Recent Benchmark Computations of Laminar Flow Around a Cylinder

S. Turek[a] *and M. Schäfer*[b]

[a]Institut für Angewandte Mathematik, Universität Heidelberg
INF 294, D-69120 Heidelberg, Germany
[b]Fachgebiet Numerische Berechnungsverfahren im Maschinenbau, TH Darmstadt
Petersenstr. 10, D-64287 Darmstadt, Germany

SUMMARY

An overview of recent benchmark computations for 2D and 3D laminar flows around a cylinder is given. These have been defined for a comparison of different solution approaches for the incompressible Navier-Stokes equations developed within the DFG Priority Research Program "Flow Simulation on High Performance Computers". The principal purpose of the benchmarks is discussed and some general conclusions which can be drawn from the (partially surprising) results are formulated. The exact definitions of the benchmarks, the numerical schemes and the computers employed by the various participating groups can be found in detail in [2]. In the end of our talk we present a new mathematical concept for rigorous error control and corresponding efficient algorithmical tools which can be applied to CFD–simulations.

1. INTRODUCTION

During the last years many solution methods for various flow problems have been developed with considerable success. In many cases, the computing times are still very long and, because of a lack of storage capacity and insufficient resolution, the agreement between the computed results and experimental data is - even for laminar flows - only qualitative in nature. If numerical solutions are to play a similar role to wind tunnels, they have to provide the same accuracy as measurements, in particular in the prediction of the overall forces.

Several new techniques such as "unstructured grids", "multigrid", "operator splitting", "domain decomposition" and "mesh adaptation" have been developed in order to improve the performance of numerical methods. To facilitate the comparison of these solution approaches, a set of benchmark problems has been defined and all participants of the DFG Priority Research Program "Flow Simulation on HighPerformance Computers" working on incompressible flows have been invited to submit their solutions. This paper presents the results of these computations contributed by altogether 17 research groups, 10 from within of the Priority Research Program and 7 from outside. The major purpose of the benchmark is to establish, whether constructive conclusions can be drawn from a comparison of these results so that the solutions can be improved. It is not the aim to come to the conclusion that a particular solution A is better than another solution B; the intention is rather to determine whether and why certain approaches are superior to others. The benchmark is particularly meant to stimulate future work.

In the first step, only incompressible laminar test cases in two and three dimensions have been selected which are not too complicated, but still contain most difficulties representative of industrial flows in this regime. In particular, characteristic quantities such as drag and lift coefficients have to be computed in order to measure the ability to produce quantitatively accurate results. This benchmark aims to develop objective criteria for the evaluation of the different algorithmic approaches. For this purpose, the participants have been asked to submit a fairly complete account of their computational results together with detailed information about the discretization and solution methods used. As a result it should be possible, at least for this particular class of flows, to distinguish between "efficient" and "less efficient" solution approaches. Since this benchmark has been proved to be successful it is now being under work to be extended to include also certain turbulent flows. In our talk we present the first official version of this turbulent benchmark.

It is particularly hoped that the (laminar) benchmark will provide the basis for reaching decisive answers to the following questions which are currently the subject of controversial discussion:

1. *Is it possible to calculate incompressible (laminar) flows accurately and efficiently by methods based on explicitly advancing momentum?*

2. *Can one construct an efficient solver for incompressible flow without employing multigrid components, at least for the pressure Poisson equation?*

3. *Do conventional finite difference methods have advantages over new finite element or finite volume techniques?*

4. *Can steady-state solutions be efficiently computed by pseudo-time-stepping techniques?*

5. *Is a low-order treatment of the convective term competitive, possibly for smaller Re numbers?*

6. *What is the "best" strategy for time stepping: fully coupled iteration or operator splitting?*

7. *Does it pay to use higher order discretizations in space or time?*

8. *What is the potential of using unstructured grids?*

9. *What is the potential of a posteriori grid adaptation and time step selection in CFD?*

10. *What is the "best" approach to handle the nonlinearity: quasi-Newton iteration or nonlinear multigrid?*

These questions appear to be of vital importance in the construction of efficient and reliable solvers, particularly in three space dimensions. Everybody who is extensively consuming computer resources for numerical flow simulation should be interested.

2. DEFINITION OF TEST CASES

This section gives a brief summary of the definitions of the test cases for the benchmark computations. We restrict in this presentation to one single 3D–test case only containing most of the relevant information. All details including precise definitions of the quantities which had to be computed and also some additional instructions which were given to the participants can be found in [2]. The fluid properties are identical for all test cases. An incompressible Newtonian fluid is considered for which the conservation equations of mass and momentum are

$$\frac{\partial U_i}{\partial x_i} = 0$$

$$\rho\frac{\partial U_i}{\partial t} + \rho\frac{\partial}{\partial x_j}(U_j U_i) = \rho\nu\frac{\partial}{\partial x_j}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{\partial P}{\partial x_i}.$$

The notations are time t, cartesian coordinates $(x_1, x_2, x_3) = (x, y, z)$, pressure P and velocity components $(U_1, U_2, U_3) = (U, V, W)$. The kinematic viscosity is defined as $\nu = 10^{-3}\,\mathrm{m}^2/\mathrm{s}$, and the fluid density is $\rho = 1.0\,\mathrm{kg/m}^3$.

For the 3D test cases the flows around a cylinder with circular (and square) cross-sections are considered. The problem configurations and boundary conditions are illustrated in Figure 0.1. The outflow condition can be selected by the user.

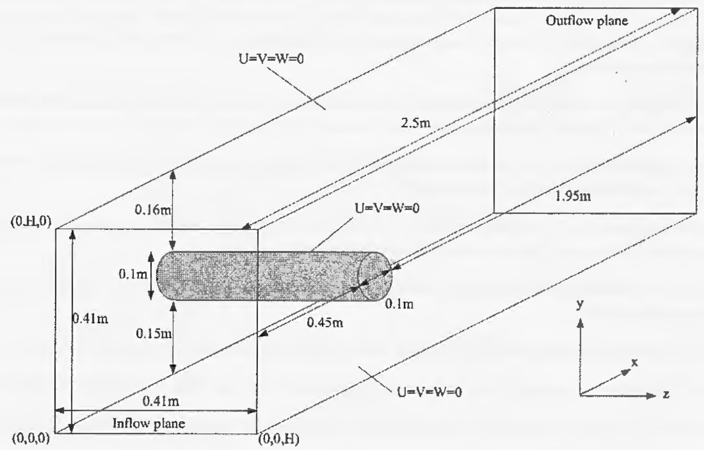

Figure 0.1: *Configuration and boundary conditions for flow around a cylinder with circular cross-section.*

Some definitions are introduced to specify the values which have to be computed. The characteristic velocities are chosen such that the Reynolds numbers can be defined as $Re = 20$ for the stationary case, resp., $Re = 100$ in the nonstationary cases. The following quantities should be computed: drag coefficient c_D, lift coefficient c_L, pressure difference ΔP and Strouhal number St. In the nonstationary cases, which were performed for a given steady and a time dependent inflow, all values should be calculated as functions in time. Especially the physically relevant values for drag and lift led to very interesting results for the evaluation.

To get a better understanding of the accuracy and efficiency of the methods used the participants were instructed to perform the computations on three successively coarsened meshes in space and time. By this instruction, 'good results by chance' on one single mesh could be excluded.

3. RESULTS

Up to now 17 groups have officially participated. Some representative results are shown in the following Tables 0.1 and 0.2 together with the computers used.

On the basis of the results obtained by these benchmark computations some conclusions can be drawn. These have to be considered with care, as the provided results depend on parameters which are not available for the authors of this report, e.g., design of the grids, setting of stopping criteria, quality of implementation, etc.

Table 0.1: Results for the stationary test

Gr.	Unknowns	c_w	c_a	ΔP	RAM	CPU	Computer
1	2426292	6.1295	0.0093	0.1693	233	2097	Fujitsu VPP500
	630564	6.1230	0.0095	0.1680	71	1238	
2	555000	6.1440	0.0074	0.1604	122	8731	IBM RS6000/370
	276800	5.8600	0.0042	0.1616	67	6094	
3	608496	6.1600	0.0095	0.1690	74	4150	Cray T3D/16
6	6303750	6.2330	-0.0040		43	221706	HP735
7	12582912	6.1932	0.0093	0.1709	3571	2630	GC/PP128
	1572864	6.1868	0.0092	0.1703	518	1120	
	196608	6.1366	0.0098	0.1673	71	460	
8	362613	6.1430	0.0084	0.1694	126	51280	IBM RS6000/590
	73262	6.0990	0.0067	0.1695	28	7178	
9	2355712	6.1800	-0.0010	0.1691		62000	IBM RS6000/590
	753664	6.1720	0.0090	0.1680		6000	
	94208	6.1310	0.0100	0.1605		950	
10	6116608	6.1043	0.0079	0.1672	700	8440	IBM RS6000/590
	771392	5.9731	0.0059	0.1605	89	1466	
	98128	5.8431	0.0061	0.1482	11	290	

For five of the ten questions above the answers seem to be clear:

1. In order to compute incompressible flows of the present (laminar) type accurately and efficiently, one should use implicit methods. The step size restriction enforced by explicit time stepping can render this approach highly inefficient, as the physical time scale may be much larger than the maximum possible time step in the explicit algorithm. This is obvious from the results for the stationary cases in 2D and 3D, and also for the nonstationary cases in 2D. For the nonstationary cases in 3D only too few results on apparently too coarse meshes have been provided, in order to draw clear conclusions. This question requires further investigation.

2. Flow solvers based on conventional iterative methods on the linear subproblems have on fine enough grids no chance against those employing suitable multigrid techniques. The use of multigrid can allow computations on workstations (provided the problem fits into the RAM) for which otherwise supercomputers would have to be used. In the submitted solutions supercomputers (Fujitsu, SNI, CRAY) have mainly been used for their high CPU power but not for their large storage capacities. For example, in the nonstationary test case the solutions 1 and 3 require with about 600,000 unknowns on supercomputers significantly more CPU time than the solution 10 with the same number of unknowns on a workstation.

3. The most efficient and accurate solutions are based either on finite element or finite volume discretizations on contour adapted grids.

4. The computation of steady solutions by pseudo time-stepping techniques is inefficient compared with using directly a quasi-Newton iteration as stationary solver.

5. For computing sensitive quantities such as drag and lift coefficients, higher order treatment of the convective term is indispensable. The use of only first order upwinding (or crude approximation of curved boundaries) does not lead to satisfactory accuracy even on very fine meshes (several million unknowns in 2D).

For the remaining five questions the answers are not so clear. More test calculations will be necessary to reach more decisive conclusions. The following preliminary interpretations of the results obtained so far may become the subject of further discussion:

Table 0.2: Results for the nonstationary test

Gr.	Unknowns	Time steps	c_w	c_a	ΔP	RAM	CPU	Computer
1	630564	800	3.2826	0.0027	-0.1117	79	156460	Fujitsu VPP500
3	608496	1600	3.2590	0.0026	-0.1072	74	76142	Cray T3D/16
	608496	800	3.2590	0.0026	-0.1157	74	50764	
6	6303750	18000	4.1600	0.0200		43	142646	HP735
7	1572864	1600	3.3011	0.0026	-0.1102	518	149923	GC/PP128
	1572864	800	3.3008	0.0026	-0.1105	518	93055	
	1572864	400	3.3006	0.0026	-0.1107	518	62026	
	196608	1600	3.3053	0.0028	-0.1066	71	63057	
8	199802	1000	3.2120	0.0122	-0.1112	105	846000	IBM RS6000/590
	98637	1000	3.2350	0.0123	-0.1114	39	243000	
10	6116608	668	3.2802	0.0034	-0.0959	840	164837	IBM RS6000/590
	6116608	272	3.3748	0.0360	-0.0603	840	77538	
	6116608	60	2.7312	0.0069	-0.0682	840	29742	
	771392	724	3.3323	0.0033	-0.0766	105	24745	
	98128	660	3.4200	0.0040	-0.0407	13	5687	

6. In computing nonstationary solutions, the use of operator splitting (pressure correction) schemes tends to be superior to the more expensive fully coupled approach, but this may depend on the problem as well as the quantity to be calculated. Further, as fully coupled methods also use iterative correction within each time step (possibly adaptively controlled), the distinction between fully coupled and operator splitting approach is not so clear.

7. The use of higher than second-order discretizations in space appears promising with respect to accuracy, but there remains the question of how to solve efficiently the resulting algebraic problems (see the results of 8 for all test cases). The results provided for this benchmark are too sparse to allow a definite answer.

8. The most efficient solutions in this benchmark have been obtained on blockwise structured grids which are particularly suited for multigrid algorithms. There is no indication that fully unstructured grids might be superior for this type of problem, particularly with respect to solution efficiency. The winners may be hierarchically structured grids which allow local adaptive mesh refinement together with optimal multigrid solution.

9. From the contributed solutions to this benchmark there is no indication that a-posteriori grid adaptation in space is superior to good hand-made grids. This, however, may drastically change in the future, particularly in 3D. Intensive development in this direction is currently in progress. For nonstationary calculations, adaptive time step selection is advisible in order to achieve reliability and efficiency (see the results of 10).

10. The treatment of the nonlinearity by nonlinear multigrid has no clear advantage over the quasi-Newton iteration with multigrid for the linear subproblems (compare the results of 7 with those of 10). Again, it is the extensive use of well-tuned multigrid (wherever in the algorithm) which is decisive for the overall efficiency of the method.

Although this benchmark has been fairly successful as it has made possible some solidly based comparison between various solution approaches, it still needs further development. Even in the laminar case, the chosen nonstationary 3D problems showed to be harder than expected. In particular, it was apparently not possible to achieve reliable reference solutions for two test cases (3D-2Q and 3D-2Z). Hence the benchmark has to be considered as still open and everybody is invited to try again.

4. A MATHEMATICAL CONCEPT

These benchmark calculations show that even for laminar flow quantitatively exact predictions cannot be guaranteed by "pure" computer power. Using modern algorithmical tools, even workstation calculations can be much faster than those using conventional iterative methods on supercomputers. In our talk we will explain how to design such efficient schemes by combining special discretization and solution techniques (*Multilevel Discrete Projection Methods*) which are implemented in our CFD–package FEATFLOW [3].

Moreover, the benchmark results demonstrate, especially in 3D, that there is no chance to get precise results in a controlled way without having a tool for a rigorous a posteriori error control. We explain these techniques (see [1]) which are under progress by our group (R. Rannacher) and a swedish group (C. Johnson). The combination of these techniques with our highly efficient solution techniques should lead to completely new CFD–software which will really be able to capture practical problems.

5. REFERENCES

[1] **Johnson, C., Rannacher, R., Boman, M.:** *Numerics and hydrodynamic stability: Towards error contol in CFD*, Preprint 1993-13, Mathematics Department, Chalmers University of Technology, 1993, to appear in SINUM

[2] **Schäfer, M., Turek, S. (with support by F. Durst, E. Krause, R. Rannacher):** *Benchmark computations of laminar flow around cylinder*, Proc. DFG Priority Research Program 'Flow Simulation on High Performance Computers', 1996, Vieweg

[3] **Turek, S.:** FEATFLOW *Finite element software for the incompressible Navier–Stokes equations: User Manual, Release 1.0*, University of Heidelberg, 1995

[4] **Turek, S.:** *Tools for simulating nonstationary incompressible flow via discretely divergence-free finite element models*, Int. J. Numer. Meth. Fluids, 18, 71–105 (1994)

[5] **Turek, S.:** *A comparative study of time stepping techniques for the incompressible Navier-Stokes equations: From fully implicit nonlinear schemes to semi-implicit projection methods*, to appear in: Int. J. Numer. Meth. Fluids