Benchmark Computations of 3D Laminar Flow Around a Cylinder with CFX, OpenFOAM and FeatFlow

E. Bayraktar*, O. Mierka and S. Turek

Institute of Applied Mathematics (LS III), TU Dortmund
Vogelpothsweg 87, D-44227, Dortmund, Germany.

E-mail: `ebayrakt@math.uni-dortmund.de`

Abstract

Numerically challenging, comprehensive benchmark cases are of great importance for researchers in the field of CFD. Numerical benchmark cases offer researchers frameworks to quantitatively explore limits of the computational tools and to validate them. Therefore, we focus on simulation of numerically challenging benchmark tests, laminar and transient 3D flows around a cylinder, and aim to establish a new comprehensive benchmark case by doing direct numerical simulations with three distinct CFD software packages. Although the underlying benchmark problems have been defined firstly in 1996, the first case which was a steady simulation of flow around a cylinder at $Re = 20$ could be accurately solved first in 2002 by John. Moreover, there is no precisely determined results for non-stationary case, the simulation of transient flow with time varying Reynolds number. The benchmark problems are studied with three CFD software packages, OpenFOAM, Ansys-CFX and FeatFlow which employ different numerical approaches to the discretization of the incompressible Navier-Stokes equations, namely finite volume method, element based finite volume method and finite element method respectively. The first benchmark test is considered as the "necessary condition" for the software tools, then they are compared according to their accuracy and performance in the second benchmark test. All the software tools successfully pass the first test and show well agreeing results for the second case such that the benchmark result was precisely determined. As a main result, the CFD software package with high order finite element approximation has been found to be computationally more efficient and accurate than the ones adopting low order space discretization methods.

*Corresponding author

1. Introduction

Despite all the developments in computer technology and in the field of numerics, the numerical solution of incompressible Navier-Stokes equations is still very challenging. Efficient and accurate simulation of incompressible flows is very important and prerequisite for the simulation of more complex applications, for instance, polymerization, crystallization or mixing phenomena. Numerous open-source or commercial CFD software packages are available to study these complex applications. Nonetheless, it is often observed that some of these software tools which produce colorful pictures as results of the simulations, fail in some benchmark tests subject to laminar flow around obstacles. Therefore, we are motivated to quantitatively compare performances of well known software packages by studying benchmark problems for laminar flow in 3D.

A set of benchmark problems had been defined within a DFG High-Priority Research Program by Schäfer & Turek [1] and these test cases have been studied by many researchers [2, 3]. One of the most studied 3D cases from the mentioned study is the flow around a cylinder with Reynolds number being 20. Although it is a low Reynolds number with steady solution problem and it had been formulated many years ago, the exact solutions could have been determined first by John in 2002 [3] and later by Braack & Richter [2]. Regarding existence of the very precise results for this benchmark test, we expect the employed software tools to accurately reproduce the results, which is considered as necessary condition for the software packages to continue with solving a second benchmark test for higher Reynolds numbers.

The second benchmark problem is unsteady and corresponds to a time-varying Reynolds number. There are not many studies on this benchmark test, one of the most recent studies is from 2005 by John [4]. However, in John's study the benchmark computation is performed to verify the developed methodology and software rather than improving the benchmark results, and his study is focused on the numerical approaches to the solution of incompressible Navier-Stokes equations. In this study, we aim to establish a new comprehensive benchmark case by doing direct numerical simulations with three distinct CFD software packages.

The benchmark problems are studied with the open-source software package OpenFOAM, the widely used commercial code ANSYS-CFX (CFX) and our in-house code FeatFlow. OpenFOAM (version 1.6) is a C++ library used primarily to crate executables, known as applications. The applications fall into two categories: solvers that are each designed to solve a specific problem in continuum mechanics; and utilities that are designed to perform tasks that involve data manipulation (see *http://www.openfoam.com*). From the available solvers, icoFoam which is designed to solve the incompressible Navier-Stokes equations with a Finite Volume approach, is employed. CFX (version 12.0 Service Pack 1) is a commercial general purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems with the element based Finite Volume Method.

The transient solver of CFX is employed for the benchmark computations. FeatFlow is an open source, multipurpose CFD software package which was firstly developed as a part of the FEAT project by Stefan Turek at the University of Heidelberg in beginning of the 1990s based on the Fortran77 finite element packages FEAT2D and FEAT3D (see *http://www.featflow.de*). FeatFlow is both a user oriented as well as general purpose subroutine system which uses the finite element method (FEM) to treat generalized unstructured quadrilateral (in 2-D) and hexahedral (in 3-D) meshes.

Studying benchmark problem with these three different software packages which employ different numerical techniques also give an insight to answer of the following questions:

1. Can one construct an efficient solver for incompressible flow without employing multigrid components, at least for the pressure Poisson equation?
2. What is the "best" strategy for time stepping: fully coupled iteration or operator splitting (pressure correction scheme)?
3. Does it pay to use higher order discretization in space or time?

These questions are considered to be crucial in the construction of efficient and reliable solvers, particularly in 3D. Every researcher who is involved in developing fast, accurate and efficient flow solvers should be interested. The authors had put their honest effort to obtain the most accurate results with the most optimal settings for all the codes; nevertheless, this benchmark study is especially meant to motivate future works by other research groups and the presented results are opened to discussion.

The paper continues with the benchmark configuration and the definition of comparison criteria. In Section 3, the software packages and the employed numerical approaches are described. The results are presented within the subsequent section and the paper is concluded with a discussion of the results.

2. Benchmark Configurations

The solvers are tested in two benchmark configurations with an incompressible Newtonian fluid whose kinematic viscosity (ν) is equal to 10^{-3} m^2/s and for which the conservation equations of mass and momentum are written as follows,

$$\begin{aligned} &\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nu \Delta \mathbf{u}, \\ &\nabla \cdot \mathbf{u} = 0 \end{aligned} \tag{1}$$

The Reynolds number is defined as $Re = UD/\nu$ where U is the mean velocity of the imposed parabolic profile on the inflow boundary and D is the diameter of the cylinder. The benchmark geometry and the corresponding 2D mesh at the coarsest level are shown in Figure 1. The 3D mesh is obtained by extruding the 2D mesh in the z direction with 4 layers of cells, however the first level mesh for the computations is obtained by two successive refinements via connecting opposite midpoints of the the coarsest mesh which yields a mesh of 6144 cells, see Figure 2. Our preliminary studies showed that this mesh offers a good balance between accuracy and computational cost.

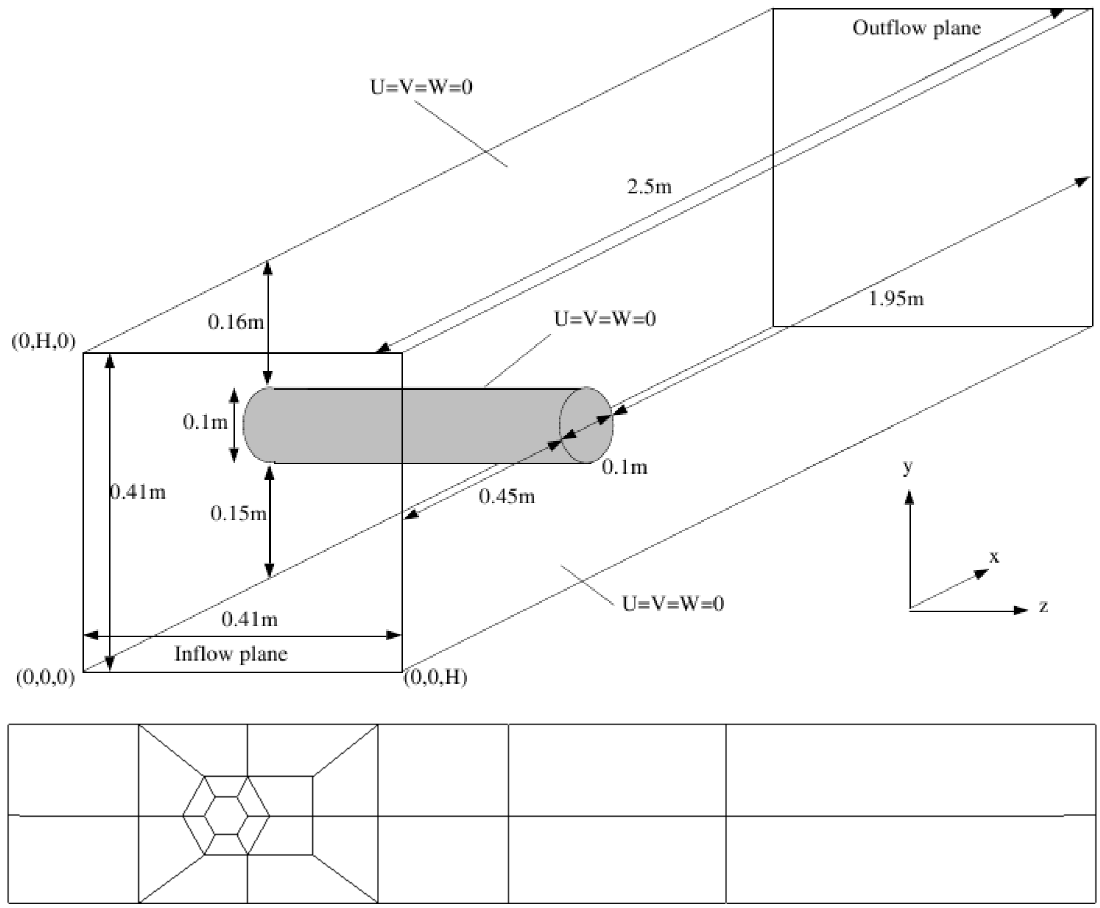

Figure 1: The geometry and the coarsest level mesh (in 2D).

The software tools employ different numerical approaches to the discretization in space which leads to different numbers of degrees of freedom (DOF) for the same problem. In the case of OpenFOAM and CFX, the numbers of DOF are comparable while FeatFlow has a greater number of DOF for the same mesh due to a high order finite element approximation. Therefore, while comparing the results, reader should keep in mind that for the same computational mesh, FeatFlow has approximately as many DOF as the others have at one level finer grid. The number of DOF is always proportional to number of cells (equivalent to number of vertices for hexahedral meshes with large number of cells) in all CFD packages. The numbers of DOF are presented with respect to the number of

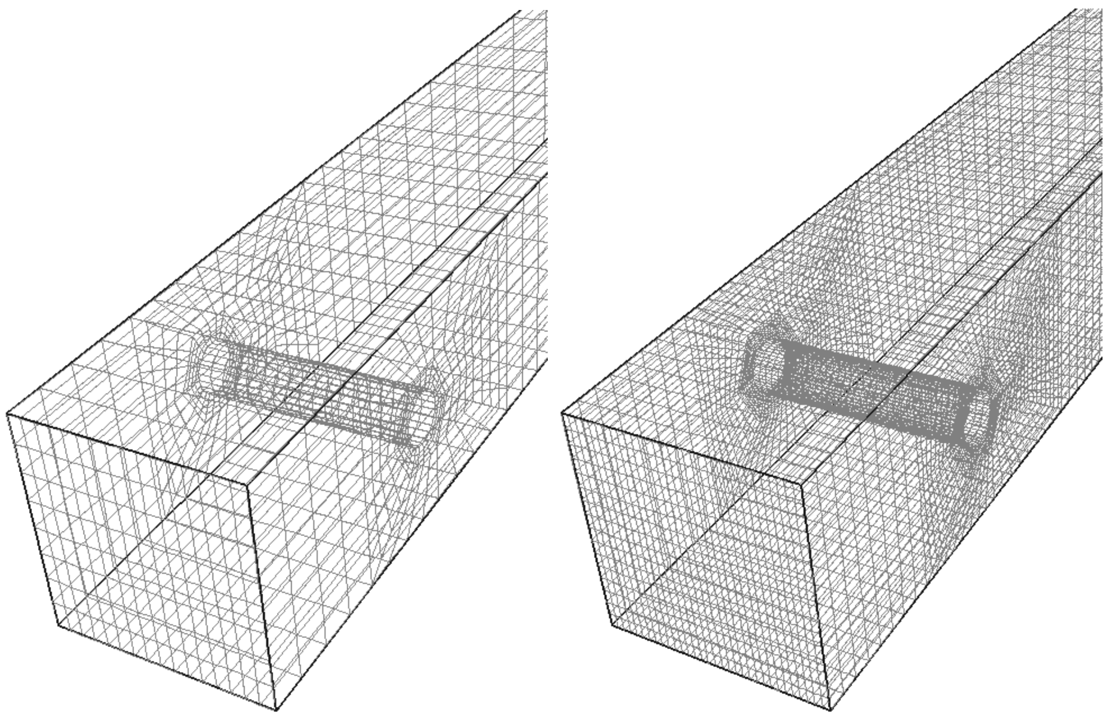

Figure 2: The first and second level meshes, with 6144 and 49152 cells respectively.

cells of the corresponding mesh levels in Table 1.

- $ndof = 3 \cdot nvt$ for velocity and $ndof = nvt$ for pressure in case of CFX.
- $ndof = 3 \cdot nel$ for velocity and $ndof = nel$ for pressure in case of OpenFOAM.
- $ndof = 24{\cdot}nvt$ for velocity and $ndof = 4{\cdot}nel$ for pressure in case of FeatFlow.

  where $ndof$, nvt and nel denote the number of DOF, the number of vertices and the number of cells, respectively.

The first benchmark problem is at $Re = 20$ and has been studied numerically by many research groups, and very accurate results have been presented. The second benchmark problem is unsteady, with a time-varying inflow profile which is very challenging and has not been rigorously studied, consequently the results have not yet been precisely determined. For the benchmark problems, no-slip boundary condition is employed on the walls and natural do-nothing boundary conditions are imposed at the outflow plane. The inflow conditions are set with $U_m = 0.45m/s$ and $U_m = 2.25m/s$ for the first and the second benchmark problem with the parabolic velocity profiles from equations (2) and (3), respectively,

$$U(0, y, z) = 16U_m yz(H - y)(H - z)/H^4, \quad V = W = 0 \tag{2}$$

Table 1: Number of unknowns.

Levels	# of Cells	Software	# of DOF u	# of DOF P	total # of DOF
		CFX	21828	7276	29104
L1	6144	OF	18423	6144	24567
		FeatFlow	174624	24576	199200
		CFX	160776	53592	214368
L2	49152	OF	147456	49152	196608
		FeatFlow	1286208	196608	1482816
		CFX	1232400	410800	1643200
L3	393216	OF	1179648	393216	1572864
		FeatFlow	9859200	1572864	11432064
		CFX	9647136	3215712	12862848
L4	3145728	OF	9437184	3145728	12582912
		FeatFlow	77177104	12582912	89760016

$$U(0,y,z) = 16U_m yz sin(\pi t/8)(H-y)(H-z)/H^4, \quad V = W = 0 \tag{3}$$

In the benchmark calculations, the dimensionless drag and lift coefficients for the cylinder are regarded as the comparison criteria which are calculated according to (4).

$$c_D = \frac{2F_w}{\rho U^2 DH}, \qquad c_L = \frac{2F_L}{\rho U^2 DH} \tag{4}$$

where F_D and F_L are defined as,

$$F_D = \int \left(\rho\nu \frac{\partial v_t}{\partial n} n_y - p n_x \right) dS, \qquad F_L = -\int \left(\rho\nu \frac{\partial v_t}{\partial n} n_x - p n_y \right) dS \tag{5}$$

with the following notations: surface of cylinder S, normal vector n on S with $x-$ and $y-$ component n_x and n_y, tangential velocity v_t on S and tangent vector $t = (n_y, -n_x, 0)$.

3. Used CFD Software Packages

We studied the benchmark problems with three software packages: open-source software package OpenFOAM (OF), commercial code ANSYS-CFX (CFX), and our in-house code FeatFlow. Each software adopts a different approach to the discretization of the

Navier-Stokes equations. The first adopts a conventional Finite Volume Method (FVM), ANSYS-CFX is developed on an element based FVM (ebFVM) and the last adopts a high order Galerkin Finite Element approach. Their approaches to the discretization of Navier-Stokes equations are one of the main differences between the codes. Moreover, their solution approach to the resulting linear and non-linear systems of equations after discretization is another distinction which influences the computational performance rather than the accuracy of the results. Nevertheless, in the case of our in-house code, the space discretization method is also effective regarding the performance of the solvers, the finite element spaces were chosen such that the linear solvers would be the most efficient.

Table 2: Discretization schemes and solver parameters for OF.

Type	**Parameters**	**Value**	**Description**
Time schemes	ddtSchemes	CrankNicholson 0.5	Euler blended (0.5) Crank-Nicholson, improved stability.
Interpolation schemes	interpolation-Schemes	linear	Linear interpolation (central differencing).
Surface normal gradientschemes	snGradSchemes	corrected	Explicit non-orthogonal correction.
Gradient schemes Divergence schemes	gradSchemes divSchemes	Gauss linear	Second order, Gaussian integration.
Laplacian schemes	laplacianSchemes	Gauss linear corrected	Unbounded, second order, conservative.
solvers p	solver smoother tolerance	GAMG Gauss-Siedel 10^{-7}	Geometric-Algebraic Multi-grid with Gauss-Siedel smoother. Convergence criteria.(*)
solvers U	Solver preconditioner tolerance	PBICG DILU 10^{-6}	Preconditioned bi-conjugate gradient solver with diagonal based incomplete LU preconditioner. (*)

($L_2 norm$ of residual normalized by $L_2 norm$ of solution, convergence criterion for the linear solver [6].)

OpenFOAM achieves the spatial discretization by using FVM on block structured meshes with Gaussian integration and linear interpolation. From the available techniques, temporal discretization is obtained with equidistant implicit Euler blended Crank-Nicolson time stepping scheme (blending factor = 0.5). Later, pressure and momentum equations are decoupled by using the PISO algorithm [5]. For the solution of the momentum equation, PBiCG is employed. PBiCG is a preconditioned bi-conjugate gradient solver for asymmetric lduMatrices using a run-time selectable preconditioner. DILU preconditioner which is a simplified diagonal-based incomplete LU preconditioner for asymmetric matrices was chosen. Then, the pressure equation is solved by using a Geometric-Algebraic Multigrid solver with a Gauss-Seidel type smoother. The details of the chosen discretization schemes and solver tolerances are presented in Table 2.

CFX employs an element based Finite Volume approach to discretize in space and high resolution scheme is chosen for the stabilization of the convective term. Time discretization is achieved by Second Order Backward Euler scheme. Tri-linear finite element based functions are used as interpolation scheme. ANSYS CFX uses a coupled solver, which solves the hydrodynamic equations (for u, v, w, p) as a single system. First, non-linear equations are linearized (coefficient iteration), then these linear equations are solved by an Algebraic Multigrid (AMG) solver. The chosen discretization schemes and solver parameters are given in Table 3.

Table 3: Discretization schemes and solver parameters for CFX.

Setting	**Value**	**Description**
Transient scheme	Second Order Backward Euler scheme	Second order, unbounded, implicit, conservative time stepping.
Interpolation scheme Pressure interpolation Velocity interpolation Shape function	FE shape functions Linear-Linear Trilinear Parametric	True tri-linear interpolation for velocity and linear-linear interpolation for pressure.
Advection scheme	High resolution	Numerical advection scheme with a calculated blending factor.
Convergence criteria Residual type Residual target	 MAX $5x10^{-}5$	Maximum value of normalized residuals. For details, see [7].

Our in-house developed open source fluid dynamics software package FeatFlow (here: module PP3D) is a transient 3D Finite Element based code parallelized on the basis of domain decomposition techniques. Velocity and pressure are discretized with the high order Q2/P1 element pair and their solution is obtained via a discrete projection method. Since this pair of elements is quite stable even in case of moderate Reynolds number flows, no additional stabilization of the advection is required which is very distinctive regarding the other software packages which employ certain stabilization schemes. The discretization in time is achieved by the second order Crank-Nicholson method which provides efficient and accurate marching in time together with the implemented adaptive time-step-control. The adopted discrete projection approach gives rise to a subsequent solution of the so called Burgers equations and the Pressure-Poisson equation, which is solved in order to enforce the incompressibility constraint. Both of these equations are solved with geometric multigrid solvers: here, SSOR/SOR and UMFPACK/SOR are employed as the solver/smoother pairs for velocity and pressure, respectively.

4. Results

The benchmark problems studied with all the codes, and while the problems were being studied, the question was not: "Is the software tool capable of solving the problem?" but "How accurate and efficient is the given software tool?". Therefore, to show the accuracy of the employed CFD softwares, the benchmark problem at $Re = 20$ is firstly studied. The results for this case obtained for 4 different mesh levels are given in Table (4).

Table 4: Results for steady test case (Re=20), reference values: $c_D = 6.18533$ and $c_L = 0.009401$ [2].

# of Cells	**Software**	c_D	c_L	**%Err** c_D	**%Err** c_L
L1	CFX	6.06750	0.01255	1.91	33
6144	OF	6.13408	0.01734	0.83	84
	FeatFlow	6.13973	0.00956	0.74	1.8
L2	CFX	6.13453	0.00817	0.82	14
49152	OF	6.19702	0.01099	0.19	17
	FeatFlow	6.17433	0.009381	0.18	0.21
L3	CFX	6.17481	0.00928	0.17	1.3
393216	OF	6.19362	0.01001	0.13	6.5
	FeatFlow	6.18260	0.009387	0.04	0.15
L4	CFX	6.18287	0.009387	0.04	0.15
3145728	OF	6.18931	0.00973	0.06	3.5
	FeatFlow	6.18465	0.009397	0.01	0.05

The qualitative results were indistinguishable even for the coarsest level calculations for the $Re = 20$ test case, thus only a sample snapshot of the results which has been obtained on the finest grid by FeatFlow is given in Figure (3).

The expected agreement of the results for the first benchmark test motivated us to go on with the second benchmark problem which is the challenging part of our benchmarking studies. The second benchmark problem has a fixed simulation time, $T = 8$ s, whereas the first benchmark problem is simulated towards the steady state solution. There is no precise unique definition of the stopping criteria for all the softwares for the first case. Therefore, while the software tools are compared with respect to their accuracy in both cases, their computational performance has been tested in the later case, as well. The coarse grid computations were performed sequentially and fine grid ones were done in parallel, based on domain decomposition method. Both sequential and parallel computations were performed on identical compute nodes: Dual-core AMD Opteron™ Processor 250 2.4 GHz with 8 GB total memory. To decrease the latency time due to memory bandwidth limitation each partition is submitted to one node. The nodes were interconnected via 1GHz ethernet connection, regarding the number of nodes it is

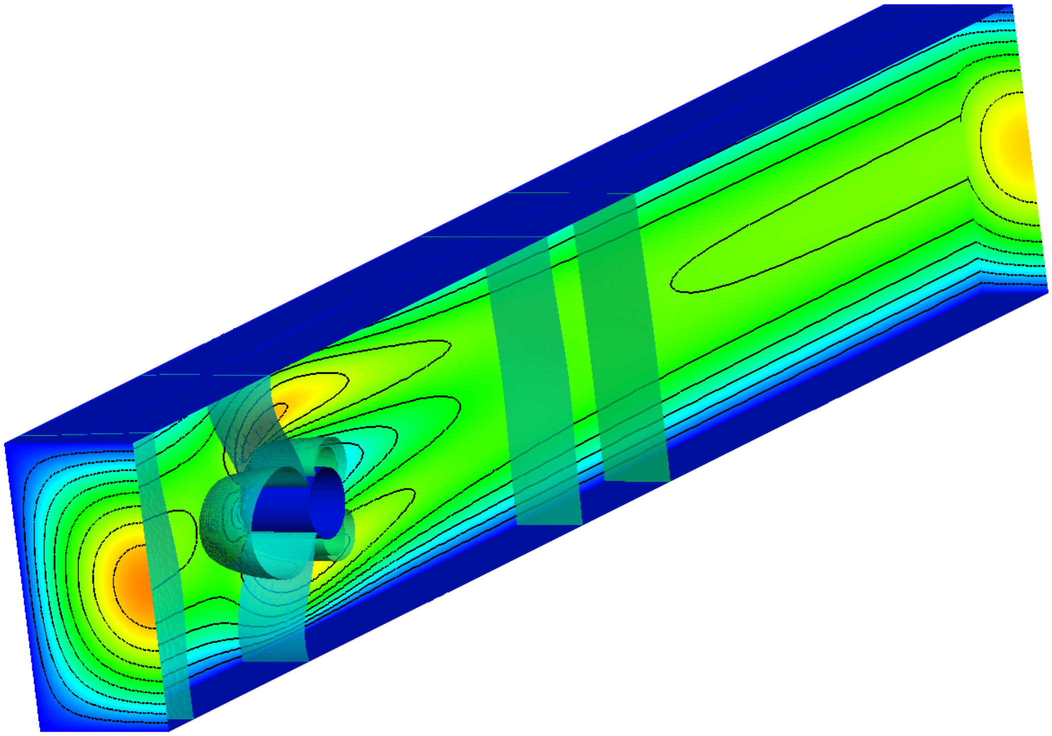

Figure 3: Snapshot for the $Re = 20$ test case with 3145728 cells.

a sufficiently fast connection. However, it has to be mentioned that the performance of the software packages will increase for higher number of partitions by using infiniband connected nodes.

In the second benchmark test, the flow is simulated for 8 seconds, a half period of the imposed inflow condition. The simulations start with zero inflow and zero initial condition at $t = 0$ s and finish at $t = 8$ s with zero inflow again, see Equation (3). Due to the transient inflow condition, adaptive time stepping technique is a good candidate for this problem. However, in preliminary studies with this technique, numerical oscillations are observed in the results which are obtained by CFX and OpenFOAM. The oscillations were nor visible in qualitative results neither in the drag coefficient results. However, when the results of a sensitive variable such as c_L are plotted, the numerical oscillations appear, see Figure (4). The results are obtained on a mesh with 393216 cells by OpenFOAM for two different values of maximum Courant numbers, maxCo, and with different tolerance values of linear solvers. Tolerance values of the velocity solver, uTol, are set to 10^{-5} or 10^{-6} and tolerance values of the pressure solver, pTol, are set to 10^{-6} or 10^{-7}.

Therefore, a fixed time step size is used in the simulations although it leads to excess of computational costs. The benchmark calculations are performed on several levels of refined meshes to obtain mesh independent results and to show convergence of the solvers with respect to the mesh size. The chosen time step sizes are maximum values for which the solution is independent of the chosen time step size. The comparison criteria are

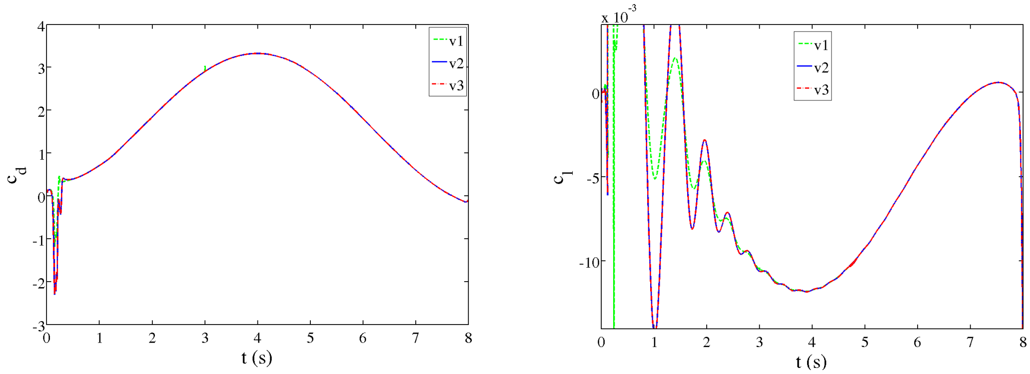

Figure 4: Numerical oscillations due to adaptive time stepping algorithm (v1: maxCo=0.25, uTol=10^{-5} pTol=10^{-6}; v2: maxCo=0.5, uTol=10^{-5} pTol=10^{-6}; v3: maxCo=0.5, uTol=10^{-6} pTol=10^{-7}).

maximum drag coefficient and minimum lift coefficient. Since the lift coefficient is more sensitive than the drag coefficient, the lift coefficient results are more representative in accuracy. When all the results are considered, it is clear that FeatFlow has the best convergence behavior, namely showing quadratic convergence, with respect to the mesh size. This result was foreseen by the authors due to the employed quadratic finite element functions. And converged results with respect to mesh size are already obtained on the 3rd level mesh by FeatFlow. In Figure 5, it is clearly shown that level 3 and level 4 results are identical and consequently the finest level results are considered as the reference through this study. Moreover, it is worth to mention that these results are the closest to one reported by John [4].

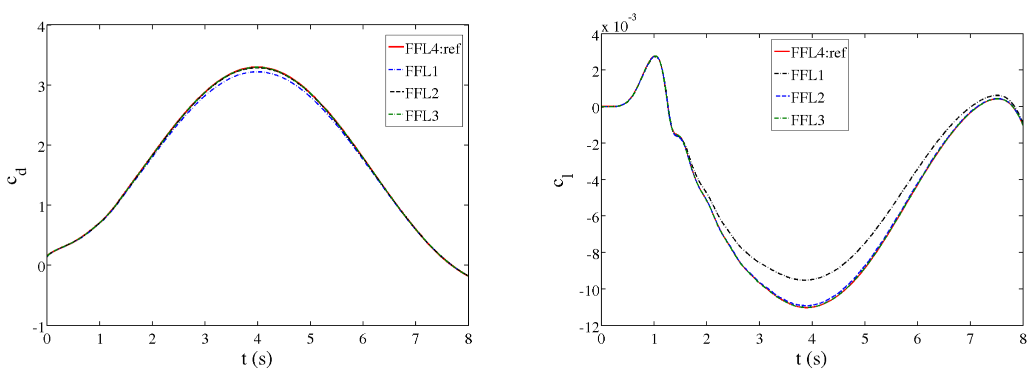

Figure 5: FeatFlow results for the second benchmark: drag coefficient (left), lift coefficient(right).

The old results for c_{Dmax} and c_{Lmin} had been given within the intervals, [3.2000, 3.3000] and [0.0020, 0.0040] respectively, by Turek et al. [1], and c_{Dmax} has been determined as 3.2968 by John [4]. These old results are not sufficient and accurate enough to establish an inclusive benchmark study. Besides, to evaluate the results of benchmark calculations with respect to only c_{Dmax} and c_{Lmin} is not much elucidating on the accuracy of the results. Hence, additional to the comparison of c_{Dmax} and c_{Lmin} values, we also compare the

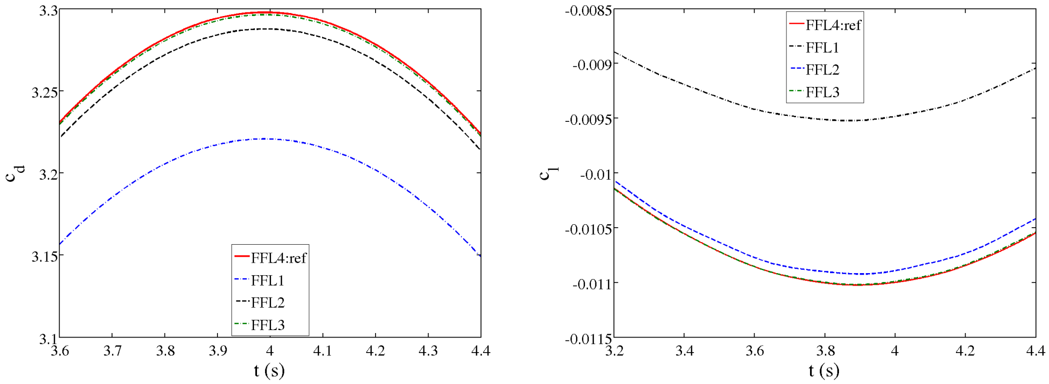

Figure 6: A closer look at the Figure 5: drag coefficient (left), lift coefficient(right).

solution regarding L_2Err and $L_\infty Err$ which denotes the L_2 *norm* of the error normalized by L_2 *norm* of the reference solution and the L_∞ *norm* of error, respectively, are obtained through the following calculation steps:

1. An equidistant discrete time step is specified for which L_2Err is independent of the step size.
2. The reference solution and the other solutions are interpolated (linear interpolation) to the specified discrete time step.
3. L_2 *norm* and L_∞ *norm* of the differences between reference solution and other solutions are calculated.

Results of the benchmark calculations are plotted in the Figures (5–9) and values of the comparison criteria, L_2Err, $L_\infty Err$ and relative errors due to the comparison criteria are given in Tables 5–10.

In Figure (11), we present the results of the benchmark calculations for the finest level, and it is obvious that the results are in agreement. An interesting finding is that the FeatFlow results at level 2 are as accurate as results of CFX or OpenFOAM at the finest level, see Table (11, 12).

Table 5: FeatFlow results. (*: clock time x number of nodes)

Case	# of Cells	c_{Dmax}	c_{Lmax}	c_{Lmin}	Tstep (s)	Time* (s)
FFL1	6144	3.2207	0.0027	-0.0095	0.010	3220 x 2
FFL2	49152	3.2877	0.0028	-0.010892	0.010	17300 x 4
FFL3	393216	3.2963	0.0028	-0.010992	0.010	35550 x 24
FFL4	3145728	3.2978	0.0028	-0.010999	0.005	214473 x 48

Table 6: Error calculations for FeatFlow results.

Case	%Err		$\%L_2Err$		$\%L_\infty Err$	
	c_{Dmax}	c_{Lmin}	c_D	c_L	c_D	c_L
FFL1	2.34	13.6	2.09	13.8	7.71	0.152
FFL2	0.31	0.91	0.29	1.05	1.01	0.013
FFL3	0.05	0.09	0.06	0.28	0.23	0.003

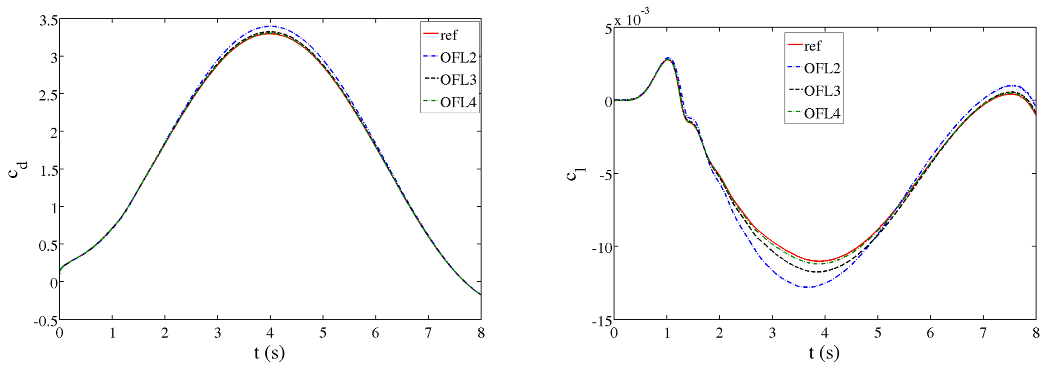

Figure 7: OpenFOAM results for the second benchmark: drag coefficient (left), lift coefficient (right).

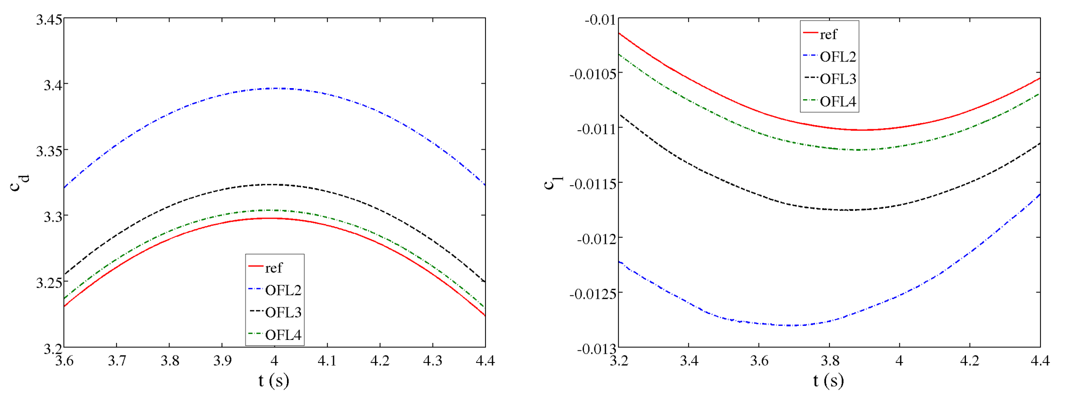

Figure 8: A closer look at the Figure 7: drag coefficient (left), lift coefficient(right).

Table 7: OpenFOAM results. (*: clock time x number of nodes)

Case	# of Cells	c_{Dmax}	c_{Lmax}	c_{Lmin}	Tstep (s)	Time* (s)
OFL2	49152	3.3963	0.0029	-0.0128	0.0025	4850
OFL3	393216	3.3233	0.0028	-0.0118	0.0010	76300 x 4
OFL4	3145728	3.3038	0.0028	-0.0112	0.0005	593500 x 24

5. Conclusions

The results obtained by this benchmark computations definitively replace the existing reference results for the second test case. FeatFlow results at mesh level 3 could be

Table 8: Error calculations for the OpenFOAM results.

Case	%Err		$\%L_2 Err$		$\%L_\infty Err$	
	c_{Dmax}	c_{Lmin}	c_D	c_L	c_D	c_L
OFL2	3.0	16	2.61	14.5	10.0	0.21
OFL3	0.8	7.3	0.67	5.91	2.56	0.08
OFL4	0.2	1.8	0.16	1.47	0.61	0.02

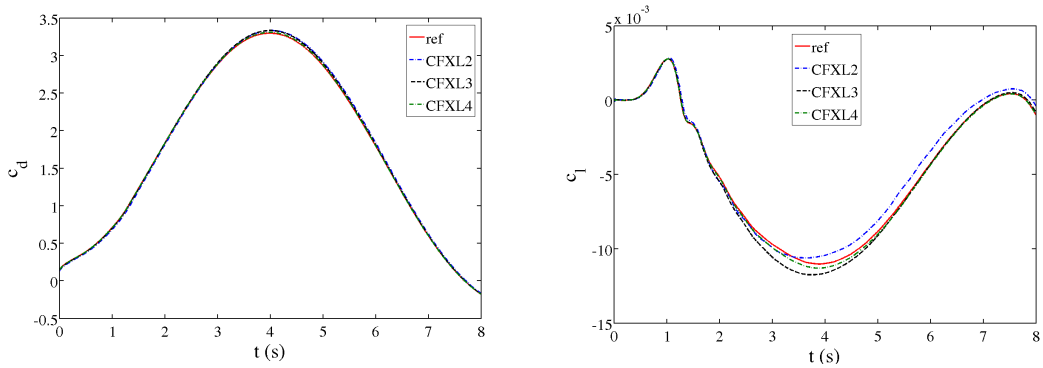

Figure 9: CFX results for the second benchmark: drag coefficient (left), lift coefficient(right).

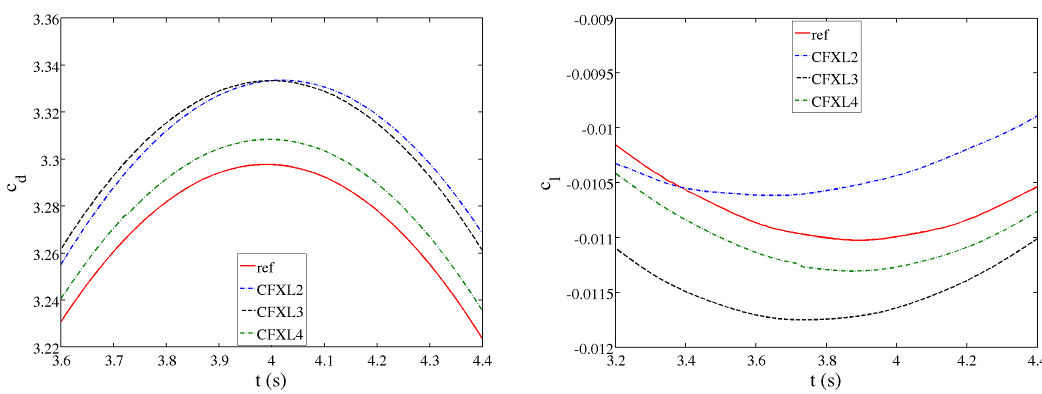

Figure 10: A closer look at the Figure 9: drag coefficient (left), lift coefficient(right).

Table 9: CFX results. (*: clock time x number of nodes)

Case	# of Cells	c_{Dmax}	c_{Lmax}	c_{Lmin}	Tstep (s)	Time* (s)
CFXL2	49152	3.3336	0.0028	-0.0106	0.010	22320
CFXL3	393216	3.3334	0.0028	-0.0118	0.005	61530 x 4
CFXL4	3145728	3.3084	0.0028	-0.0113	0.005	115300 x 24

already considered as mesh independent results and regarding the results obtained at mesh level 4, we can conclude that fully converged solution of the second benchmark test

Table 10: Error calculations for the CFX results.

Case	%Err		$\%L_2Err$		$\%L_\infty Err$	
	c_{Dmax}	c_{Lmin}	c_D	c_L	c_D	c_L
CFXL2	1.1	3.63	1.52	7.81	5.31	0.10
CFXL3	1.1	7.27	0.98	6.31	3.75	0.10
CFXL4	0.3	2.73	0.29	2.24	1.20	0.03

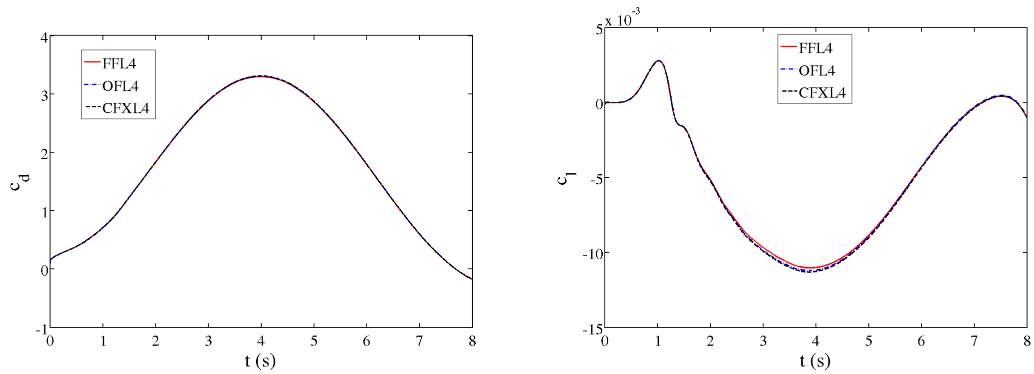

Figure 11: The finest level results for the second benchmark test: drag coefficient (left), lift coefficient(right).

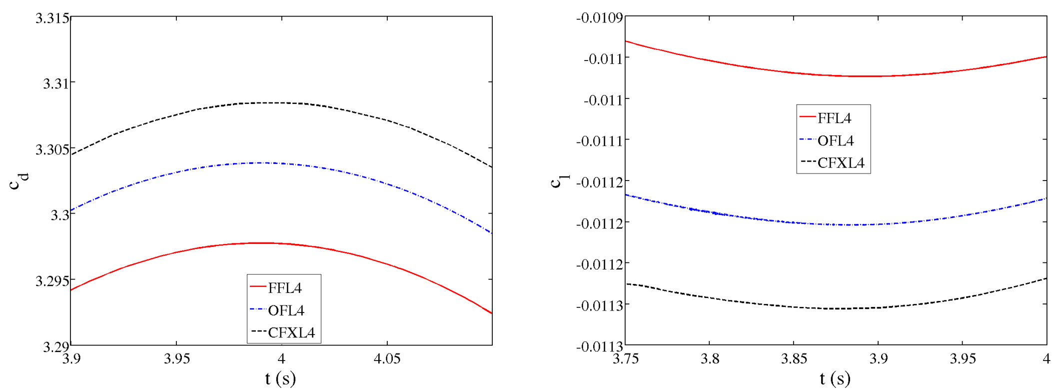

Figure 12: A closer look at the Figure 11: drag coefficient (left), lift coefficient(right).

is achieved, which has been the primary goal of the study. Moreover, we got an insight to answers of our questions in the beginning of the study with the obtained results.

1. We have observed that multigrid techniques slightly increase the performance of the segregated solvers in the solution of viscous Burger's equation due to the chosen small time steps. However, in the case of coupled solvers or the solution of the pressure-Poisson problem, there is a drastic difference between the performance of conventional (single grid) iterative methods and multigrid techniques. Consequently, it seems to be impossible to obtain efficient solvers for laminar incompressible flow problems unless suitable multigrid techniques are employed.

Table 11: The finest level results and FeatFlow results at level 2. (*: clock time x number of nodes)

Case	# of Cells	c_{Dmax}	c_{Lmax}	c_{Lmin}	Tstep (s)	Time* (s)
FFL2	49152	3.2877	0.0028	-0.0109	0.0100	17300 x 4
FFL4	3145728	3.2978	0.0028	-0.0110	0.0050	214473 x 48
OFL4	3145728	3.3038	0.0028	-0.0112	0.0002	593500 x 24
CFXL4	3145728	3.3084	0.0028	-0.0113	0.0050	115300 x 24

Table 12: Comparison of FeatFlow results at level 2 with OpenFOAM and CFX at level 4. (*: clock time x number of nodes)

Case	%Err		$\%L_2 Err$		$\%L_\infty Err$		Time* (s)
	c_{Dmax}	c_{Lmin}	c_D	c_L	c_D	c_L	
FFL2	0.31	0.91	0.29	1.05	1.01	0.01	17300 x 4
OFL4	0.18	1.82	0.16	1.47	0.61	0.02	593500 x 24
CFXL4	0.32	2.73	0.29	2.24	1.20	0.03	115300 x 24

2. Fully coupled implicit solvers (CFX) offer the advantage of using larger time step sizes, however, to be able to reach the desired accuracy, they require more nonlinear iterations. Thus, the overall computational cost has not been changed significantly. In the light of our calculations, there is not much difference between these two approaches in the solution of unsteady incompressible laminar flows; however, this question requires further investigation.

3. Using higher order discretization schemes in space leads to denser linear system of equations which can be solved more efficiently on state of the art computers. And since, FeatFlow is more accurate and efficient in the test cases (see Table (12)), we can conclude that it pays to use higher order discretization in space and time.

Regarding the comparison of the software tools, the most prominent conclusion can be drawn form Table (12): FF calculation at level 2 on 4 nodes has a similar accuracy of other codes at level 4 on 24 nodes. While FF requires ≈5 hours of computation for these calculations, CFX and OF require much more.

As a conclusion, in the $Re = 20$ case, we succeeded to obtain fully converged results with all three softwares, and although the second benchmark test was particularly challenging, a reliable reference solution has been obtained. Regarding the computational performance of the employed software packages, this benchmark should be considered as still open and a motivation for CFD software developers to join. All data files and the corresponding plots of the results obtained through this study can be downloaded from "http://www.featflow.de/en/benchmarks.html" website.

Acknowledgements

The authors like to thank the German Research Foundation (DFG) for partially supporting the work under grants Sonderforschungsbereich SFB708 (TP B7) and SPP1423 (Tu102/32-1), and Sulzer Innotec, Sulzer Markets and Technology AG for supporting Evren Bayraktar with a doctoral scholarship.

References

[1] M. Schäfer and S.Turek, *Benchmark computations of laminar flow around a cylinder* (With support by F. Durst, E. Krause and R. Rannacher). In E. Hirschel, editor, Flow Simulation with High-Performance Computers II. DFG priority research program results 1993-1995, number 52 in Notes Numer. Fluid Mech., pp.547–566. Vieweg, Weisbaden, 1996.

[2] M. Braack and T. Richter, *Solutions of 3D Navier-Stokes benchmark problems with Adaptive Finite Elements*. Computers and Fluids, 2006, **35(4)**, pp.372–392.

[3] V.John, *Higher order finite element methods and multigrid solvers in a benchmark problem for 3D Navier-Stokes equations*, Int. J. Numer. Math. Fluids, 2002, **40**, pp.755–798.

[4] V.John, *On the efficiency of linearization schemes and coupled multigrid methods in the simulation of a 3D flow around a cylinder*, Int. J. Numer. Math. Fluids, 2006, **50**, pp. 845–862.

[5] J.H Ferziger and M.Peric, *Computational Methods for Fluid Dynamics*, 3rd ed. Springer, 2002.

[6] OpenFOAM User's guide (version 1.6), 2009, http://www.openfoam.com/docs/

[7] ANSYS CFX-Solver, Release 10.0:Theory, http://www.ansys.com/cfx