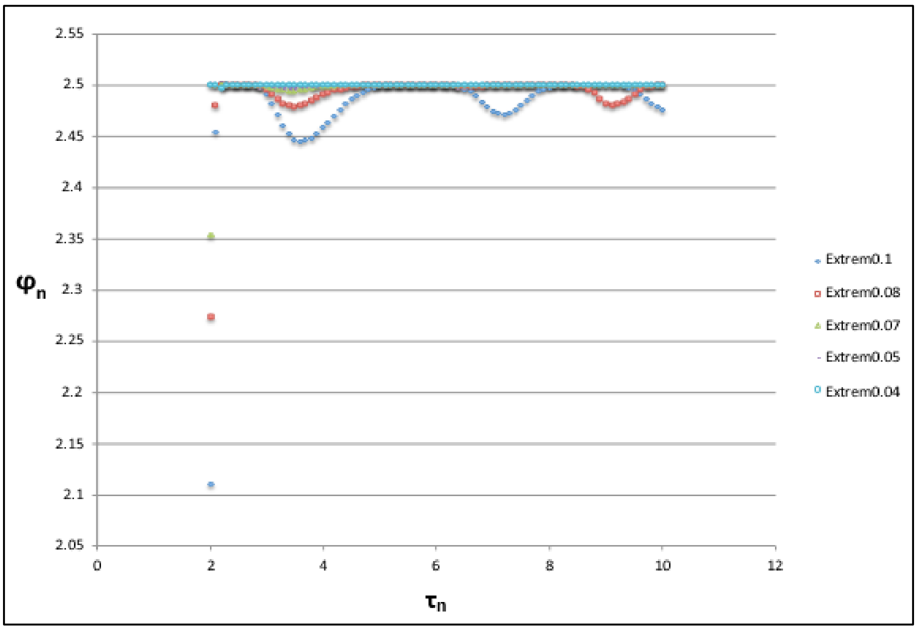

Fig 1: Temperature profile at the point (0, 2) for Re = 50, Fr = 1 and Pr = 7 and plotted as a function of time for extremely fine mesh 0.1, 0.08, 0.07, 0.05 and 0.04 respectively.

2.1. Mesh Sensitivity Test and Results

As earlier stated, the exactness of any numerical solution is also tied to the mesh size used. That is, the smaller the mesh size the more accurate the result becomes. Thus, we have taught it wise to carryout a mesh dependence test so as to ensure correctness of simulation. In order to ensure this, we have examined both the temperature distribution and velocity field at the point (0,2) at the centre line of the plume for those results by (George & Kay, 2017) at $\tau = 5$ for $Re = 50$, $Fr = 1$, $Pr = 7$ and plotted as a function of time for different mesh sizes (0.1, 0.08, 0.07, 0.05 *and* 0.04) as shown in Figure 1 and Figure 2 here. The results in Figure 1 and Figure 2 shows that both velocity field and temperature distribution at that point (0, 2) converges as mesh size decreases. For that of the temperature distribution (Figure 1), the converging behaviour is not very obvious as compared to that of the velocity field where the solution converges grad- ually to a single solution as mesh size gets smaller and smaller. This might be to the fact that the mixture requires just little mixing before attaining the same temperature as that of the ambient fluid. Conclusively, we can say at this point that the solutions are meant to be independent of the mesh used, if the mesh size is ≤ 0.05. Furthermore, with this same mesh size (0.05) using COMSOL, we have attempted to replicate some of the results by (Srinarayana *et al*. 2008) [4], especially those in Figure 3 and Figure 5 as shown in their work with Froude number $Fr = 2$ & 8, Prandtl number $Pr = 7$, Reynolds number $Re = 100$ with source temperature $\varphi = -1$ within the time range $10 \leq \tau \leq 240$ with the same set of equations as given in the work by (Srinarayana *et al*. 2008) [4] and this is shown here in Figure 3 and Figure 4. The overall behaviour of the results as recorded by (Srinarayana *et al*. 2008) [4] were also observed and also appears similar. But then, the flapping behaviour for the result as shown in Figure 3 was noticed to have occurred a bit earlier within the simulation time here when compared to the time showed by (Srinarayana *et al*. 2008) [4]. Though, this little variation might be as a result of the simulation package (code) Gerris. Their results were obtained using the Gerris open source code a quad-tree based adaptive mesh solver which uses a fractional-step projection method. The advective terms are discretised using a second-order Godunov type scheme. The remaining terms used the standard second-order schemes and the equations are solved using a semi-implicit multi-grid approach (Srinarayana *et al*. 2008) [4].

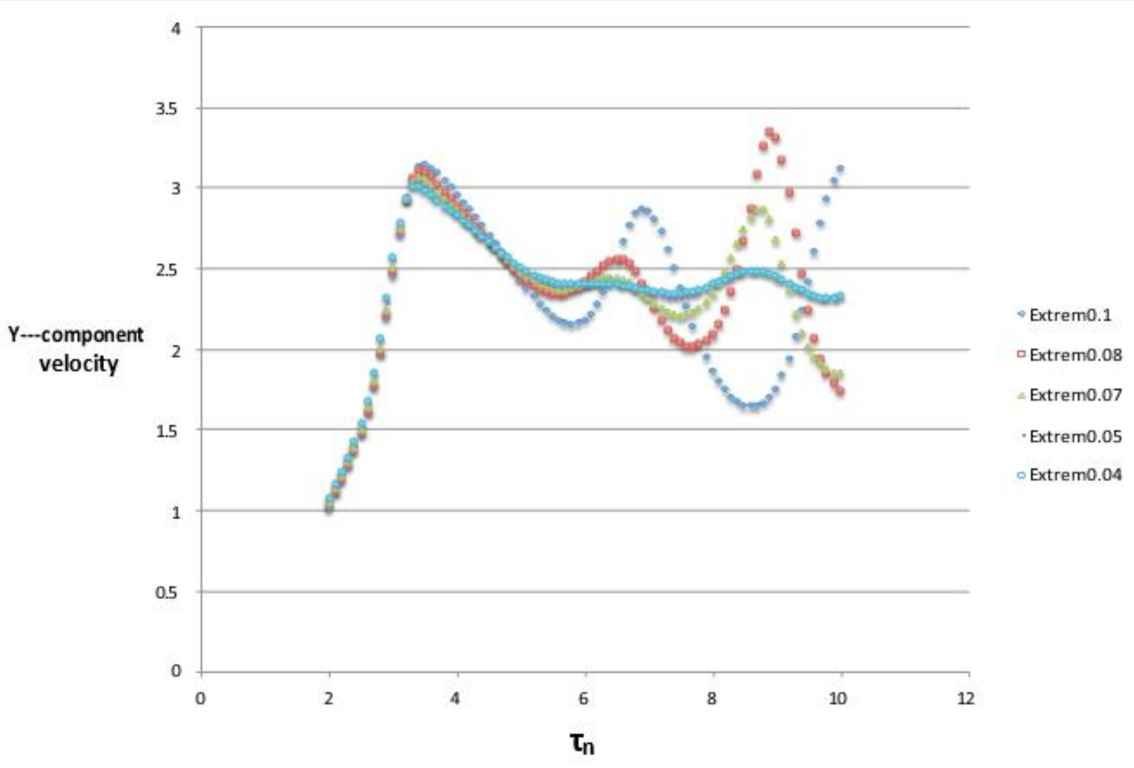

Fig 2: Vertical velocity profile at the point (0, 2) for *Re* = 50, *Fr* = 1 and *Pr* = 7 and plotted as a function of time for extremely fine mesh 0.1, 0.08, 0.07, 0.05 and 0.04 respectively.

2.2. Code verification Test and Results

The basic set of equations that were used for the simulation of those results by (George & Kay, 2017) are:

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \tag{1}$$

$$\frac{\partial U}{\partial \tau}+U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re}(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}) \tag{2}$$

$$\frac{\partial V}{\partial \tau}+U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re}(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2})+\frac{1}{Fr^2}[\phi^2-2\phi] \tag{3}$$

$$\frac{\partial \phi}{\partial \tau}+U\frac{\partial \phi}{\partial X}+V\frac{\partial \phi}{\partial Y}=\frac{1}{RePr}(\frac{\partial^2 \phi}{\partial X^2}+\frac{\partial^2 \phi}{\partial Y^2}) \tag{4}$$

The purpose of this section is to ensure that the proposed equations are solved by verifying the codes that were used for the simulation. We are aware that discretisation method in Comsol Multiphysics is the Galerkin's Method. However, it is a difficult task to specify the exact position of every mesh point to enable us determine the values at each nodal point. Instead, these values are extracted from COMSOL by the means of numerical interpolation using the finite difference method. With this, we are also believing that we can obtain closely related approximations to the equations even if a different scheme is use for the verification. This test is on the case with Reynolds number *Re* = 50, Froude number *Fr* = 1, Prandtl.

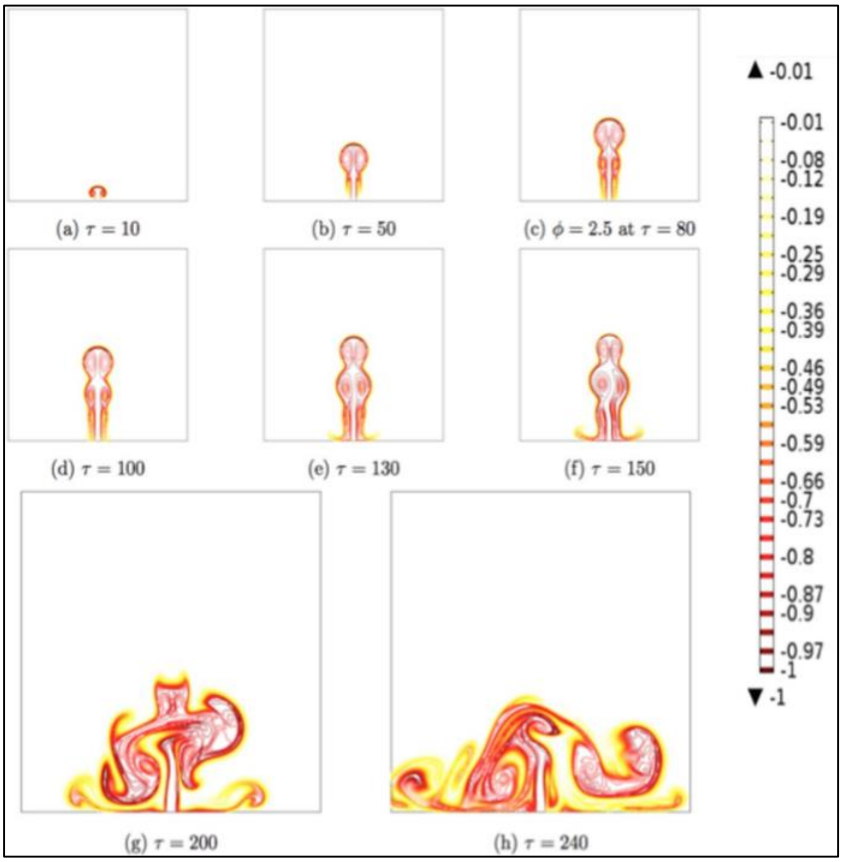

Fig 3: Evolution of temperature field for Fr = 8, Pr = 7, Re = 100 with $\varphi = -1$ at time $10 \leq \tau \leq 240$

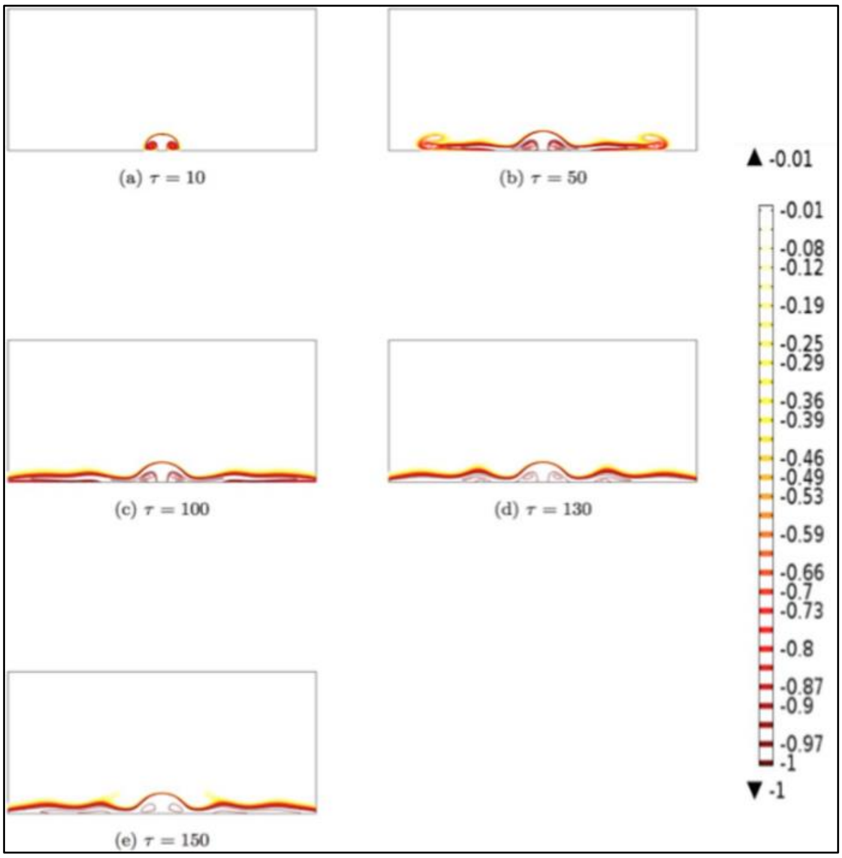

Fig 4: Evolution of temperature field for $Fr = 2$, $Pr = 7$, $Re = 100$ with $\varphi = -1$ at time $10 \leq \tau \leq 150$

Number $Pr = 7$, at point (0, 2), at the centre of the plume, and at time $\tau = 5$. The vertical ΔY and horizontal step size ΔX is the same 0.05; while that of the time step $\Delta\tau = 0.1$. After using the Forward Difference Method as shown in (5), (6), (7) and (8) for the interpolation, the following approximations were obtained for the equations in (1), (2), (3) and (4).

$$V(X,Y,\tau) = V_{i,j}^{n}; \frac{\partial V}{\partial \tau} = \frac{V_{i,j}^{n+1} - V_{i,j}^{n}}{\Delta \tau}; \frac{\partial V}{\partial X} = \frac{V_{i+1,j}^{n} - V_{i,j}^{n}}{\Delta X}; \frac{\partial V}{\partial Y} = \frac{V_{i,j+1}^{n} - V_{i,j}^{n}}{\Delta Y};$$
$$\frac{\partial^2 V}{\partial Y^2} = \frac{V_{i,j+1}^{n} - 2V_{i,j}^{n} + V_{i,j-1}^{n}}{\Delta Y^2}; \frac{\partial^2 V}{\partial X^2} = \frac{V_{i+1,j}^{n} - 2V_{i,j}^{n} + V_{i-1,j}^{n}}{\Delta X^2} \tag{5}$$

$$U(X,Y,\tau) = U_{i,j}^{n}; \frac{\partial U}{\partial \tau} = \frac{U_{i,j}^{n+1} - U_{i,j}^{n}}{\Delta \tau}; \frac{\partial U}{\partial X} = \frac{U_{i+1,j}^{n} - U_{i,j}^{n}}{\Delta X}; \frac{\partial U}{\partial Y} = \frac{U_{i,j+1}^{n} - U_{i,j}^{n}}{\Delta Y};$$
$$\frac{\partial^2 U}{\partial Y^2} = \frac{U_{i,j+1}^{n} - 2U_{i,j}^{n} + U_{i,j-1}^{n}}{\Delta Y^2}; \frac{\partial^2 U}{\partial X^2} = \frac{U_{i+1,j}^{n} - 2U_{i,j}^{n} + U_{i-1,j}^{n}}{\Delta X^2} \tag{6}$$

$$\phi(X,Y,\tau) = \phi_{i,j}^{n}; \frac{\partial \phi}{\partial \tau} = \frac{\phi_{i,j}^{n+1} - \phi_{i,j}^{n}}{\Delta \tau}; \frac{\partial \phi}{\partial X} = \frac{\phi_{i+1,j}^{n} - \phi_{i,j}^{n}}{\Delta X}; \frac{\partial \phi}{\partial Y} = \frac{\phi_{i,j+1}^{n} - \phi_{i,j}^{n}}{\Delta Y};$$
$$\frac{\partial^2 \phi}{\partial Y^2} = \frac{\phi_{i,j+1}^{n} - 2\phi_{i,j}^{n} + \phi_{i,j-1}^{n}}{\Delta Y^2}; \frac{\partial^2 \phi}{\partial X^2} = \frac{\phi_{i+1,j}^{n} - 2\phi_{i,j}^{n} + \phi_{i-1,j}^{n}}{\Delta X^2} \tag{7}$$

$$\frac{\partial P}{\partial X} = \frac{P_{i+1,j}^{n} - P_{i,j}^{n}}{\Delta X}; \frac{\partial P}{\partial Y} = \frac{P_{i,j+1}^{n} - P_{i,j}^{n}}{\Delta Y}; \tag{8}$$

The Continuity Equation as given in (1) approximates as follows:

$$\frac{\partial U}{\partial X} = -0.900347022; \ \frac{\partial V}{\partial Y} = 0.910710949 \tag{9}$$

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} \approx 0.010 \tag{10}$$

The X-Momentum equation as given in (2) also approximates as follows:

$$\frac{\partial U}{\partial \tau} = -0.001550458; \ U\frac{\partial U}{\partial X} = -0.004042746; \ V\frac{\partial U}{\partial Y} = 0.00828627; \ \frac{\partial P}{\partial X} = -0.004370912;$$
$$\frac{1}{Re}(\frac{\partial^2 U}{\partial X^2}) = -0.00030398; \ \frac{1}{Re}(\frac{\partial^2 U}{\partial Y^2}) = -0.000168113 \tag{11}$$

Then, the left hand side of the X-Momentum equation now becomes;

$$\frac{\partial U}{\partial \tau} + U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} \approx 0.003 \tag{12}$$

Whereas, the right hand side of the X-Momentum equation then gives;

$$-\frac{\partial P}{\partial X} + \frac{1}{Re}(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}) \approx -0.014 \tag{13}$$

The Y-Momentum equation as given in (3) also approximates as follows:

$$\frac{\partial V}{\partial \tau} = -0.27636697; \ U\frac{\partial V}{\partial X} = -0.000579581; \ V\frac{\partial V}{\partial Y} = 2.273645601; \ \frac{\partial P}{\partial Y} = -0.849099775;$$
$$\frac{1}{Re}(\frac{\partial^2 V}{\partial X^2}) = -0.112232389; \ \frac{1}{Re}(\frac{\partial^2 V}{\partial Y^2}) = 0.012951756; \ \frac{1}{Fr^2}[\phi^2 - 2\phi] = 1.249950585 \tag{14}$$

Then, the left hand side of the Y-Momentum equation now becomes;

$$\frac{\partial V}{\partial \tau} + U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} \approx 1.997 \tag{15}$$

Whereas, the right hand side of the Y-Momentum equation then gives;

$$-\frac{\partial P}{\partial X}+\frac{1}{Re}(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2})+\frac{1}{Fr^2}[\phi^2-2\phi] \approx 2.000 \quad (16)$$

The Temperature equation as given in (4) also approximates as follows:

$$\frac{\partial \phi}{\partial \tau}=0.0000438936 \approx 0;\ U\frac{\partial \phi}{\partial X}=-0.0000028426 \approx 0;\ V\frac{\partial \phi}{\partial Y}=-0.000634676 \approx 0;$$
$$=\frac{1}{RePr}(\frac{\partial^2 \phi}{\partial X^2})=-0.0000904332 \approx 0;\ \frac{1}{RePr}(\frac{\partial^2 \phi}{\partial Y^2})=-0.00000428228 \approx 0; \quad (17)$$

Then, the left hand side of the Temperature equation now becomes;

$$\frac{\partial \phi}{\partial \tau}+U\frac{\partial \phi}{\partial X}+V\frac{\partial \phi}{\partial Y} \approx -0.001 \quad (18)$$

Whereas, the right hand side of the Temperature equation then gives;

$$\frac{1}{RePr}(\frac{\partial^2 \phi}{\partial X^2}+\frac{\partial^2 \phi}{\partial Y^2}) \approx 0 \quad (19)$$

From the continuity equation (10), the interpolated results has shown that computation at the centre line of the plume is well resolved to some extent even though the individual terms as shown in (9) varies slightly with a maximum variation of $\approx \pm 0.010$. However, by definition, we have that the sum of these terms is equal zero. And this is also evident that the sum of these individual terms is approximately zero when rounded up within the first few decimal places. Computation from the Y-Momentum equation has also shown to have been resolved properly to some extent. Fluid flow in this direction is key for a rising plume at the first few time interval and as well as the source term in this equation being crucial to the investigation. However, the results from the sum of the individual terms from both the left and right hand sides (15) and (16) of this Y-Momentum equation showed interesting as both sides appears positive with a maximum variation of $\approx \pm 0.003$. Water at the plume's core at that point is expected to be undiluted and positively buoyant, which will in turn result in an upward movement and not negatively buoyant. There- fore, the positive values obtained from the interpolation in this direction are believed to be reasonable and the Y-Momentum equation well resolved to a reasonable extent. Meanwhile, sum of the individual terms from the interpolation in both sides of the X-Momentum equation (12) and (13) showed both positive and negative values. This discrepancies in the signs especially the right hand side (13) tells us that there was some slight mixed fluid at that point in the plume. However, the maximum variation between the left and right hand side of the X-Momentum equation is $\approx \pm 0.011$ when compared which also appears reasonable.

From the temperature equation, the results showed very insignificant but negative on the left hand side (see equation 18) and approximately zero on the right hand side (see equation 19). It is believed that temperature of the plume at the core and around the source will be very warm as compared to the temper- ature elsewhere. The fact that step size for the interpolation is small, this also resulted in the temperature variation within the various points to be very small, unless the interpolation is made with a bigger step size where variations within points may be significant. However, even as results from both left and right hand sides of the temperature equation appears to be very small, we still believe that these results are well resolved to some extent with a maximum variation of $\approx \pm 0.001$ between them. In overall, interpolations at the centre point (0, 2) of the plume for the various equations are reasonable and realistic. Though, there were some little variations when comparing the approximations from the left and right of both the momen- tum and the temperature equation but then, it is clear that the proposed set of equations are solved using the Comsol Multiphysics. The data from the Comsol file for the calculation are given below in equation (20).

Values from Comsol Multiphysics Text files at (0, 2)

$$V^n_{i,j}=2.496561179; V^{n+1}_{i,j}=2.468924482; V^{n-1}_{i,j}=2.527303807; V^n_{i+1,j}=2.490107343;$$
$$V^n_{i,j+1}=2.542096726; V^n_{i-1,j}=2.488985966; V^n_{i,j-1}=2.452644601; U^n_{i,j}=0.004490209;$$
$$U^{n+1}_{i,j}=0.004335163; U^{n-1}_{i,j}=0.004578498; U^n_{i+1,j}=-0.040527142; U^n_{i,j+1}=0.004656163;$$
$$U^n_{i-1,j}=0.049469563; U^n_{i,j-1}=0.004303241; \phi^n_{i,j}=2.499983528; \phi^{n+1}_{i,j}=2.499987917;$$
$$\phi^{n-1}_{i,j}=2.499980271; \phi^n_{i+1,j}=2.499951875; \phi^n_{i,j+1}=2.499970817; \phi^n_{i-1,j}=2.499936052;$$
$$\phi^n_{i,j-1}=2.499992492; P^n_{i,j}=-0.1259849; P^n_{i+1,j}=-0.126203446; P^n_{i,j+1}=-0.168439889 \quad (20)$$

2.3. Discussion/Concluding remarks

We have carried out an investigation on the validation of some results by (George and Kay, 2017) in their work entitled "Warm discharges in cold fresh water: 2. Numerical simulation of laminar line plumes", with the aid of a CFD tool (COMSOL MultiPhysics) through Code Verification and Mesh Sensitivity Test. We also tried to reproduce some results by (Srynarayana *et al*. 2008) in their work entitled "Height and stability of laminar plane fountains in a homogeneous fluid" with the same COMSOL MultiPhysics for code validation and correctness of the results. As highlighted above, the correctness or exactness of any numerical solution is also tied to the mesh size (the smaller the mesh size the more accurate the result becomes). Mesh dependence test was carryout where we examined both the temperature distribu- tion and velocity field at the point (0,2) at the core of the plume for those results by (George & Kay, 2017) at $\tau = 5$ for Re = 50, Fr = 1, Pr = 7 and plotted as a function of time for different mesh sizes (0.1, 0.08, 0.07, 0.05 and 0.04). The results confirmed that both velocity field and temperature distribution at the specified point converged as mesh size decreases. Though, the converging behaviour was not too clear in the temperature distribution because, mixture here requires just little mixing before attaining the same temperature as that in the ambient fluid. Whereas, when compared to the velocity field, we observed that the solution converges gradually to a single solution as mesh size gets smaller and smaller. Thus, the solutions by (George and Kay, 2017) will be independent of the mesh used, if the mesh size is ≤ 0.05.

For the code verification, we have earlier stated that the discretisation method in Comsol Multiphysics is the Galerkin's Method, and that it is a difficult task to specify the exact position of every mesh point to enable us determine the values at each nodal point. Thus, we have instead extracted these values from COMSOL by the means of numerical interpolation using the finite difference method. The results here showed that values from the individual terms in the continuity equation vary slightly with a maximum variation of $\approx \pm 0.010$ when taken their sum. This we can say is approximately zero when rounded up within the first few decimal places. Though, by definition we have that the sum of these terms is equal zero. Interpolated values from the Y-Momentum equation showed that it was resolved properly to some extent; though with a maximum variation of $\approx \pm 0.003$. Note that fluid flow in this direction is very key for a rising plume at the first few time interval. Individual terms from both the left and right hand sides of this Y-Momentum equation appeared positive. Water at the plume's core at that point is expected to be undiluted and positively buoyant, which will in turn result in an upward movement. Therefore, the positive values obtained from the interpolation in this direction are believed to be reasonable. Whereas, values from the X-Momentum equation showed both positive and negative values with maximum variation of $\approx \pm 0.011$. This indicates that there was some slight mixed fluid at some point in the plume on the right hand side and it is also reasonable. Interpolated values from the temperature equation showed very close, though, negative on the left and positive on the right but approximately zero in all. This is reasonable because, It is believed that temperature of the plume at the core and around the source is warm as com- pared to the temperature elsewhere. Though, the insignificant negative value on the left also indicate that there was some sort of negatively buoyant fluid at that point. Thus, we strongly believe that these results are well resolved to some extent with a maximum variation of $\approx \pm 0.001$ between them.

Furthermore, with this same mesh size (0.05) using COMSOL, we have replicated some of the results by (Srinarayana *et al*. 2008) [4], maintaining the same settings as they have used to produce those in Figure 3 and 5 as shown in their work within the time range $10 \leq \tau \leq 240$ and this is shown here in Figure 3 and

4. The overall behaviour of the results as recorded by (Srinarayana *et al*. 2008) [4] were also observed and ap- pears similar. Though, the flapping behaviour for the result as shown in Figure 3 here was noticed to have occurred a bit faster within the simulation time. Though, this slight variation might be as a result of the simulation package (code) Gerris. In overall, interpolations at the centre point (0, 2) of the plume for the various equations are reasonable and realistic. Though, there were some little variations when comparing the approximations from the left and right of both the momentum and the temperature equation but then, it is clear that the proposed set of equations are solved using the Comsol Multiphysics. Conclusively, the results as shown by (George and Kay, 2017) are reliable and independent of mesh size ≤ 0.05 and as well the CFD package used is reliable as this could reproduce other results from Gerris open source code.

References

1. Benay R, Chanetz B, Delery J. Code verification/validation with respect to experimental data banks. Aerospace Science and Technology. 2003;7:239–62.
2. Altaie MA, Alsarraj GR, Al-Bayati HA. Verification and validation of a software: A review of the literature. Iraqi Journal for Computer and Information. 2020;46(1):40–7.
3. George MA, Kay A. Warm discharges in cold fresh water: 2. Numerical simulation of laminar line plumes. Environmental Fluid Mechanics. 2017;17:231–46.
4. Srinarayana N, McBain DG, Armfield WS, Lin W. Height and stability of laminar plane fountains in a homogeneous fluid. International Journal of Heat and Mass Transfer. 2008;51:4717–27.
5. Vanova P, Kvocak V. The importance of mesh module in FEM analysis of push-out tests. Section Building Structures & Structural Mechanics. 2021;21(2):86–9.
6. Gupta N, Bhardwaj N, Khan MG, Dave V. Global trends of computational fluid dynamics to resolve real-world problems in the contemporary era. Current Biochemical Engineering. 2020;6(3):1–20.
7. Bistafa RS. On the development of the Navier-Stokes equation by Navier. Revista Brasileira de Ensino de Fisica. 2018;40(2):1–12.
8. Engelmann L, Hasslberger J, Baik S, Klein M, Kempf A. Direct numerical simulation of an unsteady wall-bounded turbulent flow configuration for the assessment of large eddy simulation models. Scientific Reports. 2023;13:11202.
9. Argyropoulos DC, Markatos CN. Recent advances on the numerical modelling of turbulent flows. Applied Mathematical Modelling. 2015;39:693–732.
10. Patil H, Jeyakarthikeyan VP. Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS. Materials Science and Engineering. 2018;402:012065.

11. COMSOL Multiphysics Cyclopedia. The finite element method (FEM) [Internet]. Available from: https://www.comsol.fr/multiphysics/finite-element-method?parent=physics-pdes-numerical-042-62 [Accessed 28 April 2024].
12. COMSOL Multiphysics Reference Manual. Version: COMSOL 5.2.
13. Jauregui R, Silva F. Numerical validation methods [Internet]; c2011. Available from: https://core.ac.uk/download/pdf/41765513.pdf [Accessed 28 April 2024].
14. Lintermann A. Computational meshing for CFD simulations [Internet]; c2020. Available from: https://www.researchgate.net/publication/344784554ComputationalMeshingforCFDSimulations [Accessed 28 April 2024].