Numerical studies for Flow Around a Sphere regarding different flow regimes caused by various Reynolds numbers

R. Jendrny, H. Damanik, O. Mierka, S. Turek

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

Abstract

We simulate the flow around a sphere with different Reynolds numbers and take a look at the arising flow regimes: Steady axis-symmetric, steady planar-symmetric and unsteady periodic flow. The first one arises for Reynolds numbers, which are lower than approximately 211, the second one follows up to Reynolds number approximate 270; for 300 we get the periodic flow.
The simulations are done by a full 3D flow solver and compared to a 2.5D axis-symmetric method. We see differences between both methods for high Reynolds numbers because we do not have an axis-symmetric flow for all considered ones: The full 3D flow solver simulates the flow regimes which are mentioned above; the 2.5D method always computes axis-symmetric solutions.

Keywords: Navier-Stokes equation, flow around a sphere, axis-symmetry, planar-symmetry, drag-coefficient

1. Introduction

1.1. Motivation

To simulate a three dimensional flow around a sphere it has become standard that this is done by a computational fluid dynamics (CFD) code. Of course there are a lot of possibilities to simulate this flow. In the literature ([2] or [3]) it is shown, that we will have different stationary flow regimes for different Reynolds numbers. The flow will change from axis-symmetry to planar-symmetry.

We concentrate on stationary flows. The equation we want to solve is the following one, also known as stationary Navier-Stokes equation (NSE):

$$\begin{aligned} -\nu\Delta u + u\cdot\nabla u + \nabla p &= f, \\ \nabla\cdot u &= 0. \end{aligned}$$

In general we know two different methods which can solve this equation for flow around a sphere: A full 3D flow solver and a 2.5D axis-symmetric method which is a simplified version to solve special 3D flow problems. In this paper we want to take a look at the differences between these methods and from which Reynolds number on the 2.5D method will give us wrong results.

Email addresses: robert.jendrny@math.tu-dortmund.de (R. Jendrny), hogenrich.damanik@math.tu-dortmund.de (H. Damanik), otto.mierka@math.tu-dortmund.de (O. Mierka), stefan.turek@math.tu-dortmund.de (S. Turek)

1.2. Numerical approach

To solve the stationary NSE we have to discretise the equation. We have done this with the finite element method using the LBB-stable FEM pair Q2-P1. For details we refer to [1]. It results in the well known system:

$$\begin{pmatrix} S(u_h) & kB \\ B^T & 0 \end{pmatrix} \begin{pmatrix} u_h \\ p_h \end{pmatrix} = \begin{pmatrix} g_h \\ 0 \end{pmatrix}.$$

Now we apply the two methods to the given system. The 3D method is described in detail in [4]. The 2.5D method is an adjusted method for axis-symmetric flows: In this method we extend the standard mixed FEM to axis-symmetric problems to be able to simulate flow around a sphere with 2D finite elements. With this monolithic 2.5D method we directly solve steady flow problems. On the other hand the 3D method solves unsteady flow problems via using projection scheme method. So we have to take the non-stationary NSE and calculate until convergence, i.e. the solution has to satisfy:

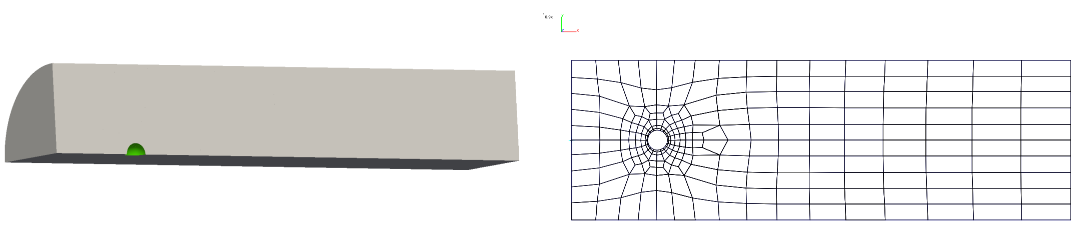
$$\text{IF } \{ \max_{i \in \{x,y,z\}} \frac{|F_i^{(k+1)} - F_i^{(k)}|}{|F_i^{(k+1)}|} \leq 10^{-4} \} \text{ AND } \{ \max_{i \in \{x,y,z\}} \frac{|F_i^{(k+1)} - F_i^{(k)}|}{\Delta t} \leq 10^{-5} \}$$
$$\text{THEN: STOP.}$$

Here F_i is the force in i-th direction acting on the sphere, i.e. drag in x direction and lift in y and z direction.

Compared to the 3D method the 2.5D method will give different solutions for certain Reynolds numbers. We are interested in that point where the 2.5D method is not able to give us reliable results for three dimensional flow problems. Therefore we take the 3D method as our reference code.

2. Test configuration for numerical studies

We know that the topic of flow around a sphere has been simulated by several scientific groups. They also give a list how the drag-coefficient with different Reynolds numbers 'should' behave. Before we start to show our results for the drag-coefficient we describe our test case:



(a) Cutting through the area (b) Coarse mesh cutplane (level 1)

Figure 1: Simulation domain

The pipe has a radius of 4 and a length of 25. At position $x = 0$ we have the inflow part and at $x = 25$ the outflow part. All boundaries except the outflow part are prescribed with a velocity of (absolute) value 1 in positive x-direction. Centered in y and z direction we have a sphere with x_M-coordinate 4.3 and a radius of 0.5. All simulations are done in this area on level 3 (two refinements). For the 2.5D method we use the twice refined cutplane shown above, which means that we have the same restriction and we can compare our results.

	3D		2.5D	
Level	NEL	NEQ	NEL	NEQ
1	1564	46882	126	1508
2	12512	362594	565	5787
3	100096	2850634	2016	22658

Table 1: Grid information

In this domain we compute the flow for several Reynolds numbers using both methods to validate our test configuration. We get the following drag-coefficients:

Re	3D	2.5D
100	1.11155220	1.1122660
110	1.05630190	1.0593010
120	1.01142340	1.0136100
130	0.97249203	0.9736551
140	0.93832462	0.9383106
150	0.90805589	0.9067336
160	0.88082690	0.8782798
170	0.85627315	0.8524479
180	0.83397473	0.8288421
190	0.81366776	0.8071454
200	0.78642173	0.7871007

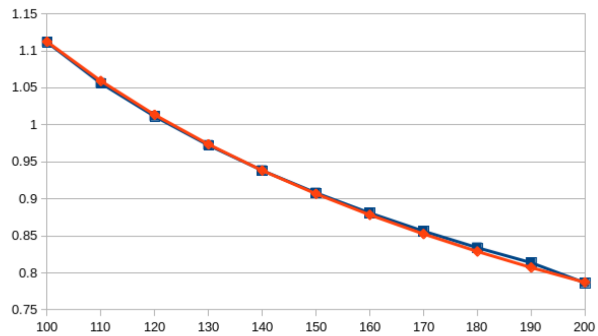


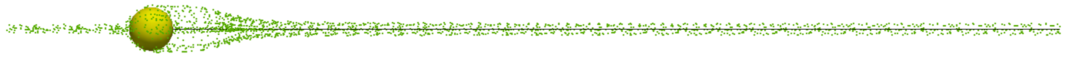
Figure 2: Drag-coefficients

Jones and Clarke ([3]) for example have calculated a drag-coefficient of value ≈ 1.1 for Reynolds number 100. They have also shown the behavior of drag-coefficients for different Reynolds numbers ([3], Fig. 1). If we compare our results to [3] (Fig. 1) or to [2], we see that they behave more or less identical. In a first step we can say that our methods are giving 'right' results for the axis-symmetric flow regime.

3. Comparison of 2.5D and 3D

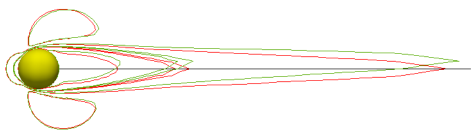
3.1. Visual Representation

Jones and Clark ([3]) also claim that the flow past the sphere has different structures: First it is a steady axis-symmetric flow until Reynolds number 211, then the axis-symmetry gets broken and they get a planar-symmetric solution which is steady as well. In our simulations we first take a look at the visual structure, if we can see that the axis-symmetry gets broken. Therefore we study the flow for Reynolds numbers 200 and 225 simulated by the 3D method.

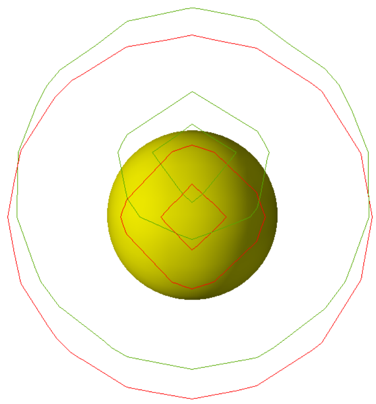


(a) Particle-tracer (Re 200, axis-symm.)

(b) Particle-tracer (Re 225, planar-symm.)



(c) contour in x-z-plane (Re 200, Re 225)



(d) contour in y-z-plane (Re 200, Re 225)

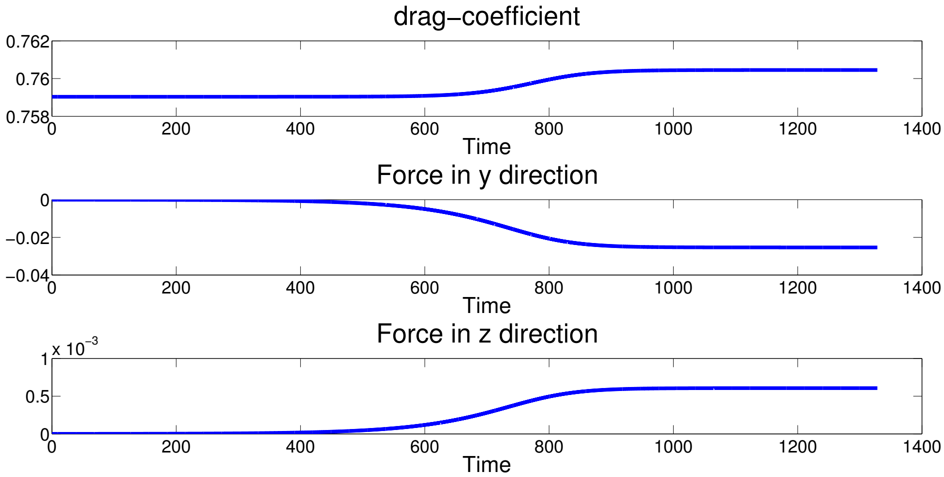
Figure 3: Velocity magnitude: Flow structure

(a) Re 200, axis-symm.

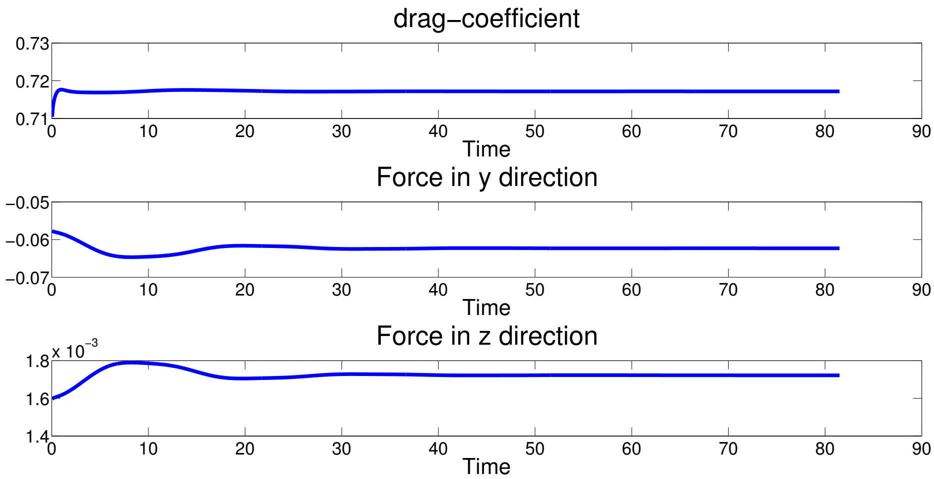
(b) Re 225, planar-symm.

Figure 4: **left**: Scaled velocity y component, **right**: scaled pressure

In our output we can clearly see that the axis-symmetry gets broken. Consequently, we can underline the results given by [3]. Furthermore we can see that we have steady solutions: The flow structure is stable. The behaviour of the acting forces underlines this steady state solution.



(a) Re 215



(b) Re 250

Figure 5: Forces getting stationary during simulation time

The forces become 'stationary' after some time. Especially the forces for Reynolds number 250 (planar-symmetry flow, [3]) become stationary during simulation. We know from other flow problems (see literature and further test simulations) that we get a steady state solution if we decrease the Reynolds number. That is why we conclude that the flow for Reynolds numbers which are lower than 250 will end up with steady state solutions.

3.2. Limitations of the 2.5D axis-symmetric code

Johnson and Patel [2] have shown that the axis-symmetry gets broken between Reynolds numbers 211 and 212. Then the flow should become planar symmetric, i.e. the useful 2.5D method should give other (wrong) results.

Re	3D			2.5D axis	error
	drag	f_y	f_z	drag	
200	0.78642173	-1.67732E-7	1.36292E-9	0.78710070	8.63E-4
201	0.78449974	-1.77456E-7	1.49998E-9	0.78517870	8.65E-4
202	0.78259611	-1.89137E-7	1.66589E-9	0.78327080	8.61E-4
203	0.78070318	-2.03506E-7	1.87136E-9	0.78137690	8.62E-4
204	0.77882883	-2.21596E-7	2.13209E-9	0.77949690	8.57E-4
205	0.77695756	-2.45101E-7	2.47136E-9	0.77763050	8.65E-4
206	0.77511288	-2.76582E-7	2.92673E-9	0.77577760	8.57E-4
207	0.77327143	-3.22048E-7	3.58837E-9	0.77393800	8.61E-4
208	0.77144896	-3.92454E-7	4.61601E-9	0.77211150	8.58E-4
209	0.76963771	-5.17058E-7	6.43913E-9	0.77029800	8.57E-4
210	0.76783776	-7.98801E-7	1.05465E-8	0.76849730	8.58E-4
211	0.76604919	-2.04676E-6	2.89822E-8	0.76670920	8.61E-4
212	0.76445687	-9.09340E-3	2.18715E-4	0.76493360	6.23E-4
213	0.76311380	-1.66513E-2	3.99156E-4	0.76317030	7.40E-5
214	0.76178082	-2.15264E-2	5.16252E-4	0.76141930	4.75E-4
215	0.76045223	-2.53423E-2	6.06943E-4	0.75968020	1.02E-3
216	0.75912207	-2.85436E-2	7.51187E-4	0.75795300	1.54E-3
217	0.75780294	-3.12995E-2	8.56142E-4	0.75623760	2.07E-3
218	0.75648893	-3.37348E-2	9.47344E-4	0.75453370	2.59E-3
219	0.75517404	-3.59288E-2	9.94907E-4	0.75284130	3.10E-3
220	0.75387099	-3.79060E-2	1.04984E-3	0.75116030	3.61E-3
221	0.75256747	-3.97195E-2	1.10014E-3	0.74949040	4.11E-3
222	0.75126999	-4.13849E-2	1.14635E-3	0.74783160	4.60E-3
223	0.74997876	-4.29217E-2	1.18887E-3	0.74618370	5.09E-3
224	0.74869402	-4.43455E-2	1.22829E-3	0.74454650	5.57E-3
225	0.74740951	-4.56755E-2	1.26511E-3	0.74292010	6.04E-3

Table 2: Differences between 3D and 2.5D axis-sym.

For the 2.5D axis-symmetric code we can only show values for the drag because the other forces have to become zero by definition of axis-symmetry. In the first few rows the drag-coefficients of both codes are identical (relative error: $\frac{|drag2.5D - drag3D|}{|drag2.5D|} < 1‰$). We can clarify that the axis-symmetry gets broken as shown in the literature, because we see a big jump in the forces across the flow direction. The forces in y and z direction are numerically zero until Reynolds number 211, after this we get a planar-symmetric solution. To point up that we get different solutions with both codes we visualize the drag-coefficients calculated with both methods.

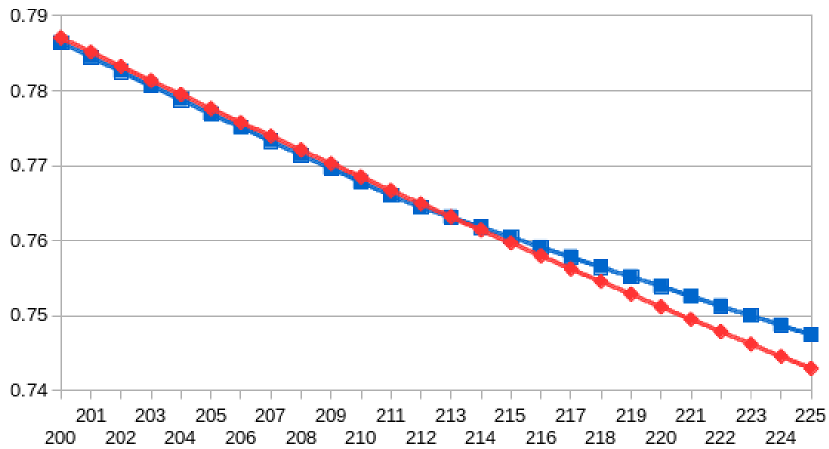


Figure 6: Differences between the 2.5D and the 3D methods

Obviously the difference between the 2.5D and the 3D code increases for higher Reynolds numbers. We can see a crossing between both curves. The crossing is caused by the forces in y and z direction. Now we want to check at which point the crossing takes place. So we refine our simulated resolution for Reynolds numbers.

RE	DRAG	f_y	f_z
210.00	0.76783776	-7.98801E-7	1.05465E-8
210.50	0.76694403	-1.13352E-6	1.54832E-8
210.75	0.76650071	-1.44874E-6	2.01084E-8
211.00	0.76604919	-2.04676E-6	2.89822E-8
211.25	0.76560532	-3.52283E-6	5.09284E-8
211.50	0.76516149	-3.79217E-4	9.07995E-6
211.75	0.76479439	-5.71566E-3	1.37909E-4
212.00	0.76445687	-9.09340E-3	2.18715E-4

Table 3: Start of planar-symmetric solution

We see a finite value for the lift starting with Reynolds number 211.5 which means that we should not use the 2.5D axis-symmetric code to simulate the flow for Reynolds numbers which are higher than 211.5 since the flow becomes planar-symmetric. But we can still work with these solutions: The 3D code needs initial solutions. In general we have taken a zero-solution for the first simulation and after that the final-solution of the previous simulation. But we can also take the solution calculated by the 2.5D code to start our 3D simulation for high Reynolds numbers.

4. Further flow regimes: Unsteady periodic flow

All shown tests, i.e. Reynolds number 50 to 250, have given steady state solutions. Now we are interested in unsteady solutions which will be given for Reynolds numbers higher than 270 ([3]). For this region we want to get the drag meanvalue to analyze if our code can still simulate such problems. We also want to analyze the behavior of the flow past the sphere. Hence, we will create some videos for these test cases. It should be clear that we only calculate the flow for these Reynolds numbers with the 3D method since the flow becomes unsteady and no more axis-symmetric. Periodic flows appear if we simulate flow with Reynolds numbers between 270 and 400 ([3]). We consider the flow with Reynolds number 300.

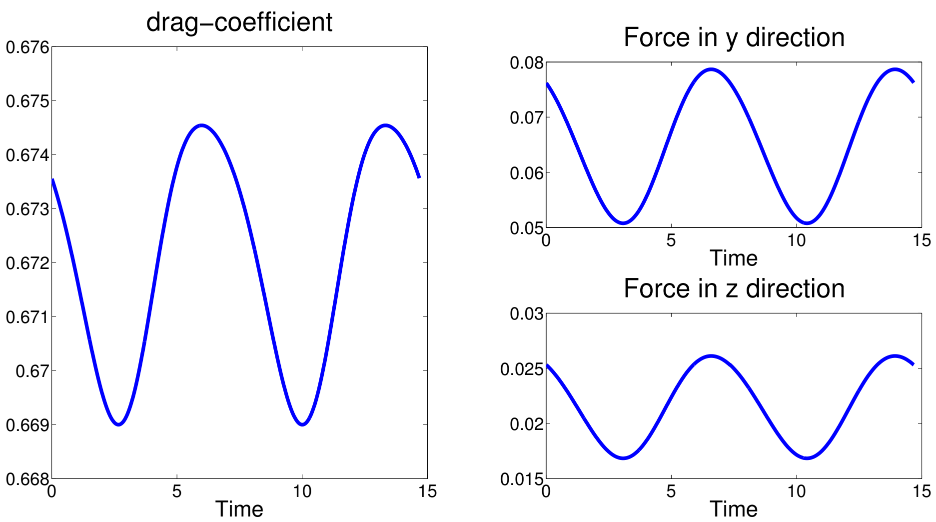


Figure 7: Forces during simulation

This figure verifies that the flow is periodic with a period of 7.34. The simulated mean-value of the drag is 0.6715. We close this test comparing our results to the one which are shown in the literature.

METHOD	DRAG-MEANVALUE	AMPLITUDE
Our test	0.672	0.003
Johnson/Patel ([2])	0.656	0.004
Jones/Clarke ([3])	0.668	0.004

Table 4: Periodic flow results

Our test results differ to the other ones: The drag has a small difference. This depends on our used mesh and the used numerical variables. We have placed the sphere 4 units behind the inflow part. Maybe it would be better if we increase this distance because we would not have 'enough' space in which the flow structure can develop itself. And we set boundary conditions that might influence the flow. In the literature it is also not clearly said which level of discretisation and which timestep Δt they have used to create their results.

5. Conclusions

In this work we have simulated the three dimensional flow around a sphere. We have seen that the three dimensional flow can be simulated by our 2.5D axis-symmetric code until a certain Reynolds number which can improve simulation time. In general we get the flow regimes:

RANGE OF RE	FLOWSTRUCTURE
20 - 211	"stationary" axis-symmetric
211.5 - 270	"stationary" planar-symmetric
$\approx$ 300	unsteady periodic flow

Table 5: Flow regimes

Until Reynolds number 211 we have a steady state solution which is axis-symmetric. There the 2.5D axis-symmetric code is much faster than the 3D code. After Reynolds number 211.5 the code is still faster but it gives wrong results because the flow is no longer axis-symmetric. We have got a steady planar-symmetric solution which cannot be solved by an axis-symmetric code by definition. But an initial solution can be computed by this axis-symmetric code for detailed three dimensional flow problems.
The 3D code, of course, can simulate steady state solutions and unsteady flow problems which can be as well symmetric as non-symmetric.

Future work

For Reynolds numbers up to 250 we have a steady state solution. But we still solve this steady flow taken as an unsteady one which needs lots of working/simulation time. Therefore we want to develop a code which can solve the stationary flow around a sphere problem efficiently even in the case of planar-symmetry.

References

[1] D. Boffi & L. Gastaldi, *On the quadrilateral Q2-P1 element for Stokes problem.* International Journal for numerical methods in fluid, pp 1001-1011, 2002.

[2] T.A. Johnson & V.C. Patel, *Flow past a sphere up to a Reynolds number of 300.* Journal Fluid Mech. 378, pp 19-70, 1999.

[3] D.A. Jones & D.B. Clarke, *Simulation of Flow Past a Sphere using the Fluent Code.* Maritime Platforms Division, Defence Science and Technology Organisation, Victoria 3207, Australia, 2008.

[4] S. Turek, *Efficient solvers for incompressible flow problems: An Algorithmic and Computational Approach.* Springer, 3-540-65433-X, 1999.