

RWTH AACHEN UNIVERSITY

PROJECT WORK

**Investigations on one-way
coupling effects of particle-laden
decaying isotropic turbulent flows**

*Julian Stemmermann, Steffen Trienekens Christian
Soika*

supervised by
Konstantin FRÖHLICH

July 16, 2017

Contents

1	Nomenclature	3
2	Introduction	4
3	Mathematical models	5
3.1	Single-phase flow	6
3.1.1	The Navier-Stokes equations	6
3.2	Particle dynamics	8
4	Numerical methods	9
4.1	Direct numerical simulation	9
5	Results	11
5.1	Boundary conditions and simulation properties	11
6	Conclusion	12
7	References	13

1 Nomenclature

2 Introduction

in Computational Methods for Multiphase Flow ist auf den Seiten 3-9 ein interessantes Beispiel.

3 Mathematical models

3.1 Single-phase flow

In this section the mathematical basics for understanding and simulating turbulent flows are discussed. However, it should be pointed out that this is no complete treatise of the mathematical and physical basics. The reader can achieve further insight on this topic by looking at different books and papers, e.g. [7].

3.1.1 The Navier-Stokes equations

The Navier-Stokes-Equations are of great importance for understanding turbulent phenomena. This set of equations exists in forms for compressible and incompressible fluids. For an infinitesimal small volume element $d\tau$ and using the cartesian coordinate system, they can be written in the so-called 'divergence form':

$$\frac{\partial \mathbf{Q}}{\partial t} + \nabla \mathbf{H} = 0 \quad (3.1)$$

∇ Nabla-operator
Q Container for conserved variables in the Navier-Stokes equations
H Container for fluctuating variables in the Navier-Stokes equations
It should be noted by the reader that this work only contains investigations about chemically inert fluids and particles, and that the simulation results only fit under this condition. The vector **Q** contains all the variables which are conserved, i.e. the density ρ , the velocity \mathbf{u} and the specific inner energy E :
Q Density
Q Three-dimensional velocity
Q Specific inner energy

$$\mathbf{Q} = \begin{pmatrix} \rho \\ \rho \mathbf{u} \\ \rho E \end{pmatrix} \quad (3.2)$$

H is the flux vector which stores all the floating variables and may be split up into two parts:

$$\mathbf{H} = \mathbf{H}^i + \mathbf{H}^v \quad (3.3)$$

Hⁱ Stores the inviscid variables in the flux-vector included in the Navier-Stokes equations
H^v Stores the viscous variables in the flux-vector included in the Navier-Stokes equations
The contents of the two vectors are displayed below:

$$\mathbf{H}^i = \begin{pmatrix} \rho \mathbf{u} \\ \rho \mathbf{u} \mathbf{u} + p \\ \mathbf{u}(\rho E + p) \end{pmatrix} \quad (3.4)$$

p Pressure

$$\mathbf{H}^v = -\frac{1}{Re} \begin{pmatrix} 0 \\ \boldsymbol{\tau} \\ \boldsymbol{\tau} \mathbf{u} + \mathbf{q} \end{pmatrix} \quad (3.5)$$

Re Reynolds number
 $\boldsymbol{\tau}$ Stress tensor
 \mathbf{q} Heat conduction
Hⁱ is called inviscid flux and contains only the variables that are independent of the fluids viscosity, it describes the way a fluid with zero viscosity would behave. In contrast, the viscous flux **H**^v represents the effects of viscosity. The Reynolds number $Re = \frac{\rho v d}{\eta}$ is defined to be the ratio of inertia to tenacity, which makes it very valuable for understanding turbulent flows. This is also due to the fact that two familiar

objects with the same Reynolds number behave similar in turbulence. One can assume that flows with $Re \ll 1$ are laminar and flows with $Re \gg 1$ are turbulent. To solve the Navier-Stokes-Equations, more information regarding some variables is required. For Calculating the specific inner Energy E and the heat conduction \mathbf{q} , the following equations are used:

$$E = e \frac{1}{2} |\mathbf{u}|^2 \quad (3.6)$$

e Specific internal energy

$$\mathbf{q} = - \frac{\mu}{Pr(\gamma - 1)} \nabla T \quad (3.7)$$

Pr Prandtl number μ Dynamic viscosity with

$$\gamma = \frac{c_p}{c_v} \quad (3.8)$$

c_p Specific isobaric heat capacity c_v Specific isochoric heat capacity and the Prandtl number

$$Pr = \frac{\mu_\infty c_p}{k_t} \quad (3.9)$$

k_t thermal conductivity using the specific heat capacities of the fluid c_v and c_p . If one could assume that the fluid is a newtonian fluid, the linear correlation between stress and the rate of strain results in:

$$\boldsymbol{\tau} = 2\mu \mathbf{S} - \frac{2}{3}\mu(\nabla * \mathbf{u})\mathbf{I} \quad (3.10)$$

\mathbf{I} Identity tensor \mathbf{S} Rate-of-strain-tensor T Temperature in which $\mathbf{S} = \frac{(\nabla \mathbf{u})(\nabla \mathbf{u})^T}{2}$ denotes the rate-of-strain-tensor. Additionally, the viscosity μ can be approximated through Sutherland's law, which is based on the ideal gas-theory:

$$\mu(T) = \mu_\infty \left(\frac{T}{t_\infty} \right)^{3/2} \frac{T_\infty + S}{T + S} \quad (3.11)$$

S Sutherland temperature R Universal gas constant S is in this case the Sutherland temperature. To achieve closure the caloric state equation $e = c_v T$ and the state equation for an ideal gas $p = \rho R T$ are used. The specific gas constant is determined by $R = c_p - c_v$. These equations form a set of partial differential equations which is used to compute the flow statistics.

3.2 Particle dynamics

Siewert: -3.1a-3.4 (spherical particles) OHNE GRAVITATION Stokes Drag/Stokes
Coefficient Filterung (Fritz) -; Viskosität durch numerischen Fehler, Smagorinsky
nicht benutzen

4 Numerical methods

To simulate flows like those described above we have two options. The direct numerical simulation (DNS) is the easier one to understand, although it is numerically very expensive. The Large-eddy simulation (LES) is numerically more capable, still we must accept certain inaccuracies. These two numerical methods are now discussed in the following chapter.

4.1 Direct numerical simulation

The basis of the direct numerical simulation (DNS) are the Navier-Stokes equations as described above. The idea is that the computer is very good at calculating and solves these equations completely. This provides us a very accurate result, as all scales of motion are being resolved. Still it requires an immense level of computational resources which increases rapidly with the Reynolds number. These computational resources were not available until the 1970s [7].

(noComputationalParticles) implizite LES (Motivation fuer LES - Pope Chapter 9, Bild 9.4), DNS

5 Results

5.1 Boundary conditions and simulation properties

The simulations were carried out using ZFS, the simulation tool developed and implemented at the Institute of Aerodynamics at RWTH Aachen University [2] [3]. The tool is capable of simulating finite-volume flows of compressible fluids. In this case the turbulence was simulated on a cubic grid using 64^3 , 96^3 , 128^3 and 256^3 . The first three cases were simulated using LES, the case in which 256^3 cells were used is carried out as DNS. Further information can be gained by looking at [7, p.344ff for DNS and p. 558ff for LES]. For simplification, the special case of isotropic turbulence was used. For this idealised flow form the statistical velocities are invariant in all directions of the grid. It follows that the flows velocity are invariant for rotations and reflections. The turbulence was initialised using a seed-based random generator. To achieve physical results, the simulation was carried out to timestep 150, at which a restart file was written out. This procedure insures a fully developed turbulent flow, which has emancipated from the initialisation. In this flow field, a specific number of spherical particles were injected. The velocities of the fluid were interpolated for each particle to match the velocities of particles and fluid as accurately as possible.

boundary conditions, for example Reynolds number?

Graphen (particleFree rot, Laden gruen)

6 Conclusion

7 References

- [1] Edited by A. Prosperetti and G. Tryggvason. *Computational Methods for Multiphase Flow*. Cambridge University Press, 2009.
- [2] W. Schröder D. Hartmann, M. Meinke. An adaptive multilevel multigrid formulation for cartesian hierarchical grid methods. *Comput. Fluids*, 2008.
- [3] W. Schröder D. Hartmann, M. Meinke. A strictly conservative cartesian cut-cell method for compressible viscous flows on adaptive grids. *Comput. Meth. Appl. Mech. Eng.*, 2010.
- [4] S. Elghobashi. Particle-laden turbulent flows: direct simulation and closure models. *Applied Scientific Research*, 1991.
- [5] S. Fritz. Simulation isotroper turbulenz. study work, 2003.
- [6] M. Meinke K. Fröhlich, L. Schneiders and W. Schröder. Validation of particle-laden large-eddy simulation using hpc systems. In *Sustained Simulation Performance 2017*, 2017. unpublished conference document.
- [7] S. B. Pope. *Turbulent Flows*. Cambridge University Press, 2010.
- [8] W. Schröder. Fluidmechanik. lecture notes 'Fluid mechanics', 2010.