Lattice Boltzmann Simulation of non-Darcy Flow in Porous Media

Manuel Hasert\textsuperscript{a,*}, Jörg Bernsdorf\textsuperscript{a}, Sabine Roller\textsuperscript{a}

\textsuperscript{a}German Research School for Simulation Sciences GmbH and RWTH Aachen University, Schinkelstr. 2a, 52062 Aachen, Germany

Abstract

Flow through porous media at low Reynolds numbers has been studied in detail with the Lattice Boltzmann Method (LBM) for applications such as groundwater flow, pollution transport or adsorption processes. In contrast to that, medium to high Reynolds number flow through porous media, which occurs in many areas of industrial engineering, has not yet widely been investigated on a microscopic level by detailed numerical simulations.

In this paper, we focus on air flow through a porous medium, because our far goal entails the simulation of acoustic excitations from the turbulent flow leaving the porous medium. We validate the LBM at Reynolds numbers beyond the limit of Darcy’s law, and compare the results of direct numerical simulation with those achieved by applying a Smagorinsky-type large eddy turbulence model. For this, we performed flow simulations through a generic (periodic) porous medium at a variety of resolutions to investigate the effect of LES modelling at lower mesh sizes, where the subgrid scale effects become important.

Keywords: Lattice Boltzmann Method, non-Darcy flow, porous medium, turbulence, aeroacoustics, TRT, LES Smagorinsky

1. Introduction

The flow behaviour through porous media has long been of interest to scientists and engineers due to the importance of its applications [1]. We focus on an industrial application, namely the noise emission of a pneumatic device with a porous medium acting as a silencer. Our far goal is to simulate the sound generation from the exhaust process, which results from the air flow being forced through the porous medium silencer. The porous medium flow and the fact that aeroacoustics is a weakly compressible phenomenon makes the Lattice Boltzmann Method (LBM) an ideal candidate for solving this problem. In this paper however, we will only address the pore scale flow and recapture, that the aeroacoustics are reproduced accurately. We will not show the actual acoustic emission of the turbulent outflow.

When solving aeroacoustic problems, one is confronted with the notion of scale separation. The main flow regime is dominated by convective forces which transport fluid fluctuations at an average velocity corresponding to the main flow. This regime is characterised by small length scales $O(\text{mm})$ and a large amount of energy. The structures are dissipated and diffused and have little influence on the fluid behaviour at a long distance. The acoustic regime captures the pressure fluctuations with little energy which are transported with the speed of sound. The dissipation is low, so the transport can occur over a large distance and the acoustic far field is calculated in the range of $O(\text{meters})$.

\textsuperscript{*}Corresponding author

Email address: m.hasert@grs-sim.de (Manuel Hasert)
The acoustic and main flow field can either be solved separately, or coupled as in direct acoustic simulations (DAS). In a separate solution of the scales, the flow of information is modelled such that it only occurs from the main flow field to the acoustic field. We focus on the direct resolution of both scales in order to model eventual responses of the acoustic to the main flow field. The fluid is pressed through the porous structure of the silencer, where energy is dissipated. The turbulent vortical outflow is the main cause of acoustic noise. This micro-pore scale adds a third regime in the order of $O(10\mu m)$. We try to solve this discrepancy between the length scales by modelling the flow through the porous medium with under-resolved channels of just three grid points. The validity of Darcy’s law, which relates the pressure drop in a porous medium linearly to an average velocity, is widely accepted for low pressure drop differences or flow velocities. This relation however, is only valid for very low Reynolds numbers $Re < 1$. Velocity terms of higher order have been added to Darcy’s equation in Forchheimer’s formulation to account for the non-linear relation between pressure drop and velocity, that dominates at higher Reynolds numbers. These higher order terms can be seen as an extension of Darcy’s law for inertial effects, vanishing for lower velocities, where viscous effects dominate. The effects in these non-Darcian regimes are recognized in many fields of applications [2], such as ground water movement [3] or oil recovery [4] and also for chemical or combustion processes. The LBM has been widely used for flow calculations through porous media. However, high Reynolds number flows in these media still remains a relatively open field, especially when it comes to turbulence affected flows. The goal of this paper is to identify the required resolution and the accuracy of the LBM extended with a turbulence model, when the porous medium is under-resolved. The remainder of this paper is structured as followed. Section 2 serves as an overview over the literature and gives a description of flow regimes occurring in porous media, Section 3 introduces the numerical method and the used turbulence model and boundary conditions. An overview over aeroacoustic considerations is given in Section 4 and turbulence description and modelling is introduced in Section 5. Validations of porous media flow for various resolutions and Reynolds numbers for a generic porous medium are presented in Section 6 and the engineering application is described in detail in Section 7. An outlook on future work is given in Section 8.

2. Overview over porous media flows

In porous media, the Reynolds number $Re_D$ is expressed with the following specific quantities. In consolidated media, the characteristic length $l_D$ takes the size of a characteristic pore size. The characteristic velocity $u_D$ is taken to be an average velocity inside the pores, the so-called seepage velocity $u_D = \frac{q}{1-\phi}$ with the flow rate $q$ and the porosity $\phi$. The porosity $\phi$ denotes the ratio of fluid or pore volume $V_f$ to total volume $V_{tot}$ in the porous medium as $\phi = V_f/V_{tot}$. The Reynolds number then takes the form

$$Re_D = \frac{u_D \cdot l_D}{\nu}$$

(1)

with the kinematic viscosity $\nu$ of the fluid. The classification of flow regimes inside consolidated media depends on a microscopic or macroscopic point of view and differs in the literature [5]. On the microscopic scale, Dybbs and Edwards [6] observed four flow regimes: a laminar regime, which can be subdivided into a steady and unsteady part, a transitional and a fully turbulent part. On the macroscopic scale, there is the linear, or Darcy regime, the quadratic or Forchheimer regime and the fully turbulent regime.

1 Viscous, linear regime. In small Reynolds number flows $Re < 1$, the inertial forces are small compared to the viscous forces and can be neglected. The Navier-Stokes equations reduce to a pressure gradient and skin friction, i.e. shear stress inside the pores. Shear stress is proportional to the velocity gradient inside a pore, which macroscopically yields a linear relation of fluid velocity and pressure drop. This leads to the law of Darcy

$$-\frac{\partial p}{\partial x} = \frac{\mu}{k}\bar{u}_D$$

(2)

with the superficial average (seepage) velocity $\bar{u}_D$, the dynamic viscosity of the fluid $\mu$ and the permeability $k$ of the porous medium. As stated above, this relation is only valid for low Reynolds numbers. On the macroscopic view, this regime equals the Darcy regime, because Darcy’s law is valid.
Flows with higher Re are dominated by non-linear interactions between the inertial, viscous and pressure forces, leading to a non-linear relation between velocity and pressure drop.

2 Weakly inertial, laminar steady regime. In this regime, the flow is still laminar and steady. With increasing Reynolds number $1 < Re < 10$, the inertial forces, which are non-linear in the Navier-Stokes equations, affect the momentum conservation. Macroscopically, a non-linear relation between the pressure drop and the flow velocity can be observed, which was first described by Forchheimer as [7]

$$\frac{\partial p}{\partial x} = \frac{\mu}{k} u' D + \beta \rho u_D^2$$

(3)

The constant factor $\beta$ is known to be mainly dependent on the flow path tortuosity and is usually recovered from experiments. In this regime, the boundary layer starts to develop while an intertial core flow grows with the Reynolds number. This core flow is responsible for the non-linearity. The core flow increases and takes a greater influence on the flow with increasing Re. This regime can persist until $Re < 150$.

3 Inertial, transitional, unsteady regime. Inertial forces grow with the Reynolds number and cause instabilities. Dybbs and Edwards [6] observed regular fluctuations and found the regime $150 < Re < 300$ and transitions to turbulence may occur.

4 Fully turbulent regime. At Reynolds numbers $Re > 300$, the inertial forces dominate over the viscous terms and create random, unsteady flow patterns and turbulence. The turbulent flow occurs with higher permeability only.

Friction coefficient. The friction factor is a dimensionless number for quantifying the pressure loss in porous media flow. It is a function of the pressure drop and the porosity $\phi$

$$f = \frac{\partial p}{\partial x} = \frac{dP}{\rho_0 u^2} \frac{\phi^3}{1 - \phi}.$$  

(4)

The friction coefficient $\Lambda$ is calculated from the friction factor by multiplication with the Reynolds number $Re_D$ as $\Lambda = f \cdot Re_D$. It can be seen as a kind of dimensionless permeability and will be used throughout this paper.

3. Numerical Method

The Boltzmann equation [8] describes the time evolution of a particle distribution function $f$, which defines the probability to find a particle at a given time $t$ at a certain position in space $\vec{x}$ with a given velocity $\vec{v}$. The Lattice Boltzmann equation is a special discretisation hereof, working on a discrete set of directions, which form the lattice for the propagation of the discrete distribution functions $f_i(\vec{x},t)$ at each time step. For the typically used D3Q19 lattice, $q = 19$ discrete velocities $\vec{e}_i$ are chosen. The simplified LBM equation without a forcing term reads

$$f_i(\vec{x} + \vec{e}_i \tau, \vec{e}_i, t + \delta t) = f_i(\vec{x}, \vec{e}_i, t) - C = \begin{cases} \frac{\delta_t}{\tau} (f_i(\vec{x}, \vec{e}_i, t) - f_i^{eq}(\rho, \vec{v})) & \text{for BGK} \\ -\frac{\delta_t}{\tau_v} n_i^+ + \frac{\delta_t}{\tau_{od}} n_i^- & \text{for TRT with } \tau_v = \tau, \tau_{od} = \frac{5}{8} \frac{\tau - 1}{\tau - 1}. \end{cases}$$

(5)

The collision operator $C$ is in the simplest form approximated with the Bhatnagar Gross Krook (BGK) model, where the distribution functions are relaxed towards the thermodynamic equilibrium distributions $f^{eq}$ with the relaxation time $\tau$. The Navier-Stokes equations are recovered in the small Knudsen number limit by the application of the Chapman-Enskog procedure. The viscosity of fluid is expressed via the collision frequency $\tau$ as

$$\nu = c_s^2 \left( \frac{\tau}{2} - \frac{1}{2} \right) \frac{\delta_t}{\delta_i}.$$  

(6)

The model is valid for isothermal and weakly compressible assumptions. The isothermal assumption relates pressure and density with the speed of sound as $p = \rho c_s^2$. 
3.1. BGK incompressible collision model

For the single-relaxation time BGK model, He and Luo [9] introduced an incompressible formulation. A local pressure distribution function \( p_i = c_s^2 f_i \) serves as an independent variable with a slightly altered local equilibrium distribution function

\[
\tilde{p}_i = w_i \left( \delta p + p_0 \left( \frac{1}{2} \frac{(\tilde{e}_i \cdot \tilde{u})^2}{c_s^4} + \frac{(\tilde{e}_i \cdot \tilde{u})^2}{2c_s^4} - \frac{\tilde{u}^2}{2c_s^2} \right) \right)
\]

using a pressure fluctuation \( \delta p \) around the mean pressure \( p_0 \). In contrary to the standard BGK formulation, this model does not include the fixed relation between pressure and density. With the standard formulation a flow rate increase can be observed under strong pressure gradients. The incompressible model, pressure and density are primarily defined by the constant rate \( p_0 \) and the flow rate is kept constant over the computational domain[10].

3.2. Two relaxation time collision model

An alternative relaxation model using two relaxation times (TRT) was proposed by Ginzburg [11]. The TRT model performs the collision on a symmetric (even) \( n^+ \) and anti-symmetric (odd) part \( n^- \) of the collision operator \( C \) with two distinct relaxation times \( \tau_{ev} \) and \( \tau_{od} \), of which \( \tau_{ev} \) can be selected freely as in the BGK model and \( \tau_{od} \) is chosen for the wall positions to be independent of \( \tau \) according to Eq. 5. The TRT model can be seen as a sub-class of the Multiple Relaxation Time (MRT) model [12]. The explicit transformation to the moment space as in the MRT model is not necessary, which decreases the computational effort comparable to BGK models. With this model, we also use an incompressible formulation, simulating the pressure variance \( \delta p \) about a mean reference value of \( p_0 \) as in the BGK incompressible model.

3.3. Boundary conditions

Due to the high superficial area in porous media, the influence of boundary condition effects has to be considered. Different wall boundary conditions have been developed [13] which differ in accuracy, mass conservation properties and the computational effort. Pan [14] has emphasized the importance of a careful choice of wall boundaries and showed the dependence of the wall position on the relaxation parameter, especially for SRT models. For the TRT model, we choose the set of relaxation parameters \( \tau_{ev}, \tau_{od} \) according to the Magic parameters in [11] which cures the dependency of the wall position. The non-slip wall boundaries which are incorporated as half way bounce-back boundaries. For the inflow we use velocity boundaries and at the outflow the corresponding pressure boundary conditions as proposed by Bouzidi [15]. These boundary conditions yield a second order accuracy, which corresponds to the order of the numerical scheme.

4. Aeroacoustic sound generation

Since Lighthill’s works [16] it is well known, that the sound created by turbulent shear flows resembles an acoustic quadrupole source. In this section we give a short overview of the validation of the numerical scheme for aeroacoustic phenomena of such type. Acoustic waves are small pressure fluctuations which are several orders of magnitude smaller than the fluctuations of the flow field, propagated with the speed of sound \( c_s \). Several authors have shown the validity of the LBM for acoustic wave propagation processes[17]. It was found to be a low dispersion and dissipation scheme which has comparable accuracy to a higher order finite difference scheme in space and time by Marić et al. [18]. The aeroacoustic capabilities of the LBM have also been shown in Wilde [19], Crouse et al. [20], Li et al. [21] and Hasert et al. [22] and will quickly be revised here.

A well studied standard test case for which resembles a quadrupole involves a pair of vortices spinning around a common center while emitting acoustic waves. The behaviour of the system is equivalent to a spinning acoustic quadrupole which is the typical sound production mechanism of turbulent shear flow. The vortices have equal circulation \( \Gamma \) and spin on a circle with radius \( r_0 \) at angular speed \( \omega = \Gamma/(4\pi r_0^2) \) and a rotating Mach number \( M_{on} = u/c_s = \Gamma/(4\pi r_0 c_s) \). Müller & Obermeier [23] derived an analytical solution for the acoustic far field, where the vortices are considered as point sources in an inviscid fluid. One solution for the incompressible near flow field and one for the acoustic far field are matched asymptotically in an intermediate domain to give an asymptotically valid solution expressed by a flow potential \( \Phi \) with \( \nabla \Phi = \vec{u} \)

\[
\Phi(z, t) = \frac{\Gamma k^2 r_0^2}{8} H_2^0(kr)e^{i2(\omega t - \theta)}
\]
with \( k = 2\omega/c_s \) and \( H_2^{(2)} \) being the second kind Hankel function of order two. The acoustic pressure fluctuations in the far field solution can be extracted from the equation above as the real part of the complex potential \( \Phi \) and the initial velocity field can be obtained by differentiating with respect to \( z \).

### 4.1. Simulation setup and numerical results

The simulation is performed on an equidistant grid with a total size of \((400r_0)^2\) with the rotational radius \( r_0 \) resolved with \( \delta_x = 10 \) grid points. The initial Mach number at the core radius \( r_c \) was \( Ma_{\text{Rot,ini}} = 5.196 \cdot 10^{-2} \) and the viscosity \( \nu = 4.168 \cdot 10^{-5} \) was chosen. The initial velocity field is derived from Eq. 8 and a constant pressure is set. This introduces errors, which dissolve after some vortex revolutions.

The analytical solution described above is compared to a temporal and spatial distribution of the acoustic pressure from the simulation results. As a direct acoustic simulation is performed, the sum of the incompressible and acoustic pressure is calculated. The acoustic pressure is obtained by subtracting the incompressible solution from the total pressure level. We use dimensionless parameters for pressure \( p'/\left( c_s^2 \rho_0 \right) \), radius \( r/r_0 \) and time \( tc_s/r_0 \).

### Spatial pressure behaviour

Figure 1(a) shows a snapshot at time step \( t = 750 \) of the acoustic pressure, from the center of rotation through a vortex center to the domain limit. Near the vortex center, the amplitude error \( \varepsilon_\alpha \) is around 30% but decreases when advancing outside from the center of spinning motion. The error near the vortex centers might be due to errors in the analytical solution [22]. In the acoustic near field, the amplitude is slightly underpredicted, whereas the phase shifts slightly.

### Temporal pressure behaviour

In Figure 1(b) the temporal acoustic pressure at a distance \( r = 150r_0 \) from the center is shown. Due to initial perturbations from the constant pressure initialization, the phase had to be matched to the analytical solution. The amplitude is slightly lower than predicted in the analytical solution. The phase matches well at \( t = 800 \) but then the wave periods increase. The reason for this increase is considered to be due to the neglected viscosity in the analytical solution. In the simulation however, the vortex revolution speed is decreased over time by the inclusion of viscosity. This results in an oscillation frequency that decreases over time in the numerical experiments. The decreased amplitude can also be attributed to the viscosity effects as the dissipation is added to the spatial energy distribution of the spreading waves.

### 5. Turbulent flow simulation in porous media

It is nowadays widely accepted that flow through porous media is prone to develop a turbulent behaviour under certain conditions (see e.g., [24]). The Reynolds number for the laminar-turbulent transition can be assumed significantly below \( Re \approx 2300 \), the starting point of transition for pipe flow. A reason for this is the flow disturbance caused by the porous media, which can trigger any kind of irregular behaviour such as vortex shedding at Reynolds numbers in the order of below \( Re = 100 \).

When assuming a turbulent flow regime, the mesh resolution of a numerical simulation must be sufficiently fine to capture all scales of turbulent structures. A good rule of thumb is the estimation of Kolmogorov for the ratio of largest to smallest vortices for with the Reynolds number

\[
\frac{L^{4/3}}{\eta} \propto Re
\]
For the considered medium Reynolds numbers of \( Re \approx 200 \) this estimation predicts that a mesh resolution must be able to cover about one order of magnitude from smallest to largest vortex structures. The LBM enables us to perform direct numerical simulations (DNS) at sufficiently high resolution in complex geometries. This allows us to gain insight into the development of complex transient flow structures at the onset of turbulence as demonstrated in the following chapter 6.

5.1. Turbulence modelling

From the large set of turbulence models the subgrid models are particularly suitable for implementation into an LBM flow solver. A widely used approach is the Smagorinsky subgrid model [25]. For this model, one assumes the existence of turbulent flow structures of different orders of magnitude in size. Due to limited mesh resolution, the smallest vortices are not resolved by the mesh applied in the numerical simulation, therefore their dissipative effect is captured by a locally increased flow viscosity. This so called turbulent viscosity \( \nu_{turb} \) results from additional dissipation caused by the turbulent eddy motion and decay. The turbulent eddy viscosity is defined as

\[
\nu_{turb} = C_s^2 \Delta x^2 \bar{Q}
\]  

where \( C_s \) is a constant defined by Smagorinsky [26] and usually takes the values \( C_s = 0.05...0.2 \). A variety of subgrid models exist, whereas the quality of the results depend on the application and the implementations [27].

The model operator \( \bar{Q} \) depends on the employed model [28], and is in the Smagorinsky model a function of the stress tensor \( S_{ij} \). These subgrid scale based turbulence models can naturally be incorporated into the LBM by altering the relaxation time \( \tau_0 \), which defines the viscosity as defined in Eq. 6 and hence, the resulting viscosity reads

\[
\nu_{total} = \nu_0 + \nu_{turb} = \frac{2\tau_0 - 1}{6} + \tau_{turb} \quad \tau_{turb} = 3\nu_{turb} = \frac{1}{2} \left( \sqrt{\tau_0 + 18C_s^2 \Delta x^2 \bar{Q}} - \tau_0 \right) \quad \bar{Q} = \frac{1}{2} \sum_{ij} \vec{Q}_{ij} \cdot \vec{Q}_{ij}
\]

We implemented the Smagorinsky model for the TRT model by additionally calculating the equilibrium distributions as in the BGK model. The strain rate calculation is then analogous to the BGK implementation.

A secondary highly welcome effect of the Smagorinsky model is the stabilization of the LBM simulation, because higher viscosity is realized via local increase of the relaxation time \( \tau \).

To the best knowledge of the authors, very few has been published concerning LES turbulence modelling with the LBM for flow through porous media. Kuwahara et al. [29] have investigated turbulence in a regular, periodic array of square cylinders with the usage of LES simulations.

A careful comparison of a properly resolved DNS with LES simulations at reduced mesh resolutions is therefore performed in the following chapter. Besides a qualitative comparison of flow patterns, the dimensionless pressure loss (friction factor) for the flow through the porous structure is taken as a measure. Ideally, an LES simulation should recover the same friction factor as the DNS, but at a much lower resolution.

6. Porous media simulations with the Lattice Boltzmann Method

There has been a vast amount of studies about flow simulations using the LBM for porous media geometries. The simple marker-and-cell method in combination with bounce-back wall boundary conditions (or more advanced approaches) easily allows setting up highly complex geometric boundaries, and therefore makes LBM a natural choice for porous media flow. Alas, high Reynolds number flow simulation are prone to instabilities with the LBM [30] and the choice of the simulation parameters is restricted:

1. The viscosity can be tuned by the relaxation time \( \tau \). Low viscosities are modelled by converging the \( \tau \) towards its stability limit, namely \( \tau \rightarrow 0.5 \). Especially when a larger amount of the flow area is occupied by solid fraction, simulations have a tendency to become unstable.
2. The compressibility error increases with the flow velocity with an order of \( O(Ma^2) \).
3. Resolution. The characteristic length of $Re$ is given in terms of lattice units. A higher resolution leads to a higher Reynolds number by $O(n)$, and relaxation times closer to the stability limit can be used. However, the computational effort increases with an order higher than $O(n^3)$.

Considering these facts, one is severely restricted in achieving a possibly high Reynolds number. MRT models can help to increase the achievable $Re$ about a factor of 4 [31]. Subgrid turbulence models like the LES Smagorinsky model locally increase the viscosity in areas where the fluid undergoes large strain rates, which helps in stabilizing the numerical scheme.

6.1. Simulation setup

We employ a scalable generic porous geometry to study the behaviour of the flow and the dependency on simulation parameters such as the resolution. The concept of unit cells is commonly used in mineralogy and describes the arrangement of atoms in the smallest regular structure. The porous array is made up of alternating body-centered (bcc) and face-centered (fcc) cells in the $x$-direction, where the domain size $g_y = g_z$ takes the dimension of the unit cell. We then set the center of cubes with edge length $d_C = 0.6 \cdot g_y$ to the atom positions. The domain is extend in $y$- and $z$-direction into infinity by applying periodicity. This leads to a porosity of $\phi = 0.5148$ with a pore channel height of $d_P = 0.4g_y$. For the calculation of the Reynolds number $Re_D$ we use the channel height $d_P$, the average velocity inside the porous medium $u_D = q/(1 - \phi)$ and the fluid viscosity $\nu$.

Before and after the porous medium, the computational domain is extended to account for inflow and outflow effects. Especially at higher Reynolds numbers, recirculation and vortex shedding occurs after the porous medium, which is why the free flow domain behind the medium is extended to around $0.4g_x$. An inlet velocity $u_{in}$ is assigned with a constant velocity profile, whereas at the outlet, a reference pressure of $p_0$ is set.

6.2. Friction coefficient and stability

The purpose of these initial simulations is the validation of our LBM implementation for flow through the generic porous media at a wide range of Reynolds numbers, as well as investigations concerning the stability. We performed simulations for a set of Reynolds numbers ($0.001 \leq Re \leq 100$) at three different mesh resolutions.

At first, we investigate the dependency of the friction coefficient $\Lambda$ from the Reynolds number. As can be seen in Figure 3(a), the linear flow regime (1) can be observed up to a Reynolds number of $Re < 1$. In the range of $1 < Re < 10$ the Forchheimer-regime (2) starts, developing into a quadratic dependency of the friction coefficient on the Reynolds number. According to Dybbs[6], the transition towards turbulent flow occurs at $150 < Re < 300$ (3). Our simulation results correspond qualitatively well to those from other comparable LBM simulations [32].

For a given resolution of the generic porous medium, the maximum achievable Reynolds number for a direct numerical simulation (DNS) is limited by stability issues, as stated above. In order to estimate the minimum required resolution for each model, we perform studies on the stability of the LBM with various inlet velocities $u_{in}$, resolutions $d_P$ and viscosities $\nu$.

The maximum Reynolds number we were able to achieve with DNS increased with the channel diameter $d_P$ of the porous media, as shown in Figure 3(b). The highest Reynolds number we could reach with DNS, $Re_{DNS,max} = 182$ at the resolution of $d_P = 40$, is used as the reference solution for our further studies. This Reynolds number corresponds to the porous flow regime 3, where transition to turbulent flow can be observed.
6.3. **LES-DNS comparison for non-Darcy flow**

In a next step, we investigated the quality of the LES simulations at different mesh resolutions in comparison with the high-resolution DNS reference result at \( Re = 182 \). The resolution in terms of the porous channel height is varied from \( d_P = 4 \) to 40 grid points. All simulations were done using the TRT collision operator, since it is known that for the BGK-scheme the exact wall position is dependent on the relaxation parameter \( \tau \) [14]. For comparison, BGK results were produced as well.

6.4. **Numerical results**

The TRT-LES model shows good agreement of the friction coefficient with the DNS solution for all resolutions (see Figure 4), with a maximum error of below 15%. Even at the coarsest resolution of \( d_P = 4 \), the deviation is as small as 11%, whereas a general trend for better results at higher resolutions can only roughly be identified from our data. This is probably due to an overlap of mesh convergence and LES modeling effects, which will be investigated in a future study. At the highest resolution of \( d_P = 40 \), the friction coefficient of the DNS and LES simulation only differ about 0.15%. The BGK model leads to a large discrepancy of \( \Lambda \) for low resolutions, the coarsest resolution of \( d_P = 4 \) has a deviation from the reference friction coefficient of 156%. With increasing resolution, also the BGK simulation converges towards the reference value, due to an increasingly smaller contribution of the non-correct wall position at higher resolutions. In Figure 6.4 we compare the flow pattern for LES and DNS in terms of velocity vectors at an x-y-plane above the central bcc cube (illustrated in Figure 2 as cutting plane A). The DNS solution at the high resolution \( d_P = 40 \) is characterised by an irregular distribution of vortices of different size, as expected at the onset of turbulence for the given Reynolds number of \( Re = 182 \). Larger eddies of around 20 grid points and smaller ones of around 5 grid points are clearly visible. The velocity profile of the low resolution LES simulation at \( d_P = 12 \) only shows larger vortices and exhibits a symmetric flow profile. As expected, small and irregular vortex structures are either not resolved by the coarse mesh or damped out by the LES scheme. As can be seen in Figure 5(c), the eddy viscosity is increased especially near walls and at the cube edges. It is worth mentioning that for the given Reynolds number, DNS simulations for lower resolutions than \( d_P = 40 \) were not stable and results could only be produced with the help of LES.

![Figure 3: Reynolds number studies with various resolutions](image-url)

![Figure 4: \( \Lambda \) for various resolutions, LES and DNS runs with TRT and BGK model.](image-url)
6.5. Discussion

For porous media flow at the onset of turbulence, at sufficiently high mesh resolution all schemes (BGK, TRT-DNS and TRT-LES) converge towards the same result for the friction coefficient. The DNS simulation resolves all vortex scales and shows an irregular, turbulence-like flow pattern. For coarser meshes, the dissipative effects of the small vortices (which cannot be resolved) are correctly modelled by the Smagorinsky approach, leading to a comparable friction coefficient as given by the high-resolution DNS. The good match between DNS and LES requires further investigation, since for our LES simulation no wall function has been applied. Such a wall function is usually employed to determine the slip velocity at the fluid nodes adjacent to the porous structure, in case the laminar boundary layer is not resolved by the computational mesh. Concerning turbulent flow in porous media, very little is known about the dimension of a laminar boundary layer, and if the established models to determine the slip velocity can be applied. A secondary welcome effect of employing LES comes from the local increase in viscosity, especially at positions where high velocity gradients are present. This leads to a stabilising effect for the whole simulation and allows computations at coarse mesh resolutions, which otherwise cannot be achieved with DNS.

7. Engineering application

We perform a preliminary study of the porous geometry in a diverging channel without recording acoustic signals. The air release through the exhaust of the valve terminal emits noise, which is being reduced by placing a porous medium silencer. Energy in the fluid is dissipated induced by the consumption of larger turbulent flow structures from inside the terminal by the small scaled pores, resulting in a decreased sound pressure level of $\approx 80 dB$ at 1m distance which we hope to reduce even further. The aluminium made silencer originally has the size of 10x5x0.5cm, of which we retrieved a $\mu$CT scan with a volume of 69.59mm$^3$ at a resolution of 6.3$\mu$m. Based on the CT scan data, a surface mesh is extracted, from which in turn the voxel mesh is created for the simulation. For the preliminary study, a volume fraction of only $(0.96mm)^3$ is used. The porous structure is placed at the edge of the diverging channel, to model the outflow to the surrounding. The computational domain consists of 70 Mio. fluid cells. The Reynolds number ranges from $Re = 1.9 \cdot 10^4 \ldots 10^5$ depending on the operating pressure of 1...8 bar. We choose the inlet velocity $u_{in} = 0.01$, channel height $h_{in} = 155\delta_x$ and viscosity $\nu = 8.4 \cdot 10^{-5}$ resulting in $Re = 18590$ and use a BGK-LES model with $C_S = 0.17$. In future studies, the solid walls of the channel behind the porous medium will be replaced by non-reflective open boundaries [33]. In Figure 6 the vorticity is plotted and the strongly vortical flow at the outflow after the porous medium is clearly visible. We are aware that the employed turbulence model without wall functions possibly does not model wall-bounded flow velocities correctly. The goal of this study was to qualitatively estimate the feasibility of
such simulations with the flow through under-resolved geometries with the usage of an LES-Smagorinsky turbulence model.

8. Conclusions and outlook

In this paper, we described steps towards the calculation of aeroacoustic sound creation by flow through a porous medium. The generation of acoustic waves was successfully demonstrated with a spinning vortex pair. Further, we evaluated the Lattice Boltzmann Method for non-Darcy flow through a generic porous structure. Lower-resolution simulations employing a Smagorinsky LES turbulence model were validated successfully against a direct numerical simulation with high mesh resolution. The TRT-LES model showed an almost mesh independent friction coefficient even for very coarse grid resolutions.

9. References

[22] M. Hasert, J. Bernsdorf, S. Roller, Towards aeroacoustic sound generation by flow through porous media, Phil. Trans. R. Soc. Lond. A.