A thesis submitted for the degree of
Master of Science

Large Eddy Simulation of an Internal combustion engine incorporating a novel boundary layer modelling approach to evaluate piston-wall interactions

Yannick Durindel
Chair of Fluid Dynamics
University of Duisburg-Essen
30 January 2026

Primary Supervisor: M. Sc. Luiz Felipe Rico Cortez
University of Duisburg-Essen
Secondary Supervisor: Prof. Dr.-Ing Kempf
University of Duisburg-Essen

Abstract

Wall-bounded flows in internal combustion engines are inherently complex due to the formation of boundary layers near multiple surfaces. The piston surface is particularly significant owing to its large area, continuous motion, and strong influence on overall heat transfer and large-scale flow structures such as the tumble vortex. Unlike steady external flows, engine flows are highly unsteady and dominated by pressure fluctuations and non-equilibrium effects, causing classical wall function assumptions to fail.

This thesis proposes a novel boundary layer modeling approach for Large Eddy Simulation (LES) that accounts for near-wall effects by introducing a locally increased molecular viscosity near the walls, which are modeled using immersed boundaries. The approach is implemented in the in-house solver PsiPhi, which uses structured Cartesian grids with immersed boundary methods for complex geometry handling.

A one-dimensional model is developed to relate the enhanced viscosity to local flow conditions and wall distance. The model is verified using standard wall functions to ensure correct behavior in equilibrium conditions before application to engine flows.

The methodology is applied to simulate the University of Duisburg-Essen optical engine. A CAD model of the engine is generated with emphasis on accurate valve seat geometry, and a numerical grid with 0.2 mm cell size is created, totaling approximately 250 million cells. LES simulations are performed first for a simplified engine configuration to conduct grid sensitivity analysis, followed by simulations of the full optical engine geometry.

Post-processing focuses on piston-wall phenomena including shear stress evolution over the piston surface, heat transfer through the piston wall, and assessment of boundary layer detachment. Results demonstrate the capability of the enhanced molecular viscosity approach to capture near-wall turbulent transport on coarse grids while maintaining compatibility with immersed boundary methods for moving geometries.

Acknowledgment

Contents

List of Tables

List of Figures

List of Code

List of Abbreviations

DNS	Direct Numerical Simulation
FVM	Finite Volume Method
LES	Large Eddy Simulation
RANS	Reynolds-Averaged Navier-Stokes
SGS	Subgrid-Scale

1. Introduction

Wall-bounded flows are inherently complex because they give rise to boundary layers. Near the wall, steep velocity and temperature gradients develop due to the no-slip condition and the difference between the wall temperature and the fluid temperature. In internal combustion engines, these layers form along multiple surfaces, with the piston being particularly significant due to its large area, continuous motion, and strong influence on overall heat transfer and large-scale flow structures such as the tumble vortex.

1.1 Motivation

1.1.1 Boundary Layers in Internal Combustion Engines

Unlike steady external flows, engine flows are highly unsteady and dominated by pressure fluctuations and non-equilibrium effects. The in-cylinder flow undergoes rapid changes during the intake, compression, combustion, and exhaust strokes, creating a complex turbulent environment where classical boundary layer assumptions often fail.

The piston surface plays a critical role in engine heat transfer and flow dynamics:

- Large surface area exposed to combustion gases
- Continuous motion affecting the boundary layer development
- Strong influence on the tumble vortex and large-scale flow structures
- Significant contribution to overall engine heat losses

Accurate prediction of wall shear stress and heat transfer at the piston surface is essential for:

- Estimating thermal efficiency and heat losses
- Predicting component temperatures and thermal stresses
- Understanding in-cylinder chemistry and emissions formation
- Optimizing combustion chamber design

1.1.2 Limitations of Classical Wall Functions

Classical boundary-layer theory, developed mainly for steady conditions, describes how shear stress evolves with distance from the wall through three distinct regions: the viscous sublayer, the buffer layer, and the logarithmic region. Wall functions have been proposed to relate wall distance to shear stress across these zones, enabling coarse grid simulations.

However, the assumptions underlying these wall functions fail under the rapidly changing and confined conditions of internal combustion engines:

- Non-equilibrium effects due to rapid pressure changes
- Strong adverse pressure gradients during compression
- Flow reversal and boundary layer separation
- Transient thermal boundary layers
- Complex geometry effects near valve seats and piston bowl

Inaccurate wall shear-stress estimation leads to errors in heat-transfer prediction, poten-

tially underestimating thermal losses and misrepresenting engine efficiency and in-cylinder chemistry.

1.2 Challenges in Engine Simulation

1.2.1 Computational Cost

Because of the high velocities and turbulence in engines, fully resolving all flow structures through Direct Numerical Simulation (DNS) is impractical. The Reynolds numbers encountered in typical engine operation would require computational resources far beyond current capabilities.

Large Eddy Simulation (LES) offers a practical approach by resolving the large, energy-containing eddies while modeling the smaller scales. However, LES relies on Subgrid-Scale (SGS) models to represent the unresolved turbulence, and the accuracy of these models near walls remains a significant challenge.

1.2.2 Near-Wall Modeling Gap

A significant knowledge gap remains in developing SGS models that can accurately capture boundary-layer formation near the piston and predict heat transfer during the engine cycle. The standard approaches either:

- Require extremely fine grids near walls (wall-resolved LES), making simulations prohibitively expensive
- Use wall functions that fail under engine-relevant conditions (wall-modeled LES)

This motivates the development of novel wall treatment approaches that can handle the non-equilibrium conditions in engines while maintaining computational efficiency.

1.3 Objectives of This Work

This project aims to improve the physical understanding of near-wall processes in internal combustion engines and propose a novel modeling approach that accounts for boundary-layer effects. The key innovation is the introduction of a locally increased molecular viscosity near the walls, which are modeled using immersed boundaries.

The specific objectives are:

1. Generation of a CAD model of the University of Duisburg-Essen optical engine, with emphasis on accurate valve seat geometry
2. Generation of a numerical grid for the optical engine with 0.2 mm cell size
3. Development of a 1D model incorporating molecular viscosity manipulation near walls
4. Verification of the model using standard wall functions
5. LES of a simplified engine configuration including the 1D model
6. Grid sensitivity analysis to identify potential model improvements
7. LES of the full optical engine including the wall treatment model
8. Testing of different numerical frameworks: time integration and discretization schemes
9. Post-processing of simulation results with focus on piston-wall phenomena

1.4 Methodology

1.4.1 Enhanced Viscosity Approach

The proposed wall treatment introduces a locally increased molecular viscosity in the near-wall region to account for unresolved turbulent transport. Unlike traditional wall functions that prescribe velocity profiles, this approach modifies the effective viscosity to produce the correct wall shear stress on coarser grids.

The enhanced viscosity is applied within cells adjacent to the wall, where the immersed boundary method is used to represent the solid surfaces. This combination allows:

- Flexible handling of complex, moving geometries (piston motion)
- Smooth transition between near-wall and bulk flow regions
- Compatibility with existing LES subgrid-scale models

1.4.2 Immersed Boundary Method

The immersed boundary method represents solid surfaces that do not conform to the computational grid. This approach is particularly suited for engine simulations where:

- The piston moves through the domain during the simulation
- Complex valve and port geometries must be captured
- Grid generation for body-fitted meshes would be challenging

The wall treatment is integrated with the immersed boundary formulation to ensure consistent application of the enhanced viscosity near all solid surfaces.

1.4.3 1D Model Development

A one-dimensional model is developed to relate the enhanced viscosity to the local flow conditions and wall distance. This model:

- Captures the essential physics of the near-wall region
- Is computationally efficient for application in three-dimensional LES
- Can be verified against standard wall function predictions
- Is calibrated to produce correct wall shear stress behavior

1.4.4 Optical Engine Application

The University of Duisburg-Essen optical engine serves as the primary application case. Optical engines provide:

- Well-characterized geometry and operating conditions
- Experimental data for validation (when available)
- Representative engine flow features (tumble, swirl, squish)
- Realistic boundary layer development on the piston surface

1.5 Focus Areas

The post-processing and analysis focus on piston-wall phenomena:

1.5.1 Shear Stress Evolution

The wall shear stress distribution over the piston surface is analyzed throughout the engine cycle. Key aspects include:

- Temporal evolution during intake, compression, and expansion
- Spatial distribution across the piston crown
- Influence of the tumble vortex on local shear stress
- Comparison with standard wall function predictions

1.5.2 Heat Transfer

Heat transfer through the piston wall is evaluated to assess:

- Total heat flux during the engine cycle
- Peak heat transfer rates during combustion
- Spatial variations in heat transfer coefficient
- Sensitivity to the wall treatment parameters

1.5.3 Boundary Layer Behavior

The boundary layer structure and behavior are examined, including:

- Boundary layer thickness evolution
- Assessment of boundary layer detachment and reattachment
- Non-equilibrium effects during rapid transients
- Interaction with large-scale flow structures

1.6 Thesis Outline

This thesis is organized as follows:

Chapter 2: State of the Art reviews the fundamental theory of turbulent flows and simulation methods. Conservation laws and the Navier-Stokes equations are presented, followed by the concepts of turbulence and the energy cascade. The three main simulation approaches—DNS, Reynolds-Averaged Navier-Stokes (RANS), and LES—are described, with particular attention to LES filtering and subgrid-scale models. Wall treatment methods, immersed boundary techniques, and engine flow characteristics are introduced.

Chapter 3: Numerical Methods describes the PsiPhi solver used in this work. The spatial and temporal discretization schemes are presented, along with the pressure-velocity coupling algorithm. The implementation of LES, immersed boundaries, and the enhanced viscosity wall treatment are detailed.

Chapter 4: 1D Model Development presents the development and verification of the one-dimensional wall treatment model. The model formulation is described, and verification against standard wall functions is shown.

Chapter 5: Simplified Engine Simulation describes LES of a simplified engine configuration to test and refine the wall treatment. Grid sensitivity analysis and parameter studies are presented.

Chapter 6: Full Engine Simulation applies the wall treatment to the complete optical engine geometry. Results for piston-wall shear stress, heat transfer, and boundary layer

behavior are presented and analyzed.

Chapter 7: Conclusion summarizes the findings of this work, discusses limitations, and suggests directions for future research.

2. State of the Art

This chapter presents the theoretical foundations required for understanding the numerical simulation of turbulent flows in internal combustion engines. Starting from the fundamental conservation laws, we derive the governing equations and introduce the concepts of turbulence modeling that form the basis of this work.

2.1 Conservation Laws

The behavior of fluid flows is governed by the fundamental conservation principles of mass, momentum, and energy. These laws, when expressed in differential form, yield the equations that describe the motion of fluids.

2.1.1 Conservation of Mass

The principle of mass conservation states that mass can neither be created nor destroyed. For a control volume, this yields the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{2.1}$$

where ρ is the fluid density and $\mathbf{u}$ is the velocity vector. For incompressible flows, where density variations are negligible, this simplifies to:

$$\nabla \cdot \mathbf{u} = 0 \tag{2.2}$$

2.1.2 Conservation of Momentum

Newton's second law applied to a fluid element yields the momentum equation. The rate of change of momentum equals the sum of all forces acting on the fluid:

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} \tag{2.3}$$

where p is the pressure, $\boldsymbol{\tau}$ is the viscous stress tensor, and $\mathbf{g}$ represents body forces such as gravity.

For a Newtonian fluid, the viscous stress tensor is related to the strain rate tensor by:

$$\boldsymbol{\tau} = \mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T - \frac{2}{3} (\nabla \cdot \mathbf{u}) \mathbf{I} \right) \tag{2.4}$$

where μ is the dynamic viscosity and $\mathbf{I}$ is the identity tensor.

2.1.3 Conservation of Energy

The first law of thermodynamics, applied to a fluid element, gives the energy equation:

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot ((\rho E + p) \mathbf{u}) = \nabla \cdot (\kappa \nabla T) + \nabla \cdot (\boldsymbol{\tau} \cdot \mathbf{u}) + \rho \mathbf{g} \cdot \mathbf{u} \tag{2.5}$$

where E is the total energy per unit mass, κ is the thermal conductivity, and T is the temperature.

2.2 The Navier-Stokes Equations

Combining the conservation laws with the constitutive relations for a Newtonian fluid yields the Navier-Stokes equations. For an incompressible fluid with constant viscosity, these take the form:

$$\nabla \cdot \mathbf{u} = 0 \tag{2.6}$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f} \tag{2.7}$$

where $\nu = \mu/\rho$ is the kinematic viscosity and $\mathbf{f}$ represents external body forces per unit mass.

The nonlinear convective term $(\mathbf{u} \cdot \nabla)\mathbf{u}$ is responsible for the complex behavior observed in fluid flows, including the phenomenon of turbulence.

2.2.1 Reynolds Number

The Reynolds number is a dimensionless quantity that characterizes the ratio of inertial forces to viscous forces:

$$Re = \frac{UL}{\nu} \tag{2.8}$$

where U is a characteristic velocity and L is a characteristic length scale. At low Reynolds numbers, viscous forces dominate and the flow remains laminar. As the Reynolds number increases beyond a critical value, the flow transitions to turbulence.

2.3 Turbulence

Turbulence is characterized by chaotic, three-dimensional, unsteady fluid motion with a wide range of length and time scales. Understanding turbulence is essential for accurate prediction of flow behavior in engineering applications.

2.3.1 Characteristics of Turbulent Flows

Turbulent flows exhibit several distinctive features:

- **Irregularity**: The flow variables fluctuate randomly in space and time
- **Three-dimensionality**: Even flows with two-dimensional mean properties have three-dimensional turbulent fluctuations
- **Diffusivity**: Enhanced mixing and transport of momentum, heat, and mass
- **Dissipation**: Kinetic energy is continuously converted to internal energy through viscous dissipation
- **Wide range of scales**: From the largest energy-containing eddies to the smallest dissipative scales

2.3.2 Reynolds Decomposition

To analyze turbulent flows, variables are decomposed into mean and fluctuating components. For any flow variable ϕ:

$$\phi = \bar{\phi} + \phi' \tag{2.9}$$

where $\bar{\phi}$ is the time-averaged (or ensemble-averaged) mean and ϕ' is the fluctuating component. By definition, $\overline{\phi'} = 0$.

Applying this decomposition to the velocity field:

$$u_i = \bar{u}_i + u_i' \tag{2.10}$$

2.3.3 Reynolds-Averaged Navier-Stokes Equations

Substituting the Reynolds decomposition into the Navier-Stokes equations and time-averaging yields the Reynolds-Averaged Navier-Stokes (RANS) equations:

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{2.11}$$

The term $\overline{u_i' u_j'}$ represents the Reynolds stress tensor, which arises from the nonlinear convective term and requires modeling. This is known as the closure problem of turbulence.

2.3.4 Energy Cascade

Richardson's concept of the energy cascade describes how energy is transferred from large to small scales in turbulent flows. Large eddies, containing most of the kinetic energy, are unstable and break down into smaller eddies. This process continues until the eddies become small enough that viscous forces dissipate their energy as heat.

Kolmogorov's theory provides scaling laws for the smallest scales of turbulence. The Kolmogorov length scale η, time scale τ_η, and velocity scale u_η are:

$$\eta = \left(\frac{\nu^3}{\varepsilon}\right)^{1/4} \tag{2.12}$$

$$\tau_\eta = \left(\frac{\nu}{\varepsilon}\right)^{1/2} \tag{2.13}$$

$$u_\eta = (\nu\varepsilon)^{1/4} \tag{2.14}$$

where ε is the turbulent dissipation rate.

2.4 Turbulence Modeling Approaches

Three main approaches exist for simulating turbulent flows, differing in their treatment of the turbulent scales.

2.4.1 Direct Numerical Simulation (DNS)

DNS resolves all scales of turbulent motion by solving the Navier-Stokes equations without any turbulence modeling. The grid must be fine enough to capture the Kolmogorov scales, requiring:

$$N \sim Re^{9/4} \tag{2.15}$$

grid points in each direction for a three-dimensional simulation. This makes DNS computationally expensive and limits its application to relatively low Reynolds numbers and simple geometries. However, DNS provides the most accurate representation of turbulent flows and serves as a reference for validating turbulence models.

2.4.2 Reynolds-Averaged Navier-Stokes (RANS)

RANS solves for the time-averaged flow quantities, modeling all turbulent fluctuations through the Reynolds stress tensor. Common RANS models include:

- k-ε **model**: Two-equation model solving for turbulent kinetic energy k and dissipation rate ε
- k-ω **model**: Two-equation model using specific dissipation rate ω instead of ε
- **Reynolds Stress Models**: Solve transport equations for each component of the Reynolds stress tensor

RANS is computationally efficient but cannot capture unsteady turbulent phenomena or provide instantaneous flow information.

2.4.3 Large Eddy Simulation (LES)

LES represents a compromise between DNS and RANS. Large, energy-containing eddies are resolved directly while small-scale turbulence is modeled through a subgrid-scale (SGS) model. This approach is based on the observation that:

- Large eddies are geometry-dependent and carry most of the energy
- Small eddies are more universal and isotropic
- Modeling small scales introduces less error than modeling all scales

2.5 Large Eddy Simulation

2.5.1 Spatial Filtering

In LES, the flow variables are decomposed using a spatial filter rather than time averaging. For a variable ϕ, the filtered quantity $\tilde{\phi}$ is defined as:

$$\tilde{\phi}(\mathbf{x}, t) = \int_{\Omega} G(\mathbf{x} - \mathbf{x}', \Delta)\phi(\mathbf{x}', t)d\mathbf{x}' \tag{2.16}$$

where G is the filter kernel and Δ is the filter width. Common filter types include:

- **Box filter**: $G(\mathbf{x}) = 1/\Delta^3$ for $|x_i| \leq \Delta/2$
- **Gaussian filter**: $G(\mathbf{x}) = \left(\frac{6}{\pi\Delta^2}\right)^{3/2} \exp\left(-\frac{6|\mathbf{x}|^2}{\Delta^2}\right)$
- **Spectral cutoff filter**: Sharp cutoff in Fourier space

In practice, the grid itself often acts as an implicit filter with Δ related to the grid spacing.

2.5.2 Filtered Navier-Stokes Equations

Applying the filtering operation to the incompressible Navier-Stokes equations yields:

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \tag{2.17}$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial (\tilde{u}_i \tilde{u}_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{p}}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}^{sgs}}{\partial x_j} \tag{2.18}$$

The subgrid-scale stress tensor τ_{ij}^{sgs} represents the effect of unresolved scales on the resolved flow:

$$\tau_{ij}^{sgs} = \widetilde{u_i u_j} - \tilde{u}_i \tilde{u}_j \tag{2.19}$$

2.5.3 Subgrid-Scale Models

The SGS stress tensor must be modeled to close the filtered equations. The most common approach uses the eddy viscosity concept:

$$\tau_{ij}^{sgs} - \frac{1}{3}\tau_{kk}^{sgs}\delta_{ij} = -2\nu_{sgs}\tilde{S}_{ij} \tag{2.20}$$

where $\tilde{S}_{ij}$ is the filtered strain rate tensor:

$$\tilde{S}_{ij} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) \tag{2.21}$$

Smagorinsky Model

The Smagorinsky model relates the SGS viscosity to the local strain rate:

$$\nu_{sgs} = (C_s\Delta)^2|\tilde{S}| \tag{2.22}$$

where $|\tilde{S}| = \sqrt{2\tilde{S}_{ij}\tilde{S}_{ij}}$ and C_s is the Smagorinsky constant, typically around 0.1–0.2. The model is simple but has limitations:

- Does not vanish at solid walls
- Cannot predict backscatter (energy transfer from small to large scales)
- Requires ad-hoc damping near walls

Dynamic Smagorinsky Model

Germano et al. proposed the dynamic procedure, which computes the model coefficient from the resolved scales during the simulation:

$$C_s^2 = \frac{\langle L_{ij}M_{ij}\rangle}{\langle M_{ij}M_{ij}\rangle} \tag{2.23}$$

where L_{ij} is the Leonard stress and M_{ij} involves the difference between strain rates at different filter levels. The angle brackets denote averaging over homogeneous directions or time.

WALE Model

The Wall-Adapting Local Eddy-viscosity (WALE) model provides correct near-wall behavior without explicit damping:

$$\nu_{sgs} = (C_w\Delta)^2\frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\tilde{S}_{ij}\tilde{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}} \tag{2.24}$$

where S_{ij}^d is the traceless symmetric part of the square of the velocity gradient tensor.

2.6 Wall Treatment in LES

The near-wall region presents particular challenges for LES due to the small turbulent scales that develop close to solid boundaries.

2.6.1 Wall-Resolved LES

In wall-resolved LES, the grid is refined to capture the viscous sublayer and buffer layer directly. This requires:

- First grid point at $y^+ \approx 1$
- Grid spacing in wall-parallel directions: $\Delta x^+ \approx 50$–100, $\Delta z^+ \approx 15$–40

where the wall units are defined using the friction velocity $u_\tau = \sqrt{\tau_w/\rho}$:

$$y^+ = \frac{y u_\tau}{\nu}, \quad u^+ = \frac{u}{u_\tau} \tag{2.25}$$

2.6.2 Wall-Modeled LES

For high Reynolds number flows, wall-resolved LES becomes prohibitively expensive. Wall-modeled LES (WMLES) places the first grid point in the logarithmic layer ($y^+ > 30$) and uses a wall model to represent the near-wall region.

Law of the Wall

The mean velocity profile near a smooth wall follows distinct regions:

- **Viscous sublayer** ($y^+ < 5$): $u^+ = y^+$
- **Buffer layer** ($5 < y^+ < 30$): Transition region
- **Logarithmic layer** ($y^+ > 30$): $u^+ = \frac{1}{\kappa}\ln(y^+) + B$

where $\kappa \approx 0.41$ is the von Kármán constant and $B \approx 5.2$ for smooth walls.

Enhanced Viscosity Approach

One approach to wall modeling involves increasing the effective viscosity near the wall to account for unresolved turbulent transport. The total viscosity becomes:

$$\nu_{eff} = \nu + \nu_t^{wall} \tag{2.26}$$

where ν_t^{wall} is a modeled turbulent viscosity that captures the effect of unresolved near-wall eddies. This approach allows coarser grids near walls while maintaining the correct wall shear stress.

2.7 Channel Flow

The plane channel flow serves as a canonical test case for wall-bounded turbulence studies due to its geometric simplicity and well-documented behavior.

2.7.1 Configuration

The flow is driven by a pressure gradient between two parallel infinite plates separated by a distance $2h$. The flow is statistically steady and homogeneous in the streamwise (x) and spanwise (z) directions.

2.7.2 Friction Reynolds Number

The friction Reynolds number characterizes the flow:

$$Re_\tau = \frac{u_\tau h}{\nu} \tag{2.27}$$

where $u_\tau = \sqrt{\tau_w/\rho}$ is the friction velocity based on the wall shear stress τ_w. This Reynolds number represents the ratio of the outer length scale h to the viscous length scale ν/u_τ.

2.7.3 Mean Velocity Profile

In wall units, the mean velocity profile exhibits the classical law of the wall behavior. The friction velocity is related to the pressure gradient by:

$$\tau_w = -h\frac{dp}{dx} \tag{2.28}$$

2.7.4 Turbulent Statistics

Key quantities for characterizing turbulent channel flow include:

- Reynolds stresses: $\overline{u_i' u_j'}$
- Turbulent kinetic energy: $k = \frac{1}{2}\overline{u_i' u_i'}$
- Root-mean-square velocity fluctuations: $u^+_{rms} = \sqrt{\overline{u'^2}}/u_\tau$

These statistics, when obtained from DNS, serve as reference data for validating LES with various wall treatments.

2.8 Immersed Boundary Methods

Immersed boundary methods provide an alternative to body-fitted grids for simulating flows around complex geometries. The solid boundaries are represented on a fixed Cartesian grid, which simplifies grid generation and enables efficient handling of moving boundaries.

2.8.1 Concept and Motivation

In traditional body-fitted approaches, the computational grid conforms to the solid boundaries, requiring complex mesh generation and potential mesh quality issues near curved surfaces. Immersed boundary methods instead use a simple background grid (typically Cartesian) and impose boundary conditions through additional forcing terms or modified discretization near the immersed surface.

The key advantages include:

- Simple grid generation regardless of geometry complexity
- Efficient handling of moving boundaries without remeshing
- Straightforward parallelization on structured grids
- Natural extension to multi-body problems

2.8.2 Classification of Methods

Immersed boundary methods can be classified into two main categories:

Continuous Forcing Approach

The original immersed boundary method by Peskin adds a forcing term to the momentum equations:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu\nabla^2\mathbf{u} + \mathbf{f}_{IB} \tag{2.29}$$

where $\mathbf{f}_{IB}$ is the immersed boundary forcing that enforces the desired velocity at the boundary. This forcing is distributed to the grid using a smoothed delta function.

Discrete Forcing Approach

Direct forcing methods modify the discrete equations near the boundary. The velocity at boundary points is directly set or interpolated to satisfy the boundary condition:

$$u_{IB} = u_{desired} \tag{2.30}$$

This approach provides sharper boundary representation and is well-suited for high Reynolds number flows.

2.8.3 Wall Treatment with Immersed Boundaries

Combining wall treatment with immersed boundaries requires special consideration. The wall model must:

- Identify cells cut by the immersed boundary
- Compute the wall distance for each affected cell
- Apply the appropriate wall treatment based on the local y^+
- Ensure smooth transition to the outer flow region

The enhanced viscosity approach is particularly compatible with immersed boundaries because it modifies a local property (viscosity) rather than imposing explicit velocity boundary conditions.

2.9 Internal Combustion Engine Flows

The flow within an internal combustion engine exhibits unique characteristics that distinguish it from canonical turbulent flows.

2.9.1 Engine Cycle and Flow Phases

A four-stroke engine cycle comprises distinct phases with different flow characteristics:

Intake Stroke

During intake, the piston moves downward, creating a pressure difference that draws fresh charge through the intake valve. The flow entering the cylinder forms a high-velocity jet that generates large-scale vortical structures.

Compression Stroke

As the piston moves upward, the in-cylinder gases are compressed. The large-scale structures generated during intake are modified by the changing volume and interact with the walls.

Expansion Stroke

Following combustion, the piston is driven downward by the expanding gases. The boundary layers on the piston and cylinder walls experience rapid changes in pressure and temperature.

Exhaust Stroke

The piston moves upward to expel the combustion products through the exhaust valve.

2.9.2 Large-Scale Flow Structures

Engine designers use intake port geometry to generate organized large-scale motions that enhance mixing and combustion:

Tumble

Tumble is a large-scale rotational motion about an axis perpendicular to the cylinder axis. It is generated by directing the intake flow toward one side of the combustion chamber. As compression proceeds, the tumble vortex is compressed and eventually breaks down into smaller-scale turbulence, enhancing the mixing before ignition.

Swirl

Swirl is rotation about the cylinder axis, generated by tangential entry of the intake flow. Swirl persists longer than tumble due to the axial symmetry and can enhance mixing during combustion.

Squish

Squish is the radial flow generated as the piston approaches the cylinder head, forcing gases from the periphery toward the center (or vice versa). Squish enhances turbulence near top dead center.

2.9.3 Non-Equilibrium Boundary Layers

Engine boundary layers differ from classical equilibrium boundary layers in several important aspects:

Pressure Gradient Effects

The rapid pressure changes during compression and expansion create strong favorable and adverse pressure gradients that affect boundary layer development:

- Favorable gradients (during expansion) thin the boundary layer
- Adverse gradients (during compression) thicken the boundary layer and can cause separation

Unsteady Effects

The boundary layer does not have time to reach equilibrium between the different phases of the engine cycle. The wall shear stress and heat transfer lag behind the bulk flow changes.

History Effects

The boundary layer at any instant depends on the flow history, not just the instantaneous conditions. This makes equilibrium-based wall functions unreliable.

2.9.4 Heat Transfer in Engines

Heat transfer in engines is characterized by:

- High peak heat flux during combustion (order of $1\,\mathrm{MW/m^2}$)
- Strong spatial variation across combustion chamber surfaces
- Cycle-to-cycle variations in heat flux
- Transient thermal boundary layers

The wall heat flux is commonly expressed using Newton's law of cooling:

$$q_w = h(T_g - T_w) \tag{2.31}$$

where h is the heat transfer coefficient, T_g is the gas temperature, and T_w is the wall temperature.

Empirical correlations such as the Woschni correlation are commonly used in engine simulation:

$$h = C \cdot B^{-0.2} \cdot p^{0.8} \cdot T^{-0.55} \cdot w^{0.8} \tag{2.32}$$

where B is the bore, p is the pressure, T is the temperature, and w is a characteristic velocity combining piston speed and combustion effects.

2.10 Optical Engines

Optical engines are research engines equipped with transparent components that allow visual access to the combustion chamber for laser-based measurements.

2.10.1 Design Features

Typical optical engine features include:

- Extended piston with quartz or sapphire window in the crown
- Transparent cylinder liner (quartz)
- Mirror below the piston for through-piston imaging
- Skip-firing operation to manage thermal loads

2.10.2 Research Applications

Optical engines enable:

- Particle Image Velocimetry (PIV) for velocity field measurement
- Laser-Induced Fluorescence (LIF) for species concentration
- High-speed imaging of combustion
- Validation data for CFD simulations

2.10.3 Limitations

Optical engine measurements have limitations:

- Lower firing rates than production engines
- Different thermal properties of transparent materials
- Optical access constraints limit geometry modifications

- Skip-firing affects cycle-to-cycle variations

Despite these limitations, optical engines provide valuable data for understanding in-cylinder processes and validating simulation methods.

2.11 The PsiPhi Solver

PsiPhi is an in-house computational fluid dynamics code developed at the University of Duisburg-Essen, primarily for the simulation of turbulent reacting flows. The solver has been applied to a wide range of combustion and engine-related problems.

2.11.1 Development History

PsiPhi originated from research activities at the Chair of Fluid Dynamics, with a focus on high-performance computing approaches for turbulent flow simulation. The code has been continuously developed and validated against experimental and benchmark data.

2.11.2 Key Features

The main characteristics of PsiPhi include:

- Structured Cartesian grid approach for computational efficiency
- Support for both DNS and LES
- Immersed boundary method for complex geometry handling
- Scalable parallelization for high-performance computing clusters
- Various subgrid-scale models for LES
- Combustion modeling capabilities for reacting flows

2.11.3 Solver Approach

PsiPhi solves the incompressible Navier-Stokes equations using a finite volume discretization on a staggered grid. The pressure-velocity coupling is handled through a projection method (fractional step), ensuring mass conservation at each time step.

The use of structured Cartesian grids enables:

- Efficient memory access patterns
- Simple implementation of high-order schemes
- Straightforward domain decomposition for parallelization
- Compatibility with immersed boundary methods for complex geometries

2.11.4 Application Areas

PsiPhi has been applied to various flow configurations:

- Turbulent jet flames and diffusion flames
- Premixed combustion
- Internal combustion engine simulations
- Fundamental turbulence studies
- Heat transfer problems

The combination of structured grids, immersed boundaries, and established turbulence modeling makes PsiPhi well-suited for the engine boundary layer studies in this work.

3. Numerical Methods

This chapter describes the numerical methods employed in this work, with particular focus on the in-house solver PsiPhi used for both DNS and LES simulations. The discretization schemes, time integration methods, and implementation of the wall treatment are presented.

3.1 The PsiPhi Solver

PsiPhi is an in-house computational fluid dynamics solver developed for the simulation of turbulent reacting flows. The solver is designed for high-performance computing environments and supports both DNS and LES approaches.

3.1.1 Governing Equations

PsiPhi solves the incompressible Navier-Stokes equations in their filtered form for LES or unfiltered form for DNS:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{3.1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu + \nu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + f_i \tag{3.2}$$

where ν_t represents the turbulent viscosity, which is zero for DNS and computed from an SGS model for LES.

3.2 Spatial Discretization

3.2.1 Finite Volume Method

PsiPhi employs the Finite Volume Method (FVM) for spatial discretization. The computational domain is divided into control volumes, and the governing equations are integrated over each control volume. For a general transport equation:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\mathbf{u}\phi) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi \tag{3.3}$$

the integration over a control volume V with surface A yields:

$$\frac{\partial}{\partial t}\int_V \phi \, dV + \oint_A (\mathbf{u}\phi) \cdot d\mathbf{A} = \oint_A (\Gamma \nabla \phi) \cdot d\mathbf{A} + \int_V S_\phi \, dV \tag{3.4}$$

3.2.2 Grid Arrangement

The solver uses a staggered grid arrangement where:

- Scalar quantities (pressure, temperature) are stored at cell centers
- Velocity components are stored at cell faces

This arrangement prevents the checkerboard pressure oscillations that can occur with collocated grids and ensures strong pressure-velocity coupling.

3.2.3 Convective Term Discretization

The convective fluxes are discretized using a combination of schemes depending on the application:

Central Differencing Scheme

For DNS and well-resolved LES, the central differencing scheme (CDS) provides second-order accuracy:

$$\phi_f = \frac{\phi_P + \phi_N}{2} \tag{3.5}$$

where ϕ_f is the face value, and ϕ_P and ϕ_N are the values at the neighboring cell centers.

Upwind Scheme

For stability in convection-dominated flows, the upwind scheme can be employed:

$$\phi_f = \begin{cases} \phi_P & \text{if } (\mathbf{u} \cdot \mathbf{n})_f > 0 \\ \phi_N & \text{if } (\mathbf{u} \cdot \mathbf{n})_f < 0 \end{cases} \tag{3.6}$$

TVD Schemes

Total Variation Diminishing (TVD) schemes provide a balance between accuracy and stability by using flux limiters. The face value is computed as:

$$\phi_f = \phi_P + \frac{1}{2}\psi(r)(\phi_N - \phi_P) \tag{3.7}$$

where $\psi(r)$ is a limiter function and r is the ratio of consecutive gradients.

3.2.4 Diffusive Term Discretization

The diffusive fluxes are discretized using central differencing:

$$\left(\Gamma \frac{\partial \phi}{\partial x}\right)_f = \Gamma_f \frac{\phi_N - \phi_P}{\Delta x} \tag{3.8}$$

For non-orthogonal grids, correction terms are added to maintain accuracy.

3.3 Pressure-Velocity Coupling

3.3.1 Projection Method

PsiPhi uses a fractional step (projection) method to enforce the incompressibility constraint. The algorithm proceeds as follows:

Step 1: Predictor step

Compute an intermediate velocity u_i^* by solving the momentum equation without the pressure gradient:

$$\frac{u_i^* - u_i^n}{\Delta t} = -\frac{\partial (u_i u_j)^n}{\partial x_j} + \nu \frac{\partial^2 u_i^n}{\partial x_j \partial x_j} + f_i \tag{3.9}$$

Step 2: Pressure Poisson equation

The pressure is obtained by solving a Poisson equation derived from the continuity constraint:

$$\nabla^2 p^{n+1} = \frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}^* \tag{3.10}$$

Step 3: Corrector step

The velocity is corrected to satisfy continuity:

$$u_i^{n+1} = u_i^* - \frac{\Delta t}{\rho} \frac{\partial p^{n+1}}{\partial x_i} \tag{3.11}$$

3.3.2 Poisson Solver

The pressure Poisson equation is solved using iterative methods. PsiPhi supports:

- Jacobi iteration
- Gauss-Seidel iteration
- Successive Over-Relaxation (SOR)
- Conjugate Gradient methods
- Multigrid acceleration

3.4 Temporal Discretization

3.4.1 Explicit Time Integration

For the convective and source terms, explicit time integration schemes are employed:

Euler Method

The first-order Euler method:

$$\phi^{n+1} = \phi^n + \Delta t \cdot F(\phi^n) \tag{3.12}$$

Runge-Kutta Methods

Higher-order accuracy is achieved using Runge-Kutta schemes. The third-order Runge-Kutta (RK3) method:

$$\phi^{(1)} = \phi^n + \Delta t \cdot F(\phi^n) \tag{3.13}$$

$$\phi^{(2)} = \frac{3}{4}\phi^n + \frac{1}{4}\phi^{(1)} + \frac{1}{4}\Delta t \cdot F(\phi^{(1)}) \tag{3.14}$$

$$\phi^{n+1} = \frac{1}{3}\phi^n + \frac{2}{3}\phi^{(2)} + \frac{2}{3}\Delta t \cdot F(\phi^{(2)}) \tag{3.15}$$

3.4.2 Time Step Restriction

The time step is limited by stability constraints:

CFL Condition

The Courant-Friedrichs-Lewy (CFL) condition for convective stability:

$$\Delta t \leq \text{CFL} \cdot \min\left(\frac{\Delta x}{|u|}, \frac{\Delta y}{|v|}, \frac{\Delta z}{|w|}\right) \tag{3.16}$$

where $\text{CFL} \leq 1$ for explicit schemes.

Viscous Stability

The diffusive stability constraint:

$$\Delta t \leq \frac{\Delta x^2}{2\nu} \tag{3.17}$$

3.5 Boundary Conditions

3.5.1 Inlet Boundary

At inlet boundaries, velocity components are prescribed:

$$u_i = u_{i,\text{inlet}} \tag{3.18}$$

For turbulent inflow, synthetic turbulence generators or recycling methods can be employed.

3.5.2 Outlet Boundary

Convective outflow conditions allow disturbances to exit the domain:

$$\frac{\partial \phi}{\partial t} + U_c \frac{\partial \phi}{\partial n} = 0 \tag{3.19}$$

where U_c is the convection velocity and n is the outward normal direction.

3.5.3 Wall Boundary

At solid walls, the no-slip condition is applied:

$$u_i = 0 \tag{3.20}$$

For the pressure, a zero normal gradient condition is used:

$$\frac{\partial p}{\partial n} = 0 \tag{3.21}$$

3.5.4 Periodic Boundary

For channel flow simulations, periodic boundary conditions are applied in the streamwise and spanwise directions:

$$\phi(x) = \phi(x + L_x), \quad \phi(z) = \phi(z + L_z) \tag{3.22}$$

3.6 Immersed Boundary Implementation

PsiPhi uses an immersed boundary method to represent complex geometries on the Cartesian grid. This section describes the implementation details.

3.6.1 Geometry Representation

The solid geometry is represented using a signed distance function $\phi(\mathbf{x})$:

$$\phi(\mathbf{x}) = \begin{cases} > 0 & \text{fluid region} \\ = 0 & \text{boundary surface} \\ < 0 & \text{solid region} \end{cases} \tag{3.23}$$

The distance function is computed from the CAD geometry (typically STL format) and stored on the grid. For moving boundaries, the distance function is updated at each time step.

3.6.2 Cell Classification

Grid cells are classified based on the signed distance function:

- **Fluid cells**: Entirely in the fluid region ($\phi > 0$ at all vertices)
- **Solid cells**: Entirely in the solid region ($\phi < 0$ at all vertices)
- **Cut cells**: Intersected by the boundary (mixed signs at vertices)

Cut cells require special treatment to enforce boundary conditions while maintaining conservation.

3.6.3 Velocity Boundary Conditions

The no-slip condition at immersed boundaries is enforced through direct forcing. For a cut cell, the velocity is interpolated to satisfy:

$$u_{IB} = u_{wall} + (u_{fluid} - u_{wall}) \cdot f(\phi) \tag{3.24}$$

where $f(\phi)$ is a blending function based on the distance to the wall.

For moving boundaries such as the piston, the wall velocity u_{wall} corresponds to the local surface velocity.

3.6.4 Pressure Treatment

The pressure in solid cells is extrapolated from the fluid region to ensure smooth pressure gradients across the immersed boundary:

$$\left.\frac{\partial p}{\partial n}\right|_{wall} = 0 \tag{3.25}$$

This Neumann condition is consistent with the no-penetration constraint at solid walls.

3.7 LES Implementation

3.7.1 Subgrid-Scale Model

For LES, PsiPhi implements the Smagorinsky model with the SGS viscosity:

$$\nu_{sgs} = (C_s \Delta)^2 |\tilde{S}| \tag{3.26}$$

where $\Delta = (\Delta x \cdot \Delta y \cdot \Delta z)^{1/3}$ is the filter width based on the local grid spacing.

3.7.2 Near-Wall Treatment

Near solid walls, the Smagorinsky model requires damping to account for the reduced turbulent length scales. The van Driest damping function is applied:

$$f_d = 1 - \exp\left(-\frac{y^+}{A^+}\right) \tag{3.27}$$

where $A^+ \approx 25$ is the damping constant.

3.8 Wall Treatment: Enhanced Molecular Viscosity Approach

The primary focus of this work is the development of a novel wall treatment for LES that accounts for boundary-layer effects by introducing a locally increased molecular viscosity near the walls. This approach is specifically designed for use with immersed boundaries.

3.8.1 Motivation

Classical wall functions fail under the rapidly changing and non-equilibrium conditions of internal combustion engines. The enhanced molecular viscosity approach offers several advantages:

- Does not assume equilibrium boundary layer profiles
- Compatible with immersed boundary methods
- Computationally efficient for complex moving geometries
- Can be calibrated and verified against standard wall functions

3.8.2 Formulation

The effective viscosity in the near-wall region is modified to account for unresolved turbulent transport:

$$\nu_{eff}(y) = \nu \cdot (1 + \alpha \cdot f(y/\delta)) \tag{3.28}$$

where:

- ν is the molecular (kinematic) viscosity
- α is the enhancement factor (to be determined)
- $f(y/\delta)$ is a profile function depending on wall distance
- δ is a characteristic length scale (boundary layer thickness or cell size)

The key distinction from turbulent viscosity models is that this approach directly manipulates the molecular viscosity coefficient, which governs the diffusive fluxes in the momentum equation.

3.8.3 1D Model Development

A one-dimensional model is developed to determine the viscosity profile across the boundary layer. The 1D model solves a simplified momentum balance in the wall-normal direction:

$$\frac{d}{dy}\left(\nu_{eff}(y)\frac{du}{dy}\right) = \frac{1}{\rho}\frac{dp}{dx} \tag{3.29}$$

This equation is integrated from the wall to the edge of the boundary layer to obtain the velocity profile and wall shear stress. The viscosity profile $\nu_{eff}(y)$ is adjusted to match the law of the wall behavior.

3.8.4 Verification Strategy

The model is verified using standard wall functions. For equilibrium boundary layers, the enhanced viscosity model should reproduce:

- Linear velocity profile in the viscous sublayer: $u^+ = y^+$
- Logarithmic profile in the log layer: $u^+ = \frac{1}{\kappa}\ln(y^+) + B$

- Correct wall shear stress: $\tau_w = \mu \frac{\partial u}{\partial y}\Big|_{wall}$

The verification ensures that the model recovers classical results before application to non-equilibrium engine flows.

3.8.5 Implementation in PsiPhi

The enhanced viscosity is implemented by modifying the diffusive term in the momentum equation for cells near immersed boundaries:

Step 1: Wall Distance Computation

For each cell near an immersed boundary, the wall distance y is computed from the signed distance function.

Step 2: Viscosity Enhancement

The local viscosity is modified according to:

$$\nu_{cell} = \nu \cdot \left(1 + \alpha \cdot f\left(\frac{y}{\Delta}\right)\right) \tag{3.30}$$

where Δ is the local grid spacing.

Step 3: Blending

A smooth transition to the standard LES viscosity is applied:

$$\nu_{total} = \nu_{cell} + \nu_{sgs} \tag{3.31}$$

The blending function f approaches zero far from the wall, ensuring that the standard LES behavior is recovered in the bulk flow.

3.8.6 Coupling with Immersed Boundaries

The enhanced viscosity wall treatment is naturally coupled with the immersed boundary method:

- The signed distance function provides the wall distance
- Cut cells receive the enhanced viscosity based on their distance to the surface
- Moving boundaries automatically update the wall distance
- No special grid requirements near the wall

This coupling enables consistent treatment of all solid surfaces in the engine, including the moving piston.

3.9 Channel Flow Configuration

3.9.1 Computational Domain

The channel flow simulations use a domain of size $L_x \times L_y \times L_z$ with:

- $L_x = 2\pi h$ in the streamwise direction
- $L_y = 2h$ in the wall-normal direction (channel half-height h)
- $L_z = \pi h$ in the spanwise direction

3.9.2 Grid Requirements

For DNS, the grid resolution must satisfy:

- $\Delta x^+ \approx 10$–15 (streamwise)
- $\Delta y^+_{wall} < 1$ (first cell at wall)
- $\Delta z^+ \approx 5$–10 (spanwise)

For LES with wall treatment:

- $\Delta x^+ \approx 50$–100 (streamwise)
- $\Delta y^+_{wall} \approx 1$–$30$ depending on wall model
- $\Delta z^+ \approx 20$–50 (spanwise)

3.9.3 Flow Driving

The channel flow is driven by a constant pressure gradient in the streamwise direction. The pressure gradient is related to the target friction Reynolds number:

$$\frac{dp}{dx} = -\frac{\rho u_\tau^2}{h} = -\frac{\rho \nu^2 Re_\tau^2}{h^3} \tag{3.32}$$

3.10 Post-Processing and Statistics

3.10.1 Temporal Averaging

Statistical quantities are computed by time-averaging over a sufficiently long period after the flow has reached a statistically steady state:

$$\langle \phi \rangle = \frac{1}{T} \int_0^T \phi(t)\, dt \tag{3.33}$$

3.10.2 Spatial Averaging

For channel flow, additional averaging is performed over the homogeneous directions (streamwise and spanwise):

$$\langle \phi \rangle_{xz} = \frac{1}{L_x L_z} \int_0^{L_x} \int_0^{L_z} \phi(x, y, z)\, dx\, dz \tag{3.34}$$

3.10.3 Computed Quantities

The following quantities are extracted for comparison:

- Mean velocity: $\langle u \rangle^+ = \langle u \rangle / u_\tau$
- Velocity fluctuations: $u^+_{rms} = \sqrt{\langle u'^2 \rangle}/u_\tau$
- Reynolds shear stress: $\langle u'v' \rangle^+ = \langle u'v' \rangle / u_\tau^2$
- Turbulent kinetic energy: $k^+ = k/u_\tau^2$

A. Appendix

B. Appendix: Additonal Results

UNIVERSITÄT DUISBURG ESSEN

Offen im Denken

Declaration in lieu of an oath

I hereby solemnly declare that I have independently completed this work or, in the case of group work, the part of the work that I have marked accordingly. I have not made use of the unauthorised assistance of third parties. Furthermore, I have used only the stated sources or aids and I have referenced all statements (particularly quotations) that I have adopted from the sources I have used verbatim or in essence.

I declare that the version of the work I have submitted in digital form is identical to the printed copies submitted.

I am aware that, in the case of an examination offence, the relevant assessment will be marked as 'insufficient' (5.0). In addition, an examination offence may be punishable as an administra- tive offence (Ordnungswidrigkeit) with a fine of up to €50,000. In cases of multiple or otherwise serious examination offences, I may also be removed from the register of students.

I am aware that the examiner and/or the Examination Board may use relevant software or other electronic aids in order to establish an examination offence has occurred.

I solemnly declare that I have made the previous statements to the best of my knowledge and belief and that these statements are true and I have not concealed anything.

I am aware of the potential punishments for a false declaration in lieu of oath and in particular of the penalties set out in Sections 156 and 161 of the German Criminal Code (Strafgesetzbuch; StGB), which I have been specifically referred to.

Section 156 False declaration in lieu of an oath

Whoever falsely makes a declaration in lieu of an oath before an authority which is competent to administer such declarations or falsely testifies whilst referring to such a declaration incurs a penalty of imprisonment for a term not exceeding three years or a fine.

Section 161 Negligent false oath; negligent false declaration in lieu of oath

(1) Whoever commits one of the offences referred to in Sections 154 to 156 by negligence incurs a penalty of imprisonment for a term not exceeding one year or a fine. (2) No penalty is incurred if the offender corrects the false statement in time. The provisions of Section 158 (2) and (3) apply accordingly.

Place, date Signature