Direct numerical simulation of turbulent pipe flow up to a Reynolds number of 61,000

To cite this article: Bendiks Jan Boersma 2011 J. Phys.: Conf. Ser. 318 042045

View the article online for updates and enhancements.

Related content
- A new formulation for the streamwise turbulence intensity distribution
  P Henrik Alfredsson, Ramis Örlü and Antonio Segalini
- A study on the influence of triggering pipe flow regarding mean and higher order statistics
  F Zimmer, E-S Zanoun and C Egbers
- Turbulent Poiseuille Flow with Wall Transpiration: Analytical Study and Direct Numerical Simulation
  V'S Avsarkisov, M Oberlack and G Khujadze

Recent citations
- Recent advances on the numerical modelling of turbulent flows
  C.D. Argyropoulos and N.C. Markatos
- Turbulent pipe flow: Statistics, Re-dependence, structures and similarities with channel and boundary layer flows
  George K El Khoury et al
- Turbulence structure and budgets in curved pipes
  Massimiliano Di Liberto et al
Direct numerical simulation of turbulent pipe flow up to a Reynolds number of 61,000

Bendiks Jan Boersma
Department of Process and Energy
Delft University of Technology
Leegwaterstraat 44
2628 CA Delft
The Netherlands
E-mail: b.j.boersma@tudelft.nl

Abstract. In this paper we will present results of several direct numerical simulations of turbulent pipe flow. The highest Reynolds number simulated in this study was 61,000. Our numerical model uses Fourier expansions in axial and circumferential directions and 6th order staggered compact finite difference in the wall normal direction. Apart form standard turbulent statistics we also will present 1D energy spectra and autocorrelation functions.

1. Introduction

From an engineering point of view turbulent pipe flow is extremely important, because of its wide range of applications. In the past many fundamental studies on wall bounded flows have been performed in a plane channel flow geometry, see for instance (Kim et al. 1987), (Moser et al. 1999) and (Del Alamo et al. 2004). Although there is a clear similarity between pipe and channel flow, for instance, in the near wall region there are also considerable differences, especially in the core region of the flow.

In the case of turbulent pipe flow only a limited number of numerical studies have been carried out, see for instance (Eggels et al. 1993), (Loulou et al. 1997) and (Wu & Moin 2008), while there is an enormous amount of experimental data for a large range of Reynolds numbers available, see for instance (McKeon et al. 2004), (Den Toonder & Nieuwstadt 1997), and (Morrison et al. 2004). There are still many unsolved questions for pipe flow, for instance it is argued by (Mochizuki & Nieuwstadt 1996) that the peak of the axial rms decreases with Reynolds number, while in the paper by (Morrison et al. 2004) it is argued that the peak should increase with Reynolds number. The goal of the present research is to develop a highly accurate numerical model that is able to simulate, by means of DNS, flows with Reynolds numbers in the range of the experiments carried out by (McKeon et al. 2004) and (Morrison et al. 2004).

The organization of the paper is as follows: In section two we will give the governing equations, in section three we will discuss the numerical method, next in section four we will present results for different Reynolds numbers and finally in section five we will draw some conclusions.
2. Governing equations and numerical method

The flow in the pipe is governed by the incompressible Navier-Stokes equations. Here we will use the formulation given by (White 2003). The governing equations are normalized with the friction velocity \( u_* \) and the pipe diameter \( D \). The friction velocity is by definition equal to the square root of the wall friction divided by the fluid’s density, i.e. \( u_* = \sqrt{\tau_w/\rho} \). The (frictional) Reynolds number is now defined as \( Re_* = u_* D/\nu \) and the bulk Reynolds number \( Re_b = (U_b/u_*)Re_* \), where \( U_b \) is the bulk velocity. The value of \( U_b/u_* \) can be calculated using one of the many available engineering correlations.\(^1\)

2.1. Spatial discretization

Most numerical simulation models for pipe flow use 2nd order accurate finite differences. This results in general in a very efficient computational algorithm. However, the accuracy of these 2nd order methods, especially for large wavenumber phenomena is not very good. Therefore, in the present study, we will use a pseudo spectral method combined with a highly accurate 6th order staggered compact finite difference method, which has been developed by us (Boersma 2005) and (Boersma 2011).

The Navier-Stokes equations written in cylindrical coordinates are discretized with the pseudo spectral (FFT based) method in the circumferential and axial direction. In the radial direction we use the 6th order staggered compact finite difference method. The grid in the radial direction is nonuniform, with the grid point slightly clustered towards the wall. The compact finite differences itself are computed on a uniform grid with grids pacing \( \Delta R \). To be able to use a non-uniform grid in the radial direction a mapping of the following form is used

\[
\frac{df}{dr} = \frac{df}{dR} \frac{dR}{dr}
\]

where \( df/dr \) is the derivative on the non-uniform grid, \( df/dR \) the derivative on the uniform grid which will be calculated with the 6th order compact finite difference, and \( dR/dr \) maps the non-uniform grid on the uniform grid. Here we use a simple a simple relation for \( R \) so that \( dR/dr \) can be calculated analytically. The derivative \( df/dR \) has been calculated with a staggered compact finite difference, see for instance (Boersma 2005),

\[
a(f'_{i+1} + f'_{i-1}) + f'_i = \frac{b}{\Delta R} (f_{i+1/2} - f_{i-1/2}) + \frac{c}{\Delta R} (f_{i+3/2} - f_{i-3/2}), \quad \text{with} \quad 2 \leq i \leq n - 2, \quad (1)
\]

In which \( f'_i \) is derivative of \( f \) with respect to \( x \) at point \( i \), \( \Delta R \) is the (uniform) grid spacing and \( a, b, \) and \( c \) are yet unspecified coefficients. The coefficients \( a, b, \) and \( c \) can be obtained from a Taylor expansions around grid point \( i \). With the three coefficients \( a, b, \) and \( c \) in equation (1) we can obtain an 6th order accurate formulation. The values for \( a, b, \) and \( c \) for this 6th order scheme are, see for instance (Nagarajan et al. 2003):

\[
a = 9/62, \quad b = 63/62, \quad c = 17/186, \quad 2 \leq i \leq n - 2
\]

Close to the boundary at points \( i = 1 \) and \( i = n - 1 \) this sixth order formulation can not be used because information from outside the domain would be required. Therefore we use a smaller stencil for these points:

\[
a = 1/22, \quad b = 12/11, \quad c = 0, \quad O(\Delta R)^4
\]

\(^1\) A common relation between \( u_* \) and \( U_b \) is given by the Blasius correlation

\[
\frac{u_*}{U_b} = 8 \left( \frac{U_b D/\nu}{\tau_w} \right)^{0.25}
\]
We use a 3rd or a 4th order accurate formulation: 

\[ f'_0 + 23 f'_1 = \frac{1}{\Delta R} (-25 f_{1/2} + 26 f_{3/2} - f_{5/2}) + O(\Delta R)^3 \]  
\[ f'_0 + \frac{331}{15} f'_1 = \frac{1}{\Delta R} \left( \frac{8677}{360} f_{1/2} + \frac{4531}{180} f_{3/2} - \frac{11}{10} f_{5/2} + \frac{1}{36} f_{7/2} + \frac{1}{360} f_{9/2} \right) + O(\Delta R)^4 \]  

The equation above in for the point \( i = 0 \), for the point \( i = n \) similar equations can be derived.

2.2. Temporal discretization

The time integration is split into two steps. In the first step the velocity is integrated to an intermediate level \( u^* \) with help of 3rd order Adams-Bashforth method

\[ u^{n+1} = u^* - \Delta t \frac{1}{\rho} \nabla p^{n+1/2} \]  

Where \( \Delta t \) is the time step, \( f(R^{n-j}) \) denotes all the spatial derivatives in the governing equation at time \( t = (n-j)\Delta t \). Subsequently in the pressure correction scheme, the pressure at time level \( n + 1/2 \) is used to calculate the velocity at time level \( n + 1 \) (see for instance (Wesseling 2001)): 

\[ \nabla \cdot u^{n+1} = \nabla \cdot u^* - \frac{\Delta t}{\rho} \nabla \cdot \left( \nabla p^{n+1/2} \right) = 0. \]  
\[ \nabla \cdot u^* = \frac{\Delta t}{\rho} \nabla \cdot \left( \nabla p^{n+1/2} \right) \]  

After the solution of the pressure \( p^{n+1/2} \) from the Poisson equation, equation (7), the final velocity \( u^{n+1} \) can be computed with help of equation (5). It should be noted that for a consistent formulation it is essential to use the form given by equation (7) and not to replace the term on the right hand side of equation (7) by Laplacian of \( p^{n+1/2} \), see for instance (Peric 2002). The algorithm above is well known and has with an explicit advection and diffusion step in principle third order time accuracy for the velocity and second order time accuracy for the pressure. The Poisson equation for the pressure is solved with help of Fourier Transforms in the axial and circumferential direction. The compact discretization in the radial directions results in full matrix vector system with is solved with dgetrf/dgetrs routines from the LAPACK library. For a more detailed description of the algorithm and the solution of the Poisson equation we refer to (Boersma 2005). For an explicit method in a cylindrical system the time step is in general limited to the grids pacing \( r\Delta \theta \) close to the centerline where \( r \approx 0 \). In previous studies this limitation is overcome by using in an implicit time integration method for the circumferential direction, see for instance (Eggels et al. 1993) and (Wu & Moin 2008). Here we have followed a different approach. From the work of (Lessen & Singh 1973) it is known that at \( r = 0 \) there is only a single Fourier mode, for larger values of \( r \) there are multiple Fourier modes. In the region close to \( r = 0 \) the number of Fourier modes in the Fourier expansion in the circumferential direction is gradually reduced from \( N_0 \) to 1. In this way the time step is not limited to the \( r\Delta \theta \) but to the \( r \) value of the first grid point, i.e. \( \Delta r/2 \).
Figure 1. The axial rms profiles normalized with $u_*$ as a function of $r$, in all cases $Re_*=360$.

**Code validation**

The first direct numerical simulation of pipe flow has been performed by (Eggels et al. 1993). The Reynolds number based on the friction velocity in the simulation was equal to 360. This DNS has been repeated several times by various researchers (Unger 1994) (Loulou et al. 1997) and recently (Wu & Moin 2008). The well resolved simulation of (Wu & Moin 2008) shows slightly different results that (Eggels et al. 1993), especially in the profiles for the axial rms profile. We will use the data of (Wu & Moin 2008) and (Eggels et al. 1993) to validate our code and get a feeling about the necessary resolution in the radial direction. Following Eggels (Eggels et al. 1993) our simulations are driven by a constant pressure gradient

$$\frac{1}{\rho} \frac{\partial p}{\partial z} = 4 \frac{u_*^2}{D}$$

where $u_*$ is the friction velocity and $D$ is the pipe diameter. In Figure 1 we show the axial root mean square profiles given by (Eggels et al. 1993), (Wu & Moin 2008) together with two simulations performed by us on a resolution of $54 \times 128 \times 256$ using a non-uniform grid in the radial direction and a simulation with $96 \times 128 \times 256$ using a uniform grid in the radial direction. Our simulations are considerable courser than the simulations of (Wu & Moin 2008). The good agreement between our results on a low resolution and the results of (Wu & Moin 2008) on a much higher resolution is an indication of the superior performance of our numerical method. In the case with the non-uniform grid the grids pacing near the wall is 1.3 wall units and near the centerline 4.4 wall units. Note that due to the staggered arrangement of the grid the first point is located $1.3/2 = 0.65$ wall units away from the wall. Later on when we will perform simulations for higher Reynolds numbers we will consider these numbers as the maximum allowed grid spacing. The difference between our results and the result of (Eggels et al. 1993) is likely caused by the to course numerical resolution in the radial direction and the low order numerical method used by (Eggels et al. 1993). Furthermore, we observe that a too low numerical resolution and/or accuracy results in an over prediction of the axial rms profiles. This is in general also observed in Large Eddy Simulations which tend to over predict the axial rms in pipe/channel flows, see for instance (Hamba 2003) and (Morinishi et al. 1998).

**3. Results**

In this section we will present results for three different Reynolds numbers. In general it takes a considerable amount of time before a simulation is fully developed. From earlier simulations and also from literature results it is estimated that a statistical steady state is only reached
after a simulation time \( t > 10D/u_* \). Especially on large computational meshes this will require a large amount of CPU time. To reduce the CPU time we run the simulations first on a course mesh, for roughly \( 10D/u_* \) after these 10 times scales the results are interpolated to a finer mesh on which the computations are continued for \( 0.5D/u_* \). These results are again interpolated to the final mesh. On this mesh the computations are continued for another \( 2D/u_* \). After these two timescales 60 data fields each separated in time by \( 0.05D/u_* \) are stored for further data analysis. With this procedure we save a considerable amount of CPU time.

### 3.1. \( Re=24590 \)

This Reynolds number has been chosen because it matches the experiment performed by (Den Toonder & Niewstadt 1997). Den Toonder reports a frictional Reynolds number of 1382. This number is used as input parameter for our simulations. The (final) computational mesh consisted of \( 224 \times 384 \times 512 \) grid point in the radial, circumferential and axial direction respectively. The simulations for this Reynolds number have been performed on 32 CPUs of a 48 Opteron system. A time step took approximately 20 seconds. (Den Toonder & Niewstadt 1997) reports a bulk Reynolds number of \( \approx 24850 \). If we compute the bulk Reynolds number from our simulations we find a value of 24590 which is very close to the experimental value. In Figure 2 we show the 1D energy spectra in the axial direction obtained from the DNS. The spectra show that the flow is fairly well resolved.

In Figure 3 the mean velocity profile obtained from the DNS and the experimental data of (Den Toonder & Niewstadt 1997) as a function of the non dimensional wall distance have been plotted. The agreement between simulation and experiment is excellent. In Figure 3 The root mean square profiles from the DNS and the experiment of (Den Toonder & Niewstadt 1997) are reported.

Den Toonder also reports results for the skewness and flatness of the radial and axial velocity components. In Figure 4 we reported the results from our DNS and we compare these with the data of (Den Toonder & Niewstadt 1997). There has been some discussion about the high value of the flatness close to the wall. In (Xu et al. 1996) it is argued that localized extremely high values of \( u'_r(>10\sqrt{u'_r u'_r}) \) could be responsible for this. In an experiment these values could be created by external disturbances (vibrations). If these observations are observed in a simulation there validity is questionable, because the extremely high energy levels in these areas will be poorly resolved because the grid resolution is not designed for it. Therefore we have chosen to
Figure 3. Left: The mean velocity profile obtained from the DNS ($Re_b = 24590$) together with the experimental data reported by (Den Toonder & Niewstadt 1997) ($Re_b = 24850$). Right: The rms and Reynolds shear stress profiles obtained from the DNS together with the experimental data reported by (Den Toonder & Niewstadt 1997).

Figure 4. Top Left: The skewness of the radial velocity component. Top right: The flatness of the radial velocity component. Bottom Left: The skewness of the axial velocity component. Bottom right: The flatness of the axial velocity component (lines DNS and symbols the experiment of (Den Toonder & Niewstadt 1997)).
Figure 5. The autocorrelations of the radial, circumferential and axial velocity component at different radial locations, Re=24590.

In the Figures 5 and 2 we present the autocorrelations and energy spectra at various radial locations. The autocorrelations halfway the computational domain at \( z = 4D \) are very small which shows that the computational domain is sufficiently long. The 1D energy spectra show that at least 2-3 decades of the turbulent kinetic energy in the flow are resolved. Both the autocorrelations and spectra are a good indication for the quality of this simulation. The results in this section give us sufficient confidence to study a higher Reynolds number.

3.2. Re=60800

In the previous sections we have validated our code against the DNS of (Wu & Moin 2008) and against the experimental data of (Den Toonder & Nieuwstadt 1997). In this section we will present results for a pipe flow with a Reynolds number of 60800. In this case the grid consisted of \( 430 \times 512 \times 1024 \) points in the radial, circumferential and axial direction. The grid spacing in the radial direct in non dimensional wall units is comparable to the one we have used for the Re = 5300 case. In Figure 6 we show the skin friction obtained from the three simulations at Re = 5300, Re = 24500 and Re = 61000. The near wall structure shows clearly a streak spacing of approximately 100 wall units and the and the length of the near wall structures seems to decrease with the Reynolds number. In Figure (Den Toonder & Nieuwstadt 1997) the mean velocity and rms profiles are shown together with the experimental results of (Den Toonder & Nieuwstadt 1997). Although we have only a limited amount of Reynolds number we do not observe the increase in avail as suggested by (McKeon et al. 2004) in their first
The local skin friction normalized by $\rho u_\tau^2$ obtained from the different simulations. The horizontal and vertical scales are given in non dimensional wall units.

Figure 6. The local skin friction normalized by $\rho u_\tau^2$ obtained from the different simulations. The horizontal and vertical scales are given in non dimensional wall units.

principal conclusion and we find support for the suggestion of decreasing peak value as stated by (Mochizuki & Nieuwstadt 1996). Finally, we show in Figure 8 the autocorrelations for this case. We observe quite some difference in the axial correlation between the $Re = 24800$ and the $Re = 61000$ case. This correlation indicates that in the high Reynolds number case long structure are present which are not present in the $Re = 60800$. This issue clearly needs further investigation.

4. Conclusion

In this paper we have reported results of a direct numerical simulation of turbulent pipe flow at a Reynolds number of approximately 24500 and 61000 using our newly developed and highly accurate DNS code. The results at the Reynolds number of 24500 show excellent agreement with the data of (Den Toonder & Nieuwstadt 1997). The high Reynolds number case shows a non vanishing auto correlation function n for the axial velocity component, while this is not the case in the low Reynolds (Re=24500) case. This could be an indication for the presence of long
**Figure 7.** Left: The mean velocity obtained from the DNS at $Re = 60800$, together with the experimental data of (Den Toonder & Niewstadt 1997). Right: The root mean square profiles obtained from the DNS at $Re = 60800$, together with the experimental data of (Den Toonder & Niewstadt 1997).

**Figure 8.** The autocorrelations of the radial, circumferential and axial velocity component at different radial locations, $Re=60800$. 
streamwise structures. Furthermore, we observe a slight decrease in the peak value of the axial rms which does not agree with the observation of (Morrison et al. 2004) but this is supported by the review article by agreement with the observation of (Mochizuki & Nieuwstadt 1996).

References


