Experimental validation of computational fluid dynamics for solving isothermal and incompressible viscous fluid flow

Bilen Emek Abali¹ · Ömer Savaş²

Received: 29 January 2020 / Accepted: 23 July 2020 / Published online: 11 August 2020
© The Author(s) 2020

Abstract

In order to validate a computational method for solving viscous fluid flows, experiments are carried out in an eccentric cylindrical cavity showing various flow formations over a range of REYNOLDS numbers. Especially, in numerical solution approaches for isothermal and incompressible flows, we search for simple experimental data for evaluating accuracy as well as performance of the computational method. Verification of different computational methods is arduous, and analytic solutions are only obtained for simple geometries like a channel flow. Clearly, a method is expected to predict different flow patterns within a cavity. Thus, we propose a configuration generating different flow formations depending on the REYNOLDS number and make the experimental results freely available in order to be used as an assessment criterion to demonstrate the reliability of a new computational approach.

Keywords Cavity flow · Experimental validation · PIV · CFD · FEM

1 Introduction

The computational study of a physical problem is accepted as reliable, if the employed implementation is validated by using the experimental results. Especially, in fluid dynamics simulations, since the computational methods are still being developed, the experimental results with clearly identified initial and boundary conditions are of paramount importance. Experimental data are used for validating the numerical results, specifically for cases where analytic solutions are not known. Ample studies are available in the literature comparing the experimental and numerical results for NEWTONIAN fluids like water [1, 2], blood [3, 4], even for very thick fluids [5], during casting [6], and also for geometries with an obstacle creating vortex shedding [7–9] or even for a mixture [10]. Especially, in engineering science, a commercial software or an in-house code is employed for a numerical analysis of a simplified system.

Assumptions have to be made for the simplification; thus, the reliability of the used software is challenging to justify. Furthermore, we stress that different types of partial differential equations make the coupled system difficult to solve [11]. Hence, various methods and implementations exist and are emerging; examining their reliability is of great interest. In benchmarking new numerical schemes [12], the reference results are generated by using direct numerical simulations in order to design a computationally challenging configuration [13–16] for testing the robustness of the numerical implementation. For some special applications, experimental data are utilized for a specific geometry and used to validate the numerical results [17, 18]. For systems with lacking the experimental results, direct numerical simulations are used for generating test cases to verify the accuracy of a numerical implementation [19, 20].

Generally speaking, a numerical implementation needs to be robust and the used method is expected to deliver

✉ Bilen Emek Abali, bilenemek@abali.org | ¹Institute of Mechanics, Faculty V of Mechanical Engineering and Transport Systems, Technische Universität Berlin, Einsteinufer 5, 10587 Berlin, Germany. ²Department of Mechanical Engineering, Fluid Mechanics Laboratory, University of California, Berkeley, 140 Hesse Hall, Berkeley, CA 94720-1740, USA.

reliably the accurate results. A computationally challenging configuration examines the robustness of the numerical implementation. An experimental setup provides a realistic setting such that we can quantify the accuracy of the method. We propose a simple yet effective geometry generating different flow formations that are challenging for a numerical implementation to simulate reliably. Experimental data are recorded, processed, and made public in [21] to be used for testing a new or existing implementation. The setup is simple, a cavity flow of a NEWTONIAN fluid, and we compare the computed velocities with the experimental results directly. By changing the inlet velocity, the flow pattern changes completely such that developers may find these experimental results useful for examining performance of their numerical implementations.

In this work, we collect the velocity data of a NEWTONIAN (linear viscous) fluid in the case of an incompressible flow. We refer to [22–24] for studies with a similar experimental setup for different systems. Herein, we present an experimental study on an eccentric cylindrical cavity, in which the flow is studied by means of particle image velocimetry (PIV) [25]. This standard technique is often employed for laminar flows [26–28], in order to study the transient response [29], or behind an obstacle [30, 31]. In this work, fluorescent particles are used both for flow visualization (FV) and PIV at the central plane of a symmetric flow cavity. The 2-D measurements at this plane are compared to the results obtained from 3-D computations. As long as the computational results match the experimental results on the symmetry plane, we expect the implementation being reliable. We examine the method proposed by Abali [32] based on the finite element method by using open-source libraries developed under the FEniCS project [33]. The proposed approach is already verified by exact solutions for simple geometries, and the method attains a local monotonic convergence for a three-dimensional computational domain. Herein, we validate the method by using the experimental results obtained in this work.

We propose a simple experimental setup providing different flow formations depending on the REYNOLDS number. This experimental data are novel for examining a computation implementation since the same configuration generates various flow formations within the laminar regime by just changing the inflow velocity. A detailed comparative analysis is undertaken between the experimental data and computational results. Computational fluid dynamics (CFD) uses often the finite volume method (FVM), although we use herein the finite element method (FEM) that allows incorporating multiphysics more efficiently.

There are mainly three approaches like projection methods [34–37], stabilization with limiting strategies [38–40], and using enriched or mixed formulations for solving pressure and velocity fields [41–44]. FEM is still emerging for fluid dynamics, and we propose to use the experimental results for validating new computational approaches. We demonstrate qualitative and quantitative studies and make the experimental results and numerical implementation—CAD model, used mesh, and code in FEniCS to be used under GNU public license [45]—publicly available (see Abali [21]) in order to encourage scientific exchange.

2 Experimental setup

2.1 Flow loop

The experimental setup is sketched in Fig. 1, and the camera from the top records the flow within the cavity through a high-pass filter as shown in pictures.

The flow geometry is a short aspect ratio cylindrical cavity, which is cast out of clear silicone. The cavity is manufactured in two steps. First, a mold assembly is machined of four aluminum parts to form the flow cavity. A low-temperature metal alloy (Cerroflow 117, melting point 47.5°C) is cast into this cavity. Then, the metal model is removed from the aluminum mold assembly and placed in a silicon gel bath (Sylgard 184, Dow Corning Inc., USA), hardened after curing, and the alloy is removed from the model by melting, leaving a transparent flow model.

The cavity is of diameter $D = 19$ mm and length $L = 25.4$ mm with a right-handed reference system (xyz) as illustrated in Fig. 2. The flow in the feed and discharge tubes is in the x -direction. The laser light sheet is in the xy -plane that is the symmetry plane of the geometry and the camera looks into the flow model along the z -direction.

It is fed and discharged through eccentric in-line cylindrical pipes; both of diameter $d = 6.35$ mm and tangent to the cylindrical wall of the main flow cavity. The reason for the choice of this particular geometry is that it provides a combination of challenging boundary conditions, even for laminar flows. For example, the two-dimensional backward facing step geometry, both unbounded and bounded forms above, have been usually taken as a reference geometry in flow simulations (see, for example [46]). The current geometry presents *bounded, three-dimensional forward and backward facing steps at once*.

The working fluid is isopropyl alcohol of density $\rho = 0.785$ g/cm³ and kinematic viscosity of $\nu = 0.031$ cm²/s. Its index of refraction is $n = 1.378$, which is near, but not a perfect match, to that of the silicone casting ($n = 1.41$) out of which the flow model is built. The fluid is seeded with 6 μ m fluorescent particles (Duke Scientific Corporation, Red

Fig. 1 Experimental setup. Left top: sketch of the flow loop and the experimental setup for recording the flow field. Right: picture of the camera positioned above the cavity. Left bottom: the cavity as seen with the naked eye under the laser light through a high-pass filter

Fig. 2 Drawing of the cylindrical cavity (d, D, L) = (0.635 cm, 1.905 cm, 2.54 cm). The flow in the inlet and outlet pipes is along the x-axis, and it is measured on the symmetry, xy-plane

Fluorescent Polymer Microspheres) for flow visualization and PIV measurements.

The flow pumping system described by Tsai and Savaş [47], which is comprised of a piston and a gear pump in series, is employed here to drive the fluid in the system. In the current application, the piston pump is turned off and the servomotor of the gear pump is replaced with a microstepper motor (25 000 pulses per revolution) to achieve stable operation at very low speeds. The gear pump capacity is 10.0 cm³/revolution. The gear pump speed ranged from 0.005 (200 s per revolution) to 1 revolution per second (rps). Flows are identified by their REYNOLDS number based on the flow rate in the inlet pipe,

$$\text{Re}_d = \frac{U_{\text{ave}} d}{\nu} = \frac{4 Q}{\pi \nu d}, \quad (1)$$

where Q is the volumetric flow rate. At very slow rotation rates of the pump, i.e., 0.005 and 0.010 rps, the flow rate was actually determined from the PIV data, as the static leakage of the gear pump introduces large uncertainties in flow rates determined from the displacement volume of the gear pump. For the steady inflow experiments presented here, the REYNOLDS numbers ranged from $\text{Re}_d = 2.7$ –650. Later on for comparison, we will use REYNOLDS numbers all computed from the PIV data.

2.2 Optics

The optical components include a digital camera Kodak Megaplug ES1.0 for imaging (1008 × 1018 pixel resolution), a cemented spherical–cylindrical lens pair for laser light sheet generation, and a removable flat mirror for laser beam selection. The cemented pair consists of a +400 mm FL plano-convex lens and −20 mm FL cylindrical lens. Light beams form the PIV lasers, NewWave Gemini-30, dual-pulsed YAG laser, wavelength $\lambda = 0.532 \mu\text{m}$, and flow visualization laser, a continuous wave (CW) argon-ion laser, operated at green wavelength $\lambda = 0.5145 \mu\text{m}$, share the same optical path into the flow cavity. The removable mirror shown in the figure is put into position when the PIV lasers are needed. The thickness of the light sheet at the flow cavity is estimated to be about 50 μm . The camera, fitted with a Canon FD 50 mm, f1:1.4 lens, a 5-mm extension tube, and a C-mount adapter, is used for both FV and PIV image acquisitions. The STOKES's shift of the fluorescent seed particles is 50 nm. Therefore, a high-pass filter with a cutoff wavelength of 0.560 μm was used to record the light emitted from the fluorescent seed particles. Experimental

Table 1 Summary of experiments

Case	Flow parameters			FV (CW laser)		PIV (pulsed lasers)	
	Pump speed (rps)	U_{ave} (cm/s)	Re_d	Imaging rate (fps)	Exposure (ms)	Sample rate (samples/s)	Δt (ms)
(a)	0.005	0.132*	2.7*	20	10	10	12.5
(b)	0.010	0.272*	5.6*	20	10	10	12.5
(c)	0.025	0.789 [†]	16 [†]	20	10	10	12.5
(d)	0.050	1.58 [†]	32 [†]	20	10	10	12.5
(e)	0.100	3.16 [†]	65 [†]	20	10	10	10
(f)	0.250	7.89 [†]	160 [†]	20	10	10	4
(g)	0.500	15.3 [†]	320 [†]	20	10	10	2
(h)	1.000	31.6 [†]	650 [†]	20	10	10	1

FV-flow visualization, PIV-particle image velocimetry. (*)*: estimated from the PIV data, ([†]): estimated after pump calibration

setup details, including FV and PIV imaging parameters, are shown in Table 1.

Image acquisition and processing techniques are similar to those described and used in [22, 48, 49]. In particular, the PIV algorithm developed in house is used for correlation velocimetry. Postprocessing is done using routines written in IDL (Interactive Data Language).

3 Computational method

Isothermal flow of viscous fluid is given by balance equations of mass and momentum:

$$\frac{\partial \rho}{\partial t} + (v_i \rho)_{,i} = 0, \quad \frac{\partial (\rho v_j)}{\partial t} + (v_i \rho v_j - \sigma_{ij})_{,i} = \rho g_j, \quad (2)$$

respectively, where x_i indicate the physical coordinates with index i from 1 to 3 and we employ the usual comma notation for spatial derivatives $()_{,i} = \partial()/\partial x_i$ as well as EINSTEIN'S summation convention over repeated indices. All field variables are expressed in Cartesian coordinates. The unknowns to be solved are the mass density, ρ , and the velocity, v_i , of fluid particles occupying x_i in space at the instant t . The specific supply term, g_j , is because of gravitational forces, and its value is known, but this term is negligible in the simulations, as we know that the flow is predominantly caused by the pressure difference. We assume that isopropyl alcohol is a NEWTONIAN fluid and model the flow by using a linear constitutive equation for CAUCHY'S stress:

$$\sigma_{ij} = (-p + \lambda d_{kk})\delta_{ij} + 2\mu d_{ij}, \quad d_{ij} = \frac{1}{2}(v_{i,j} + v_{j,i}), \quad (3)$$

with the material constants λ , μ , and p , the hydrostatic pressure. This model is also called NAVIER-STOKES equation,

as it has been proposed by both scientists in a phenomenological way, we refer to [50] for historical remarks. Under the usual assumption of an incompressible flow and the fluid has been homogeneous initially, the mass density remains constant in time and space. Then, from Eq. (2), we obtain

$$v_{i,i} = 0, \quad \rho \frac{\partial v_j}{\partial t} + \rho (v_i v_j)_{,i} - \sigma_{ij,i} - \rho g_j = 0. \quad (4)$$

In this case, namely for an incompressible flow, the mechanical pressure $-\frac{1}{3}\sigma_{ii}$ is identical to the hydrostatic pressure, p ; hence, p corresponds to the (mechanical) pressure generated by a pump. Since the mass density is given, we use Eq. (4)₁ as the governing equation of p . We emphasize that usually Eq. (4)₁ is inserted into Eq. (4)₂, which we circumvent for the sake of numerical stability. Various computational implementations exist in the literature for solving the governing equations (4) resulting in velocity and pressure fields. A straight-forward solution strategy in a two-dimensional continuum by using the finite element method is described in [51, Sect. 1.7]. The strategy relies on generating a so-called weak form out of governing equations. Trying to use this weak form in the case of a three-dimensional continuum starts causing numerical instabilities [52–54]. For circumventing these numerical problems, a great effort has been undertaken, for a brief review, we refer to [55]. As a remedy to these numerical problems, stabilization methods have been proposed in [56–58], and these methods are advanced [59–62] and implemented [63–65]. Also different numerical methods have been suggested for a robust implementation [66–70]. Another approach has been taken by changing the weak formulation [71–74]. Novel approaches have been proposed for advancing meshing algorithms as well as developing better solvers for fluid dynamics [75–78]. Also novel

developments exist in hybrid methods as in [79–81] leading to research codes as in [82] as well as incorporating semi-analytical methods [83] or by modeling jump conditions on element boundaries in [84–86]. All these different theoretical approaches and various numerical implementations indicate that a computational method accepted by all communities is unlikely to occur in the near future.

We employ the method developed and verified by Abali [32] in order to generate a weak form for computing velocity and pressure as follows:

$$\text{Form}|_{\Omega} = F_1 + F_2 + F_3, \quad (5)$$

with

$$\begin{aligned} F_1 &= \int_{\Omega} v_{i,j} \delta p \, dv, \\ F_2 &= \int_{\Omega} \left(\rho \frac{v_j - v_j^0}{\Delta t} \delta v_j - \rho g_j \delta v_j + \rho (v_i v_j)_{,i} \delta v_j + p_{,i} \delta v_i + \tau_{ij} \delta v_{j,i} \right) dv, \\ F_3 &= \int_{\Omega} \left((v_j - v_j^0) - \Delta t g_j + \Delta t (v_i v_j)_{,i} - \frac{\Delta t}{\rho} (-p \delta_{ij} + \tau_{ij})_{,i} \right) \delta p_{,j} \, dv. \end{aligned} \quad (6)$$

The boundary term, $n_i \tau_{ij}$, vanishes in the case of an incompressible fluid since the prescribed pressure p against the plane outward, $t_i = -p n_i$, leads to the formulation on the boundary, $n_i \tau_{ij} = n_i (\sigma_{ij} + p \delta_{ij}) = t_j + n_j p = 0$.

We use continuous linear elements being the standard formulation for the finite element method [87]. For each node, four variables $\{p, v_1, v_2, v_3\}$ are computed in three-dimensional space belonging to

$$\mathcal{V} = \{p, v_i \in [\mathcal{H}^n(\Omega)]^4 : p, v_i|_{\partial\Omega} = \text{given}\}, \quad (7)$$

where $[\mathcal{H}^n]^4$ is a four-dimensional HILBERTIAN SOBOLEV space of class C^n . We use $n = 1$, i.e., linear form functions for pressure and also for velocity. We circumvent any numerical problems affected by the LADYZHENSKAYA–BABUSKA–BREZZI condition [88]. The additional governing equation restricting the pressure gradient enables using the same space for both fields (velocity and pressure). No stabilizing methods or limiting procedures are used in the implementation such that the approach attains monotonous convergence, in other words, using a finer mesh improves the accuracy without affecting the stability. We emphasize that g, s, mm are chosen as units leading to $\mu\text{N} = \text{g mm/s}^2$ and $\text{Pa} = \mu\text{N/mm}^2$. The preprocessing has been realized in Salome¹ by using NetGen algorithms² for the triangulation. The mesh is of tetrahedron elements and is

transformed as explained in [51][Appendix A.3] in order to implement in a Python code using packages developed under the FEniCS project,³ which is wrapped in C++, and run on a Linux machine using Ubuntu.⁴

4 Results and discussion

Figure 3 shows the flow patterns for the cases listed in Table 1. The traces of the fluorescent particles recorded at long exposure under continuous illumination show the steady flow pattern at the central plane of the cavity. Each picture is constructed by adding subsequent images. High-speed regions show long streak, whereas the low-speed recirculation zones show dotted trajectories. All flows are steady, with the possible exception of that at $\text{Re} = 650$ where the corresponding video sequence shows signs of instability at the unconstrained side of the inflow jet. Blurred signatures of these instabilities are marked with arrows in Fig. 3h. At the lowest two REYNOLDS numbers, the flow spreads into the cavity filling it, and, then, converges to the exit port. At the higher REYNOLDS number, the flow attains a *ballistic regime* where the jet from the inlet port heads straight to the exit port with little distortion in the velocity profile. A large circulation zone is set up in the cavity driven by the ballistic jet. At intermediate REYNOLDS numbers (16 and, to some extent, 32), the transition from cavity filling regime to the ballistic regime is apparent.

Computational solution of the same geometry has been obtained on a mesh as in Fig. 4 with 53,208 degrees of freedom, which is a *coarse mesh* for such a study, selected on purpose, demonstrating the robustness of the implementation.

Then, we used a *fine mesh* as depicted in Fig. 5 resulting in 345,972 degrees of freedom and obtained very similar results; we present herein the results obtained with the coarse mesh.

We stress that the local monotonous convergence as demonstrated in [32] assures an improved accuracy for refined mesh choices. Because of this monotonous convergence, verification of the numerical results by an error quantification as in [89] is not necessary since we validate the computational results by comparing them to the experimental results. An important challenge is to maintain the incompressibility; hence, we use the ratio of mass flow at the inlet per outlet as a measure to quantify the accuracy of the incompressibility condition. Space and time discretizations affect this measure, by choosing the

¹ Salome, the open-source integration platform for numerical simulation, <http://www.salome-platform.org>.

² Netgen Mesh Generator, <https://sourceforge.net/projects/netgen-mesher/>.

³ The FEniCS computing platform, <https://fenicsproject.org/>.

⁴ Ubuntu, open source software operating system, <https://www.ubuntu.com/>.

Fig. 3 Short time-averaged images for the cases compiled in Table 1. The arrows in (h) show blurred signatures of instabilities developing at the free edge of the jet

Fig. 4 Coarse mesh, left: the outer surface, right: cut in the middle along the elements for the sake of visualization

Fig. 5 Fine mesh, left: the outer surface, right: cut in the middle along the elements for the sake of visualization

aforementioned degrees of freedom with a time step of 1 ms, we have obtained an approximate error of 0.25%. The simulation lasts nearly 30 h on the coarse mesh and 600 h on the fine mesh by using 40 cores⁵ in parallel by utilizing *mpirun* using the default backend *PETSc* [90] with the mesh partitioner *SCOTCH* [91] and by sharing vertices in separating the mesh to threads. Especially, the solver parameters are the key for a successful computation in incompressible flow simulations; therefore, we emphasize that the code is made public as well in [21]. A standard *NEWTON-RAPHSON* solver with consistent linearization is used. Because of using small time steps and the computed values for initial values for the algorithm, less than five iterations have been observed to reach a tolerance of 10^{-3} in the aforementioned units, and tolerance means the *EUCLIDIAN* norm of changes in each iteration. For each iteration, *KRYLOV* subspace method is used for the linearized problem, and we have selected the generalized minimal residual method *gmres* for the underlying non-symmetric operator. We have used an algebraic multigrid *amg* preconditioner [92] as implemented in *FEniCS* by using *PETSc* algorithms. Specifically, we use an unsmoothed aggregation. The un restarted *gmres* with a tolerance of 10^{-4} has converged for coarse and fine meshes roughly the same manner such that the preconditioner *amg* bounds the

condition number of the problem independent on the mesh size. The success of the preconditioner depends on the chosen boundary conditions [93]. We emphasize that on all boundaries, *DIRICHLET* conditions of either unknowns are used. The experimental setup is modeled by setting the velocity profile at the inlet of radius $r = d/2$ along the inlet pipe axis,

$$v_m \left(1 - \frac{x^2 + y^2}{r^2} \right), \quad (8)$$

providing a parabolic velocity distribution directed inward the cylindrical cavity. The maximum velocity, v_m , at the center is given by,

$$v_m = \frac{\nu \text{Re}}{r}, \quad (9)$$

using the *REYNOLDS* number, *Re*, up to 600 linearly increased in 600 s under the assumption that the change is slow enough to attain the steady-state solution for each time step. We stress that the computation is transient and we compare the results by choosing the instants depending on the maximum velocity obtained at the same location from the PIV data and computation. Specifically, at the symmetry plane, $z = 0$, we have chosen the center of inlet pipe, $x/L = -0.1$ and $y/L = 0.625$ such that the velocity is directed along the *x*-axis. The time values chosen for the comparison read 3 s, 6 s, 17 s, 34 s, 65 s, 145 s, 290 s, and 550 s, and we present in Table 2 maximum velocities,

⁵ Intel Xeon Broadwell-EX series E7-4850 operating under Ubuntu 18.04

Table 2 Choice of time values for comparison, the experimental results are obtained from the PIV data, and maximum velocity is calculated in experimental as well as computational data at the same location, $v_{\max} = v_1(-0.1L, 0.625L, 0)$, giving REYNOLDS number Re

Case	Experimental results		Computational results		
	$v_{\max}$ in mm/s	Re	Time in s	$v_{\max}$ in mm/s	Re
(a)	2.6	2.7	3.0	2.8	2.9
(b)	5.4	5.6	6.0	5.6	5.8
(c)	15.3	15.7	17.0	16.0	16.4
(d)	31.3	32.1	34.0	32.1	32.9
(e)	61.3	62.8	65.0	61.3	62.7
(f)	137.5	140.8	145.0	136.7	140.1
(g)	281.2	288.0	290.0	274.2	280.8
(h)	529.2	542.0	550.0	521.9	534.6

In the figures, the experimental results are shown on left side and the numerical ones on the right side. Selected velocity profiles are shown on the top row and streamline patterns on the bottom row. For the computational postprocessing, the line integral convolution technique in ParaView has been employed [94]. Therefore, we compare streamlines on the symmetry plane where measurements were taken. Two important phenomena are captured in these simulations. First, the pressure drop because of the cavity results in a circulation beyond a threshold value between $Re = 6 - 16$, and this phenomenon is visible by comparing streamlines of cases (b) and (c). Second, the circulation center moves from left to right with an increasing REYNOLDS number, see streamlines for cases (c) to (h). Both phenomena are predicted by simulations; hence, we conclude that, quali-

Fig. 6 Comparison of measurement and calculations: Case (a) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

$v_{\max} = v_1(-0.1L, 0.625L, 0)$, calculated from the experimental and computational results.

On all walls, the velocity is assumed to vanish—simply the no-slip and no-penetration condition is applied by setting zero velocities—throughout the simulation. At the outlet, the reference pressure of 1 bar is set as a DIRICHLET boundary condition. Initially, zero velocity and reference pressure are defined in the whole domain.

Figures 6, 7, 8, 9, 10, 11, 12, and 13 show side-by-side comparison of measurements and calculations at the symmetry plane of the flow cavity corresponding to the flows in Fig. 3.

tatively, the agreement is adequate for all REYNOLDS numbers. We stress that the chosen time values are slightly changing the streamlines. Therefore, by choosing the instants we introduce an uncertainty to the comparison. We also denote that in higher REYNOLDS numbers even if the instant is chosen where the maximum velocity of the computation is lower than the experimental, plots show that the velocity is overestimated in computations. The exact location of the circulation center is not captured precisely, we repeat that the boundary conditions are set to increase the inlet velocity slowly so the

Fig. 7 Comparison of measurement and calculations: Case (b) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

Fig. 8 Comparison of measurement and calculations: Case (c) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

simulation represents the steady state, and this consideration may lead to inaccuracies. In order to check this assumption, another simulation has been done where the maximum $Re = 600$ is reached in 1800 s instead of 600 s. (In the same machine with same amount of cores,

this simulation takes 81 h.) After choosing appropriate time values, we obtain no significant deviations to the presented results. Therefore, we conclude that no-slip boundary conditions may be too restrictive for accurately modeling the experimental results for slow

Fig. 9 Comparison of measurement and calculations: Case (d) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

Fig. 10 Comparison of measurement and calculations: Case (e) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

speeds. Additionally, we emphasize that the 3-D flow profile is sophisticated, for showing a better insight, and we refer to Fig. 14 presenting streamlines on a cut plane orthogonal to the symmetry plane.

The pressure distribution from the computation is reasonable—despite the fact that the mesh is coarse—providing a numerical robustness difficult to obtain in an FEM computation.

Fig. 11 Comparison of measurement and calculations: Case (f) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

Fig. 12 Comparison of measurement and calculations: Case (g) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

For a quantitative comparison of the results, we demonstrate in Fig. 15 velocity profiles as the REYNOLDS number increases.

Specifically, along selected lines along y at $x/L = \{0.2, 0.5, 0.8\}$, velocities have been extracted from

the PIV data and directly compared to the computational results. For postprocessing the computational results, velocity profiles are normalized by using the maximum velocity, $v_{\max}$, given in Table 2 in order to scale all the results in a single plot. The agreement provides a measure

Fig. 13 Comparison of measurement and calculations: Case (h) PIV left column and CFD right column. Selected velocity profiles (top row) and streamlines (bottom row) are presented

Fig. 14 Computational results are presented for case g, in color the pressure distribution and streamlines emanating at the central plane of the inlet tube

for reliability of the computation within the great range of REYNOLDS numbers presented herein. Especially, in low REYNOLDS numbers, the computed velocity profile is slightly higher than the experimental profile. This result relies on the chosen instant indicating a higher REYNOLDS number as well, see Table 2. In high REYNOLDS numbers, the computation overestimates the experimental results, we stress that experimental results are obtained by evaluating steady flows in the cavity. At REYNOLDS numbers beyond 650, the highest considered here, the flow in the cavity becomes unsteady, and turbulent at even higher REYNOLDS

number. Even the case of $Re = 650$ discussed here shows occasional signs of unsteadiness. The quantitative agreement at $x/L = 0.5$ demonstrates that the computational results approximate the motion of the recirculation center appropriately. However, the exact position of the recirculation center is not captured leading to a less accurate agreement at $x/L = 0.2$ and $x/L = 0.8$. This issue is partly related to the chosen time values for the evaluation. Computational implementation from [32] has been used to obtain all the results. By selecting the numerical solver with the aforementioned settings, a transient computation of the nonlinear and coupled system has been performed without any numerical instabilities. The flow formation phenomena have been correctly predicted such as recirculation starting beyond a threshold velocity as well as its center moving from left to right. The exact location of the recirculation center is not detected by the computation. For judging the reliability of the numerical implementation, the overall agreement is adequate.

5 Closing remarks

An experimental validation of the viscous fluid flow computation has been presented in order to assess the reliability of a new approach or implementation. The setup configuration is designed for reducing the comparison to a (symmetry) plane, in which a flow pattern formation is observed in the steady state for REYNOLDS numbers of a

Fig. 15 Comparison of flow profiles at increasing REYNOLDS numbers: measurements and calculations at the symmetry plane, along $x/L = 0.2$ (top), $x/L = 0.5$ (middle), and $x/L = 0.8$ (bottom). Veloc-

ity profiles are normalized by $v_{\max}$ for corresponding REYNOLDS numbers, Re , of cases (a)–(h) given in Table 2

sizeable range. By using this setup, validating a numerical implementation is simple; we have demonstrated such a procedure by using a numerical implementation based on the finite element method. For a qualitative analysis, streamline patterns are examined; for a quantitative verification, velocity profiles are compared along selected lines

on the symmetry plane. Geometry and configuration are given in detail, and all experimental results and also its computational implementation used herein are publicly available in [21] to be used under GNU public license [45]. We hope that the data presented here may prove useful for testing new computational ideas.

Acknowledgements Open Access funding provided by Projekt DEAL. B. E. Abali's work is partly funded by a grant from the Daimler and Benz Foundation.

Compliance with ethical standards

Conflict of interest On behalf of all authors, the corresponding author states that there is no conflict of interest.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit <http://creativecommons.org/licenses/by/4.0/>.

References

- Vakhrushev A, Wu M, Ludwig A, Nitzl G, Tang Y, Hackl G, Wincor R (2017) A water experiment benchmark to evaluate numerical models for the motion of particles in continuous casting tundish. *Steel Res Int* 88(1):1600276
- Yanxia F, Yuan J, Yuan S, Pace G, d'Agostino L, Huang P, Li X (2015) Numerical and experimental analysis of flow phenomena in a centrifugal pump operating under low flow rates. *J Fluids Eng* 137(1):011102
- Sigüenza J, Pott DP, Mendez S, Sonntag SJ, Kaufmann TAS, Steinseifer U, Nicoud F (2018) Fluid-structure interaction of a pulsatile flow with an aortic valve model: a combined experimental and numerical study. *Int J Numer Methods Biomed Eng* 34(4):e2945
- Arzani A (2018) Accounting for residence-time in blood rheology models: do we really need non-newtonian blood flow modelling in large arteries? *J R Soc Interface* 15(146):20180486
- Sarghini F, Romano A, Masi P (2016) Experimental analysis and numerical simulation of pasta dough extrusion process. *J Food Eng* 176:56–70
- Viswanath A, Manu MV, Savithri S (2017) Numerical simulation and experimental validation of free surface flows during low pressure casting process. *J Mater Process Technol* 244:320–330
- Wang Z, Huang B, Zhang M, Wang G, Zhao X (2018) Experimental and numerical investigation of ventilated cavitating flow structures with special emphasis on vortex shedding dynamics. *Int J Multiphase Flow* 98:79–95
- Liang Y, Liu P, Zheng N, Shan F, Liu Z, Liu W (2019) Numerical investigation of heat transfer and flow characteristics of laminar flow in a tube with center-tapered wavy-tape insert. *Appl Therm Eng* 148:557–567
- Tiwari SS, Pal E, Bale S, Minocha N, Patwardhan AW, Nandakumar K, Joshi JB (2020) Flow past a single stationary sphere, 1. Experimental and numerical techniques. *Powder Technol* 365:115–148
- D'Bastiani C, Alba JL, Mazzarotto GT, de Farias Neto SR, Beal LL, Reynolds A, Kennedy D (2019) Multiphase cfd modelling and piv validation of a uasb reactor. In: AIP conference proceedings, vol 2116, pp 030007. AIP Publishing
- Morton KW (2019) Revival: numerical solution of convection–diffusion problems (1996). CRC Press, Boca Raton
- Schäfer M, Turek S, Durst F, Krause E, Rannacher R (1996) Benchmark computations of laminar flow around a cylinder. In: *Flow simulation with high-performance computers II*, pp 547–566. Springer
- Turek S, Hron J, Madlik M, Razzaq M, Wobker H, Acker JF (2011) Numerical simulation and benchmarking of a monolithic multigrid solver for fluid-structure interaction problems with application to hemodynamics. In: *Fluid structure interaction II*, pp 193–220. Springer
- Ruiz AM, Lacaze G, Oefelein JC, Mari R, Cuenot B, Selle L, Poinot T (2015) Numerical benchmark for high-reynolds-number supercritical flows with large density gradients. *AIAA J* 54(5):1445–1460
- Boileau E, Nithiarasu P, Blanco PJ, Müller LO, Fossan FE, Hellevik LR, Donders WP, Huberts W, Willemet M, Alastruey J (2015) A benchmark study of numerical schemes for one-dimensional arterial blood flow modelling. *Int J Numer Methods Biomed Eng* 31(10):e02732
- Flemisch B, Berre I, Boon W, Fumagalli A, Schwenck N, Scotti A, Stefansson I, Tatomir A (2018) Benchmarks for single-phase flow in fractured porous media. *Adv Water Resour* 111:239–258
- Bouillot P, Brina O, Ouared R, Yilmaz H, Lovblad K-O, Farhat M, Pereira VM (2016) Computational fluid dynamics with stents: quantitative comparison with particle image velocimetry for three commercial off the shelf intracranial stents. *J Neurointerv Surg* 8(3):309–315
- Piro MHA, Wassermann F, Grundmann S, Tensuda B, Kim SJ, Christon M, Berndt M, Nishimura M, Tropea C (2017) Fluid flow investigations within a 37 element candu fuel bundle supported by magnetic resonance velocimetry and computational fluid dynamics. *Int J Heat Fluid Flow* 66:27–42
- Merzari E, Fischer P, Yuan H, Van Tichelen K, Keijers S, De Ridder J, Degroote J, Vierendeels J, Doolaard H, Gopala VR et al (2016) Benchmark exercise for fluid flow simulations in a liquid metal fast reactor fuel assembly. *Nucl Eng Des* 298:218–228
- Roussel N, Gram A, Cremonesi M, Ferrara L, Krenzer K, Mechtcherine V, Shyshko S, Skocec J, Spangenberg J, Svec O et al (2016) Numerical simulations of concrete flow: a benchmark comparison. *Cem Concr Res* 79:265–271
- Abali BE (2019) Supply code. <http://bilenemek.abali.org>. Accessed 1 Aug 2020
- Bale-Glickman J, Selby K, Saloner D, Savaş Ö (2003) Experimental flow studies in exact-replica phantoms of atherosclerotic carotid bifurcations under steady input conditions. *J Biomech Eng* 125(1):38–48
- Sturgeon V, Savaş Ö, Saloner D (2006) An experimental study of transitional behavior in physiological flow regimes. In: *ASME 2006 international mechanical engineering congress and exposition*. American Society of Mechanical Engineers, pp 259–268
- Tsai WW, Savaş Ö, Maitland D, Ortega J, Small W, Wilson TS, Saloner D (2006) Experimental study of the vascular dynamics of a saccular basilar aneurysm. In: *Proceedings of the IMECE2006 ASME international mechanical engineering congress and exposition*, 5–10 Nov 2006; Chicago, vol 4662
- Raffel M, Willert CE, Scarano F, Kähler CJ, Wereley ST, Kompenhans J (2018) Particle Image velocimetry: a practical guide. Springer
- Fahy P, McCarthy P, Sultan S, Hynes N, Delassus P, Morris L (2014) An experimental investigation of the hemodynamic variations due to aplastic vessels within three-dimensional phantom models of the circle of willis. *Ann Biomed Eng* 42(1):123–138
- Bucs SS, Linares RV, Marston JO, Radu AI, Vrouwenvelder JS, Picioreanu C (2015) Experimental and numerical

- characterization of the water flow in spacer-filled channels of spiral-wound membranes. *Water Res* 87:299–310
28. Zhu G, Wei Y, Yuan Q, Yang J, Yeo JH (2019) Piv investigation of the flow fields in subject-specific vertebro-basilar (va-ba) junction. *BioMed Eng Online* 18:93
 29. Dai Z, Zheng Z, Fletcher DF, Haynes BS (2015) Experimental study of transient behaviour of laminar flow in zigzag semi-circular microchannels. *Exp Therm Fluid Sci* 68:644–651
 30. Abed AH, Shcheklein SE (2018) Investigation of hydrodynamic characteristics of laminar flow condition around sphere using piv system. In: *Journal of Physics: conference series*, vol 1015, pp 032001. IOP Publishing
 31. Durhasan T, Pinar E, Ozkan GM, Aksoy MM, Akilli H, Sahin B (2018) Piv measurement downstream of perforated cylinder in deep water. *Eur J Mech B Fluids* 72:225–234
 32. Abali BE (2019) An accurate finite element method for the numerical solution of isothermal and incompressible flow of viscous fluid. *Fluids* 4(1):5
 33. Langtangen HP, Logg A (2016) Solving PDEs in python: the FEniCS tutorial I. Springer, Berlin
 34. Alexandre Joel Chorin (1969) On the convergence of discrete approximations to the Navier–Stokes equations. *Math Comput* 23(106):341–353
 35. J-L Guermond and Luigi Quartapelle (1998) On the approximation of the unsteady navier-stokes equations by finite element projection methods. *Numer Math* 80(2):207–238
 36. Gresho PM (1990) On the theory of semi-implicit projection methods for viscous incompressible flow and its implementation via a finite element method that also introduces a nearly consistent mass matrix. part 1: Theory. *Int J Numer Methods Fluids* 11(5):587–620
 37. Gresho PM, Chan ST (1990) On the theory of semi-implicit projection methods for viscous incompressible flow and its implementation via a finite element method that also introduces a nearly consistent mass matrix. part 2: Implementation. *Int J Numer Methods Fluids* 11(5):621–659
 38. Franca LP, Hughes TJR (1988) Two classes of mixed finite element methods. *Comput Methods Appl Mech Eng* 69(1):89–129
 39. Tezduyar TE (1991) Stabilized finite element formulations for incompressible flow computations. In: Hutchinson JW, Wu TY (eds) *Advances in applied mechanics*, vol 28. Elsevier, Amsterdam, pp 1–44
 40. Förster C, Wall WA, Ramm E (2007) Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. *Comput Methods Appl Mech Eng* 196(7):1278–1293
 41. Baiocchi C, Brezzi F (1993) Virtual bubbles and galerkin-least-squares type methods (ga. ls). *Comput Methods Appl Mech Eng* 105(1):125–141
 42. Matthies G, Tobiska L (2002) The inf-sup condition for the mapped $Q_k - P_{k-1}^{disc}$ element in arbitrary space dimensions. *Computing* 69(2):119–139
 43. Arnold D, Boffi D, Falk R (2002) Approximation by quadrilateral finite elements. *Math Comput* 71(239):909–922
 44. Brezzi F, Fortin M (2012) Mixed and hybrid finite element methods, vol 15. Springer, Berlin
 45. GNU Public (2007) Gnu general public license. <http://www.gnu.org/copyleft/gpl.html>. Accessed 1 Aug 2020
 46. Speziale CG (1991) Analytical methods for the development of Reynolds-stress closures in turbulence. *Annu Rev Fluid Mech* 23(1):107–157
 47. Tsai W, Savaş Ö (2010) Flow pumping system for physiological waveforms. *Med Biol Eng Comput* 48(2):197–201
 48. Sholl M, Savaş Ö (1997) A fast Lagrangian piv method for study of general high-gradient flows. In: *35th aerospace sciences meeting and exhibit*, p 493
 49. Tsuei L, Savaş Ö (2000) Treatment of interfaces in particle image velocimetry. *Exp Fluids* 29(3):203–214
 50. Darrigol O (2002) Between hydrodynamics and elasticity theory: the first five births of the Navier–Stokes equation. *Arch Hist Exact Sci* 56(2):95–150
 51. Abali BE (2017) Computational reality, solving nonlinear and coupled problems in continuum mechanics. In: *Advanced structured materials*, vol 55. Springer
 52. Donea J, Huerta A (2005) Finite element methods for flow problems. Wiley, Chichester
 53. Hoffman J, Johnson C (2007) Computational turbulent incompressible flow, applied mathematics: body and soul, vol 4. Springer, Berlin
 54. Blazek J (2015) Computational fluid dynamics: principles and applications. Butterworth-Heinemann, Oxford
 55. Morton KW (1982) Finite element methods for non-self-adjoint problems. In: Turner P (ed) *Topics in numerical analysis*, vol 965. Springer, Berlin, pp 113–148
 56. Hughes TJR, Brooks A (1982) A theoretical framework for Petrov–Galerkin methods with discontinuous weighting functions: application to the streamline-upwind procedure. *Finite Elem Fluids* 4:47–65
 57. Hughes TJR, Franca LP, Balestra M (1986) A new finite element formulation for computational fluid dynamics: circumventing the Babuska–Brezzi condition: a stable Petrov–Galerkin formulation of the Stokes problem accommodating equal-order Interpolation. *Comput Methods Appl Mech Eng* 59:85–99
 58. Tezduyar TE, Mittal S, Ray SE, Shih R (1992) Incompressible flow computations with stabilized bilinear and linear equal order interpolation velocity pressure elements. *Comput Methods Appl Mech Eng* 95:221–242
 59. Rannacher R (2004) Encyclopedia of computational mechanics, chapter 6 incompressible viscous flows, vol 3. Wiley, London, pp 155–181
 60. Hughes TJR, Scovazzi G, Franca LP (2004) Encyclopedia of computational mechanics, chapter 2 multiscale and stabilized methods, vol 3. Wiley, London, pp 5–59
 61. Gravemeier V, Kronbichler M, Gee MW, Wall WA (2011) An algebraic variational multiscale-multigrid method for large-eddy simulation: generalized- α time integration, fourier analysis and application to turbulent flow past a square-section cylinder. *Comput Mech* 47(2):217–233
 62. Codina R, Badia S, Baiges J, Principe J (2018) Variational multiscale methods in computational fluid dynamics. In: *Encyclopedia of computational mechanics*, 2nd edn, pp 1–28
 63. Jansson J, Degirmenci NC, Hoffman J (2017) Adaptive unified continuum fem modeling of a 3d fsi benchmark problem. *Int J Numer Methods Biomed Eng* 33(9):e2851
 64. Chen L, Asai K, Nonomura T, Xi G, Liu T (2018) A review of backward-facing step (bfs) flow mechanisms, heat transfer and control. *Therm Sci Eng Prog* 6:194–216
 65. Hansen KB, Arzani A, Shadden SC (2019) Finite element modeling of near-wall mass transport in cardiovascular flows. *Int J Numer Methods Biomed Eng* 35(1):e3148
 66. Schofield SP, Christon MA, Dyadechko V, Garimella RV, Lowrie RB, Swartz BK (2010) Multi-material incompressible flow simulation using the moment-of-fluid method. *Int J Numer Methods Fluids* 63(8):931–952
 67. Zanolini O, Fambri F, Dumbser M, Hidalgo A (2015) Space-time adaptive discontinuous galerkin finite element schemes with a posteriori sub-cell finite volume limiting. *Comput Fluids* 118:204–224
 68. Canales D, Leygue A, Chinesta F, González D, Cueto E, Feulvarch E, Bergheau J-M, Huerta A (2016) Vademecum-based gfem (v-gfem): optimal enrichment for transient problems. *Int J Numer Methods Eng* 108(9):971–989

69. Borker R, Farhat C, Tezaur R (2017) A discontinuous galerkin method with lagrange multipliers for spatially-dependent advection–diffusion problems. *Comput Methods Appl Mech Eng* 327:93–117
70. Larios A, Rebholz LG, Zervas C (2019) Global in time stability and accuracy of imex-fem data assimilation schemes for Navier–Stokes equations. *Comput Methods Appl Mech Eng* 345:1077–1093
71. Gresho PM (1991) Incompressible fluid dynamics: some fundamental formulation issues. *Annu Rev Fluid Mech* 23(1):413–453
72. Abali BE, Müller WH, Georgievskii DV (2013) A discrete-mechanical approach for computation of three-dimensional flows. *ZAMM J Appl Math Mech* 93(12):868–881
73. Charnyi S, Heister T, Olshanskii MA, Rebholz LG (2017) On conservation laws of Navier–Stokes galerkin discretizations. *J Comput Phys* 337:289–308
74. John V, Knobloch P, Novo J (2017) Finite elements for scalar convection-dominated equations and incompressible flow problems: a never ending story. *WIAS Preprint*, 2410
75. Zhang B, Gu C, Dai Z (2010) An adaptive control strategy for proper mesh distribution in large eddy simulation. *J Hydrodyn Ser* 22(6):865–870
76. Elman HC, Silvester DJ, Wathen AJ (2014) Finite elements and fast iterative solvers: with applications in incompressible fluid dynamics. Oxford University Press, Oxford
77. Elman HC, Ramage A, Silvester DJ (2014) A computational laboratory for investigating incompressible flow problems. *SIAM Rev* 56(2):261–273
78. Silvester D et al (2019) Balanced iterative solvers for linear non-symmetric systems and nonlinear systems with pde origins: efficient black-box stopping criteria. *J Sci Comput* 81(1):271–290
79. Shuangzhang T, Aliabadi S (2007) Development of a hybrid finite volume/element solver for incompressible flows. *Int J Numer Methods Fluids* 55(2):177–203
80. Calgaro C, Creusé E, Goudon T (2008) An hybrid finite volume-finite element method for variable density incompressible flows. *J Comput Phys* 227(9):4671–4696
81. Wan T, Aliabadi S, Bigler C (2009) A hybrid scheme based on finite element/volume methods for two immiscible fluid flows. *Int J Numer Methods Fluids* 61(8):930–944
82. Christon MA (2011) Hydra-th theory manual. Los Alamos National Laboratory, Los Alamos
83. Rosi G, Giorgio I, Eremeyev VA (2013) Propagation of linear compression waves through plane interfacial layers and mass adsorption in second gradient fluids. *ZAMM J Appl Math Mech-Mathematik und Mechanik* 93(12):914–927
84. Tan H, Pillai KM (2009) Finite element implementation of stress-jump and stress-continuity conditions at porous-medium, clear-fluid interface. *Comput Fluids* 38(6):1118–1131
85. Yu P, Zeng Y, Lee TS, Bai HX, Low HT (2010) Wake structure for flow past and through a porous square cylinder. *Int J Heat Fluid Flow* 31(2):141–153
86. Schofield SP, Christon MA (2012) Effects of element order and interface reconstruction in fem/volume-of-fluid incompressible flow simulation. *Int J Numer Meth Fluids* 68(11):1422–1437
87. Zohdi TI (2018) Finite element primer for beginners. Springer, Berlin
88. Babuska I, Rheinboldt WC (1978) Error estimates for adaptive finite element computations. *SIAM J Numer Anal* 15(4):736–754
89. Stern F, Wilson RV, Coleman HW, Paterson EG (2001) Comprehensive approach to verification and validation of cfd simulations—part 1: methodology and procedures. *J Fluids Eng* 123(4):793–802
90. Balay S, Abhyankar S, Adams M, Brown J, Brune P, Buschelman K, Dalcin L, Dener A, Eijkhout V, Gropp W (2019) Petsc users manual. Argonne National Laboratory
91. Chevalier C, Pellegrini F (2008) Pt-scotch: a tool for efficient parallel graph ordering. *Parallel Comput* 34(6–8):318–331
92. Ruge JW, Stüben K (1987) Algebraic multigrid. In: McCormick SF (ed) Multigrid methods. SIAM, pp 73–130
93. Richardson CN, Sime N, Wells GN (2019) Scalable computation of thermomechanical turbomachinery problems. *Finite Elem Anal Des* 155:32–42
94. Loring B, Karimabadi H, Rortershteyn V (2014) A screen space gpgpu surface lic algorithm for distributed memory data parallel sort last rendering infrastructures. Technical report, Lawrence Berkeley National Lab. (LBNL), Berkeley, CA (United States)

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.