Study of a transitional backward facing step flow with boundary layer manipulations

Günter Bärwolff
Technische Universität Berlin, 10623 Berlin, Germany

Abstract. With the aim of the drag reduction or a decreasing reattachment length in the wake of the step, acoustic manipulations of the boundary layer in front of the step were performed by experiments and numerical simulations ([1], [3] and [5]). The numerical investigations were done as direct numerical simulations and large eddy simulations. Beside the acoustic boundary layer manipulation over slits with certain frequencies, experiments were done with oblique backward facing steps or with moving boundaries to simulate oblique geometries. Based on the experiences of the above-discussed numerical simulation of a straight backward facing flow large eddy simulations with moving boundaries are under consideration.

1 Mathematical model

We consider a backward facing step channel and we investigate the flow for a Reynolds number of 3000 built with the velocity $u_0$ of the block inflow profile and the step height $H$ (see fig. 1). The top of the channel is considered as a slip wall and in the lateral direction a periodic behavior is assumed.

To describe the flow we get from the nondimensional Navier-Stokes equation the equation system for the filtered quantities

$$\frac{\partial \overline{\mathbf{u}}}{\partial t} + \nabla \cdot \overline{\mathbf{u} \mathbf{u}} = -\nabla \psi + \nabla \cdot 2\nu \overline{\mathbf{S}}$$

$$\nabla \cdot \overline{\mathbf{u}} = 0$$
where \( \nu \) is the total eddy-viscosity
\[
\nu = \frac{1}{Re} + \nu_t.
\]
\( \mathbf{u} = (u_1, u_2, u_3) \) is the velocity vector, \( \psi \) is the "pseudo"-pressure with
\[
\psi = \overline{p} + \frac{1}{3} \tau_{ij}, \quad \tau_{ij} = \overline{u_i u_j} - \overline{u_i} \overline{u_j},
\]
and for the strain rate tensor \( \overline{S} \) we have
\[
\overline{S} = (\overline{S}_{ij}), \quad \overline{S}_{ij} = \frac{1}{2} \left( \frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right).
\]
In the case of a direct numerical simulation we have \( \nu_t = 0 \) and there is no effect of filtering, and the equations (1) and (2) are the classical Navier-Stokes equations with \( \overline{\mathbf{u}} = \mathbf{u} \) and \( \overline{\mathbf{p}} = \mathbf{p} \) for an incompressible fluid.

If we don’t resolve all small structures by the spatial discretization \( \nu_t \) is the eddy-viscosity of a subgrid-scale model. We use a subgrid-scale model of Germano-type following Akselvoll/Moin [4]. For \( \nu_t \) we have
\[
\nu_t = C \overline{\Delta}^2 \overline{S}, \quad \overline{S} = \sqrt{2 \overline{S}_{ij} \overline{S}_{ij}}.
\]
For the Germano-type subgrid scale model we need two different filters to handle the equation of motion, the so-called grid filter \( \overline{G} \) and the test filter \( \overline{\tilde{G}} \), with the filter with \( \overline{G} \) of the test filter, assumed to be larger than that of the grid-filter. The quantity \( C \overline{\Delta}^2 \) we set
\[
C \overline{\Delta}^2 = -\frac{1}{2} \frac{L_{ij} M_{ij} >_y}{2} < \frac{M_{kl} M_{kl} >_y}{2},
\]
where,
\[
L_{ij} = \overline{u_i u_j} - \overline{u_i} \overline{u_j}, \quad M_{ij} = (\overline{\Delta}^2)^3 \overline{S}_{ij} \overline{S}_{ij} - \overline{S}_{ij} \overline{S}_{ij}.
\]
\( >_y \) indicates an average taken over the homogeneous spanwise direction.

The result of the subgrid-scale modelling is concentrated in the variable eddy-viscosity. The equation of motion is of the same type than the Navier-Stokes equation.

With the above-discussed boundary conditions which will be specified in the following section and appropriate initial conditions now we have a mathematical model for the transitional flow over a backward facing step.

2 Fluid physical task

The figures 2 and 3 show the two kinds of used manipulations, which results in Dirichlet-boundary conditions.
controlled backward facing step-flow

For the streamwise length $L_x$, the spanwise width $L_y$ and the vertical height $L_z$ we set $(L_x, L_y, L_z) = (22H, 6H, 12H)$. The choice of the vertical height is based on experiences concerning the dependency of the reattachment length on $L_z$. Only beyond $11H$ the dependency of the reattachment length can be neglected. The inlet section or the step has the length $L_s = 5H$. The inflow profile was assumed as a block profile with the velocity $u_{inflow} = \text{const}$. Non-uniform grid spacings for the streamwise and vertical directions are used. In the $x$- and the $z$-directions we consider a refined grid around the step. In $z$-direction fine spacings are used near the channel bottom. In the spanwise direction uniform grid spacings are used.

Because of the absence of detailed information about the blowing and suction during the acoustic control it was simulated by a sine function of the form $u_{jet} = A \sin(2\pi f t) \left( u_1 = u_2 = u_{jet} \right)$ and calculations for $A = 0.1 \ u_{inflow}$, $A = 0.01 \ u_{inflow}$ and $A = 0.001 \ u_{inflow}$ using a fixed frequency of $f = 50 \ Hz$ were done. Beside the acoustic boundary layer manipulation over slits with certain frequencies experiments are done with oblique backward facing steps or with moving boundaries to simulate oblique geometries. Based on the experiences of the numerical simulation of a straight backward facing flow direct and large eddy simulations with moving boundaries are under consideration.

We investigate both a moving upstream boundary in front of the step (lateral velocity $u_u$) and a moving boundary behind the step in the downstream region of the bottom of the channel (lateral velocity $u_d$, see fig. 3):
The simulations are done for a wide range of lateral boundary velocities, i.e.,
from \(v_{\text{initial}} = 0.5 u_0\) to \(v_{\text{final}} = 2.0 u_0\) with \(u_0 = u_{\text{inflow}}\) as the inflow profile velocity.

3 Numerical solution procedure

Equations (1) and (2) may be discretized in spatial dimensions by a finite volume method on staggered grids for the velocity components \(u_i\) of the velocity vector and the pressure \(\psi\). The finite volume discretization results in a ode system for the velocity components at every grid point

\[
\frac{\partial u_h}{\partial t} + \nabla_h \cdot u_h u_h = -\nabla_h \psi_h + \nabla_h \cdot 2\nu \nabla_h \psi_h, \quad \nabla_h \cdot u_h = 0, \tag{5}
\]

written in vector form. The approximation is conservative and is of second order \((O(h^2))\). The time integration is done either by a leapfrog method or an Adams-Bashforth method. Thus, we have to solve in every time step the equation system

\[
\frac{u^{n+1}_h - u^n_h}{\tau} = -\beta \nabla_h \psi^{n+1}_h, \quad \nabla_h \cdot u^{n+1}_h = 0, \tag{7}
\]

where \(u^n_h\) is a given result of an estimation by a predictor step, \(\tau\) is the time step and \(\beta\) is a constant depending on the time integration method used (in the case of the Adams-Bashforth method \(\beta = 1\), see also [2]). The equations (7) are equivalent to the equation:

\[
-\Delta_h \psi^{n+1}_h = -\frac{1}{\tau \beta} \nabla_h \cdot \hat{u}_h. \tag{8}
\]

\(u^{n+1}_h\) is then given by an explicit fill-in step following (7). Thus, \((u^{n+1}_h, \psi^{n+1}_h)\) can be found either iteratively or by the solution of a Poisson equation for \(\psi^{n+1}\) followed by an explicit fill-in step to get \(u^{n+1}\). Both possibilities are implemented in the sequential codes (see [2]). In our parallel code implementation the iterative solution method to get \((u^{n+1}_h, \psi^{n+1}_h)\) is realized as a solver for the equation system (7). The numerical simulations are done with a finite volume method on staggered grids which is parallelized on a Cray T3D/T3E using fast Cray-specific shared memory transfer utilities or the platform independent MPI library. The method is of second order in space and time. The mass conservation per time step was realized by a pressure-velocity iteration method.

Non-uniform grid spacings for the streamwise and vertical directions are used. In the \(x\)- and the \(z\)-directions we consider a refined grid around the step. In

\footnote{For a compact description we renounce the lines over \(u, \psi\).}
controlled backward facing step-flow

z-direction fine spacings are used near the channel bottom. In the spanwise direction uniform grid spacings are used.

The parallelized numerical model (DNS/LES) was validated by comparisons of the numerical DNS/LES results of [5] and the experimental data of [1] and [6]. The following figures 4-7 show the result of comparison for the mean velocities and the rms-values. For these comparisons a neutral, non-manipulated backward facing step flow for the transitional Reynolds number $3000$ was considered. For the direct numerical simulations about 10 million grid points are needed for the spatial discretization ($512 \times 132 \times 196$). In the case of the large eddy simulations it is possible to work with about 500 thousands of spatial grid points ($260 \times 68 \times 84$). The experimental results were produced by LDA-measurements.

The performance of the parallelized code ($1.3 \times 10^7$ gridpoints, 100 time steps, 25 pressure-velocity iterations) on different mpp systems is shown in table 1. In addition to the named mpp systems we are also very successful using the SGI O 200 for the solution of moderate problems with less than $10^6$ grid points.

![Fig. 4. $u_{mean}$ at $x = 5H$](image1)

![Fig. 5. $w_{mean}$ at $x = 5H$](image2)

![Fig. 6. $u_{rms}$ at $x = 5H$](image3)

![Fig. 7. $w_{rms}$ at $x = 5H$](image4)

4 Results

The results of the reattachment length reduction in the case of the moving upstream boundary in front of the step show a good agreement with
Table 1. Performance of the FV code on different mpp systems

<table>
<thead>
<tr>
<th>#procs</th>
<th>time t [sec]</th>
<th>mflops</th>
<th>$t_{CPU}/t$</th>
<th>$S_p = t_{CPU}/t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>J90</td>
<td>1</td>
<td>16428</td>
<td>92</td>
<td>0.27</td>
</tr>
<tr>
<td>J90</td>
<td>4</td>
<td>4173</td>
<td>357</td>
<td>1.03</td>
</tr>
<tr>
<td>J90</td>
<td>8</td>
<td>2244</td>
<td>673</td>
<td>1.95</td>
</tr>
<tr>
<td>J90</td>
<td>16</td>
<td>1390</td>
<td>1087</td>
<td>3.15</td>
</tr>
<tr>
<td>C90</td>
<td>1</td>
<td>4380</td>
<td>338</td>
<td>1.00</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>#procs</th>
<th>proc $x$ * proc $y$</th>
<th>time t [sec]</th>
<th>mflops</th>
<th>$t_{CPU}/t$</th>
<th>$S_p = t_{CPU}/t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>T3D</td>
<td>32</td>
<td>329</td>
<td>443</td>
<td>1.31</td>
<td>1.00</td>
</tr>
<tr>
<td>T3D</td>
<td>64</td>
<td>1671</td>
<td>886</td>
<td>2.62</td>
<td>1.99</td>
</tr>
<tr>
<td>T3D</td>
<td>128</td>
<td>871</td>
<td>1700</td>
<td>5.03</td>
<td>3.82</td>
</tr>
<tr>
<td>T3D</td>
<td>256</td>
<td>437</td>
<td>3387</td>
<td>10.62</td>
<td>7.61</td>
</tr>
</tbody>
</table>

the known experimental measurements. The large eddy simulations with the moving boundary behind the step are in the beginning and the first analysis shows no remarkable influences of the lateral boundary velocity $v_d$ to the position of the reattachment point $X_r$. A significant reduction of the reattachment length was not found. The figures 8 and 9 show the resulting mean wall shear stress distribution ($u_\tau$) of a large eddy simulation with a moving upstream boundary (emulated oblique step flow) using the lateral wall velocity $v_{lateral} = u_{inflow}$ compared to $u_\tau$ of the unmanipulated step flow (neutral step flow). There were no visible differences in the wall shear stress distribution of a LES and a DNS. The investigation of the shear layer was focused to the experimental found lateral structures in the region $x = 0 \, H$ (position of the step) to $x = 3 \, H$. Both the LES and the DNS could reproduce the observed structures. But a detailed investigation is necessary, because in our averaging strategy the lateral direction was considered as a homogeneous one to increase the averaging process and thus we don’t have time averaged information about the velocity and pressure in the lateral direction. The fig-

Fig. 8. $u_\tau$ of the “oblique” step flow    Fig. 9. $u_\tau$ of the “neutral” step flow
Figures 10 and 11 show the pressure contours in the plane $x = 1 \, H$ from instantaneous snapshots from the LES and DNS.

![Contour plot P](image1)

**Fig. 10.** Pressure contours at $x = 1 \, H$, LES

![Contour plot P](image2)

**Fig. 11.** Pressure contours at $x = 1 \, H$, DNS

The figures 12 and 13 show the pressure contours behind the step in the plane $z = 1 \, H$ parallel to the channel bottom from an instantaneous snapshot of the LES and DNS.

Further results especially colour figures of pressure and vorticity contours of the investigated backward facing step flow are presented on my webpage [http://www.math.tu-berlin.de/~baerwolf](http://www.math.tu-berlin.de/~baerwolf).
The comparison of the LES and DNS of the neutral backward facing step flow without a manipulation \( A = 0 \) to the controlled cases with a fixed frequency of \( f = 50 \, \text{Hz} \) and a Reynolds number of 3000 for an amplitude of \( A = 0.01 \, u_{\text{inflow}} \) gives a decrease of the reattachment length \( X_r \) of more than 30%. This agrees with the experimental experiences of Huppertz (1994).

The experimental found decrease of the reattachment length \( X_r \) and the development of lateral structures behind the step in the case of a manipulation by a moved boundary were confirmed by the numerical simulations. Because of the good agreement of the large eddy and direct numerical simulation results with the experimental data of [1] for qualitative parameter studies the LES will be sufficient.

To do further investigations of the lateral structure it is necessary to have pure time averaged informations of the flow \( \langle u_{\text{mean}} \rangle \) and \( \langle p_{\text{mean}} \rangle \) and that’s what we have to do next.
References


