Development of a Robust Workflow for a CFD Analysis of External Aerodynamics in a Virtual Wind Tunnel

Diploma Thesis

By

Alexander Gnech

Supervisors: Dr.-Ing. Marc Ratzel
Dipl.-Ing. Lutz Pauli
Prof. M. Behr, Ph.D.

Aachen, March 9, 2012
Affidavit:

I hereby declare that I wrote this thesis on my own and without the use of any other than the cited sources and tools and all explanations that I copied directly or in their sense are marked as such, as well as that the thesis has not been available to any audit authority yet.

Eidesstattliche Erklärung:

Ich versichere, dass ich die Diplomarbeit selbstständig und ohne Benutzung anderer als der angegebenen Quellen und Hilfsmittel angefertigt habe und alle Ausführungen, die wörtlich oder sinngemäß übernommen wurden, als solche gekennzeichnet sind, sowie dass die Diplomarbeit in gleicher oder ähnlicher Form noch keiner anderen Prüfungsbehörde vorgelegt wurde.

Aachen, March 9, 2012

Alexander Gnech
Abstract

Automotive aerodynamics is an important field of research, as most of the engine power is required to overcome the aerodynamic drag force. Thus, aerodynamics represents the efficiency and environmental compatibility of vehicles. Furthermore, the reduction of fuel consumption becomes more and more important in the automotive industry. However, the increase in efficiency in terms of improved aerodynamics is often opposed by design specifications. These contrary factors hinder development processes in automotive industry. Therefore, Altair Engineering Inc. develops a vertical software solution referred to as the virtual wind tunnel tool. This software solution combines aerodynamic CFD analyses and design optimizations within a single development environment. Many stages of computer-aided development processes are automated and adapted to specific requirements in this field of application. Thus, the virtual wind tunnel tool enables both engineers and designers, to reduce the product development time and cost.

The CFD analyses in the virtual wind tunnel tool are performed using the general-purpose finite element based CFD flow solver AcuSolve™. Turbulent flow characteristics of usual passenger cars at highway speeds require accurate prediction of unsteady wake effects, as these effects determine the pressure induced drag force. Furthermore, a friction induced drag is produced in the boundary layer of the vehicle.

The aim of this thesis is the evaluation of requirements on the volume mesh to capture these forces accurately. A mesh sensitivity analysis is performed using the ASMO, and the results of the simulations are validated with experimental data. The results are compared to mesh requirements of a real passenger car from the automotive industry in order to attempt to discover parametrization possibilities for generic meshing rules. To enable realistic road simulations, boundary conditions for simulating a moving ground and rotating wheels are introduced. Furthermore, time-stepping issues are addressed.

The results of the simulations and gained insights into flow physics are used to improve the workflow in the virtual wind tunnel tool in terms of robustness, speed and accuracy.
# Contents

List of Figures iii

List of Tables v

Notation vi

1 Introduction

1.1 Motivation 1

1.2 Goal 1

1.3 Outline 2

2 Background

2.1 External Aerodynamics 3

2.2 Turbulence 6

2.3 Mathematical Modeling

2.3.1 Governing Equations 7

2.3.2 Turbulence Modeling 7

2.3.3 LES Filter 8

2.3.4 Turbulence Closure Model 10

2.3.5 Near-Wall Modeling 12

2.3.6 Boundary Conditions 14

2.4 Numerical Formulation

2.4.1 Finite Element Discretization 14

2.4.2 Galerkin-Least Squares Formulation 17

2.4.3 Time Discretization 18

3 Simulation Process

3.1 Model Setup 22

3.1.1 Wind Tunnel Domain 22

3.1.2 ASMO Model 22

3.1.3 Sedan Model 24

3.1.4 Boundary Conditions 26

3.2 Mesh Generation 28

3.2.1 General Mesh Requirements 26

3.2.2 Meshing Process 33
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Schematic of automotive aerodynamics [Geppert (2006)]</td>
</tr>
<tr>
<td>2.2</td>
<td>Boundary layer and adverse pressure gradient [Anderson (2006)]</td>
</tr>
<tr>
<td>2.3</td>
<td>Distributions in a flat-plate boundary layer: (solid) $- \frac{u}{u_\infty}$; $- f_d$; $- r_d$ [Spalart et al. (2005)]</td>
</tr>
<tr>
<td>3.1</td>
<td>ASMO: Surface mesh.</td>
</tr>
<tr>
<td>3.2</td>
<td>ASMO: Virtual wind tunnel.</td>
</tr>
<tr>
<td>3.3</td>
<td>Simple sedan: Model features.</td>
</tr>
<tr>
<td>3.4</td>
<td>Complex sedan: Model features.</td>
</tr>
<tr>
<td>3.5</td>
<td>Sedan model: Wind tunnel and flow characteristics.</td>
</tr>
<tr>
<td>3.6</td>
<td>Moving ground: Boundary conditions for different configurations: green: inlet, red: outlet, grey: no-slip condition, blue: symmetry condition, yellow: moving ground</td>
</tr>
<tr>
<td>3.7</td>
<td>Mesh Settings: Comparison of boundary layer meshing types</td>
</tr>
<tr>
<td>3.8</td>
<td>Mesh Settings: Features</td>
</tr>
<tr>
<td>3.9</td>
<td>Mesh Settings: ROI volume mesh</td>
</tr>
<tr>
<td>4.1</td>
<td>ASMO: Time history of the drag coefficient.</td>
</tr>
<tr>
<td>4.2</td>
<td>ASMO: Influence of grid size.</td>
</tr>
<tr>
<td>4.3</td>
<td>ASMO: Time history of $y^+$.</td>
</tr>
<tr>
<td>4.4</td>
<td>ASMO: Skin friction coefficient on the bottom in symmetry plane.</td>
</tr>
<tr>
<td>4.5</td>
<td>ASMO: Roof pressure in symmetry plane.</td>
</tr>
<tr>
<td>4.6</td>
<td>ASMO: Front pressure in symmetry plane.</td>
</tr>
<tr>
<td>4.7</td>
<td>ASMO: Base pressure in symmetry plane.</td>
</tr>
<tr>
<td>4.8</td>
<td>ASMO: Problematic discretization in the wake for fine resolution.</td>
</tr>
<tr>
<td>4.9</td>
<td>ASMO: Underbody pressure in symmetry plane.</td>
</tr>
<tr>
<td>4.10</td>
<td>ASMO: Time history of x-velocity for different core element sizes.</td>
</tr>
<tr>
<td>4.11</td>
<td>ASMO: Transition from RANS to LES turbulence model.</td>
</tr>
<tr>
<td>4.12</td>
<td>ASMO: Time history of residual ratio.</td>
</tr>
<tr>
<td>4.13</td>
<td>ASMO: Time history of solution increment ratio.</td>
</tr>
<tr>
<td>4.14</td>
<td>Moving ground: Time history of the drag coefficient for different configurations.</td>
</tr>
<tr>
<td>4.15</td>
<td>Moving ground: Time history of drag coefficient for particular components.</td>
</tr>
<tr>
<td>4.16</td>
<td>Moving ground: Comparison of $y^+$ over time.</td>
</tr>
<tr>
<td>4.17</td>
<td>Moving ground: Comparison of the pressure coefficient.</td>
</tr>
<tr>
<td>4.18</td>
<td>Simple sedan: Mesh control parameter validation.</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>4.19</td>
<td>Simple Sedan: Time history of $y^+$.</td>
</tr>
<tr>
<td>4.20</td>
<td>Simple sedan: Convergence and $c_d$ of steady-state run.</td>
</tr>
<tr>
<td>4.21</td>
<td>Simple sedan: Convergence and time-stepping with variable time step.</td>
</tr>
<tr>
<td>4.22</td>
<td>Simple sedan: Comparison of transient solution strategies with constant and variable time step size.</td>
</tr>
<tr>
<td>4.23</td>
<td>ASMO: Comparison of the drag signal for fixed and moving ground conditions, $C_f = C_s = 3$</td>
</tr>
<tr>
<td>4.24</td>
<td>Simple sedan: Comparison of solution strategies for $C_f = 3$.</td>
</tr>
<tr>
<td>4.25</td>
<td>Complex sedan: Time history of $c_d$, $y^+$, $\hat{c}_d$, $\hat{\epsilon}_d$ and $\epsilon$.</td>
</tr>
<tr>
<td>4.26</td>
<td>Complex sedan: Surface pressure distribution compared to simple sedan.</td>
</tr>
<tr>
<td>A.1</td>
<td>Virtual Wind Tunnel Tool.</td>
</tr>
<tr>
<td>A.2</td>
<td>ASMO: Pressure distribution paths in symmetry plane</td>
</tr>
<tr>
<td>A.3</td>
<td>ASMO: Vorticity Field at different time steps</td>
</tr>
<tr>
<td>A.4</td>
<td>ASMO: Base pressure at different time steps</td>
</tr>
<tr>
<td>A.5</td>
<td>Complex sedan: DDES Transition</td>
</tr>
<tr>
<td>A.6</td>
<td>Simple sedan: Moving ground</td>
</tr>
</tbody>
</table>
# List of Tables

2.1 Spalart-Allmaras constants. .................................................. 11
3.1 Moving ground: Specification of wheel velocity. ..................... 28
3.2 ASMO: mesh control parameters .............................................. 37
4.1 ASMO: Influence of time step size. ........................................ 40
4.2 ASMO: Integrated $y^+$ value over the vehicle body. ................ 42
4.3 ASMO: Integrated $y^+$ value over the wind tunnel ground. ........ 43
4.4 ASMO: Partial drag coefficient. .............................................. 43
4.5 ASMO: Comparison of grid densities in the wake. ..................... 45
4.6 ASMO: Comparison of refined wake lengths. ............................ 45
4.7 ASMO: Influence of boundary layer transition. ......................... 47
4.8 ASMO: Influence of far field. ............................................... 47
4.9 ASMO: Steady-state compared to transient analysis. .................. 56
4.10 Moving ground: Dependency on moving ground settings. .......... 57
4.11 Moving ground: Partial drag amount. ..................................... 58
4.12 Simple sedan: Drag coefficient for different mesh control settings. 62
4.13 Simple sedan: Drag coefficient for different mesh control settings. 64
4.14 Simple sedan: Partial drag amount - fixed ground. .................. 65
4.15 Simple sedan: Comparison of core length. .............................. 66
4.16 Simple sedan: Required time in flow startup phase. ................. 66
4.17 Simple sedan: Drag coefficient for different time steps. .......... 66
A.1 ASMO model dimensions. .................................................... 86
A.2 ASMO: Virtual wind tunnel dimensions. ................................ 86
A.3 Sedan model dimensions. ................................................... 89
A.4 Sedan: Virtual wind tunnel dimensions. ................................ 90
A.5 Simple sedan: Investigated mesh parameters. ......................... 91
A.6 Complex sedan: Improved mesh control parameters. ................. 91
## Notation

### Lowercase Latin Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>b</td>
<td>diffusive eddy viscosity flux in S-A model</td>
</tr>
<tr>
<td>$c_{b1}, c_{b2}$</td>
<td>basic S-A model constants for free shear flows</td>
</tr>
<tr>
<td>$c_d$</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>$c_f$</td>
<td>friction coefficient</td>
</tr>
<tr>
<td>$c_l$</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>$c_p$</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>$c_{w1}, c_{w2}, c_{w3}$</td>
<td>S-A model constants for destruction term</td>
</tr>
<tr>
<td>$d$</td>
<td>distance normal to a surface</td>
</tr>
<tr>
<td>$\tilde{d}$</td>
<td>modified distance normal to a surface</td>
</tr>
<tr>
<td>$d_w$</td>
<td>wheel diameter</td>
</tr>
<tr>
<td>f</td>
<td>external force vector</td>
</tr>
<tr>
<td>$f_d$</td>
<td>DDES blending function</td>
</tr>
<tr>
<td>$f_{v1}, f_{v2}, S$</td>
<td>S-A auxiliary function for near wall flow regions</td>
</tr>
<tr>
<td>$f_w$</td>
<td>S-A auxiliary function for destruction term</td>
</tr>
<tr>
<td>g</td>
<td>Dirichlet boundary condition for the velocity</td>
</tr>
<tr>
<td>$g, r$</td>
<td>S-A auxiliary functions for destruction term</td>
</tr>
<tr>
<td>h</td>
<td>Neumann boundary condition for the momentum equation</td>
</tr>
<tr>
<td>$h_{\text{global}}$</td>
<td>abstract global element size</td>
</tr>
<tr>
<td>$h_{\text{global}}$</td>
<td>global element size for the ASMO</td>
</tr>
<tr>
<td>$h_{\text{global}}$</td>
<td>global element size for the sedan model</td>
</tr>
<tr>
<td>$h_{\text{local}}$</td>
<td>first layer height</td>
</tr>
<tr>
<td>$h_{\text{local}}$</td>
<td>abstract local element size</td>
</tr>
<tr>
<td>$h_{\text{local}}^{\text{core}}$</td>
<td>local element size in the wake (core)</td>
</tr>
<tr>
<td>l</td>
<td>turbulent length scale</td>
</tr>
<tr>
<td>n</td>
<td>normal vector</td>
</tr>
<tr>
<td>$n_{\text{bot}}$</td>
<td>Octree mesh parameter for the bottom first layer height</td>
</tr>
<tr>
<td>$n_{\text{el}}$</td>
<td>number of elements</td>
</tr>
<tr>
<td>$n_{\text{flh}}$</td>
<td>Octree mesh parameter for the first layer height</td>
</tr>
<tr>
<td>$n_i$</td>
<td>abstract Octree mesh parameter</td>
</tr>
<tr>
<td>$n_p$</td>
<td>number of nodes</td>
</tr>
<tr>
<td>$n_z$</td>
<td>Octree mesh parameter for core element size</td>
</tr>
<tr>
<td>p</td>
<td>pressure</td>
</tr>
<tr>
<td>$\bar{p}$</td>
<td>large scale pressure</td>
</tr>
<tr>
<td>$\hat{p}$</td>
<td>filtered pressure (LES)</td>
</tr>
</tbody>
</table>
List of Tables

$p_\infty$ freestream pressure
$q$ pressure weighting function
$r_d$ DDES parameter
t time
t$_{init}$ initial time step for variable time step
$u_\infty$ freestream velocity
$\mathbf{u}$ velocity vector
$\mathbf{u}$ large scale velocity vector
$u'$ small scale velocity vector
$\mathbf{u}_||$ velocity vector parallel to a surface
$u_I$ inflow velocity
$\mathbf{u}_I$ initial condition for the velocity field
$\mathbf{u}^*$ friction velocity vector
$\mathbf{u}^+$ dimensionless velocity vector
$v$ Neumann boundary condition for the continuity equation
$\mathbf{w}$ velocity weighting function
$x$ space vector
$y^+$ dimensionless distance normal to the wall
$y^*$ integrated dimensionless distance normal to the wall

Uppercase Latin Symbols

$A$ surface area of the body
$A_{A}, A_S$ frontal area of ASMO, sedan model
$A_{WT,A}, A_{WT,S}$ wind tunnel cross section (ASMO, sedan model)
$B$ Neumann boundary condition for the turbulence equation
$C_{A}, C_S$ ground clearance of ASMO, sedan model
$C_f, C_s$ filter width, sample with for drag convergence assessment
$C_{DES}$ detached eddy simulation constant
$D$ drag force
$G$ Dirichlet boundary condition for the eddy viscosity
$H_{A}, H_S$ height of ASMO, sedan model
$H_{WT,A}, H_{WT,S}$ wind tunnel height of ASMO, sedan model
$I$ identity tensor
$I$ turbulence intensity
$I_{WT,A}, I_{WT,S}$ clearance from inlet to vehicle model (ASMO, sedan)
$L$ lift force
$L_{A}, L_S$ length of ASMO, sedan model
$L_{WT,A}, L_{WT,S}$ wind tunnel length of ASMO, sedan model
$M$ moment on the body
$M_i, N_i$ finite element shape functions
$O_{WT,A}, O_{WT,S}$ clearance from vehicle model (ASMO, sedan) to outlet
$P$ Engine power
$Re$ Reynolds number
### List of Tables

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>force on the body</td>
</tr>
<tr>
<td>$St$</td>
<td>Strouhal number</td>
</tr>
<tr>
<td>$T$</td>
<td>simulation time interval, end time</td>
</tr>
<tr>
<td>$T_s$</td>
<td>vortex shedding period</td>
</tr>
<tr>
<td>$\mathbf{T}$</td>
<td>viscous stress tensor</td>
</tr>
<tr>
<td>$\hat{T}$</td>
<td>filtered viscous stress tensor (LES)</td>
</tr>
<tr>
<td>$W_{A,W}$, $W_S$</td>
<td>width of ASMO, sedan model</td>
</tr>
<tr>
<td>$W_{WT,A,W_{WT,S}}$</td>
<td>wind tunnel width of ASMO, sedan model</td>
</tr>
<tr>
<td>$X_r$</td>
<td>distance from the rear face of the vehicle to the reattachment point</td>
</tr>
</tbody>
</table>

### Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\beta$</td>
<td>turbulent viscosity ratio</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>LES filter width (implicit: grid spacing)</td>
</tr>
<tr>
<td>$\Delta$</td>
<td>local element length scale</td>
</tr>
<tr>
<td>$\Delta t$</td>
<td>time step size</td>
</tr>
<tr>
<td>$\Delta T_f$</td>
<td>signal convergence filter width</td>
</tr>
<tr>
<td>$\Delta T_s$</td>
<td>signal convergence sample width</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>rate of strain tensor</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>termination condition tolerance for drag convergence</td>
</tr>
<tr>
<td>$\zeta_{WT,A}, \zeta_{WT,S}$</td>
<td>blockage ratio in ASMO, sedan model</td>
</tr>
<tr>
<td>$\eta$</td>
<td>abstract spatial coordinate</td>
</tr>
<tr>
<td>$\vartheta$</td>
<td>basic S-A model constant for free shear flows</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Von Kármán constant</td>
</tr>
<tr>
<td>$\Gamma$</td>
<td>boundary of computational domain</td>
</tr>
<tr>
<td>$\Gamma_{Entrance}$</td>
<td>slip bottom surface in the entrance section</td>
</tr>
<tr>
<td>$\Gamma_{Fixed}$</td>
<td>no-slip bottom surface without velocity</td>
</tr>
<tr>
<td>$\Gamma_g$</td>
<td>Dirichlet boundary</td>
</tr>
<tr>
<td>$\Gamma_{g,Wall}$</td>
<td>boundary surface with no-slip condition</td>
</tr>
<tr>
<td>$\Gamma_{g,Bottom}$</td>
<td>boundary surface of the bottom</td>
</tr>
<tr>
<td>$\Gamma_{g,Inflow}$</td>
<td>boundary surface of the inlet</td>
</tr>
<tr>
<td>$\Gamma_{g,Wheels}$</td>
<td>boundary surface of the wheels</td>
</tr>
<tr>
<td>$\Gamma_h$</td>
<td>Neumann boundary</td>
</tr>
<tr>
<td>$\Gamma_{Moving}$</td>
<td>moving bottom surface</td>
</tr>
<tr>
<td>$\Gamma_v, \Gamma_{g,Body}$</td>
<td>boundary surface of the vehicle</td>
</tr>
<tr>
<td>$\mu$</td>
<td>molecular viscosity</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>(dynamic) eddy viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinematic viscosity</td>
</tr>
<tr>
<td>$\nu_t$</td>
<td>(kinematic) eddy viscosity</td>
</tr>
<tr>
<td>$\nu_{in}$</td>
<td>eddy viscosity at the inlet</td>
</tr>
<tr>
<td>$\nu_0$</td>
<td>initial condition for the eddy viscosity</td>
</tr>
<tr>
<td>$\breve{\nu}$</td>
<td>auxiliary eddy viscosity</td>
</tr>
<tr>
<td>$\xi$</td>
<td>abstract spatial coordinate</td>
</tr>
</tbody>
</table>
### List of Tables

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\phi$</td>
<td>eddy viscosity weighting function</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>total stress tensor</td>
</tr>
<tr>
<td>$\bar{\sigma}$</td>
<td>large scale total stress tensor</td>
</tr>
<tr>
<td>$\hat{\sigma}$</td>
<td>filtered total stress tensor (LES)</td>
</tr>
<tr>
<td>$\tau, \tau_e, \tau_m, \tau_s$</td>
<td>GLS stabilization parameters</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>wall shear stress vector</td>
</tr>
<tr>
<td>$\tau^{SGS}$</td>
<td>“sub-grid” stress tensor</td>
</tr>
<tr>
<td>$\chi$</td>
<td>S-A auxiliary function for near wall flow regions</td>
</tr>
<tr>
<td>$\psi$</td>
<td>abstract quantity in convergence assessment algorithm</td>
</tr>
<tr>
<td>$\Psi$</td>
<td>S-A RHS source terms</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>computational domain</td>
</tr>
<tr>
<td>$\omega$</td>
<td>angular velocity vector</td>
</tr>
</tbody>
</table>

### Calligraphic Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mathcal{G}$</td>
<td>filter kernel</td>
</tr>
<tr>
<td>$\mathcal{L}$</td>
<td>differential operator</td>
</tr>
<tr>
<td>$\mathcal{O}$</td>
<td>mathematical order</td>
</tr>
<tr>
<td>$\mathcal{P}_I$</td>
<td>space of linear polynomials</td>
</tr>
<tr>
<td>$\mathcal{Q}$</td>
<td>pressure function space</td>
</tr>
<tr>
<td>$\mathcal{R}$</td>
<td>residual of the differential equation</td>
</tr>
<tr>
<td>$\mathcal{S}_0$</td>
<td>eddy viscosity test function space</td>
</tr>
<tr>
<td>$\mathcal{S}_G$</td>
<td>eddy viscosity trial function space</td>
</tr>
<tr>
<td>$\mathcal{V}_0$</td>
<td>velocity test function space</td>
</tr>
<tr>
<td>$\mathcal{V}_g$</td>
<td>velocity trial function space</td>
</tr>
<tr>
<td>$W$</td>
<td>perturbation operator</td>
</tr>
</tbody>
</table>

### Superscripts

- $Exp$ experimental data
- $h$ discretized variable
- $local$ local element size
- $global$ (abstract) global element size
- $SGS$ sub-grid scale
- $Sim$ simulated data

### Subscripts

- $A$ ASMO model
- $e, el$ element
- $flh$ first layer height
- $z$ (core) zone
List of Tables

bot wind tunnel ground  
p node points  
i index for abstract variables  
S sedan model  
WT Wind tunnel

Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASMO</td>
<td>Aerodynamisches Studienmodell</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer Aided Engineering</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>DDES</td>
<td>Delayed-Detached Eddy Simulation</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>GLS</td>
<td>Galerkin/Least-Squares</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>MOL</td>
<td>Boundary layer meshing using constant element growth rate</td>
</tr>
<tr>
<td>NBL</td>
<td>Boundary layer meshing using constant number of element layers</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds averaged Navier-Stokes equations</td>
</tr>
<tr>
<td>ROI</td>
<td>Region Of Interest</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds averaged Navier-Stokes equations</td>
</tr>
<tr>
<td>VWT</td>
<td>Virtual Wind Tunnel</td>
</tr>
</tbody>
</table>
1 Introduction

1.1 Motivation

Due to environmental and economical considerations, the reduction of fuel consumption is a central challenge in the automotive industry. A main aspect is the improvement of aerodynamics. This results in a decrease of the engine power, because the aerodynamic drag force is a major constituent of the total driving resistance [Aronson et al. (2000)]. Even drag increments of one thousandth are important in the automotive industry [Cosner (2000)]. As external aerodynamics has been analysed by engineers for a long time, a variety of adequate methods for both experimental fluid dynamics - commonly in wind tunnels - and computational fluid dynamics (CFD) has been developed. The use of computational fluid dynamics has advantages in design development processes: the replacement of physical experiments by computational simulations reduce e.g. the cost of development and failure risks. Furthermore it leads to a detailed understanding of flow physics and allows rapid evaluation of design alternatives and optimization. Therefore, CFD allows to enhance aerodynamics and thus a decrease in drag [Shaw (2002)].

Besides the formulation of adequate, stable numerical methods in a CFD-solver, which are required to capture the flow physics accurately, the software architecture must be designed for parallel execution on shared and distributed memory computer systems. Furthermore, the embedding in a flexible, user-friendly computer aided engineering (CAE) environment with associated pre- and post-processing software enhances its usability. Only this concurrence empowers the engineer to gain the maximum benefit of CFD, i.e. facilitates the generation of reliable results, efficient validation of design alternatives and thereby the aerodynamic optimization process.

1.2 Goal

This thesis is embedded in the development project of a vertical software solution - namely the virtual wind tunnel tool of Altair Engineering Inc. (see Appendix A.2 Fig. A.1). It enables both, engineers and designers, to perform CFD analyses and design optimizations within one framework. The advantage of a vertical software solution is the limitation of use cases to a specific field of application, thereby it allows a higher level of automation and standardization. Thus, the virtual wind tunnel tool contains automated domain dimensioning, mesh generation, solver setup and simulation, post-processing, result evaluation and design optimization.
The used CFD-solver in this application is AcuSolve™, a commercially available finite element flow solver. AcuSolve™ is based on the Galerkin/Least-Squares finite element formulation, and is second-order accurate in space and time [cf. Donea and Huerta (2003)]. The implemented robust iterative solver obtains rapid convergence on large unstructured meshes even when high aspect ratio and badly distorted elements are present [Godo et al. (2011)].

In this thesis, the usability of AcuSolve™ in external aerodynamics is evaluated. Therefore, the requirements to perform an accurate analysis are considered. The investigated adaptation properties mainly consists of the volume mesh control parameters and time integration settings. Furthermore, the turbulence modeling is addressed. As previous studies investigated the same problem with a generic car model and compared it to experiments, the same model and equivalent initial and boundary conditions are used. A specification of boundary conditions for realistic road simulations is analysed in order to evaluate possibilities which increase the degree of realism. The results of the aerodynamic simulations are validated in terms of the drag coefficient and pressure distribution over the vehicle.

In addition to the accurate prediction of aerodynamic coefficients, the determination of inherent correlations between wind tunnel properties and both, meshing and time-stepping control parameters, is of interest. The specified mesh control parameters are validated using two variations of a real sedan model.

To verify the reliability of the results, a multitude of simulations must be performed. Appropriate mesh settings as well as solution strategies are considered, and potential bottlenecks in the simulation process are evaluated. Thereby, the manual effort of the simulation setup is decreased and the robustness of the CFD-analysis is improved.

This results in a proper base for the software development of the virtual wind tunnel tool.

1.3 Outline

First of all, Chapter 2 provides the necessary background about the underlying physics and its mathematical modeling. Furthermore, the numerical formulation in AcuSolve™ is described. Chapter 3 describes the entire simulation setup, which consists of the introduction of the utilized models, a detailed description of the meshing process, and a short summary of the solver settings. In Chapter 4, the results of the CFD analyses are studied and evaluated in detail. Finally, Chapter 5 gives an overview of the gathered insights and states concluding remarks.
2 Background

2.1 External Aerodynamics

External aerodynamics in general deals with the flow around a solid object. For a start, the virtual wind tunnel tool is constructed for analyses in the automotive industry. Thus, this thesis deals with a flow around a vehicle as shown in Fig. 2.1. In common wind tunnels, a moving vehicle on a road is approximated using a fixed vehicle at an inflow speed, which replaces the vehicle velocity. Far upstream of the vehicle, the undisturbed airflow is referred to as freestream. The freestream velocity is denoted by $u_\infty$. The flow reaches the vehicle at the stagnation point. At this point, the velocity of the fluid stagnates to zero, and thus the pressure exerted on the vehicle is at its maximum value.

As the fluid moves past the vehicle, the molecules very close to the surface adhere to the surface, their velocity is zero. Due to the air viscosity, these adhered molecules decrease the velocity of adjacent molecules. The farther the fluid is away from the vehicle surface, the less is the effect of the surface and the velocity increases. At a certain point, the viscous forces are negligible compared to inertial forces. The small region between the surface and this point is referred to as boundary layer (see Fig. 2.2). This region is characterized by huge velocity gradients, which cause the friction drag.

Depending on the body shape, the pressure gradient increases in flow direction. As the fluid in the boundary layer is slower, it is more affected by this so-called adverse pressure gradient. The motion of the fluid in the boundary layer is already retarded...
by friction. Additionally, it has to overcome the increased pressure in flow direction, which tends to further reduce its velocity. At the point, where the velocity gradient normal to the wall is zero, the flow reverses. Thus, the boundary layer can detach from the surface, which leads to an immediate change in the pressure distribution on the surface. However, if no pressure induced boundary layer separation occurs, the flow separates at least at the trailing edges of the vehicle.

![Boundary layer and adverse pressure gradient](image)

Figure 2.2: Boundary layer and adverse pressure gradient [Anderson (2006)].

The resulting detached shear layers enclose a wake, a turbulent flow region behind the vehicle. In most cases, the flow behavior in the wake is highly unsteady, but the flow motion depends on the respective shape and geometrical details of the rear part of the vehicle. Due to the rotational character of the wake flow, the pressure at the rear face can not increase.

The main concern in external aerodynamics is the prediction of forces and moments on the vehicle. Independent of the complexity of the vehicle shape, the forces and moments on the vehicle arise due to the two aforementioned sources:

- The pressure distribution over the vehicle surface $\Gamma_v$
- The shear stress distribution over the vehicle surface $\Gamma_v$

Both pressure and shear stress have dimensions of force per unit area, but pressure acts normal to the surface and the shear stress tangential to the surface. The pressure and shear stress distributions are integrated over the whole body. Hence, an aerodynamic
2.1 External Aerodynamics

force $\mathbf{R}$ and moment $\mathbf{M}$ on the vehicle results. Several experiments have shown, that the pressure induced part of the drag force $\mathbf{R}$ is due to the unsteady effects in the wake, which prevent the pressure recovery at the rear face [Duell and George (1992); Ishihara and Takagi (1999)].

In this thesis, the evaluation is limited to the determination of the force $\mathbf{R}$, which is then defined by:

$$\mathbf{R} = \int_{\Gamma_v} \sigma \cdot \mathbf{n} \, d\Gamma_v, \quad (2.1)$$

where $\sigma$ denotes the total stress tensor, and $\mathbf{n}$ denotes the outward pointing normal unit vector to the surface.

The resultant force can be split into two components respective to their acting direction:

- The component of $\mathbf{R}$ acting perpendicular to $u_\infty$ is referred to as lift $L$
- The component of $\mathbf{R}$ acting parallel to $u_\infty$ is referred to as drag $D$

Different dimensionless coefficients are used to capture the contributions of different forces:

- Drag coefficient: $c_d = \frac{D}{\frac{1}{2}\rho A u_\infty^2}$,
- Lift coefficient: $c_l = \frac{L}{\frac{1}{2}\rho A u_\infty^2}$,
- Pressure coefficient: $c_p = \frac{p-p_\infty}{\frac{1}{2}\rho u_\infty^2}$,
- Skin friction coefficient: $c_f = \frac{\tau_w}{\frac{1}{2}\rho u_\infty^2}$,

where $A$ denotes the frontal cross-section reference area of the given body shape, $p_\infty$ and $\rho$ denote the freestream pressure and density, respectively.

For a Newtonian fluid like air the wall shear stress is defined by:

$$\tau_w = \mu \left[ \frac{\partial u_{||}}{\partial n} \right]_{d=0}, \quad (2.2)$$

where $u_{||}$ denotes the velocity parallel to the solid boundary, $\mu$ denotes the molecular viscosity, and $d$ denotes the distance normal to the surface.

In automotive aerodynamics two main concerns are drag reduction and prevention of undesired lift forces. As already mentioned, the majority of the engine power required to overcome the drag force:

$$P \sim R_x u_\infty = \frac{1}{2} c_d \rho A u_\infty^3, \quad (2.3)$$
2.2 Turbulence

Thus, apart from the driving speed, the possibilities to improve vehicle aerodynamics are limited to the drag coefficient $c_d$ and the frontal area of the vehicle $A$. A wide range of studies in the past led to extensive knowledge in automotive aerodynamics. However, due to specific design instructions, the improvement of vehicle aerodynamics is a tedious process.

2.2 Turbulence

A compact definition of turbulence has been formulated by Corrsin (1961): “Incompressible hydrodynamic turbulence is a spatially complex distribution of vorticity which advects itself in a chaotic manner [...] The vorticity field is random in both space and time, and exhibits a wide and continuous distribution of length and time scales.” Turbulence occurs at flows with high Reynolds numbers, when viscosity is not able to dampen the turbulent fluctuations. The anisotropic low-frequency large-scale structures in a turbulent flow are bounded by geometrical quantities, while the isotropic high-frequency small-scale structures, denoted as Kolmogorov scales, are bounded by viscosity. To visualize this spectrum of scales one often refers to turbulent eddies. An turbulent eddy can be seen as a local turbulent swirling motion over a certain region, whose characteristic dimension is in the local scale. Turbulent flows are characterized by its dissipative mechanisms, enhanced diffusion, and inherent three-dimensional properties as e.g. vortex stretching.

Depending on the Reynolds number, a boundary layer can be laminar or turbulent. Furthermore, a boundary layer can start in a laminar state and then undergo a transition into a turbulent boundary layer. However, the Reynolds numbers within this work are very high. Thus, the investigated boundary layers are turbulent on the entire surface.

Turbulent boundary layers are thicker, and the velocity throughout the boundary layer up to a point close to the surface remains almost at the freestream velocity. However, from this point the velocity rapidly decreases to zero at the surface. Thus, the velocity gradients in a turbulent boundary layer are larger and the shear stresses are increased [Anderson (2006)].

Due to the increased transport of momentum from the freestream to the surface, a turbulent boundary layer is enabled to overcome the adverse pressure gradient on the vehicle. Thus, a turbulent boundary layer remains longer attached, and the dimensions of the wake region decrease.

2.3 Mathematical Modeling

2.3.1 Governing Equations

The fluid in a subsonic, external aerodynamic flow can be considered as a continuum. Even the smallest eddies are significantly larger than the molecular length scales \[\text{Tennekes and Lumley (1983)}\]. In addition, the airflow around a vehicle at low Mach numbers can be simplified to an incompressible flow \[\text{Anderson (2006)}\], i.e. the density is assumed to be constant. Furthermore, it is assumed that the internal energy or temperature does not influence the flow field. Thus, the energy equation is omitted.

Hence, the turbulent flow around a vehicle in a bounded domain \(\Omega \subset \mathbb{R}^3\) can be described using the unsteady viscous incompressible form of the Navier-Stokes equations:

\[
\nabla \cdot \mathbf{u} = 0 \quad \text{on } \Omega \times (0, T), \\
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \mathbf{f} \right) - \nabla \cdot \mathbf{\sigma} = 0 \quad \text{on } \Omega \times (0, T),
\]

where \(\rho\) and \(\mu\) are the density and viscosity, respectively. \(\mathbf{u}(x, t)\) is the velocity vector, \(p(x, t)\) the pressure and \(\mathbf{f}(x, t)\) an external, e.g., gravitational, force field. \(\mathbf{\sigma}(x, t)\) is the stress tensor and defined by:

\[
\mathbf{\sigma} = -p\mathbf{I} + \mathbf{T}, \quad \mathbf{T} = 2\mu \varepsilon(\mathbf{u}),
\]

where \(\mathbf{T}(x, t)\) denotes the viscous stress tensor and \(\varepsilon(\mathbf{u}) = \frac{1}{2} \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right)\) the fluid rate of strain tensor.

2.3.2 Turbulence Modeling

In the case of a highly turbulent problem, it is not feasible to solve the system of equations \[2.4, 2.5\] accurately. To cover the whole spectrum of turbulent scales in space and time, the total computing cost scales as \(O(Re^3)\) \[\text{Tennekes and Lumley (1983)}\]. This so-called Direct Numerical Simulation (DNS) is prohibitive for any flow with a high Reynolds number.

Alternative methods of turbulent flow prediction reduce the computing cost by only resolving a part of the turbulent scales. Therefore, mainly two distinct modeling approaches have been established; the filtering and Reynolds-averaging.

The Large Eddy Simulation (LES) bases on the theory of Kolmogorov, that the largest scales contain the majority of the energy and dominate the transport process \[\text{Kolmogorov (1941)}\]. Hence LES uses the filtering approach and computes only the large, geometry dependent turbulent eddies in time and space. The small scales...
are left to be modelled using a sub-grid-scale model. Since the smallest turbulent oscillations are less affected by boundary conditions and make a small contribution to both the energy and transport processes, LES is expected to be very accurate.

Although LES reduces the computing cost compared to DNS, LES requires a very fine grid resolution in the boundary layers, an issue that remains even with successful wall-layer modeling [Squires (2004)]. Thus, LES demands high computational resources for high Reynolds number flows and complex geometries. For a more comprehensive insight into LES refer to [Sagaut (2006)].

In the Reynolds-averaging approach the quantities in the Navier-Stokes equations are decomposed into its averaged and fluctuating quantities. The outcome of this approach are the Reynolds-averaged Navier-Stokes equations (RANS), which introduce additional unknowns, the so-called Reynolds stress tensor.

Only the mean flow is computed using these RANS equations, while the description of the turbulent fluctuations using the Reynolds stress tensor requires an appropriate turbulence closure model. The closure is obtained with a constitutive equation relating the Reynolds stress tensor e.g. to the velocity gradients. A lot of different relations have been proposed, which shall not be further discussed here.

Because all turbulent scales are modelled, the required mesh resolution is far lower than in DNS or LES. Hence RANS reduces the computing cost significantly. However, the statistical models of RANS methods have been developed on the basis of mean parameters for thin turbulent shear flows [Squires (2004)]. RANS methods have various difficulties in predicting unsteady effects in massively separated flows at high Reynolds numbers [Spalart (2000a); Perzon and Davidson (2000)].

Although studies have shown that LES allows fairly accurate prediction of unsteady wake forces [Nakashima et al. (2008)], LES is not appropriate due to its requirements on fine boundary layer resolutions. In [Krajnovic and Davidson (2001)] it is shown, that the wall normal resolution at the rear face has little influence on the accurate prediction of the pressure induced drag.

Therefore, the Detached Eddy Simulation (DES) is introduced. DES applies RANS for the prediction of the attached boundary layer and LES for the resolution of the unsteady large turbulent scale in separated flow regions. In [Favre and Efraimsson (2011)] the DES modification Delayed-DES (DDES) is validated for unsteady cross-wind conditions and different mesh types containing polyhedral or hexahedral elements. The results lead to the conclusion that DDES is a reliable turbulence model for transient simulations with a reasonable effort of computation time. However, a challenging task is the appropriate transition between these two turbulence modeling approaches [cf. Spalart (2001); Squires (2004); Mockett et al. (2008)].

As the DES modification Delayed-DES (DDES) is used throughout this thesis, this turbulence modeling approach is further addressed in the next sections.
2.3 Mathematical Modeling

2.3.3 LES Filter

To use LES, a low-pass filter $G$ must be applied on the Navier-Stokes equations. This filter performs the scale separation using a prescribed filter width $\Delta$. All scales larger than $\Delta$ are actually resolved, and the smaller scales are modeled using a Sub-Grid-Scale (SGS) model.

The idea is to decompose the quantity, e.g. velocity, into the single contributions $\bar{u}$ and $u'$ of the large and small scale respectively

$$ u = \bar{u} + u' . $$

(2.7)

In general, the filter operation is a convolution of a function with a filter kernel $G$:

$$ \bar{u}(\xi) = \oint u(\eta)G(\xi, \eta, \Delta) d\eta . $$

(2.8)

This operation eliminates all scales below the filter width $\Delta$. It is worth noting that AcuSolve™ utilizes implicit filtering, i.e. the grid itself defines the filter width $\Delta$. Thus, the filter process itself is not treated further. For a comprehensive insight about the LES filter process refer to [Ghosal and Moin (1995)].

Due to the nonlinearity of the advection term, an additional “sub-grid” stress tensor $\tau_{SGS}$ is introduced into the Navier-Stokes equations, which relates the resolved to unresolved scale.

$$ \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} - \nabla \cdot \sigma - \nabla \cdot \tau_{SGS} = \bar{f} \quad \text{on } \Omega \times (0, T) , $$

(2.9)

The sub-grid stress tensor must be related to resolved quantities. Therefore, usually the Boussinesq hypothesis is employed. The sub-grid stress tensor is related to the resolved strain by introducing an eddy viscosity model with the eddy viscosity $\nu_t$:

$$ \tau_{SGS} - \frac{1}{3} \text{tr} (\tau_{SGS}) I = -2\nu_t \varepsilon(\bar{u}) . $$

(2.10)

The pressure absorbs the non-deviatoric component of the stress tensor

$$ \hat{p} = \bar{p} + \frac{1}{3} \rho \text{tr} (\tau_{SGS}) I . $$

(2.11)

Consequently, the filtered Navier-Stokes equations can be written as:

$$ \nabla \cdot \bar{u} = 0 \quad \text{on } \Omega \times (0, T) , $$

(2.12)

$$ \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} - \nabla \cdot \hat{\sigma} = \bar{f} \quad \text{on } \Omega \times (0, T) , $$

(2.13)

with

$$ \hat{\sigma} = -\frac{1}{\rho} \hat{p} I + \hat{T} , \quad \hat{T} = 2(\nu + \nu_t) \varepsilon(\bar{u}) , $$

(2.14)

where $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity. In the following, the sub- and superscripts are omitted for notational simplicity.
2.3.4 Turbulence Closure Model

The utilized DDES model is based upon the common Spalart-Allmaras (S-A) one-equation model [Spalart and Allmaras (1992)]. The S-A model is a one equation eddy-viscosity model, which in general requires the assumption of isotropic, homogeneous turbulence. It is an appropriate RANS model for the attached boundary layer improving the prediction of flows with adverse pressure gradients compared with $k-\epsilon$ and $k-\omega$ models [Bardina et al. (1997)].

This DDES formulation in [Spalart et al. (2005)] is similar to a proposal for the shear stress transport model in [Menter and Kuntz (2004)], which is not completely implemented in AcuSolve™ at this time.

The S-A model relates the eddy viscosity to a computed viscosity, which satisfies the following transport equation:

$$\frac{\partial \tilde{\nu}}{\partial t} + \mathbf{u} \cdot \nabla \tilde{\nu} - \frac{1}{\nu} \left\{ \nabla \cdot \left[ (\nu + \tilde{\nu}) \nabla \tilde{\nu} \right] \right\} = c_{b1} \tilde{\nu} S \tilde{\nu} + \frac{1}{\nu} c_{b2} (\nabla \tilde{\nu})^2 - c_{w1} f_w \left[ \frac{\tilde{\nu}}{d} \right]^2 , \quad (2.15)$$

with

$$f_{v1} = \frac{\chi^3}{\chi + c_{v1}}, \quad \chi = \frac{\tilde{\nu}}{\nu} ,$$

$$f_{v2} = 1 - \frac{\chi}{1+\chi f_{v1}}, \quad r = \frac{\tilde{\nu}}{S \kappa^2 d^2} ,$$

$$f_w = g \left( \frac{1+c_{w1}}{g^2+c_{w2}} \right)^{\frac{1}{6}}, \quad g = r + c_{w2} (r^6 - r)^{\frac{1}{6}},$$

$$S = |\nabla \times \mathbf{u}| + \frac{\tilde{\nu}}{\kappa^2 d^2} ,$$

where $d$ is the distance to the nearest no-slip surface. The eddy viscosity is then defined by $\nu_t = \tilde{\nu} f_{v1}$. The constants are listed in Table 2.1.

To enhance readability, Eq. 2.15 is rewritten as follows:

$$\frac{\partial \tilde{\nu}}{\partial t} + \mathbf{u} \cdot \nabla \tilde{\nu} - \nabla \cdot \mathbf{b} = \Psi , \quad (2.16)$$

where $\mathbf{b}$ denotes the diffusive flux:

$$\mathbf{b} = \frac{1}{\nu} (\nu + \tilde{\nu}) \nabla \tilde{\nu} , \quad (2.17)$$

and $\Psi$ the source terms on the right hand side:

$$\Psi = c_{b1} \tilde{\nu} S \tilde{\nu} + \frac{1}{\nu} c_{b2} (\nabla \tilde{\nu})^2 - c_{w1} f_w \left[ \frac{\tilde{\nu}}{d} \right]^2 . \quad (2.18)$$
Table 2.1: Spalart-Allmaras constants.

<table>
<thead>
<tr>
<th></th>
<th>Value</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_{b1}$</td>
<td>0.1355</td>
<td>$\vartheta = 2/3$</td>
</tr>
<tr>
<td>$c_{b2}$</td>
<td>0.622</td>
<td>$\kappa = 0.41$</td>
</tr>
<tr>
<td>$c_{w2}$</td>
<td>0.3</td>
<td>$c_w = 7.1$</td>
</tr>
<tr>
<td>$c_{w3}$</td>
<td>2.0</td>
<td>$c_{w1} = c_{b1}/\kappa^2 + (1 + c_{b2})/\vartheta$</td>
</tr>
</tbody>
</table>

The DDES model is obtained by a modification of the length scale in the (S-A)-RANS model. Therefore, the distance $d$ normal to the wall and parameter $r$ in Eq. (2.15) are replaced by $\tilde{d}$ and $r_d$ respectively, which are defined as follows:

$$
\tilde{d} = d - f_d \max(0, d - C_{DES}\Delta),
$$

with

$$
f_d = 1 - \tanh \left( [8r_d]^3 \right), \quad r_d = \frac{\nu_t + \nu'}{\sqrt{(\nabla u)^2 \kappa^2 d^2}},
$$

where $\nabla u$ denotes the velocity gradient, $\Delta = \max(\Delta x, \Delta y, \Delta z)$ is the local element length scale and $C_{DES} = 0.65$ the DES constant.

The above definition of the length scale deviates from the original formulation of the DES length scale, which is defined by:

$$
\tilde{d} = \min(d, C_{DES}\Delta).
$$

As a consequence, DDES avoids the appearance of the grid-dependent phenomena “Modelled Stress Depletion” (MSD) and “Grid Induced Separation” (GIS). MSD occurs in cases of thick boundary layers or shallow separation regions, if the grid spacing parallel to the wall is not well-chosen. In these cases, the grid spacing parallel to the wall may become smaller than the boundary layer thickness, either through grid refinement or boundary layer thickening. Hence, the LES limiter $C_{DES}\Delta$ is activated in the boundary layer, which reduces the eddy viscosity, and therefore, the modeled Reynolds stress. Unfortunately, the grid is not fine enough to replace the modeled stress with a resolved Reynolds stress, derived from the velocity fluctuations. Thus, the depleted stresses reduce the skin friction, which can lead to premature separation (GIS) [Spalart et al. (2005)].

The DDES modification (2.19), (2.20) overrides the original DES limiter. The crucial effect of this modification is, that the length scale also depends on the eddy viscosity, which is introduced in $r_d$. Thus, the length scale $\tilde{d}$ is time dependent.

Similar to $r$ in the S-A Model, $r_d$ equals 1 in the outer part of the boundary layer, and drops to 0 at the edge of the boundary layer. $r_d$ is used in the blending function $f_d$, which is designed to be 0 everywhere except in the LES region, where $f_d$ equals 1 (see
2.3 Mathematical Modeling

Figure 2.3: Distributions in a flat-plate boundary layer: (solid) \(-\frac{u}{u_\infty}\); \(-\cdot f_d\); \(-\cdot r_d\); \(\cdots\) [Spalart et al. (2005)].

Fig. 2.3. Combined with the new definition of the length scale \(2.19\), a transition to LES is inhibited, if \(f_d\) determines a point inside the boundary layer.

In the DDES the RANS behavior does not depend on the ratio of the boundary layer thickness and the mesh spacing parallel to the wall. Therefore, the full RANS functionality in the boundary layer is maintained, while LES behavior in massively separated regions is still achieved [Spalart et al. (2005)]. Thus, DDES might be the most appropriate turbulence model for the purpose of massively separated flows in external aerodynamics [Mockett et al. (2008)]. For an in-depth analysis of DDES refer to [Spalart et al. (2005)].

Nevertheless, the utilization of DDES still requires an appropriate grid generation.

2.3.5 Near-Wall Modeling

The importance of the near-wall flow has been shown in section 2.1, 2.3.2 and 2.3.4, thus the mathematical modeling of the wall must be considered separately.

First of all, the dimensionless distance and velocity are introduced:

\[ y^+ = \frac{u^*d}{\nu}, \quad (2.22) \]

\[ u^+ = \frac{u_||}{u^*}, \quad (2.23) \]

where \(y^+\) denotes the dimensionless distance normal to the wall, \(u_||\) denotes the velocity parallel to the boundary, and

\[ u^* = \sqrt{\tau_w}, \quad (2.24) \]
is called friction velocity. The wall shear stress is defined by:
\[
\tau_w = (\nu + \nu_t) \left[ \frac{\partial u_i}{\partial n} \right]_{d=0}.
\] (2.25)

In general, the turbulent boundary layer can be divided into three domains:

- **Viscous sub-layer:** \( y^+ \leq 5 \).
  The viscous sub-layer is a very thin region close to the no-slip wall. The flow velocity is decreased, thus the turbulent fluctuations are damped and the flow obtains a laminar flow profile. Thus, the eddy viscosity is close to zero.

- **Buffer layer:** \( 5 < y^+ < 30 \).
  In the Buffer layer both viscous and turbulent stresses are present. However, the ratio between both is unknown.

- **Logarithmic layer:** \( y^+ \geq 30 \).
  In this outer part of the boundary layer, the average velocity at a certain point is proportional to the logarithm of the distance from that point to the solid boundary. In the log-layer, the viscous stresses are negligible compared to the Reynolds stresses.

As it is requested to capture the flow physics accurately including adverse pressure gradients and thus, flow separation, the near wall gradients must be resolved. Therefore, the mesh spacing in the boundary layer must be considered carefully, which is further described in section 3.2. The wall function is only valid for non-separated flows and mild pressure gradients. Thus, this approach is not suitable here. However, the wall handling implemented in AcuSolve has no lower limit on the grid spacing. It is formulated to approximate the resolved solution, if a (local) \( y^+ \) value is in the laminar sub-layer.

The law of the wall holds the relationship:
\[
\begin{align*}
  u^+ &= y^+, & \text{with} & \quad y^+ < 5, \\
  u^+ &= \frac{1}{\kappa} \log(y^+) + 5.5, & \text{with} & \quad y^+ \geq 30.
\end{align*}
\] (2.26)

In the buffer layer, neither law holds. However, the linear and logarithmic velocity profile intersects around \( y^+ \sim 11 \). In simple approaches, this \( y^+ \) value is used to switch between the linear and logarithmic velocity profile.

Consider Eq. (2.22) and (2.23), the relationship (2.26) takes the form:
\[
 f(\tau_w, n, u_{i|}, \nu) = 0.
\] (2.27)

The Newton method is applied to solve this nonlinear system of equations. At next, the magnitude of the wall traction is calculated. Finally, the total viscosity within the volume element close to the wall is adjusted, such that this relation still holds at the
This wall-modeling approach approximates to the resolved solution. For small $y^+$ values, the wall model has no influence. The flow in the boundary layer is resolved. If a $y^+$ value increases, the adaptation of the viscosity leads to the fulfilment of Eq. (2.26). Hence, this wall model enhances the accuracy in case of increasing $y^+$ values.

It is worth noting, that the $y^+$ values considered in later chapters are always the integrated $y^+$ value of the respective component surface, e.g.:

$$y^+ := \frac{\int_{\Gamma_v} y^+ d\Gamma_v}{\int_{\Gamma_v} d\Gamma_v} \quad (2.28)$$

### 2.3.6 Boundary Conditions

To obtain a mathematically closed system for Eqs. (2.13), (2.12), (2.15) appropriate initial and boundary conditions are required. The boundary of the flow domain $\Omega$ is partitioned into two disjoint types, the Dirichlet boundary $\Gamma_g$ and Neumann boundary $\Gamma_h$. On the Dirichlet boundary a fixed value is prescribed for the velocity vector and the eddy viscosity:

$$u(x, t) = g \quad \text{on } \Gamma_g, \quad (2.29)$$
$$\nu_t(x, t) = G \quad \text{on } \Gamma_g, \quad (2.30)$$

On the Neumann boundary the total stress as well as the diffusive flux of the eddy viscosity normal to the boundary is specified:

$$\sigma \cdot n = h \quad \text{on } \Gamma_h, \quad (2.31)$$
$$u \cdot n = v \quad \text{on } \Gamma_h, \quad (2.32)$$
$$b \cdot n = B \quad \text{on } \Gamma_h. \quad (2.33)$$

Additionally, the flow domain $\Omega$ requires an appropriate initial condition, defined by

$$u(x, 0) = u_0 \quad \text{on } \Omega, \quad (2.34)$$
$$\nu_t(x, 0) = \nu_0 \quad \text{on } \Omega, \quad (2.35)$$

which satisfies

$$\nabla \cdot u_0 = 0. \quad (2.36)$$

### 2.4 Numerical Formulation

#### 2.4.1 Finite Element Discretization

To solve the system of Navier-Stokes equations (2.12), (2.13), (2.29), (2.31), (2.34) numerically using the finite element method (FEM), one has to transform this so-called strong formulation into a weak form. Therefore, the momentum equation (2.13)
and continuity equation (2.12) are multiplied by the weighting functions \( w \in V_0 \) and \( q \in Q \), respectively. The function spaces \( V_0 \) and \( Q \) consist of all test functions \( w \) and \( q \), respectively, whose zeroth and first order derivatives have to be square-integrable. While \( w \) has to vanish on the Dirichlet boundary, the so-called trial function \( u \in V_g \) must satisfy the Dirchlet conditions on \( \Gamma_g \). Hence, the function spaces \( V_0 \), \( V_g \) and \( Q \) in (2.41) are first-order Sobolev spaces \( H^1 \) \[ Donea and Huerta (2003) \]:

\[
\begin{align*}
V_g &= \{ u \mid u \in (H^1(\Omega))^3 : u = g \text{ on } \Gamma_g \} , \\
V_0 &= \{ w \mid w \in (H^1(\Omega))^3 : w = 0 \text{ on } \Gamma_g \} , \\
Q &= \{ p \mid p \in (H^1(\Omega)) \} .
\end{align*}
\]

Both, the momentum equation and continuity equation, are integrated by parts, thereby generating the Neumann boundary condition on \( \Gamma_h \).

The resulting integral equation can be written as:

\[
\begin{align*}
\int_{\Omega} w \left( \frac{\partial u}{\partial t} + u \cdot \nabla u - f \right) & d\Omega + \int_{\Omega} \nabla w : \sigma \ d\Omega \\
\quad - \int_{\Gamma_h} w \cdot h - qv \ d\Gamma - \int_{\Omega} \nabla q \cdot u \ d\Omega &= 0 .
\end{align*}
\]

The continuity equation is integrated by parts to provide discrete conservation of mass, and the pressure term is integrated by parts to provide symmetry with the continuity equation, which is discussed in detail in \[ Gresho and Sani (1998) \].

The Galerkin spatial discretization replaces \( V_g, V_0 \) and \( Q \) by finite-dimensional subspaces \( V_h, V_0^h \) and \( Q^h \). Therefore, the flow domain \( \Omega \) is partitioned into \( n_{el} \) disjoint convex subdomains \( \Omega^e \) with a piecewise smooth boundary \( \partial \Omega^e \).

The local approximations for the trial functions \( u_h, p^h \), and the corresponding weighting functions \( w^h, q^h \) have to fulfill the above mentioned requirements on the spatial subdomains, thus they are in the discrete Sobolev space \( H^{1h} \):

\[
\begin{align*}
V^h_g &= \{ u^h \mid u^h \in (H^{1h}(\Omega))^3, u^h|_{x \in \Omega_e} \in P_1(\Omega^e)^3 : u^h = g^h \text{ on } \Gamma^h_g \} , \\
V^h_0 &= \{ w^h \mid w^h \in (H^{1h}(\Omega))^3, w^h|_{x \in \Omega_e} \in P_1(\Omega^e)^3 : w^h = 0 \text{ on } \Gamma^h_g \} , \\
Q^h &= \{ p^h \mid p^h \in (H^{1h}(\Omega)), p^h|_{x \in \Omega_e} \in P_1(\Omega^e) \} ,
\end{align*}
\]

where \( P_1(\Omega^e) \) is the space of all linear polynomials defined on \( \Omega^e \).

The Galerkin form of the filtered Navier Stokes equations (2.13) reads:

Given \( f, g \) and \( h \), find \( u^h \in V^h_g, p^h \in Q^h \) such that \( \forall w^h \in V^h_0, q^h \in Q^h \) :

\[
\begin{align*}
\int_{\Omega_h} w^h \left( \frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h - f \right) & d\Omega + 2 \int_{\Omega_h} (\nu + \nu_l) \varepsilon(w^h) : \varepsilon(u^h) \ d\Omega \\
\quad - \int_{\Omega_h} w^h \cdot p^h \ d\Omega - \int_{\Gamma_h^h} w^h \cdot h^h - q^h v^h \ d\Gamma - \int_{\Omega_h} \nabla q^h \cdot u^h \ d\Omega &= 0 ,
\end{align*}
\]

(2.39)
2.4 Numerical Formulation

with

\[ \mathbf{u}^h(\mathbf{x}) = \sum_{i=1}^{n_p} N_i(\mathbf{x}) u_i \quad \text{and} \quad p^h(\mathbf{x}) = \sum_{i=1}^{n_p} M_i(\mathbf{x}) p_i, \]

(2.40)

where \( N_i(\mathbf{x}) \) as well as \( M_i(\mathbf{x}) \) are the element shape functions. \( n_p \) is the number of nodal points, \( u_i \) and \( p_i \) denote the nodal unknowns in the resulting system of equations.

This general Galerkin method suffers from stability problems for advection-dominated flows, because the advection operator is not self-adjoint [Brooks and Hughes (1982)]. In the considered problem the non-linear advection term dominates over the viscous term. Thus, the standard Galerkin method produces spurious and growing oscillations appearing in the vicinity of steep gradients, which pollute the entire domain. Furthermore, a solution of (2.39) using the Galerkin method with equal-order interpolation of velocity and pressure does not satisfy the necessary LBB-condition [Donea and Huerta (2003); Hughes et al. (1986)]. Although, an equal-order nodal interpolation is not necessary, it is computationally convenient, especially when working with higher-order discretizations [Whiting (1999)].

To circumvent these two problems a stabilization technique can be applied, which utilizes tightly controlled artificial diffusion.

Stabilized finite element formulations have been shown to be stable, robust and accurate for steady and unsteady Navier-Stokes equations, large eddy simulations and Reynolds averaged simulations of turbulent flows [Whiting (1999)]. A comprehensive review of the so-called Petrov-Galerkin stabilization approaches is given in [Fries and Matthies (2004)].

As AcuSolve™ uses the Galerkin/Least-Squares (GLS) finite element formulation, the stabilization for (2.39) is described in section 2.4.2.

Similarly to the procedure with the Navier-Stokes equations, the weak form of the eddy viscosity transport equation (2.16) is obtained by multiplying the trial function with weighting functions \( \phi \), integrating over the domain, applying the chain rule and the divergence theorem and replacing the function spaces by finite dimensional subspaces:

\[
\begin{align*}
S_G^h & = \{ \mathbf{\tilde{v}}^h \mid \mathbf{\tilde{v}}^h \in H^1(\Omega), \mathbf{\tilde{v}}^h|_{x \in \Omega_e} \in \mathcal{P}_1(\Omega_e) : \mathbf{\tilde{v}}^h \cdot \mathbf{n}^h = G^h \text{ on } \Gamma_y \} , \\
S_0^h & = \{ \phi^h \mid \phi^h \in H^1(\Omega), \phi^h|_{x \in \Omega_e} \in \mathcal{P}_1(\Omega_e) : \phi^h = 0 \text{ on } \Gamma_g \} , \\
\end{align*}
\]

(2.41)

Then, the Galerkin formulation of the advective-diffusive transport equation reads:

Find \( \mathbf{\tilde{v}}^h \in S_G^h \) such that \( \forall \phi^h \in S_0^h \):

\[
\begin{align*}
\int_{\Omega^h} \phi^h \left[ \frac{\partial \mathbf{\tilde{v}}^h}{\partial t} + \mathbf{u}^h \cdot \nabla \mathbf{\tilde{v}}^h \right] d\Omega + \int_{\Omega^h} \nabla \phi^h \cdot \mathbf{b}^h d\Omega \\
- \int_{\Gamma^h} \phi^h B^h d\Gamma + \int_{\Omega^h} \phi^h \Psi^h d\Omega = 0
\end{align*}
\]

(2.42)
2.4 Numerical Formulation

Eq. (2.42) is unstable due to the appearance of an advective term. The stabilization of the transport equation with the Galerkin/Least-Squares Approach is analogous to the stabilization of the Navier-Stokes equations, which is addressed in the next section.

2.4.2 Galerkin-Least Squares Formulation

In general, a consistent stabilization of the Galerkin formulation in Eq. (2.39) is obtained by adding a stabilization term to the weak form, i.e.

\[ \sum_e \int_{\Omega^e} J(w^h) \tau R(u^h) d\Omega, \]  

(2.43)

where \( J(w^h) \) is a perturbation operator for the test function, \( \tau \) the stabilization parameter and \( R(u^h) \) the residual of the differential equation.

The important stabilization parameter \( \tau \) is separately introduced for each equation. Several proposals for \( \tau \) are well documented in [Franka and Frey (1992); Shakib (1989)]. As the solver converges, the residual falls to zero and the discretized differential equations are exactly satisfied.

The Galerkin/Least-Squares stabilization [Hughes et al. (1989); Shakib (1983)] is an element-by-element weighted least-squares residual of the original differential equation, i.e.

\[ R(u^h) = L(u^h) - f. \]

(2.44)

Thus, the operator applied to the test function for the momentum equation reads as follows:

\[ L(u^h) = \frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h + \nabla p^h - \nabla (2(\nu + \nu_t) \varepsilon(u^h)). \]

(2.45)

With the definition of \( L(u^h) \) in (2.45), the GLS formulation of the filtered Navier-Stokes problem (2.13), (2.12) results in:

Given \( f, g \) and \( h \), find \( u^h \in V^h_g, p^h \in Q^h \) such that \( \forall w^h \in V_0^h, q^h \in Q^h \)

\[ \int_{\Omega} w^h \left( \frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h - f \right) d\Omega + 2(\nu + \nu_t) \int_{\Omega} \varepsilon(w^h) : \varepsilon(u^h) d\Omega \]

\[ - \int_{\Omega} \nabla \cdot w^h p^h d\Omega - \int_{\Omega} \nabla q^h \cdot u^h d\Omega - \int_{\Gamma} w^h \cdot h^h - q^h v^h d\Gamma \]

\[ + \sum_{e=1}^{ne} \int_{\Omega^e} \left\{ u^h \cdot \nabla w^h + \nabla q^h - \nabla \left( 2(\nu + \nu_t) \varepsilon(u^h) \right) \right\} \tau_m \]

\[ \cdot \left\{ \frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h - f + \nabla p^h - \nabla \left( 2(\nu + \nu_t) \varepsilon(u^h) \right) \right\} d\Omega_e \]

\[ + \sum_{e=1}^{ne} \tau_e \nabla \cdot w^h \nabla \cdot u^h d\Omega_e = 0. \]
The stabilization of the momentum term using $\tau_m$ remedies the two numerical difficulties, which were already discussed in section 2.4.1. Furthermore, the stabilization of the continuity equation using $\tau_c$ provides stability to the velocity field at high Reynolds numbers.

The GLS formulation of the turbulence model is straightforward. The stabilization term, i.e.

$$\sum_{e=1}^{n_{el}} \int_{\Omega_e} \mathcal{J}(\phi^h) \tau_s R(\bar{\nu}^h) \, d\Omega_e (2.47)$$

is added to Eq. (2.42), where $R(\bar{\nu}^h)$ is the unstabilized residual of the scalar equation 2.16:

$$R(\bar{\nu}^h) = \mathcal{L}(\bar{\nu}^h) - \Psi^h = \frac{\partial \bar{\nu}^h}{\partial t} + \mathbf{u}^h \cdot \nabla \bar{\nu}^h - \nabla \cdot \mathbf{b}^h - \Psi^h (2.48)$$

The stabilization process of the transport equation is not discussed in further detail.

The implemented Galerkin/Least-Squares formulation provides second order accuracy for spatial discretization of all variables. The stabilization operators implemented in AcuSolve$^{\text{TM}}$ ensure both, global conservation and local conservation for individual elements. For further information regarding GLS refer to [Hughes et al. (1989)] and [Shakib (1989)]. Additional details about the specific implementation in AcuSolve$^{\text{TM}}$ are summarized by [Johnson and Bittorf (2002)].

Both, the Navier-Stokes equations and the scalar eddy viscosity equation are time-dependent. Therefore, the time discretization is addressed in the next section.

### 2.4.3 Time Discretization

By default, AcuSolve$^{\text{TM}}$ uses the generalized-$\alpha$ method [Jansen et al. (2000)] for time integration of transient problems$^1$.

The generalized-$\alpha$ time integrator is a predictor-multicorrector algorithm and has been verified as being second-order accurate in time [Lyons et al. (2007)]. In the following, the superscript $h$ is omitted for notational simplicity.

Two difficulties arise with the time integration of the Navier-Stokes equations. First of all, the pressure has no explicit time dependence. Thus, the pressure is iterated to remain consistent with the velocity which is integrated with the generalized-$\alpha$ method. Furthermore, due to the advection term a nonlinearity is introduced into the system of equations (2.46).

$^1$ The time step size for the steady-state analyses is $10 \times 10^{10}$, such that the time derivative tends to zero.
2.4 Numerical Formulation

The system of equations (2.46) can be written as a vector of nodal values of the nonlinear residuals. Velocity and its time derivative are evaluated at the intermediate time steps \( t_{n+\alpha_f} \) and \( t_{n+\alpha_m} \), respectively, i.e.:

\[
\mathbf{R} \left( \mathbf{u}_{n+\alpha_f}, \dot{\mathbf{u}}_{n+\alpha_m}, p_{n+1} \right) = 0 ,
\]

where \( \dot{\mathbf{u}} \) denotes the time derivative of the velocity field and \( n \) is the subscript for the current time step \( t_n \). To advance the solution in time, the update equation relates the velocity to its time derivative from a time \( t_n \) to a time \( t_{n+1} \) as follows:

\[
\mathbf{u}_{n+1} = \mathbf{u}_n + \Delta t \dot{\mathbf{u}}_n + \gamma \Delta t (\dot{\mathbf{u}}_{n+1} - \dot{\mathbf{u}}_n) .
\]

The temporal locations at \( n \) and \( n + 1 \) are related to the intermediate time step values \( n + \alpha_f \) and \( n + \alpha_m \) by:

\[
\dot{\mathbf{u}}_{n+\alpha_f} = \dot{\mathbf{u}}_n + \alpha_f (\dot{\mathbf{u}}_{n+1} - \dot{\mathbf{u}}_n) ,
\]

\[
\mathbf{u}_{n+\alpha_f} = \mathbf{u}_n + \alpha_f (\mathbf{u}_{n+1} - \mathbf{u}_n) .
\]

Proper selection of \( \alpha_f \), \( \alpha_m \), and \( \gamma \) determine the accuracy of the solution, as well as the stability of the method [Whiting (1999)].

The nonlinearities in the advection term are appropriately handled by a predictor-multicorrector algorithm. This algorithm begins with a predictor stage:

\[
\mathbf{u}_{n+1}^{(0)} = \mathbf{u}_n ,
\]

\[
\dot{\mathbf{u}}_{n+1}^{(0)} = \frac{\gamma - 1}{\gamma} \dot{\mathbf{u}}_n ,
\]

\[
P_{n+1}^{(0)} = p_n .
\]

The superscript (inside parentheses) represents the corrector iteration number. The predictor (2.53) is consistent with (2.51), such that 2\textsuperscript{nd} order accuracy is preserved. It is worth noting that other choices of predictors are also possible (e.g. refer to [Jansen et al] (2000)).

Afterwards, the loop of multi-corrector passes is entered. The computed intermediate solutions at \( t_{n+\alpha_f} \) and \( t_{n+\alpha_m} \), i.e.

\[
\dot{\mathbf{u}}_{n+\alpha_m}^{(i)} = \dot{\mathbf{u}}_n + \alpha_m (\dot{\mathbf{u}}_{n+1}^{(i-1)} - \dot{\mathbf{u}}_n) ,
\]

\[
\mathbf{u}_{n+\alpha_m}^{(i)} = \mathbf{u}_n + \alpha_m (\mathbf{u}_{n+1}^{(i-1)} - \mathbf{u}_n) ,
\]

\[
\dot{\mathbf{u}}_{n+\alpha_f}^{(i)} = \dot{\mathbf{u}}_n + \alpha_f (\dot{\mathbf{u}}_{n+1}^{(i-1)} - \dot{\mathbf{u}}_n) ,
\]

\[
\mathbf{u}_{n+\alpha_f}^{(i)} = \mathbf{u}_n + \alpha_f (\mathbf{u}_{n+1}^{(i-1)} - \mathbf{u}_n) ,
\]

19
enable the evaluation of Eq. (2.49). For small iteration numbers, the residual can be expected to be far away from its desired value of 0\(^2\). Thus, it is required to improve the values of \(\dot{\mathbf{u}}_{n+\alpha_m}\) and \(\mathbf{u}_{n+\alpha_f}\). Due to the non-linearity of the residual at the intermediate time steps, it is linearized. The linearization yields a sparse matrix system, which has to be solved for the velocity and pressure increments:

\[
\begin{pmatrix}
  \mathbf{K}^{(i)} & \mathbf{G}^{(i)} \\
  \mathbf{D}^{(i)} & \mathbf{C}^{(i)}
\end{pmatrix}
\begin{pmatrix}
  \Delta \dot{\mathbf{u}}_{n+1}^{(i)} \\
  \Delta \mathbf{p}_{n+1}^{(i)}
\end{pmatrix}
= - \begin{pmatrix}
  \mathbf{R}_{m}^{(i)} \\
  \mathbf{R}^{(i)}
\end{pmatrix}.
\] (2.59)

The matrix entries are approximations of the tangent matrices of the residual vectors, i.e.:

\[
\mathbf{K}^{(i)} \approx \frac{\partial \mathbf{R}^{(i)}_{m} \left( \dot{\mathbf{u}}_{n+\alpha_m}^{(i)}, \mathbf{u}_{n+\alpha_f}^{(i)}; \mathbf{p}_{n+1}^{(i)} \right)}{\partial \mathbf{u}_{n+1}^{(i)}},
\] (2.60)

\[
\mathbf{G}^{(i)} \approx \frac{\partial \mathbf{R}^{(i)}_{m} \left( \dot{\mathbf{u}}_{n+\alpha_m}^{(i)}, \mathbf{u}_{n+\alpha_f}^{(i)}; \mathbf{p}_{n+1}^{(i)} \right)}{\partial \mathbf{p}_{n+1}^{(i)}},
\] (2.61)

\[
\mathbf{D}^{(i)} \approx \frac{\partial \mathbf{R}^{(i)}_{c} \left( \dot{\mathbf{u}}_{n+\alpha_m}^{(i)}, \mathbf{u}_{n+\alpha_f}^{(i)}; \mathbf{p}_{n+1}^{(i)} \right)}{\partial \dot{\mathbf{u}}_{n+1}^{(i)}},
\] (2.62)

\[
\mathbf{C}^{(i)} \approx \frac{\partial \mathbf{R}^{(i)}_{c} \left( \dot{\mathbf{u}}_{n+\alpha_m}^{(i)}, \mathbf{u}_{n+\alpha_f}^{(i)}; \mathbf{p}_{n+1}^{(i)} \right)}{\partial \mathbf{p}_{n+1}^{(i)}}.
\] (2.63)

These matrix entries determine the direction to increment the solution \(\Delta \dot{\mathbf{u}}_{n+1}^{(i)}\) and \(\Delta \mathbf{p}_{n+1}^{(i)}\), respectively.

To solve the velocity equation, a Preconditioned Generalized Minimum Residual (PGMRES, \cite{Saad1986}) solution method is applied. The preconditioning is performed using an technique similar to algebraic multigrid domain decomposition techniques, and can not be deepened within this thesis. A conjugate gradient projection method (CGP) is used to solve the pressure equation. The efficiency of element-by-element GMRES solution techniques has been shown by \cite{Shakib1991}. Thus, the iterative solution strategies are not described in detail.

Once this matrix is solved, the values of the solution variables are updated.

\[
\dot{\mathbf{u}}_{n+1}^{(i+1)} = \dot{\mathbf{u}}_{n+1}^{(i)} + \Delta \dot{\mathbf{u}}_{n+1}^{(i)},
\] (2.64)

\[
\mathbf{u}_{n+1}^{(i+1)} = \mathbf{u}_{n+1}^{(i)} + \gamma \Delta \mathbf{u}_{n+1}^{(i)},
\] (2.65)

\[
\mathbf{p}_{n+1}^{(i+1)} = \mathbf{p}_{n+1}^{(i)} + \Delta \mathbf{p}_{n+1}^{(i)}.
\] (2.66)
If neither the maximum number of iterations nor the convergence of the nonlinear solver is reached, the next corrector pass is entered. Otherwise, the time step is completed.

The GLS formulation of the turbulence equation (2.42) is solved separately from the flow equations. However, the time integration is performed analogous to the previous procedure, and thus, it is not further deepened within this thesis.
3 Simulation Process

In this chapter, the simulation process of the virtual wind tunnel tool is described. The different vehicles and the wind tunnel design are introduced in Section 3.1. In Section 3.2 the background for the mesh setup and the parametrized meshing process is described. The specification of the solver settings in Section 3.3 completes the preprocessing. Section 3.4 provides insight into the postprocessing procedure.

3.1 Model Setup

3.1.1 Wind Tunnel Domain

In a computational analysis, the flow domain must be defined carefully to ensure the reliability of the results. First of all, the vehicle must be placed in the virtual wind tunnel, such that there is enough space between the inlet, the vehicle and the outlet. Then, the dissipation of vortices downstream from the vehicle does not disturb the solution upstream, and the pressure at the stagnation point evolves reasonably.

Several proposals for appropriate object placement exist in literature, but a common recommendation is described e.g. in [Lanfran (2005)]. This guideline states, that there should be at least three vehicle lengths of space in front of the car, and five vehicle lengths behind.

Furthermore, one has to consider that the ratio of the vehicle cross section to the wind tunnel cross section is within a certain range. This so-called blockage ratio in the wind tunnel has to be less than 6% [West and Apelt (1982)] or even less than 2% [Lanfran (2005)]. As a result, the effects of the wind tunnel walls on the pressure distribution and thus, the drag coefficient are small. Otherwise, the flow field around the car is disturbed by wall influences.

3.1.2 ASMO Model

The ASMO ("Aerodynamisches Studien MOdell") was created more than ten years ago in the Daimler Benz research department. The intention was to have a neutral body for the validation of CFD-codes, which is not related to their actual car development. Furthermore, it was attempted to create a feasible body, which produces a very low drag force. In Fig. 3.2b 3.1b the surface mesh of the asmo is shown from two perspectives. The ASMO body has some geometrical simplifications: a square back rear, smooth surface and boat tailing. No pressure induced boundary layer separation
3.1 Model Setup

(a) Vehicle
(b) Vehicle back

Figure 3.1: ASMO: Surface mesh.

occurs and thus, the flow does not detach before it reaches the rear end of the vehicle [Perzon and Davidson (2000)]. Therefore, a lower contribution of the pressure induced drag force is to be expected [Aronson et al. (2000)]. However, the geometrical shape in the back is obviously much closer to a hatchback than a notchback. This geometrical configuration favors an oscillating wake flow [Seibert et al. (2002)]. Complex geometrical details like, e.g., exterior mirrors are not present which facilitates an easy mesh generation.

The computational domain describes a virtual solid wall wind tunnel. The placement of the vehicle in the domain is in accordance to the specified guideline in section 3.1.1, except the clearance between the vehicle and outlet. As the wake extent of the ASMO is not to be expected as large as of real passenger cars, this slight difference can be neglected.

Furthermore, the cross section ratio of the ASMO model to the wind tunnel results in a blockage ratio $\zeta_{WT,A}$ of 2.1%. Thus, no disturbance of the solution due to wind tunnel walls is to be expected.

The dimensions of the ASMO model and the virtual wind tunnel are summarized in Appendix A.1 Table A.1 and Table A.2, respectively.

For the mesh refinement, a rectangular box around the car is specified; in the following referred to as core zone. All tetrahedral elements in this zone except the boundary layer and adjacent element layers will have a uniform size, which is in the following referred to as core element size. The core zone extends to 0.45 m in vertical- and 0.4 m in spanwise direction$^1$. In front of the vehicle, the core zone begins at a distance of 0.2 m to the first car surface mesh point, i.e. the stagnation point. In flow direction, the core zone length behind the vehicle varies with $L_{WL,A}$ which is defined in subsequent sections. For the sake of clarity, this core length behind the vehicle is in the following referred to as core length.

$^1$ The y-direction normal to the ground plane is referred to as vertical direction, the z-direction normal to the symmetry plane is referred to as spanwise direction.
3.1 Model Setup

(a) Surface mesh

(b) Volume mesh (core zone around the vehicle)

Figure 3.2: ASMO: Virtual wind tunnel.

3.1.3 Sedan Model

The Sedan model is a real passenger car model containing various details. The flow separation is geometrically well defined, such that the total drag is minimized. However, due to the non-streamlined shape and the hatchback configuration an increase of the pressure-induced contribution to the drag is to be expected. The Sedan model is analysed for two different configurations.

The first model, in the following referred to as simple sedan, has been provided by an automotive company. The objectives of the simple sedan analysis are both to review the reliability of mesh control settings for arbitrary vehicle shapes and to investigate time stepping issues. The surface mesh of the simple sedan model has been modified within this thesis. The originated variant is referred to as complex sedan model. An engine is included into the motor compartment and a simplified radiator grill is modeled into the surface mesh in the front region. Thus, the flow through the engine compartment is taken into account. An exhaust system is connected to the engine which enhances the complexity of the flow in the underbody region. Therefore, the shape of the surface mesh is changed at different locations.

(a) Rear wheel

(b) Underbody view

Figure 3.3: Simple sedan: Model features.
3.1 Model Setup

Additionally, another more detailed set of wheels is inserted into this model which allows to investigate a first layer height for a different wheel configuration. Furthermore, the mesh requirements in the engine compartment and underbody region are analysed. Another objective of this analysis is to evaluate the capability of the underlying meshing scheme regarding thin gaps in complex models. The model details are shown in Fig. 3.4. The virtual wind tunnel for the sedan model is also a solid wall wind tunnel. The vehicle dimensions, wind tunnel properties and core zone dimensions are listed in Appendix A.3, Table A.3 and Table A.4 respectively.

Figure 3.4: Complex sedan: Model features.

Figure 3.5: Sedan model: Wind tunnel and flow characteristics.
3.1.4 Boundary Conditions

3.1.4.1 Fixed Ground

The boundary conditions in the virtual wind tunnel are as follows. Recalling Eq. 2.29, the Dirichlet boundary condition $\Gamma_g$ specifies all components of the velocity and the eddy viscosity. The associated boundary is divided into two parts $\Gamma_{g,Wall}$ and $\Gamma_{g,Inflow}$. The boundary $\Gamma_{g,Wall}$ thereby is composed as follows:

$$\Gamma_{g,Wall} = \Gamma_{g,Bottom} \cup \Gamma_{g,Body} \cup \Gamma_{g,Wheels} ,$$  \hspace{1cm} (3.1)

For $\Gamma_{g,Wall}$ a no-slip condition is imposed:

$$\mathbf{u}(\mathbf{x}, t) = 0 , \quad \text{on } \Gamma_{g,Wall} ,$$ \hspace{1cm} (3.2)

$$\nu_t(\mathbf{x}, t) = 0 , \quad \text{on } \Gamma_{g,Wall} ,$$ \hspace{1cm} (3.3)

The Inlet boundary $\Gamma_{g,Inflow}$ used to specify a non-zero inflow velocity and eddy-viscosity:

$$\mathbf{u}(\mathbf{x}, t) = \mathbf{u}_{in} , \quad \text{on } \Gamma_{g,Inflow} ,$$ \hspace{1cm} (3.4)

$$\nu_t(\mathbf{x}, t) = \nu_{t,in} , \quad \text{on } \Gamma_{g,Inflow} ,$$ \hspace{1cm} (3.5)

A symmetry boundary 2.31 imposes a zero normal component of velocity and eddy viscosity on the roof and side walls of the wind tunnel.

On the outlet a zero pressure condition

$$p(\mathbf{x}, t) = 0 , \quad \text{on } \Gamma_{g,Outflow} ,$$ \hspace{1cm} (3.6)

is imposed. The fluxes of the tangential traction $\mathbf{T}$ and diffusive turbulence part $\mathbf{b}$ are computed from the flow field. To ensure that the mass flux at the outflow is properly directed and does not reverse, the wind tunnel domain must be defined properly.

The initial conditions from Eq. 2.34, $\mathbf{u}_0$ and $\nu_0$, equal the inflow conditions, i.e.:

$$\nu_0 \equiv \nu_{t,in} , \quad \mathbf{u}_0 \equiv \mathbf{u}_{in} .$$ \hspace{1cm} (3.7)

It is worth noting that in the discretized flow problem $\Gamma_g$ and $\Gamma_h$ are not necessary disjoint because they have a common edge, i.e. share nodes at the outer edges of the wind tunnel domain. Therefore, the no-slip condition has precedence, if two boundary conditions are imposed at the same node.

3.1.4.2 Moving Ground

Today, the experiments and simulations in automotive aerodynamics are very advanced. In order to analyse realistic road conditions, many wind tunnels today are equipped with a moving belt and the wheels of the cars have the possibility to rotate.
3.1 Model Setup

The vehicle is actually not moving. However, the ground below the vehicle is moving and the wheels rotate on this moving ground. This setup enables the analysis of the interaction of the vehicle body with the road. Therefore, this wind tunnel setup is reproduced in the virtual wind tunnel tool and additional boundary conditions must be defined.

The investigated set of boundary conditions for different wind tunnel configurations is shown in Fig. 3.6.

![Figure 3.6: Moving ground: Boundary conditions for different configurations: green: inlet, red: outlet, grey: no-slip condition, blue: symmetry condition, yellow: moving ground](image)

The bottom of the wind tunnel can be divided as follows:

\[ \Gamma_{g, Bottom} = \Gamma_{Entrance} \cup \Gamma_{Moving} \cup \Gamma_{Fixed} \]

Realistic real road simulations require the prevention of a developed boundary layer at the ground, as the vehicle is moving at high speeds and not the air itself. Therefore, two different boundary conditions for the entrance section are considered, a no-slip and slip condition. The slip condition is closer to the physical problem, because it avoids the development of a boundary layer. The results of both variants are compared.

The wind tunnel model is equipped with a moving ground below the vehicle, which has the same extents as the core zone\(^2\). The velocity of the moving ground equals the inflow velocity \( u_{in} \). Thus, no boundary layer may develop on this moving ground component.

The tangential velocity on the wheels is prescribed using a rotational reference frame. Therefore, a center of rotation is defined in the center of the wheels. An angular velocity \( \omega \) proportional to the inflow velocity and the diameter of the wheels \( d_w \) is prescribed to the wheel components:

\[ \omega = \frac{u_{in}}{d_w} \quad (3.8) \]

Table 3.1 shows the resulting angular velocities for both models. As a rotational reference frame is used, the mesh is not moving compared to a sliding mesh method. This approach facilitates the mathematical modeling.

---

\(^2\) Another possible configuration would be the movement of the entire wind tunnel ground. However, further configurations are not evaluated within this thesis.
Table 3.1: Moving ground: Specification of wheel velocity

<table>
<thead>
<tr>
<th></th>
<th>wheel diameter [m]</th>
<th>angular velocity [rad]</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASMO</td>
<td>0.046</td>
<td>1099</td>
</tr>
<tr>
<td>Sedan</td>
<td>0.346</td>
<td>144.5</td>
</tr>
</tbody>
</table>

3.1.4.3 Turbulence Boundary Conditions

The definition of appropriate boundary conditions for the turbulence modeling in external aerodynamics is essential. Physically incorrect definitions can cause an unrealistic solution or even divergence. Different turbulent quantities can be used to define the initial and boundary conditions.

In the studies Perzon and Davidson (2000); Aronson et al. (2000) a turbulence intensity $I$ of 0.1% is prescribed both on the inlet and the initial flow field. In AcuSolve\textsuperscript{TM}, an eddy viscosity is prescribed using the relationship:

$$\nu_{t,\text{in}} = \sqrt{\frac{3}{2}} u_\infty I,$$

where $l$ is the turbulent length scale. In external flows, $l$ is hard to predict. Therefore, the value of the minimal displacement thickness on the ground of the empty wind tunnel (2.6mm) is taken from the experimental setup in Aronson et al. (2000). Using Eq. (3.9), a value of $\nu_{t,\text{in}} = \nu_0 \sim 5 \times 10^{-5} m^2/s$ results. This value is used throughout this thesis, as it is within a proper bound for wind tunnel flows: $\frac{\nu}{\nu} < 10$ [Lanfran](2005).

3.2 Mesh Generation

Adequate grid design for a highly turbulent unsteady external aerodynamic flow using the DDES turbulence model is not trivial. Thus, one has to consider requirements of the mesh to cover the various physical effects in the different flow regions.

Therefore, the contribution of both the friction force and pressure-induced force on the vehicle must be considered separately. The accuracy of their prediction is mostly determined by two important mesh settings, the first layer height evaluated in terms of $y^+$ and the element size in the core zone. This investigation is expensive due to repeated simulation with appropriate grids which contain at least several millions of elements. But grid variation is the only conclusive approach both, to test sensitivity of the results and to estimate the residual error [Spalart (2001)].

\footnote{The first layer height describes the thickness of the first volume element normal to the surface element.}
The results of the mesh sensitivity analysis are used to determine a correlation of mesh parameters to wind tunnel properties. Optimal mesh parameters of this particular problem are stated. Then, the mesh settings are validated using the Sedan model.

The objective is to provide three different default mesh settings, “coarse”, “medium”, and “fine”.

The coarse one serves as an evaluation option of the correct simulation set-up. The user should be enabled to adapt the grid or solver settings before starting a computationally intensive run. Thus, even a “coarse” grid should capture the qualitative effects in a wind tunnel. The “medium” setting should yield a reasonable physical behavior in a quantitative manner, using the optimal parameters evaluated in this work. The option “fine” produces a very fine grid to ensure a sufficient accurate solution. This third option is no main objective in this thesis because producing huge grids with sufficiently refinement is feasible by the use of meshing guidelines. Furthermore, a very fine grid demands huge computing resources.

3.2.1 General Mesh Requirements

The majority of studies utilizing a DES turbulence model propose an appropriate mesh generation only for their particular problems (cf. [Favre and Efraimsson (2011), Lyons et al. (2007)]). Many refer to a comprehensive grid generation guide for a detached eddy simulation by [Spalart (2001)]. Thus, it provides the basis for the mesh parameter definition.

According to this guideline, one has to outline the different flow regions. We must consider both the local requirements individually as well as the transition between these flow regions in order to gather a coherent grid design.

Because the main focus is on the prediction of forces on the vehicle, the underlying surface mesh must be considered. The contribution of the pressure drag component is influenced by the number of surface grid points [Lopes and Carvalheira (2003)]. Thus, a surface mesh with a good quality is a preliminary requirement. To accurately capture the friction component one has to create an appropriate boundary layer mesh. On a streamlined shape like the ASMO, a noticeable part of the drag is due to friction. Therefore, one has to pay high attention to the grid design near the wall.

The most important parameter relating the underlying physics to mesh properties is the dimensionless distance of the first mesh point off of the nearest wall, expressed in terms of $y^+$ (Eq. 2.22). Former studies have shown the major importance of an appropriate choice of $y^+$ to obtain reasonable results of the drag coefficient, where the value is within a range of:

- $y^+ \propto 1$ [Bardina et al. (1997)],
- $y^+ \propto 2.5$ [Lopes and Carvalheira (2003); Favre and Efraimsson (2011)].
3.2 Mesh Generation

- \( y^+ \propto 5 \) [Spalart (2001)],
- \( y^+ \propto 8 \) [Corson (2007)],

if the flow near the vehicle surface is of primary interest. This indicates, that the exact optimal value depends on the particular problem. Thus, in some cases the first layer height must be chosen such that \( y^+ \) is far below 1 [Corson (2007)]. However, for a streamlined car shape, [Lopes and Carvalheira (2003)] observed an increase of the drag coefficient, if \( y^+ \) decreases below a value of 2.

Furthermore, the importance of the first layer height on the bottom is investigated. According to the definition in [Favre and Efraimsson (2011)], a \( y^+ \) value around 30 is reasonable.

The first layer height itself can not ensure the accuracy of the drag prediction. To resolve the fluctuations of the flow quantities in the outer layer, the different sublayers within the boundary layer must be captured accurately. Therefore, the ratio of adjacent element heights in normal direction to the wall, in the following referred to as growth rate, is important. The recommended definition from [Spalart (2001)] is utilized and the growth rate is set close to 1.3 throughout this thesis.

The underlying meshing algorithm of AcuSolve supports a boundary layer mesh option called “match outer layer”. This setting ensures the creation of as element layers as required to keep the element growth rate constant even for the transition from the last outer boundary layer elements to the adjacent element layer. Hence, this results in a smooth transition of the mesh spacing to adjacent elements. This meshing scheme provides enough elements throughout the boundary layer [Corson (2007)]. Thus, even the outer sublayer is ensured to be captured accurately [Spalart (2000b)]. This boundary layer handling results in a non-uniform distribution of boundary layers elements around the body. However, the resulting number of layers is similar to the preferred choice in other cited studies (e.g. refer to [Favre and Efraimsson (2011)]).

Despite this comfortable meshing control, the effect of a specified constant number of boundary layer elements is investigated in this thesis. This approach leads to an observable decrease in the total numbers of elements and thus, in computing time. However, this also may result in a sudden jump in the element size from the last boundary layer element to adjacent elements.

A comparison of these boundary layer meshing types is shown in Fig. 3.7. It can be clearly seen, that the utilization of the “match outer layer“ option leads to an observable increase of cells. The difference in the underbody region is significant. It is expected that this smooth transition leads to a better numerical behavior.

The flow near the wall is directly influenced by the Reynolds number, i.e. the number of boundary layers should increase proportionally to the Reynolds number [Nikitin et al (2000)]. However, the car velocities in automotive aerodynamics are in
3.2 Mesh Generation

(a) smooth transition (MOL): underbody

(b) steep transition (NBL): underbody

Figure 3.7: Mesh Settings: Comparison of boundary layer meshing types

a small range. Thus, the appropriate number of boundary layers in dependence of the Reynolds number can be neglected.

Furthermore, the experimental data is only available for an absolute air flow velocity value of 50 m/s, which corresponds to a vehicle speed of 180 km/h.

The DDES turbulence model does not restrict the first mesh spacing parallel to the wall, as described in section 2.3.4. However, a finer mesh spacing parallel to the wall can be easily achieved by a refinement of the surface mesh.

The utilization of DDES simplifies the treatment of grid spacing in different directions. Furthermore, the instabilities sustained by the free shear layers even have larger wave-lengths compared to their thickness and thus, generate enough LES content \([\text{Spalart et al. (2005)}]\). Nevertheless, one has to consider the transition from the RANS region into the LES region carefully. For this particular problem, the LES “focus region” (cf. \(\text{Spalart (2001)}\)) specifies the wake behind the vehicle. Due to its impact on the pressure drag, the turbulent motions in this recirculating region must be resolved very accurately. Thus, a very fine grid resolution is required.

Unfortunately, there is no unique way to specify an appropriate \(h^\text{local}_z\) resolving the very different length scales. Thus, a reliable DDES simulation has to be repeated with different element sizes \(h^\text{local}_z\) in the focus region.

The high mesh resolution in the focus region is mainly responsible for the final grid size. Thus, one is very interested in the minimal dimensions which are required to capture the wake accurately. Therefore, it is attempted to determine the reattachment point, which depends on the grid spacing, the chosen time step, and global instabilities due to separation. Thus, the accurate determination of the reattachment point may be difficult.
Therefore, it is convenient to vary both the mesh spacing in the refined focus zone $h_{local}$ and the dimensions of the core zone. A common length for the refinement in the wake is found in [Favre and Efraimsson (2011); Lanfrit (2005)]. Thus, the length of the core zone for the ASMO model $L_{WL,A}$ is varied in terms of the vehicle length $L$ for $0.25L$, $0.75L$, and $1.25L$. However, as the wake is formed due to the width $W$ and height $H$ of the obstacle, the description of the wake dimensions with respect to these extents is more natural. Thus, the required core zone dimensions are compared to the vehicle dimensions in all directions. As the wake length is extremely shape-dependent, the core zone for the Sedan model must be considered individually. However, selected cases are simulated with the same core lengths $L_{WL,S}$ in terms of the Sedan vehicle length. Thus, differences of the required core length can be evaluated systematically. Nevertheless, the reattachment point is evaluated for comparable steady-state and transient cases.

The region behind the wake is referred to as departure region and the mesh size is allowed to coarsen far beyond $h_{local}$ [Spalart (2001)] (see Fig. 3.8a). A coarse mesh provides the required dissipation of the vortices before the flow reaches the outflow boundary. Thus, the mesh spacing should smoothly evolve to a similar spacing like in the far field which is known as the Euler region. Therefore, the chosen mesh algorithm presented in section 3.2.2 is an appropriate choice.

As we are interested in a grid with as few elements as possible, the definition of a maximal global element size $h_{global}$ is not desired.

Although, little contribution of the far field is to be expected, the number of elements between the inlet and the stagnation point at the surface of the car must be at least 100, otherwise the pressure at the stagnation point can be overpredicted [Lanfrit (2005)].

![Figure 3.8: Mesh Settings: Features](image-url)
3.2 Mesh Generation

In difference to other commercial solvers, AcuSolve is highly insensitive to the mesh quality e.g. high aspect ratios and badly distorted elements [cf. Lyons et al. (2007); Godo et al. (2011)]. Thus, the element quality is not considered in this thesis. Even the boundary layer consists of tetrahedral elements (see Fig. 3.8b). This property enhances the possibilities to define common and robust meshing rules.

3.2.2 Meshing Process

Several meshing approaches are utilized in the meshing process which are well suited for this particular problem. As mentioned before, the input of a surface mesh with a good quality is a preliminary requirement, although the meshing capabilities even allow appropriate surface meshing. However, in a typical use case, the shell mesh of the car already has been created before aerodynamic issues are evaluated, thus this topic is not further addressed in this work.

First of all, the boundary layer mesh is extruded using the Advancing Front method [Jin and Tanner (2005)], which extrude layers consisting of tetrahedral elements from the surface faces into the specified core zone. Therefore, the first layer height and the growth rate must be specified. This propagation is either determined by a specified number of element layers or by the constraint of a constant growth rate even in the adjacent element layers. The specified core zone is filled with uniform isotropic elements. An appropriate choice for the core element size is described in the following paragraphs.

Next, the whole fluid domain is filled with elements using the Octree meshing method, which is a spatial subdivision algorithm. An initial tetrahedron encloses the whole flow domain, and is successively refined, up to the boundary of the core zone. Thus, the required refinement in certain regions close to the bounding surfaces is ensured while larger elements in the majority of the flow domain are maintained. This meshing scheme complies with the mesh requirements in the different flow regions.

In a final step, the interface regions between boundary layer, surface mesh and the entire volume region are optimized and smoothed using various techniques which are not further explained at this point.

In this thesis, a pure tetrahedral volume mesh is utilized. Thus, one gains a conformal, but unstructured grid which enables a flexible discretization of arbitrarily complex geometry. Furthermore, it avoids unnecessarily slow element growth from the core zone into the far field.

The Octree mesh method determines the size of the volume elements using the formula:

\[ h_{local} \cdot 2^{n_i} = h_{global}. \]  (3.10)
By default, $h_{\text{global}}$ and $h_{\text{local}}$ determine the circumsphere of the largest volume element and a specific local element size, respectively. However, it is desired to achieve as few elements as possible in the far field. Utilizing the introduced meshing schemes, the mesh development throughout the entire wind tunnel can be defined by a given surface mesh, a specified element growth rate, the core element size and dimensions. Thus, another definition of $h_{\text{global}}$ is applied within this thesis. To ensure a model dependent mesh generation, $h_{\text{global}}$ is chosen in dependence of a specific model property, which is selected in the next section.

According to this constraint, the mesh generation is parametrized. The mesh control parameters are expressed in terms of $n_i$. Particular attention is paid to the core element size, expressed via the parameter $n_z$, as the core zone primarily determines the amount of elements. If the same $n_z$ leads to appropriate solutions for different problems, the mesh generation can be easily automated. Otherwise, recommended ranges for the investigated mesh parameters are stated. The first layer height is mostly independent of specific model properties and the boundary layer is meshed using the Advanced Front method. However, for the sake of simplicity it is also expressed in terms of the exponent $n_{flh}$.

Summing up, this procedure results in a mesh as already shown in Fig. 3.2b. It is worth noting, that the vertical extent of this zone cannot be chosen optimal due to the natural wake development behind the vehicle. Furthermore, the mesh contains unnecessary refined zones in the upper front part of the vehicle.

Therefore, an appropriate choice would be a region of influence (ROI) which determines the refinement zone dependent on the body shape (see Fig. 3.9). Unfortunately, at this time the required normal vector calculation demands extensive computing resources and thus, time. The ROI determination will be improved in future versions and then used in the virtual wind tunnel, to determine optimal wake dimensions.

However, even for the region of influence an internal element size and length must be chosen. Thus, this work concentrates on these issues using a rectangular refinement box.

3.2.3 Definition of Mesh Control Parameters

In the following and in accordance to the summaries in section 3.2.1 and 3.2.2, a set of mesh control parameters is defined.

First of all, an absolute value for the mesh parameter $h_{\text{global}}$ must be chosen with regard to a model property, which influences the maximum element size in the far field.

For the ASMO simulation, it is observed that the total number of elements is constant.
for all global element sizes greater than the longest edge length of a volume element in the wind tunnel which equals 0.5 m. This is the lower bound for $h_{\text{global}}$, as it is not desired to specify a maximum element size. The most appropriate choice for the definition of $h_{\text{global}}$ combines the knowledge about both wind tunnel and model dimensions. The cross section combines the wind tunnel height and width to the model properties, as an bound for the blockage ratio should not be exceeded [cf. Sec. 3.1.1]. Thus, the global size is defined by

$$h_{\text{global}} \approx \sqrt{A_{\text{WT}}},$$  \hspace{1cm} (3.11)

throughout this thesis. We restrict the global element size $h_{\text{global}}$ to be a power of two, to enhance the comparability of the results. This yields $h_{\text{ASMO}} \approx 2 \text{m}$ for the ASMO and $h_{\text{Sedan}} \approx 8 \text{m}$ for the sedan model.

As the turbulent motions in the wake of the vehicle are created due to the cross section dimensions of the vehicle, it is feasible to relate the core element size to this model property. Furthermore, the scale of a turbulent eddy is related to its frequency, which is further described in section 3.3.1. Thus, if this frequency is known, the order of the required grid spacing can be estimated. In the preliminary studies, the core zone around the car and its wake has been refined with $h_{\text{local}} = 0.006$ which is twice the smallest edge length of the ASMO surface mesh on the rear side. This choice resulted in an appropriate prediction of the drag. Hence $n_z$ is varied around this value, described in Table 3.2. Furthermore, the dimensions of the core zone are evaluated,
and it is considered which dimensions are required to capture the effects in the wake, accurately.

The first layer height depends primarily on the freestream velocity and rather not on specific model properties which becomes clear from the \( y^+ \) estimation of a flat plate, i.e.:

\[
h^{local}_{flh} = 5.81 y^+ \left( \frac{\nu}{U_\infty} \right) Re_x^{0.1} .
\]

(3.12)

It can be clearly seen that the first layer height weakly depends on the Reynolds number, which contains the specific length scale.

In a first step, this equation is used to roughly estimate the first layer height of the boundary layers.

Therefore, one must calculate the Reynolds number. The choice of the length scale is not uniform in automotive aerodynamics. Some studies defined the reference length as the length of the vehicle \( L \), in other studies the reference length is the square root of the frontal cross section area \( A \).

However, the Reynolds number for both the ASMO and Sedan model are computed for both reference lengths. The comparison yields:

\[
\frac{Re_{L,ASMO}}{Re_{L,Sedan}} \sim \frac{Re_{A,ASMO}}{Re_{A,Sedan}} \sim 0.164 .
\]

(3.13)

Thus, the common and intuitive definition is chosen, and the reference length is defined by the vehicle length \( L \) throughout this thesis. For \( y^+ \geq 5 \) the estimator yields a value of \( h^{local}_{flh} \simeq 4.4 \cdot 10^{-5} \). Therefore, the first layer height is varied in a range of \( 1.5 \cdot 10^{-5} < h^{local}_{flh} < 1.22 \cdot 10^{-4} \) which is equivalent to \( 14 < n_{flh} < 17 \). In the same way two first layer heights of the bottom are estimated, yielding \( y^+ \) values around 60 and 30, respectively.

It is worth noting, that the parameters \( n_{flh} \) and \( n_{bot} \), respectively, are just introduced for simplicity. Both vary with the model dimensions, while \( y^+ \) values changes due to physical properties. However, this relationship just intends to express, that if the parameter is increased by one, the first layer height is halved.

3.2.3.1 ASMO Mesh Settings

The introduced mesh control parameters result in following values:
3.3 Solver Setup

Elementary solver settings are summarized in Appendix [A.6]

3.3.1 Time Step

Another important issue is the time step specification. As an semi-implicit time stepping scheme is used, the time step is chosen for accuracy, because no CFL based stability limit must be considered. The highest frequencies occur in the wake and thus in the core zone. In relation to numerical requirements, the time step is related to the core element size by the CFL condition, i.e.:

$$\frac{u \Delta t}{h_{local}} \leq C.$$  \hspace{1cm} (3.14)

In an optimal case, the CFL number equals 1. The time step size is defined by Spalart [Spalart (2001)] according to a CFL number of 1:

$$\Delta t = \frac{h_{local}}{u_{max}}.$$  \hspace{1cm} (3.15)

We apply a less strict condition by specifying $u_{max} \approx u_{in}$.

However, later analyses indicated that even higher CFL numbers can produce accurate results. Thus, it is intended to formulate the time step size based on physical
3.3 Solver Setup

properties. By default, the time step is based on the calculated vortex shedding period which is defined by:

\[ T_s = \frac{1}{f} = \frac{H}{St \cdot u_\infty}, \quad St = \frac{f \cdot H}{u_\infty} \quad (3.16) \]

where \( f \) and \( St \) denote the shedding frequency and Strouhal number respectively. It is to be expected that the vortex shedding is due to the horizontal extent \( H \) of the vehicle. However, the turbulent wake motions and the vortex shedding of a three-dimensional vehicle can be highly irregular Bearman (1980). Thus, no dominating shedding frequency can be observed. In this case, it is recommended to choose a sufficiently small time step size.

The Strouhal number can be computed by the shedding frequency, if a computed drag signal is already available. Otherwise, the Strouhal number must be approximated reasonably to obtain an estimation for the shedding frequency.

Furthermore, it is important to consider how much time steps within a shedding period are required to provide a good temporal resolution. Thus, two influencing factors vary the range of an appropriate time step choice. Therefore, the appropriate time-stepping size is evaluated intensively in the next chapter.

---

4 The Strouhal number \( St \) describe oscillating flow mechanisms as e.g. periodical vortex shedding.
4 Evaluation and Discussion

4.1 ASMO Model: Identification of Mesh Requirements

The most important output quantity of an automotive aerodynamics study is the drag coefficient and thus, the pressure distribution on the vehicle surface. The aim of this section is to evaluate the accuracy of performed simulations regarding different mesh control parameter specifications. For the majority of vehicle shapes, the lift coefficient does not play an important role, except e.g. racing vehicles. Furthermore, no experimental data for the lift coefficient is available, thus, the lift force is not considered further within this thesis.

4.1.1 Preliminary Results

4.1.1.1 Grid Convergence

In the preliminary work of this thesis both steady-state and transient simulations with about 22 million and 70 million elements, respectively were performed. For a time step size of $t = 3 \times 10^{-4}$, both analyses yielded an average drag of $c_d = 0.164$. Hence, a grid-independent solution was obtained. The deviation to the experiments is defined by:

$$
\text{error } e = \frac{|c_{d}^{\text{Sim}} - c_{d}^{\text{Exp}}|}{c_{d}^{\text{Exp}}} \quad (4.1)
$$

These two simulations have a deviation of 1.2 %. The simulation with 22 million elements is listed in Fig. 4.2.

4.1.1.2 Time Step Dependency

A time step study revealed that an appropriate time step choice is of major importance to capture the unsteady effects accurately, see Table 4.1. Utilizing a time step of $3 \times 10^{-4}$, AcuSolve produced accurate results. This time step coincides to a CFL number of 1 for the coarser mesh. However, the finer mesh varies in a similar manner. This motivates the definition of the time step size due to physical properties, see Eq. 3.16. The height of the obstacle is defined by the height of the ASMO model $H_A$. A shedding frequency of $f \sim 100Hz$ arise from the analysis of the drag signal. Thus, the vortex shedding period is $T \sim 0.01s$, and the Strouhal number equals $St = 0.5$. As the results at a time step size of $\Delta t_{\text{ASMO}} \sim 3 \times 10^{-4}s$ are accurate, it is concluded that 30 time steps per vortex shedding period are appropriate to resolve the unsteady effects accurately.
Table 4.1: ASMO: Influence of time step size.

<table>
<thead>
<tr>
<th># Elements [Mio]</th>
<th>$\Delta t [sec]$</th>
<th>$c_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>70 (12 x 10^6 nodes)</td>
<td>$3 \times 10^{-4}$</td>
<td>0.164</td>
</tr>
<tr>
<td></td>
<td>$6 \times 10^{-4}$</td>
<td>0.166</td>
</tr>
<tr>
<td></td>
<td>$12 \times 10^{-4}$</td>
<td>0.180</td>
</tr>
<tr>
<td>22 (4 x 10^6 nodes)</td>
<td>$3 \times 10^{-4}$</td>
<td>0.164</td>
</tr>
<tr>
<td></td>
<td>$6 \times 10^{-4}$</td>
<td>0.177</td>
</tr>
<tr>
<td></td>
<td>$12 \times 10^{-4}$</td>
<td>0.183</td>
</tr>
</tbody>
</table>

At this point, it is not clear if more than 30 time steps per shedding period would enhance the solution up to $c_d \sim 0.162$. This issue is not addressed for the ASMO analyses, and all simulations are performed using a time step size of $3 \times 10^{-4}$ seconds.

Furthermore, an appropriate time interval for the ASMO analyses has been determined. The averaged drag did not change beyond a computed time interval of 0.54 seconds. This time interval equals three freestream particle paths through the whole wind tunnel or 100 particle paths related to the car height, respectively. These path intervals are determined by $\frac{L_{WT}}{u_\infty} = 0.18 \text{ s}$ and $\frac{H_A}{u_\infty} = 0.0054 \text{ s}$, respectively. Therefore, all ASMO analyses in this chapter are performed until $T = 0.54 \text{ s}$.

4.1.1.3 Steady-state Evaluation

In addition to the transient simulations, a steady-state simulation was performed on the finest mesh and with the Spalart-Allmaras turbulence model.

The calculated drag of $c_d = 0.173$ deviates significantly from the transient result of $c_d = 0.164$. This inaccuracy coincides with former findings, that transient analyses are required to capture the drag force on passenger cars accurately (cf. Aronson et al. (2000), Perzon and Davidson (2000)).

As a consequence, pure steady-state solution strategies are not considered further. However, in Sec. 4.1.5 steady state simulations are evaluated according to their potential of overcoming the startup phase of the flow. As AcuSolve obtains rapid steady-state solutions, this is an efficient way to generate appropriate initial conditions for transient analyses.
4.1.2 Drag Coefficient

In the analysis of the ASMO model, drag stabilizes within a certain range around 0.1 seconds at the latest, see Fig. 4.1. This corresponds to the time required of the first air particles to pass the vehicle: \( \frac{L}{W_A + L_A} \sim 0.1 \) s However, the drag is averaged from 0.18 to 0.54 seconds, as it is safer to overestimate the initialization time of the flow field.

Even though, the drag is stabilized within a certain range, steep oscillations of different frequencies throughout the simulation interval represent the highly unsteady behavior of the forces on the vehicle. The highest frequency with comparable small amplitudes is probably due to numerical reasons, e.g. insufficient resolved gradients at the separation edges of the vehicle. The medium frequency is the shedding frequency of the vortices in the wake of the vehicle. The source of the low-frequency oscillations with higher amplitudes can not be determined exactly, but they result most likely from model properties of the wind tunnel. Nevertheless, these oscillations decrease according to the decrease of the element size in the core zone. Thus, a finer resolution around the vehicle model affects the stability of the drag signal. Furthermore, it is observed that both the standard deviation of the drag and the extremal values of the fluctuations are nearly mesh-independent. The statistical properties of the drag signal are further evaluated in later chapters.

![Figure 4.1: ASMO: Time history of the drag coefficient.](image)

4.1.2.1 Grid Convergence

Fig. 4.2 contains all simulations performed with the ASMO model and compare the precision of the computation to the problem size. Obviously, the mesh size has huge impact on the accuracy of the results. However, several simulations with different grid sizes result in the same drag value. The grid independent solution is obtained for at least 22 million elements, and some even finer meshes lead to worse results. Thus, a finer mesh does not necessarily improve the accuracy. This reveals the need for a separate investigation of the mesh parameters.
4.1 ASMO Model: Identification of Mesh Requirements

4.1.2.2 First Layer Height

The first layer height is one of the major factors on the accuracy of the analyses, because the decrease in $y^+$ results in an enhanced prediction of the friction-induced drag. The four investigated mesh sizes are expressed in terms of the meshing parameter $n_{flh} = \{14, 15, 16, 17\}$. The results are summarized in Table 4.2. The decrease of $y^+$ leads to a reasonable increase of accuracy until a value of about $y^+ \sim 4$ is achieved. The change of $y^+ \sim 8$ to $y^+ \sim 4$ leads to the maximum increase in accuracy. Further reduction of the first layer height does not result in a significant improvement of the drag prediction.

One may conclude, that an adequate $y^+$ value rather follows the assumptions of [Spalart (2001)].
The modification of the first layer height on the bottom of the wind tunnel results in a decrease from $y^+ = 60$ to $y^+ = 30$. Decreasing the first layer height leads to a feasible improvement of the drag prediction. Furthermore, the number of elements just slightly increases.

Thus, there is evidence that a fine boundary layer mesh on the wind tunnel ground has to be taken into account.

Although this variation does not allow further statements, the moving ground simulations are performed with the first layer height according to $y^+ \sim 30$.

### 4.1.2.3 Drag Composition

The major part of the drag originates from the ASMO body. This coincides with observations in [Lopes and Carvalheira (2003)](#), that for high Reynolds numbers the body represents the major portion of the drag.

The drag composition for different first layer heights is summarized in Table 4.4. It is recognized, that the drag portion of the front wheels is slightly higher. This force distribution does not agree with former results in [Mercker et al. (1991)](#). As we will see later, the force distribution turns to a correct physical behavior with the utilization of a moving ground. The vortices which shed off of the wheels are exemplary shown in Appendix A.2, Fig. A.3.

The development of the different $y^+$ values for the car body as well as for the front and rear wheels can be seen in Fig. 4.3. The $y^+$ values at the rear wheels are lower.

#### Table 4.3: ASMO: Integrated $y^+$ value over the wind tunnel ground.

<table>
<thead>
<tr>
<th>$n_{bottom}$</th>
<th>$\sim y^+$</th>
<th># Elements</th>
<th>error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>60</td>
<td>$1.46 \times 10^6$</td>
<td>6.7</td>
</tr>
<tr>
<td>13</td>
<td>30</td>
<td>$1.26 \times 10^6$</td>
<td>4.9</td>
</tr>
</tbody>
</table>

#### Table 4.4: ASMO: Partial drag coefficient.

<table>
<thead>
<tr>
<th>$n_{flh}$</th>
<th>total $C_d$</th>
<th>$C_{d, \text{Body}}$</th>
<th>$C_{d, \text{WheelRF}}$</th>
<th>$C_{d, \text{WheelRR}}$</th>
<th>$C_{d, \text{WheelLF}}$</th>
<th>$C_{d, \text{WheelLR}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>17</td>
<td>0.165</td>
<td>0.138</td>
<td>$7.5 \times 10^{-3}$</td>
<td>$6.2 \times 10^{-3}$</td>
<td>$7.3 \times 10^{-3}$</td>
<td>$6.1 \times 10^{-3}$</td>
</tr>
<tr>
<td>16</td>
<td>0.166</td>
<td>0.138</td>
<td>$7.7 \times 10^{-3}$</td>
<td>$6.3 \times 10^{-3}$</td>
<td>$7.7 \times 10^{-3}$</td>
<td>$6.2 \times 10^{-3}$</td>
</tr>
<tr>
<td>15</td>
<td>0.171</td>
<td>0.142</td>
<td>$8.6 \times 10^{-3}$</td>
<td>$6.4 \times 10^{-3}$</td>
<td>$8.6 \times 10^{-3}$</td>
<td>$6.4 \times 10^{-3}$</td>
</tr>
<tr>
<td>14</td>
<td>0.175</td>
<td>0.143</td>
<td>$9.3 \times 10^{-3}$</td>
<td>$6.5 \times 10^{-3}$</td>
<td>$9.9 \times 10^{-3}$</td>
<td>$6.3 \times 10^{-3}$</td>
</tr>
</tbody>
</table>
than at the front wheels, which is reasonable due to the decreased velocity in the wind shadow of the front wheels. The $y^+$ value of the rear wheels is more fluctuating which is in accordance to the results in [Damiani et al. (2004)]. The magnitude of the fluctuations decrease notably with a finer first layer height, as the gradients at the walls are better resolved.

However, the $y^+$ value of the vehicle body is more affected by decreasing the first layer height. Therefore, decreasing the first layer height brings $y^+$ of the body close to the $y^+$ values of the wheels. The best compromise between accuracy and grid size can be evaluated, if $y^+_{Body}$ is slightly above the range of $y^+$ values of the front wheels.

4.1.2.4 Wake Refinement

Besides the first layer height, the mesh refinement in the wake of the vehicle is of major importance.

The results for different core element sizes are listed in Table 4.5. A finer mesh resolution in the core zone results in enhanced accuracy of almost 6%, if $n_z$ increases from $n_z = 7$ to 8. The grid size increases about 3.3 million elements. However, an
4.1 ASMO Model: Identification of Mesh Requirements

Table 4.5: ASMO: Comparison of grid densities in the wake.

<table>
<thead>
<tr>
<th>$n_z$</th>
<th>element size [m]</th>
<th># Elements</th>
<th>$c_d$</th>
<th>error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>0.0156</td>
<td>$12.3 \times 10^6$</td>
<td>0.175</td>
<td>8.0</td>
</tr>
<tr>
<td>8</td>
<td>0.0078</td>
<td>$15.6 \times 10^6$</td>
<td>0.166</td>
<td>2.1</td>
</tr>
<tr>
<td>9</td>
<td>0.0039</td>
<td>$36.0 \times 10^6$</td>
<td>0.164</td>
<td>1.2</td>
</tr>
</tbody>
</table>

Additional increase from $n_z = 8$ to $n_z = 9$ does not lead to a significant improvement of the drag prediction. Furthermore, the mesh contains more than twice the number of elements. Therefore, considering the appropriate choice of $h^{global} = 2m$, the optimal element size in the core zone is obtained by defining the mesh parameter $n_z$ to 8.

The results show the impact of the wake resolution on the drag prediction even though the ASMO model has no pressure induced separation.

4.1.2.5 Core Zone Dimensions

An appropriate selection of core zone dimensions is essential to capture the turbulent region behind the vehicle, properly. The dimensions of the core zone are equal except the core length $L_{WL}$ behind the vehicle.

The results for the variation of the refined wake length are listed in Table 4.6. It can be clearly seen, that the accuracy is not affected by the core length. However, it is quite clear that a non-sufficient core length would be similar to a non-sufficient core element size, e.g. $n_z = 7$ in Table 4.5.

Therefore, all investigated core zones cover the wake region. As the mesh with the shorter core length contains less elements, it accentuates the need for the determination of the minimal dimensions. This may be accomplished by the determination of the reattachment point or free stagnation point, respectively.

Considering the skin friction on the ground, reattachment occurs at the point where the skin friction changes its sign from negative to positive. As already mentioned, it can be difficult to localize the reattachment point accurately. Fig. 4.4 shows the
skin friction coefficient of a steady-state and the transient analysis. All skin friction coefficients have been extracted from analyses using the same mesh\footnote{Mesh control parameters: $n_z = 9, n_{ph} = 16, n_{bot} = 13, MOL, L_{WL,A} = 0.75L$}. The skin friction coefficient of the transient analysis is averaged over all time steps and does not change its sign.

The steady-state simulation obtained reattachment at $x = 1.03045 \text{m}$, which can be written as a position relative to the vehicle position

$$\frac{X_r}{H_A} \approx L_A = 3H_A \approx 3W_A$$

Thus, the wake dimension depends on the vehicle height or width, respectively\footnote{$X_r$ is a relative measure, i.e.: the $X_r$ describes the distance from the rear face of the vehicle to the reattachment point. As the vehicle width is nearly equivalent to the vehicle height, this relationship can not be evaluated further.}. This correlation is considered for the sedan model in later sections.

As the determined reattachment point is just outside the shortest core length, the mesh is still sufficient fine enough. Thus, all utilized core lengths would be appropriate, which coincides with the results. However, the correctness of this point can not be further verified.

The other two curves show the time steps of the transient analysis with the minimal and maximal point of sign change along the symmetry plane. The reattachment point fluctuates in the range $0.15L_A < X_r < 0.41L_A$ or $0.45H_A < X_r < 1.25H_A$, respectively. This indicates, that for the majority of simulations the core length has been chosen too generous. Although it is difficult to locate the reattachment point accurately, it is possible to determine a certain range. The steady-state is almost the average within this range.
The determination of the required refinement dimensions is crucial to obtain an optimal ratio of accuracy to problem size. However, it is not ensured that the reattachment point can be determined using a steady-state analysis.

### 4.1.2.6 Boundary Layer Meshing Scheme

The influence of the transition from the cells in the boundary layer into the adjacent element layers is evaluated. The preliminary expectation is that a constant number of element layers and thus, a sudden transition from fine to coarse elements is not usable, because it worsens the results more than reducing the problem size. The utilization of a fixed number of boundary layers yields a sudden local coarsening in the element size of the adjacent element layers. This results in about 15% less elements in the grid. However, the decrease in the accuracy of the drag coefficient prediction is significant, as shown in Table 4.8.

<table>
<thead>
<tr>
<th>$n_z$</th>
<th>constant # Elements error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Growth rate $12.34 \times 10^6$ 8.0</td>
</tr>
<tr>
<td></td>
<td>Number of Layers $10.29 \times 10^6$ 9.9</td>
</tr>
<tr>
<td>8</td>
<td>Growth rate $15.57 \times 10^6$ 2.4</td>
</tr>
<tr>
<td></td>
<td>Number of Layers $13.69 \times 10^6$ 4.9</td>
</tr>
</tbody>
</table>

Therefore, a fixed number of boundary layers in the coarse grid presetting and a constant growth rate in the medium grid presetting, respectively, will be utilized in the default settings of the virtual wind tunnel mesh generator. As shown in later sections, the choice of the boundary layer meshing type also depends on the region of the vehicle, where the boundary layer mesh should be applied.

### 4.1.2.7 Far Field

The influence of a smooth transition from the core zone into the far field is not as small as the actual influence of the far field. A second, coarser refinement box with larger dimensions is utilized. This results in a comprehensible improvement of the accuracy, the number of elements does not increase significantly, though. Therefore,

<table>
<thead>
<tr>
<th>$n_z$</th>
<th># Elements error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 / -</td>
<td>$14.46 \times 10^6$ 6.7</td>
</tr>
<tr>
<td>8 / 7</td>
<td>$15.42 \times 10^6$ 5.5</td>
</tr>
</tbody>
</table>

this phenomenon should be considered. Nevertheless, it does not affect the solution.
in the same manner as the first layer height or core element size. As it is rather simple to create a sufficient fine mesh with a generous extended core zone, this thesis concentrates on the minimum requirements to predict a reasonable $c_d$. Because the accuracy of the current simulations is already sufficient and a systematic investigation of several refinement zones demands varying the different zone dimensions several times, this is not further evaluated in this thesis.

4.1.3 Pressure Profile

The accurate prediction of the surface pressure distribution accompanies with the reasonable capture of the unsteady wake effects (cf. Sec. 2.1 and Sec. 2.3.2). Due to the impact of the wake effects the surface pressure distribution is investigated in detail. The pressure distribution is evaluated at paths on the symmetry plane, shown in Appendix A.1, Fig. A.2.

The pressure profile of the ASMO was measured at Daimler Benz and Volvo (e.g. refer to Perzon and Davidson (2000)), the experimental setup of the Volvo measurements is explained in Aronson et al. (2000). The Volvo model scale wind tunnel is a slotted wall wind tunnel with a blockage ratio of 6%. This wind tunnel type is impossible to simulate with a reasonable amount of cells Aronson et al. (2000). Nevertheless, it is comparable to a solid wall wind tunnel with a lower blockage ratio which is used within this work.

4.1.3.1 Time Dependence

First of all, we consider the time dependence of the pressure distribution.

The pressure distribution on the rear face of the vehicle is mostly affected by severe unsteady effects. For fixed ground simulations, no periodicity is observed in the pressure oscillations. The base pressure profile at distinct time steps is shown in Appendix A.2, Fig. A.4. The pressure at the underbody fluctuates slightly and in a small range. On the front and roof no pressure oscillations are present.

Thus, even steady-state simulations are able to compute a valid pressure distribution in this surface regions. Furthermore, a fine discretization in this regions is not required to compute the pressure coefficient accurately, clearly visible in Fig. 4.5. However, usually it is required to compute the pressure coefficient over the entire vehicle body in order to determine the pressure-induced drag force.

In the following, all pressure profiles are time-averaged over the transient time steps, beginning at 0.18 seconds.
4.1.3.2 Roof

The prediction of the pressure distribution on the roof, Fig. 4.5, is overall accurate as well as mesh-independent. In agreement with observations in [Perzon and Davidson (2000)], very small differences to the experiments are found close to the trailing edge. The boundary layer growth in this region affects the base pressure at the upper rear end of the vehicle. Furthermore, the steady-state calculation deviates slightly from the transient analyses, which are conform on the entire roof.

![Figure 4.5: ASMO: Roof pressure in symmetry plane.](image)

4.1.3.3 Front

As with the roof pressure, the pressure distribution over the front surface of the car is overall predicted satisfactorily, see Fig. 4.6.

However, the pressure near the stagnation point differs from the experimental data. This inaccurate behaviour in this region has also been observed in [Perzon and Davidson (2000)].

The main difference is to be found due to the boundary layer meshing type, see Fig. 4.6a. Contra-intuitively, the stagnation pressure is overpredicted for all analyses utilizing a constant element growth rate from the boundary layer through the adjacent cell layers. The grids with abrupt coarsening off of the boundary layer approximate the stagnation pressure more accurate. Nevertheless, the clear deviations from the experimental data are independent from the core resolution and first layer height.

Furthermore, the computed data deviates slightly from the experiments in the range of $1m < y < 1.2m$, which is only of minor interest.
4.1 ASMO Model: Identification of Mesh Requirements

4.1.3.4 Base

The importance of the base pressure and its accurate prediction has already been explained. In Fig. 4.7, the base pressure distributions are shown for different mesh configurations. In general, the enhanced base pressure prediction accompanies with the utilization of a fine core element size in a transient analysis. The influence of the core element size is shown in Fig. 4.7a. The mesh with a coarse core element size ($n_z = 7$) can not resolve all significant scales and thus, is not able to reproduce the pressure distribution accurately. Steep gradients occur in the saddle point region between the two largest, counterrotating vertex structures. The pressure is underpredicted in the upper part and overpredicted in the lower part of the rear surface, respectively.

An analysis utilizing a core element size of $n_z = 8$ leads to a better approximation of the pressure distribution, even if there are still comprehensible deviations. However, as the calculated drag is a good approximation of the experimental result, this deviations are within an acceptable range.

Furthermore, both experimental datasets deviates significantly from each other, which indicates that this region is highly sensitive to e.g. wind tunnel properties, which can not be evaluated in this thesis.

The simulation with the finest core resolution ($n_z = 9$) avoids the pressure loss at the upper rear end. However, it can not reproduce the slight pressure rise in the lower part, which is described by the experimental data and approximated by the medium grid. The steady state simulation overpredicts the pressure along the whole base, in accordance to [Perzon and Davidson (2000)].

All simulations have shown the same oscillations at the trailing edges of the body, having observed in [Nakashima et al. (2008)]. They slightly decrease with a finer mesh resolution. However, these oscillations result from the insufficient resolution of very huge gradients at the separation points, which can not be avoided.

The base pressure distribution is also evaluated for the variation of other mesh parameters. The comparison of the boundary layer meshing scheme shows some dif-
ferences. Although it is not clearly visible in Fig. 4.7b, it is assumed that the analysis with a constant element growth rate performs slightly better.

No differences are observed between the analyses utilizing different core lengths, see Fig. 4.7c. This is in accordance to previous findings in this work. The mesh with the finer boundary layer resolution on the wind tunnel ground leads to a feasible pressure increase near the bottom. However, all grids with the finest core element size contains even this fine ground discretization, and exhibit no increase in the region near to the bottom.

The influence of the first layer height is depicted in Fig. 4.7d. It can be clearly seen, that even the first layer height has to be chosen appropriately to achieve a reasonable pressure distribution. This fact does not agree with the statement in [Nakashima et al. (2008)], that the wall normal resolution has little influence on the pressure distribution. In Fig 4.8 it can be clearly seen that the boundary layer meshing scheme utilizing a constant growth rate can be problematic. This undesired element growth into the wake region occurs in cases, where the core element size is smaller than the grid spacing of the surface mesh. Therefore, it is necessary either to choose a constant number of element layers or to refine the surface mesh. As in the utilized meshing scheme the core element size and the shell element size is known a-priori, the first approach requires less effort. However, these problematic issues illustrate that the requirement
4.1 ASMO Model: Identification of Mesh Requirements

(a) Constant number of element layers
(b) Constant growth rate

Figure 4.8: ASMO: Problematic discretization in the wake for fine resolution.

of an appropriate input surface mesh of the vehicle is crucial.

4.1.3.5 Underbody

The pressure distribution in the underbody is important due to its contribution to the overall drag (cf. [Aronson et al. (2000)]). Thus, the underbody is further investigated for two sedan models with different underbody variants.

The behavior in the underbody region is qualitatively reasonable, but not overall accurate. First of all, the pressure at the front of the car drops too much. Then, none of the meshes is able to capture the steep pressure rise after the front radius ($0m < x < 0.2m$), which has been measured in the Volvo experiments [Perzon and Davidson (2000)]. However, the computed data are in satisfying agreement with the Daimler experiments. No general conclusion regarding the accuracy can be produced, because the experiments deviate clearly from each other. The utilization of a fixed number of boundary layers and thus a sudden jump in the element growth results in a less accurate resolution in the range of ($0.1m < x < 0.3m$), which confirms the benefit of a smooth element transition in this region.

In the following evolution all transient models overpredict the pressure in the range of ($0.5m < x < 0.6m$). One may conclude that the runtime of the transient runs is not sufficient. At next, the transient simulations are not able to reproduce the pressure drop at $0.6m < x < 0.7m$. However, even the experimental data are not close to each other.

At the changeover to the rear face of the vehicle, the pressure distribution becomes more reliable, except the mesh utilizing a coarse core zone and a specified number of boundary layers.
4.1 ASMO Model: Identification of Mesh Requirements

4.1.3.6 Interim Conclusion

The analysis of the pressure distribution has shown, that different surface regions require specific meshing strategies. At the front surface, a fix number of boundary layers should be specified, because the utilization of a constant growth rate results in an overpredicted stagnation pressure. Quite contrary is the situation in the underbody region, in which a smooth transition of element size enhances the accuracy feasible. An accurate prediction of the pressure distribution on the roof is easily achieved. Thus, the large roof region contains great potential for saving a huge amount of elements. As suspected, the most challenging part of the vehicle is the rear side. A very small core element size as well as first layer height is required. However, it is assumed that the present results can even be improved by decreasing the shell element size. Therefore, also the inaccuracy in the underbody region may be caused due to insufficient refinement of the surface mesh.

4.1.4 Flow Field

The objective of this section is to provide further insights into the flow physics in particular, which affect the calculation of the investigated quantities.
4.1 ASMO Model: Identification of Mesh Requirements

4.1.4.1 Wake Oscillations

The following Fig. 4.10 shows the streamwise velocity at predefined probe locations in the flow field for different core element sizes. It is worth noting, that the medium and coarse grid in Fig. 4.10 have a refinement length of $1.25L_A$, while the wake in the finest grid is only refined until $0.75L_A$ rearward the vehicle. Unlike the fine grid, the coarse grid is not able to reproduce severe velocity fluctuations near the rear face of the vehicle oscillating at the shedding frequency. The fluctuations are damped due to the insufficient grid resolution. The oscillations captured with the medium grid are within a similar frequency range as in the fine grid.

As the flow moves downstream, the vortices dissipate and the flow stabilizes (cf. Fig. 4.10b). However, the coarser meshes show more extensive oscillations.

This indicates, that the mesh spacing in the coarse grid is not fine enough. But even the medium grid shows few extensive peaks, and the resulted drag is fairly accurate. This confirms the assumption, that only the wake flow near the vehicle boundary must be properly refined.

Thus, the coarse and medium grid are refined until $x \sim 1.8m$, while the refinement zone of the fine grid reaches just until $x \sim 1.4m$. 

---

Figure 4.10: ASMO: Time history of x-velocity for different core element sizes.
In Fig. 4.10d the evolution of the streamwise velocity near the outflow is visualized. The startup time of the flow can be observed. The first peak occurs around 0.1 seconds, which is the time from which the drag stabilizes within a certain range.

4.1.4.2 RANS - LES Transition

In Fig. 4.11a-4.11b, the transition from the Reynolds averaged S-A turbulence model in the boundary layer to the LES-region of resolved turbulence is shown. As supposed, the DDES method uses the RANS modeling throughout the boundary layer of the vehicle, the wheels and the ground.

However, Fig. 4.11a might indicate the reason of the poor pressure distribution in the frontal part of the underbody region (see Fig. 4.9). Here, the solver operates in RANS method within the entire distance from the ground to the vehicle bottom. In the wheel plane (Fig. 4.11b), a thin LES-region is found, but the flow upstream the front wheels is primarily computed using the RANS method. Although DDES is less affected by the grid spacing, this behaviour is found for all core element sizes. A separate consideration of this behavior is outstanding.

![Figure 4.11: ASMO: Transition from RANS to LES turbulence model.](image)

(a) Symmetry plane: z=0.0 m  
(b) Wheel plane: z= 0.11 m

4.1.5 Steady-state Evaluation

The steady-state solution does not predict the pressure distribution on the body as accurate as the transient solution. Furthermore, the steady-state simulation in Sec. 4.1.1 resulted in large errors even with very fine grids. Though, as shown in the previous section its overall physical behavior is qualitatively reasonable. Especially in regions, where no time dependency is observed. Because steady-state simulations are more than one order faster in computing time, the properties of a steady-state simulation are considered in detail.

The steady state problem is solved with mesh settings, which result in an accurate so-
solution in the transient case. In Table 4.9 it can be clearly seen that the steady-state run predicts the drag more accurate with much less runtime. This result is contrary to the experiences described in Sec. 4.1.1. Thus, it is possible to gain accurate results with steady-state simulations in case of streamlined shapes as the ASMO model. Nevertheless, no grid convergence can be stated, and thus a predictable accuracy of the required quantities is not feasible. However, it is well known that steady-state solutions are an appropriate choice to overcome the majority of the startup phase of the flow which required almost 6 hours of computation time in this case.

We consider the convergence of both steady-state and transient solutions, with respect to the residual and solution increment ratio, defined in Appendix A.6. The residual ratio is formed as the ratio of the residual norm over the norm of the forces in the systems and describes how well the computed solution approximate the discretized equations.

The solution increment ratio describes how the approximation changes during the iterations. The ratio is defined as the solution increment norm over the solution norm.

Steady state can be declared, if the pressure and velocity residual ratios fall below $1 \times 10^{-3}$ and velocity solution increment ratio goes below $1 \times 10^{-2}$, which is obviously not the case. This supports the well-known fact, that this problem is highly

\[
\begin{array}{|c|c|c|}
\hline
\text{run} & c_d \text{ error} [\%] & \text{elapsed time} [\text{h}] \\
\hline
\text{steady} & 0.6 & 2.2 \\
\hline
\text{transient} & 2.4 & 29.2 \\
\hline
\end{array}
\]

Figure 4.12: ASMO: Time history of residual ratio.

4 $n_z = 8, n_{flh} = 16, n_{bot} = 12$, constant element growth rate, reference length = 1.25$L$. This mesh consists of 15.57 million elements (2.66 million nodes).
unsteady. Thus, a steady solution can not guarantee an accurate solution. This is also demonstrated in Fig. 4.13. In the steady-state solution, the solution increment ratio

![Graph showing solution ratio over time](image)

Figure 4.13: ASMO: Time history of solution increment ratio.

of the velocity and pressure exceed 0.1, which marks a feasible change in the solution from iteration to iteration. Therefore, the accuracy of a steady-state solution is rather random, as it depends on manifold factors as e.g. the time increment and the number of time steps, which are taken into account for the averaged drag coefficient.

### 4.1.6 Moving Ground Integration

The moving ground simulation is performed using the boundary conditions defined in Sec. 3.1.4.2. It can be clearly seen, that the drag requires a smaller time period to stabilize within a certain time interval. Fig. 4.14 shows that a slip-boundary condition in the entrance section reduce the irregular fluctuations. As in real road conditions no boundary layer evolves on the road, this configuration models the real flow physics more accurate. With both settings, the drag decreases slightly, see Table 4.10. However, this result can not be evaluated further, as the change in drag due to a moving ground plane depends primarily on the ground clearance of the model and its shape [Krainovic and Davidson (2005)].

In Sec. 4.1.2.3 it has been found that the force distribution on the wheels in case of a fixed ground is not reasonable. This behavior is undergoing a turn-around with the utilization of a moving ground. Hence, the forces exerted on the rear wheels are

<table>
<thead>
<tr>
<th>Run</th>
<th>Setup</th>
<th>$c_d$</th>
<th>Decrease compared to fixed ground</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td>$n_z = 8, n_{th} = 15, n_{bot} = 13$</td>
<td>0.170</td>
<td>—</td>
</tr>
<tr>
<td>Moving</td>
<td>No-slip entrance</td>
<td>0.162</td>
<td>4.8 %</td>
</tr>
<tr>
<td></td>
<td>Slip entrance</td>
<td>0.164</td>
<td>3.7 %</td>
</tr>
</tbody>
</table>
4.1 ASMO Model: Identification of Mesh Requirements

Figure 4.14: Moving ground: Time history of the drag coefficient for different configurations.

Figure 4.15: Moving ground: Time history of drag coefficient for particular components.

higher, see Table 4.11. This indicates, that the utilized definition of a moving ground performs well, as the same effect has been observed in [Damiani et al. (2004)]. Utilizing a moving ground, the $y^+$ value of the wheels increases notably, in particular if a slip condition is used in the entrance section, see Fig 4.16.
Table 4.11: Moving ground: Partial drag amount.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Total $c_d$</th>
<th>$c_{d,\text{Body}}$</th>
<th>$c_{d,\text{WheelRF}}$</th>
<th>$c_{d,\text{WheelRR}}$</th>
<th>$c_{d,\text{WheelLF}}$</th>
<th>$c_{d,\text{WheelLR}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fixed</td>
<td>1.71e-001</td>
<td>1.42e-001</td>
<td>8.6e-003</td>
<td>6.4e-003</td>
<td>8.6e-003</td>
<td>6.4e-003</td>
</tr>
<tr>
<td>Slip E.</td>
<td>1.64e-001</td>
<td>1.36e-001</td>
<td>6.3e-003</td>
<td>7.8e-003</td>
<td>6.0e-003</td>
<td>8.5e-003</td>
</tr>
<tr>
<td>No slip E.</td>
<td>1.61e-001</td>
<td>1.14e-001</td>
<td>5.3e-003</td>
<td>6.7e-003</td>
<td>5.6e-003</td>
<td>6.7e-003</td>
</tr>
</tbody>
</table>

This fact is comprehensible due to the movement of the ground with airflow velocity. Thus, the fluid velocity in vicinity to the ground is not decelerated as in the fixed ground case. As a result, the mass flux between the vehicle and the ground is increased and thus, the friction velocity at the lower part of the wheels increases.

Therefore, the first layer height of the wheels must be adapted, if moving ground boundary conditions are applied. However, as the vehicle is not moving, the boundary layer on the underbody is comparable in all cases, and thus $y^+$ does not change. The $y^+$ values on the bottom are in a comparable range. This is contra-intuitive, as the wall shear stress should be noticeably reduced. However, as the overall behavior of this moving ground setup is appropriate, this recovered detail must be analyzed in further studies.

The impact of the boundary condition of the entrance section is clearly visible in Fig. 4.16. The moving ground using a no-slip condition in the entrance section can be seen as transition between the fixed ground and the moving ground with slip entrance configuration.

In case of a moving ground, the $y^+$ fluctuations of all wheels are more distinct and periodical fluctuations can be observed. As the flow which passes the front wheels is undisturbed, the $y^+$ oscillations are damped significantly. The fluctuations at the rear wheels have comparable amplitudes. However, the drag signal only shows periodical fluctuations at the vortex shedding frequency; and other oscillations disappeared.

The reason for this behavior is the rotating movement of the rear wheels, which whirl the airflow behind the wheels along the rear face of the vehicle and thus, disturb the development of large eddies in the wake. Therefore, the drag signal is less affected...
by unsteady wake movements. The observed behavior is in very good agreement with the analysis in [Krajnovic and Davidson (2005)], which led to qualitative equal results for the comparison of a moving ground to a fixed ground.

In the following, the pressure distribution is compared. The analysis using the no slip condition on the bottom in the entrance section is neglected, because the slip condition models the real physics better. Therefore, this configuration is recommended for future issues. The pressure on the underbody changes slightly, see Fig. 4.17a. This modification is also not able to capture the underbody pressure distribution overall accurately.

As assumed, the moving ground provides the pressure recovery at the rear side of the vehicle, see Fig. 4.17b. This is due to the aforementioned prescribed angular velocity on the wheel boundary. The increased pressure at the rear face is the major reason for the drag decrease.

It is worth noting, that the moving ground configuration resolves the base pressure distribution more accurate than any fixed ground simulation, even though it has not the finest core element size. The pressure at the stagnation is not as overpredicted as for the fixed ground case (see Fig. 4.17c). The velocity fluctuations in the wake has been monitored, and no significant difference to the fixed ground configuration (see Fig. 1.10) can be observed. This is in good agreement with the findings in Krajnovic and Davidson (2005).

Summing up, these results encourage the utilization of a moving ground in further automotive aerodynamic analyses.
4.2 Sedan Model: Validation of Mesh Parameters and Time Step Strategies

The objective of the Sedan analysis is to consider the generality of mesh control parameters and several time step issues. The latter one is subdivided in several tasks: the review of the required flow startup time, an analysis of the time step size, and a comparison of different solution strategies.

Furthermore, the sedan analyses should provide insights into the flow physics due to the shape change. An increased contribution of the pressure-induced drag due to transient wake effects is to be expected.

4.2.1 Preparation of Simulation Settings

The mesh study using the ASMO model has determined appropriate values for mesh control parameters. In the following, the reliability of these recommended values is validated.

It is worth noting, that the ratio of vehicle and wind tunnel dimensions of the Sedan model do not equal the ASMO model and its wind tunnel design. It is desired to ensure the performance of the virtual wind tunnel tool on both arbitrary wind tunnel and vehicle extents.

As already mentioned, the global element size for the Sedan model is determined by $h_{\text{global}} \approx \sqrt{A_{\text{WT},S}} \approx 8\text{m}$. The subsequent specified mesh control parameters are listed in Table A.5 in Appendix A.4.

First of all, the time step size is $\Delta t = 6 \times 10^{-4}$ for all simulations. This time step size coincides with a CFL number of 1 for the finest grid. Note that this is a very secure choice, which later has been shown to be unnecessary. If the same physical properties as in the ASMO analysis are assumed (Strouhal number $St = 0.5$, 30 time steps per shedding period), the time step size becomes $\Delta t = 0.00192$.

Three freestream particle paths through the wind tunnel require 2.64 seconds, and 100 particle paths using the car height as reference quantity require 2.88 seconds. However, the time interval for the initial runs first has been chosen just twice as long as for the ASMO analysis ($T = 1.08 \text{s}$). It should be sufficient, to evaluate the startup-time and grid convergence of the drag coefficient in a qualitative manner.

Therefore, the first simulations are compared to a simulation with a very fine grid of 65 million elements, which calculated a drag coefficient of $c_d = 0.30$. This simulation has been performed before this study.
4.2 Sedan Model: Validation of Mesh Parameters and Time Step Strategies

4.2.2 Grid Convergence

To investigate the grid convergence, the simple sedan is investigated using 3 different mesh sizes. Only the core element size and core length are varied, because these parameters primarily specify the number of cells. The first layer height influences the total number of elements more than in the ASMO case. Furthermore, it is to be expected that the friction induced drag force contributes only a minor part to the total drag. Thus, all simulations are performed using the first layer heights related to the mesh control parameters $n_{flh} = 16$ and $n_{bot} = 13$, respectively. In Fig. 4.18a, the time history of the drag coefficient is shown. It is noticed, that the drag stabilize only for the fine core element size within a certain range.

In general, the drag has a significantly increased startup time. Due to the model dimensions, this behavior has been expected. However, the extensive oscillations and comparably short simulation time interval does not allow to conclude, whether the drag is actually converged.

Thus, several issues arise with the determination of the startup time, the total simulation time interval and the specification, within which range the drag has to stabilize to conclude convergence. These issues are considered in detail in Sec. 4.2.3.

At this point, we assume that the chosen time interval covers the startup time, and the solution remains approximately in a certain range, i.e. the drag converges. In the following sections, these assumptions are approved.

Thus, the averaging of the drag is initiated after the characteristic startup time, when the initial particles from the inlet have passed the vehicle, i.e.:

$$\frac{I_{WT,S} + L_S}{u_\infty} \simeq 0.4s$$  \hspace{1cm} (4.3)\]

The results are listed in Table 4.12. The amplitudes are comparably higher than in the ASMO analysis. Utilizing the finest core element size, the drag stabilizes around 0.293. The analysis with the coarse core element size does not yield a drag stabilizing within a certain range. The medium core element size accords to the mesh control

<table>
<thead>
<tr>
<th>Mesh</th>
<th># Elements</th>
<th>$c_d$</th>
<th>Core element size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>$14 \times 10^6$</td>
<td>0.280</td>
<td>$n_z = 7$</td>
</tr>
<tr>
<td>Medium</td>
<td>$20 \times 10^6$</td>
<td>0.286</td>
<td>$n_z = 8$</td>
</tr>
<tr>
<td>Fine</td>
<td>$30 \times 10^6$</td>
<td>0.293</td>
<td>$n_z = 9$</td>
</tr>
<tr>
<td>Reference</td>
<td>$65 \times 10^6$</td>
<td>0.300</td>
<td>$n_z \sim 10$</td>
</tr>
</tbody>
</table>
4.2 Sedan Model: Validation of Mesh Parameters and Time Step Strategies

Parameter \( n_z = 8 \). Unfortunately, utilizing the accordant element size in the core zone, the drag also slightly decreases during the whole time interval. Neither stability nor instability can be observed clearly. Although the approximation towards the result of the reference simulation is clearly visible, a grid convergent solution could not be obtained within this thesis.

It is reasonable, that the optimal ratio of the core element size to a model property is different for the Sedan vehicle type. It has already been discussed in Sec. 3.2.1, that the finest grid spacing \( h_{local} \) has to be specified for each particular problem, individually. However, it is to be expected that the required core element size is connected to the vehicle dimensions which are responsible for the vortex generation. In relation to the model properties of the vehicle and wind tunnel, this analyses roughly indicate that a realistic sedan shape requires about twice as fine resolution of the wake structures than a streamlined body.

4.2.2.1 First Layer Height

It is considered, how different vehicles and wheel configurations affect the integrated \( y^+ \) values. The comparable ASMO cases using the same first layer height are shown in 4.3a and 4.3b.

Table 4.13 shows the two different first layer height settings. As expected, the first layer height has no major influence to the total drag, but increases the mesh size significantly.

Fig. 4.19 shows integrated \( y^+ \) values in the last 0.5 seconds of the simulations. The airflow around the wheels is highly unsteady due to the vertex shedding behind the wheels, pressure oscillations in the wheel-housing, and interactions between the wheel vortex structure and the ground boundary layer. This behaviour has also been ob-
4.2 Sedan Model: Validation of Mesh Parameters and Time Step Strategies

Table 4.13: Simple sedan: Drag coefficient for different mesh control settings.

<table>
<thead>
<tr>
<th>$n_{flh}$</th>
<th>$n_{tot}$</th>
<th># Elements</th>
<th>$c_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>13</td>
<td>$30 \times 10^6$</td>
<td>0.293</td>
</tr>
<tr>
<td>17</td>
<td>15</td>
<td>$36 \times 10^6$</td>
<td>0.294</td>
</tr>
</tbody>
</table>

served e.g. in [Damiani et al. (2004)]. Equivalent values of the first layer height $h_{flh}$ on the ASMO body lead to relatively lower $y^+$ values of the sedan car body. As the influence of the Reynolds number and thus the model length is weak, the low integrated $y^+$ value is primary due to the shape of the Sedan model. However, the local $y^+$ values show huge differences, depending on the respective part of the vehicle surface. Therefore, it is recommended to utilize different first layer heights on the vehicle body, which depend on the local wall shear stresses. In difference to the ASMO, the integrated $y^+$ values of the wheels are slightly higher than for the car body. Thus, in case of real wheel-wheelhouse configurations, the first layer height must be adapted such that it is smaller than the first layer height of the vehicle body.

4.2.2.2 Drag Composition

The drag composition of the Sedan shows a lower contribution of the wheels to the total drag. In difference to the ASMO model, the forces exerted on the rear wheels are higher, which is coincident with former studies. The drag composition of the finest run is summarized in Table 4.4.

4.2.2.3 Core Zone Dimensions

The core zone analysis of the ASMO in Sec. 4.1.2.5 does not yield different results for varying core lengths. However, the reattachment point has been determined in a small
range within the shortest refinement zone. Only two sedan analyses are comparable directly, because their configuration is equivalent except the length of the core zone behind the vehicle. Table 4.15 shows clearly, that the core length is essential. Compared to the ASMO, the refinement zone of the simple sedan model in terms of the vehicle length must be longer. Unfortunately, neither the steady-state nor transient calculation does give evidence about the reattachment point. However, the minimal value of the predicted friction coefficient $c_f$ can be located at a position behind the vehicle, which can be expressed in terms of the vehicle height, i.e.: $X_r \sim H_S \times L_S$. Based on this data, an optimal core length can not be determined clearly. This agrees with the common experience, that the dimensions of a refinement zone must be carefully evaluated for each particular problem.

4.2.3 Time Step Strategies

4.2.3.1 Steady-state Evaluation

A rough definition of the startup time of the flow has already been evaluated in Sec. 4.2.2. In the most cases, there is no specific interest in this startup phase, and it is desired to overcome this period as fast as possible. Thus, a steady-state solution of the simple sedan problem is evaluated.

As assumed, the calculation of the drag is very inaccurate (Fig. 4.20a), its value changes noticeably from time step to time step. After 25 outer iterations, velocity and pressure are almost converged, see Fig. 4.20b. The comparison of the CPU time is shown in the Table 4.16. The number of iterations for the transient startup phase is determined by the startup time in Eq. (4.3). However, at this point it is not clear, whether the steady-state solution is able to circumvent the entire startup phase, or just shorten it. This issue is evaluated in the following sections. Although this steady-state computation is not accurate, it serves as a better prediction for the initial flow field. Thus, the solution is used as initial condition for the transient runs performed in the subsequent time step study.

4.2.3.2 Time Step Study

Similar to the considered time step issues for the ASMO analysis in Sec. 4.1.1, a time step study using the sedan model is performed. An appropriate time step must be small enough to resolve the turbulent motions accurately. On the other hand $\Delta t$ must

| Table 4.14: Simple sedan: Partial drag amount - fixed ground. |
|----------------|----------------|----------------|----------------|----------------|
| Total $c_d$   | $c_d$, Body    | $c_d$, WheelRF | $c_d$, WheelRR | $c_d$, WheelLF |
| 0.293         | 0.254          | 0.007          | 0.009          | 0.010          |

Mesh configuration: $n_{ph} = 17$, $n_{bot} = 15$, $n_z = 9$, MOL
be chosen as big as possible, such that the run time is not blown up. The objective is to consider whether the physical based assumptions in Sec. 4.2.1 are valid. Therefore, two time steps are chosen above and below $\Delta t = 0.00192$, respectively.

The simulations are initialized with the steady-state solution, and one tunnel sweep (0.88 s) is performed. The drag of the different time step studies is listed in Table 4.17.

Table 4.17: Simple sedan: Drag coefficient for different time steps.

<table>
<thead>
<tr>
<th>Time step</th>
<th>$\sim$ CFL</th>
<th>$c_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$6e \times 10^{-4}$</td>
<td>1</td>
<td>0.294</td>
</tr>
<tr>
<td>$12 \times 10^{-4}$</td>
<td>2</td>
<td>0.292</td>
</tr>
<tr>
<td>$24 \times 10^{-4}$</td>
<td>4</td>
<td>0.291</td>
</tr>
<tr>
<td>$36 \times 10^{-4}$</td>
<td>6</td>
<td>0.291</td>
</tr>
</tbody>
</table>
4.2 Sedan Model: Validation of Mesh Parameters and Time Step Strategies

Surprisingly, the computed drag differs just slightly. Thus, an open issue regarding an appropriate time step size remains. This problem is probably caused by an insufficient time interval, although the drag signal fluctuates around a certain value beyond a comparable small startup time. However, the startup time is not completely eliminated by the steady-state initial run. Thus, averaging over the complete transient time interval may be inappropriate. This issue is addressed in section 4.2.4 where an specific algorithm is applied to evaluate both, the startup time and drag convergence, based on the statistical properties of the drag signal.

4.2.3.3 Variable Time Step

Besides the common strategy using an initial steady-state analysis followed by a transient computation, another solution strategy utilizes a variable time step. Therefore, the analyses must not be divided into two separate computation tasks. The variable time step is obtained by multiplying the initial time step $t_{\text{init}} = 0.03$ with a piecewise linear function $g(t)$, until $t_s = 6 \times 10^{-4}$ is reached at the end of the startup time:

$$ g(t) = \begin{cases} 
\frac{1}{2} \left( 0.4 - 0 \right) + \frac{1}{2} \left( t - 0.3 \right) & t \in (0, 0.3] \\
0.02 & t \in (0.3, 0.4] \\
0.02 & t \in (0.4, 1.08) 
\end{cases} \quad (4.4) $$

The time increment and CPU time are shown in Fig. 4.21a. Obviously, the CPU time is higher for a higher time increment. Fig. 4.21b shows the residual ratio and number of iterations of the linear solvers. The pressure is mostly affected by the coarse time step, and thus, the conjugated gradient method requires a multiple of iterations. However, as supposed, the iterations of the GMRES solver are only slightly increased during the startup phase. A significant speedup due to this solution strategy is to be expected. In the next section, the different solution strategies are compared in terms of the required computation time and accuracy.

Figure 4.21: Simple sedan: Convergence and time-stepping with variable time step.
4.2.4 Solution Strategy Benchmark

Three different solution strategies, which have been introduced in the previous sections, are discussed:

- transient analysis with constant time step size
- transient analysis with variable time step size
- initial steady-state analysis with subsequent transient restart

All computations are performed using 16 cores on a distributed memory compute cluster. The transient analysis with constant time step size is the most inappropriate methodology due to the expensive startup time.

Due to the use of a variable time step size, the simulation time is reduced by about 36 hours of CPU time. In Fig 4.22a, the drag signal show spurious oscillations, which are absent in case of a fixed time step size. Although these disturbances decay during the time interval, they are observed again at the end of the time interval. This indicates, that the utilization of a variable time step may distort the solution entirely. For further investigation, the time interval must be extended. This has not been done within this thesis, as the steady-state initial run with a subsequent transient restart performed overall well. The transient restart using a time step size of $\Delta t = 6 \times 10^{-4}$ leads to nearly the same computation time per sweep as the stand-alone transient run using the same time step size. Thus, the transient restart using $\Delta t = 12 \times 10^{-4}$ is analysed, as this time step is still resolved nearly 50 times within one vortex shedding period.

Assuming that the steady-state analysis eliminates the startup time of 0.4 seconds, the transient restart is carried out for 0.7 seconds. Obviously, the CPU time is reduced
significantly due to the shortened time interval and increased time step size. However, Fig. 4.22a shows clearly that the amplitudes of the oscillations increase considerably. Furthermore, Fig. 4.22a can not give evidence whether the startup time is eliminated or shortened. As this problem has been discovered throughout the last sections, the discussion of this issue is crucial. The next section addresses this problem describing a structured approach both to determine the startup time and to declare, whether the computed time interval is sufficient or not.

4.2.5 Criteria for Drag Convergence Assessment

As CFD analyses are often used prior to physical experiments or even replace them completely, the solution can not be compared to experimental results. Therefore, the VWT tool must establish the convergence and stability of the force quantities based on the statistical properties of the signal. It is worth noting, that the following procedure has not been developed within this thesis, but within the AcuSolve™ solver development team, after the issues from the previous sections came up. In this work, the usability of the procedure for the computed drag signals is evaluated.

Therefore, the procedure analyzes the drag signal based on its statistical properties as follows:

1. Calculate the vortex shedding period $T_s$ from Eq. (3.16)

2. Filter the signal of the considered quantity $\psi$ using a backward bandpass filter of the width $\Delta T_f = C_f \times T_s$. Therefore, high frequencies due to numerical noise and low frequencies due to blockage effects of the wind tunnel boundary conditions can be eliminated.

3. Divide the filtered signal $\hat{\psi}$ in $n$ intervals $T_i$ having the sample width $\Delta T_s = C_s \times T_s$.

4. For each interval, calculate the average $\bar{\hat{\psi}}_i$ and standard deviation $E(\hat{\psi}_i)$.

5. If $E(\hat{\psi}_i) > 0.05 \bar{\hat{\psi}}_i$, discard the interval. Therefore, the startup time is not factored into convergence calculation.

6. Calculate the running average $\bar{\hat{\psi}}$ and standard deviation $E(\hat{\psi})$ based on the average $\bar{\hat{\psi}}_i$ of each valid interval. The error bar is calculated as $1.96 \times E(\hat{\psi})/\sqrt{N}$, $N$ is the number of valid intervals.

7. The CFD analysis is terminated, if the error $e_d = \frac{1.96 \times E(\hat{\psi})}{\sqrt{N} \times \bar{\hat{\psi}}_{\text{run}}}$ < $\epsilon$, where $\epsilon$ is a user-specified convergence tolerance. Therefore, the running average of the drag signal is compared to its standard deviation, and the convergence of the signal can be observed.
Although this method should provide an quantitative convergence criteria for transient simulations, it still requires appropriate input values for $C_f$, $C_s$, and $\epsilon$.

First of all, the procedure is applied to two ASMO analyses using a fixed and moving ground condition, respectively. As a sufficient time interval is known for the ASMO analysis, appropriate filter and sample widths can be determined. Furthermore, differences between the fixed and moving ground condition can be evaluated in a quantitative manner.

![Graphs showing drag signal comparisons](image)

Figure 4.23: ASMO: Comparison of the drag signal for fixed and moving ground conditions, $C_f = C_s = 3$

Within this work, $C_f$ equals $C_s$, and the criteria has been applied using various values for the filter and sample widths. However, for the sake of clarity only one distinct parameter value of $C_f = 3$ is analyzed in detail. As the convergence is not considered whilst the simulation is running, no $\epsilon$ must be specified.

If a filter and sample width of $C_s = 1$ or $C_s = 2$, respectively, is utilized, even those
time intervals are not discarded, which belong obviously to the initial phase. Thus, these choices are not further deepened here.

The minimal filter and sample widths leading to appropriate results have the value $C_f = C_s = 3$. The result of the statistical evaluation is shown in Fig. 4.23a-4.23b. The predicted startup time (cf. Sec. 4.1.2) of about 0.1 seconds is discarded. Furthermore, the drag coefficient of the fixed ground simulation is almost completely converged at the end of the time interval, i.e. the error remains nearly constant at the simulation end time. Increased values for both, the sample and the filter width, does not lead to noticeable changes of the results. Thus, it is assumed that these values are appropriate for statistical drag convergence assessment.

As already mentioned in Sec. 4.1.6, in case of a moving-ground simulation the fluctuations of the drag coefficient are more conform and the signal is less disrupted by unsteady wake effects. Therefore, the startup time of the flow is significantly decreased, and the deviation from the average value is small throughout the entire

---

Figure 4.24: Simple sedan: Comparison of solution strategies for $C_f = 3$.

\footnote{drag signal - red line; filtered drag signal - solid black line; interval mean - green points; error bars - blue bars.}
This drag convergence analysis is applied to the drag signals of the sedan model using different time step strategies. The error of the first valid interval of the constant time step simulation (Fig. 4.24a) is clearly below the error of the variable time step analysis (Fig. 4.24b). However, the error of the variable time step decreases faster, but the unfiltered signal show the already mentioned numerical problems. Thus, both the filtered and unfiltered signal must be evaluated, to establish the suitability of the results.

The error in Fig. 4.24d show the differences of the startup time using a steady-state initial run. This indicates, that the startup time is more than halved. Of course, the error using the coarser time step size of \( \Delta t = 12 \times 10^{-4} \) is higher. Furthermore, this analysis requires more than the considered time interval to ensure convergence.

Unfortunately, it can only be assumed and not be certainly concluded that the considered time interval of the simple sedan analysis is sufficient. Although the error of the two pure transient analyses is below 1%, a restart of these simulation is required to ensure convergence utilizing wider sample widths.

However, the results of these analyses are overall suitable. Furthermore, the usability of the convergence algorithm has been evaluated to be a feasible statistical method for appropriate filter and sample widths.
4.3 Complex Sedan Model: Parametrization Capabilities

In this section, the analysis of the complex sedan model is evaluated. The model properties are described in section 3.1.3.

The major objective is to determine appropriate meshing parameters for the complex underbody flow surrounding the exhaust system and the flow within the engine compartment. Therefore, it is considered if the applied core element size is appropriate to model the unsteady effects within the wake and engine compartment. The $y^+$ values are also investigated.

As the time step specification depends on model dimensions, which are not changed within this model, the impact of enhanced model complexity is evaluated.

First of all, the complex sedan model was analysed using the same mesh settings as the simple sedan in Sec. 4.1.2. Independent of the grid density, no stabilization of the drag has been observed in the considered time interval. With regard to the results in Sec. 4.2.2 it is to be expected that the failure is due to insufficient settings either of the mesh space or the time interval. Furthermore, the complex flow field within the engine compartment may excite the drag forces to oscillate with higher amplitudes.

However, as stability can not even observed nearly, no convergence algorithm is applied to these runs. These runs have been used to get insight into the flow field and adapt the mesh settings, which are described in the following section.

4.3.1 Mesh Settings

A second even finer refinement box has been added in proximity to the trunk. Furthermore, the first layer height of the exhaust system has been increased, as its $y^+$ value has been nearly three-digit. The fine first layer heights on both the vehicle and bottom as well as the smooth growth in element size ensure an appropriate grid spacing in the vertical direction.

As the pressure drag is the major drag source, the grid spacing of the vehicle shell mesh both in the underbody- and rear region could be too coarse, and thus distort the solution. However, a decreased shell element size in the complex underbody region would lead to an enormous increase of elements. Thus, this aspect is not considered within this work.

Low flow velocity and thus, low skin friction within the engine compartment is to be expected. Thus, the engine first layer height is specified equal the bottom first layer height.

It can be assumed, that even no boundary layer meshing on the engine component is suitable. However, the boundary layer mesh is applied to obtain an appropriate $y^+$
value for an arbitrary use case, in which an engineer demands the accurate prediction of flow in proximity to the complicated engine geometry e.g. due to cooling aspects.

Therefore, the new mesh summarized in Appendix A.4 Table A.6 consists of nearly 53 million elements (8.9 million nodes). A steady-state analysis has been carried out to yield optimized initial conditions. Due to limited computing resources, the mesh size turned out being too large to obtain drag convergence using a transient analysis within an acceptable time interval (see Fig. 4.25c - 4.25d). Therefore, the major objective is limited to the investigation of mesh requirements within the engine compartment and complex underbody region. The \( y^+ \) values and surface pressure distributions are compared to the results of the simple sedan model.

**4.3.2 Comparison of Sedan Configurations**

First of all, the qualitative drag evolution in time is considered. The oscillations of all components are superposed by the vortex shedding frequency and irregular frequencies on larger scales. Extensive vortex shedding starts more or less after a time interval of 0.4 seconds which already has been determined in Sec. 4.2.2.

Fig. 4.25d shows that the startup phase of the complex sedan after a steady-state initial run requires a larger time interval than in the simple sedan model. This indicates that the model complexity must be taken into account for time stepping settings. However, as the drag convergence algorithm will terminate the simulation if the error bar remains in a certain range, the user does not have to take care about specifying a sufficient time interval.

It is assumed, that the drag coefficient of the engine component is close to zero, because the small outlet placed below the engine compartment induces a homogeneous pressure increase within the entire engine compartment.

Surprisingly, also the rear wheels obtain a negative drag in more or less regular time intervals. However, due to the fully opened wheel configuration, the appearance of this effect may be explicable. The \( y^+ \) values of the complex sedan model demonstrate significant differences compared to the simple sedan type. First of all, the relatively low airflow velocity within the engine compartment decreases the integrated \( y^+ \) value of the body as the interior wall of the engine compartment also belongs to the body component. Fig. 4.25b shows clearly that the utilization of the opened wheel type leads to a significant decrease of integrated \( y^+ \) values. This is reasonable due to the area of low wall shear stress on the inner wheel surfaces. Furthermore, the wheel \( y^+ \) values are less fluctuating compared to the simple sedan configuration (refer to Fig. 4.19).

Considering an equal first layer height, the \( y^+ \) value of the engine is half the \( y^+ \) value of the bottom. Assuming the same first layer height at the engine component and the vehicle body, their \( y^+ \) values would be in a comparable range.

It can be clearly seen, that the \( y^+ \) value of the exhaust system is without an accept-
4.3 Complex Sedan Model: Parametrization Capabilities

Figure 4.25: Complex sedan: Time history of $c_d$, $y^+$, $\tilde{c}_d$, $\bar{\tilde{c}}_d$ and $\epsilon$.

able range. It oscillates at the vertex shedding frequency and with high amplitudes, as the position of the exhaust system enables the flow to surround it completely.

As the first layer height equals that of both the body and wheels, it can be concluded that the first layer height on distant underbody components must be chosen about four times higher than the rest. Therefore, most modern cars have an underseal. However, these results instruct how to proceed with details in the underbody region, which stick out of the underbody plane. The surface pressure distributions are shown in Fig. 4.26a - 4.26d. Obviously, the pressure distribution over the roof is nearly the same, except few details near the front radius, a slightly pressure loss near the trunk deck and a small range of increased pressure around $x \sim 1.5m$.

In Fig. 4.26b, the reasonable pressure loss at the height of the radiator grill ($1.05m < z < 1.15m$) can be clearly seen. In opposite, the immense pressure loss on the engine hood near the front shield ($z > 1.75m$) can not be explained.

The base pressure drop of the complex sedan model coincides with the increased drag. This is probably due to modeling the exhaust in the surface mesh, thereby creating
The underbody pressure distribution is splitted due to the outlet in the bottom of the engine compartment which intends to enable a regular air flow avoiding reverse flow through the radiator grill. The first pressure distribution near the front radius is qualitatively reasonable, as the air flow leaves the engine compartment and thereby prevents the pressure loss upstream the outlet. The pressure downstream the outlet obtains a mostly uniform distribution over the surface, and increases slightly towards the rear side.
5 Summary

The aim of this thesis was the analysis of both space and time discretization requirements of AcuSolve\textsuperscript{TM} in order to perform accurate highly turbulent CFD simulations in a virtual wind tunnel. A mesh sensitivity analysis using different mesh control parameters was performed, and the results were validated against values of the drag coefficient and the surface pressure distribution obtained by experiments. Different requirements regarding the various vehicle and wind tunnel regions was evaluated. Most wind tunnels in use today apply real road conditions in their experiments. In order to increase the degree of realism, the simulation of these real road conditions was investigated in detail. Therefore, a set of boundary conditions for a moving ground simulation was specified. The results were compared to a fixed ground analysis. The generality of previously defined mesh control parameters was validated using two variations of a sedan car model. Furthermore, an appropriate choice of the time step size was analysed in detail. Different time step strategies were compared in terms of computing time, and the usability of a statistical procedure to determine the startup phase and termination time was evaluated.

In general, the performed simulations approximated the experimental data accurately by applying the DDES turbulence model, a well-defined tetrahedral mesh and a time step size based on the vortex shedding frequency.

The mesh sensitivity analysis using the ASMO model yielded appropriate values for the considered mesh control parameters, which could be partly expanded to the sedan analysis.

Due to its considerable effect on drag, the first layer height on the vehicle body must be chosen such that $y^+ \sim 4$ is reached. The integrated $y^+$ values of the wheels depend on the specific wheel-type and must be considered individually. If an accurate resolution of the near wall flow around the engine block is required, the first layer height must be chosen equivalent to the body. Distant details in the underbody region demands a four times higher first layer height to receive $y^+$ values in a recommended range.

Using a constant element growth boundary layer meshing scheme leads to volume meshes of good quality, which enhances the accuracy noticeably. However, the number of total elements is significantly increased compared to a constant number of boundary element layers. In general, this meshing type has to be applied as long as the element size of the underlying vehicle surface mesh is smaller than the surrounding volume mesh element sizes (cf. Sec. 4.1.3).

Regarding the constraint of the Octree meshing algorithm, the wind tunnel and
vehicle dimensions are connected to the mesh refinement in the core zone.

Based on the Octree meshing rule, an optimal core element size has been determined for the ASMO. This could not be expanded to the sedan model. However, regarding the dimensions of the sedan model and the corresponding wind tunnel, the core element size must be a factor of two smaller in order to yield accurate results.

A systematic a-priori determination of the wake dimensions is not possible, as the reattachment point and free stagnation point must be computed from the flow field. Although the wake dimension has not been determined systematically, the required length of the core zone in the investigated cases correlates to the product of height and length of the vehicle. However, this rule of thumb can definitely not be applied in general.

The pressure distribution revealed that the mesh requirements differ noticeably with regard to the considered region. The accurate pressure prediction on the roof is trivial even without accurate grid resolution in the boundary layer. The pressure in the stagnation point is affected by the boundary layer meshing scheme. Although the overall pressure prediction is accurate, some prediction problems have been found both in the base and underbody region (cf. Sec. 4.1.3.6). In general, they can be addressed using a small core element size, an appropriate first layer height, a constant element growth rate, and a sufficiently fine surface mesh. If the latter requirement is not fulfilled, the boundary layer meshing scheme must create a constant number of boundary layers.

All in all, the overall prediction of the pressure distribution on the vehicle reproduces the experimental data in the majority of the surface regions.

The DDES turbulence model has proven to work just fine, except for the ASMO-underbody region near the front radius, where RANS mode is obtained in an unforeseen manner (cf. Sec. 4.1.4.2).

To simulate real road conditions, the freestream velocity has been prescribed on the ground below the vehicle, and a rotating reference frame has been applied at the wheels. Coincident with former studies, the embedding of specific boundary conditions in the ASMO analysis increases the degree of realism considerably (cf. Sec. 4.1.6). The flow field is less disturbed, the flow startup time is decreased significantly, and the drag signal is corrected from spurious oscillations and dominated by an almost periodical vortex shedding frequency. As the mass flux between the vehicle underbody and the moving ground increases, the first layer height of the wheels has to decrease in order to yield acceptable $y^+$ values.

A comparison of time step strategies gave evidence about numerical problems when utilizing a variable time step. Furthermore, it was evaluated that the estimation of the flow startup time using the wind tunnel and model properties is only valid for closed vehicles. Due to an insufficient chosen time interval, a time step study did not yield specifiable results.

The quality of the evaluated statistical procedure determining the startup time and
termination time depends on the adaptable sample width and filter width. Nevertheless, the performed drag convergence analyses lead to the conclusion, that the utilized procedure is appropriate to determine drag convergence during runtime.

The widely-used steady-state initial run with subsequent transient restarts is the most efficient strategy to shorten the required calculation time during the startup phase. However, this strategy does not shorten the total computation time interval, as a minimum time interval depending on the model properties must be simulated in order to achieve a converged drag signal.

To summarize, the results of this thesis show that the specified meshing and time-stepping rules enable accurate transient CFD analyses in the virtual wind tunnel. However, the efficiency can be enhanced using ROI-determined wake refinement, adaptive shell mesh refinement with regard to the requirements of specific flow regions, and as a result a subdivision of the vehicle components to address these requirements specifically.

5.1 Outlook

Several issues must be addressed to improve the efficiency and accuracy of CFD simulations in the virtual wind tunnel tool. First of all, future work should focus on the determination of the refinement zone dimensions using the ROI-approach to reduce the number of elements significantly. This meshing scheme would address the inherent requirements of the different flow regions.

Also, the introduced mesh control parameters should be varied further in order to obtain grid convergence of the sedan model. In addition, the sedan car simulation must be performed with an enlarged time interval to obtain drag convergence.

The time step size for the ASMO simulation must be decreased in order to evaluate if the utilized time step size of $3 \times 10^{-4}$ is sufficiently small. This would give insight, how much time steps per vortex shedding period are required to capture the unsteady effects in the wake accurately.

The frequency spectrum of the drag signal must be analysed in detail in order to investigate the origin of both, the larger and smaller frequencies, observed in all drag signals of fixed ground simulations.

Further research is required for the accurate definition of the wake dimensions, as no satisfactory results have been generated within this thesis. Additional simulations must be performed with a refined surface mesh of the ASMO model in order to determine the effect on accuracy of the base pressure prediction.

The utilized connection of the model properties to the definition of element sizes in the turbulent regions can be seen as a first attempt to parametrize and automate the mesh generation. Further investigations containing additional real passenger cars with different levels of grid densities of both, the surface and volume mesh, are required.

In addition, simulations using other turbulence models as e.g. DES-SST should be performed. Although the DDES turbulence method operates overall well, in case of
the ASMO a RANS modeled zone has been determined between the underbody and
the wind tunnel ground. This issue must be considered in further studies.

The degree of realism of the moving ground simulation can be further improved.
Therefore, advanced simulations containing the rolling-resistance of the wheels by uti-
lizing a sliding mesh approach with deformable wheels and complex material behavior
should be performed.

In order to increase the robustness of the workflow, automated surface mesh refine-
ment for the rear part of the vehicle surface mesh should be implemented. Although
it is very challenging to define appropriate mesh sizes for the resolution of turbulent
regions accurately, rough estimations based on the vehicle extents are possible. This
can be done analogously to the estimation of the volume and time step sizes.

Although real passenger car shapes contain various features with specific require-
ments on the volume mesh, it is rather easy to categorize vehicle shapes and thus,
create component databases containing mesh requirements and guidelines for vehicle
components. This definitely has an impact onto efficient design optimization.
Acknowledgments

First of all, I would like to express my deepest gratitude and respect to all of my supervisors, Dr. Marc Ratzel from Altair Engineering, as well as Lutz Pauli and Prof. Marek Behr from the Chair for Computational Analysis of Technical Systems at RWTH Aachen University for supervising this thesis, guiding me through all stages of the study and sharing their extensive knowledge in the field of Computational Fluid Dynamics and Finite Element Analysis.

Furthermore, this thesis would not have been possible unless my friend Simo Schmidt, for many days rather than hours of proof-reading, always reminding me to keep it simple. Many errors in English grammar and spelling would have never been found without him.

I am indebted to many of my colleagues from Altair Germany, especially everybody in the office Cologne, for supporting me during the last months with hours of HyperMesh selections, organizational issues and qualified conversations. All of this enabled me to focus on my thesis.

Furthermore, I would like to thank the people from the AcuSolve™ team, in particular David Corson, Ravi Lanka, Marlow Springer, Yoshifumi Suzuki, and Yaser Khalighi, for answering so many questions regarding AcuSolve or CFD in general, showing great patience and supporting me during various stages of my study.

Finally, I am endless grateful to my family, friends, and my girlfriend Jolanda. Without their incredible support, patience, belief and love, this thesis would not have driven to such success.
Bibliography


Appendix A

A.1 ASMO analysis - Model Properties

Table A.1: ASMO model dimensions.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frontal area $A_A$</td>
<td>0.0783 $[m^2]$</td>
</tr>
<tr>
<td>Length $L_A$</td>
<td>0.810 $[m]$</td>
</tr>
<tr>
<td>Height $H_A$</td>
<td>0.270 $[m]$</td>
</tr>
<tr>
<td>Width $W_A$</td>
<td>0.290 $[m]$</td>
</tr>
<tr>
<td>Ground clearance $C_A$</td>
<td>0.033 $[m]$</td>
</tr>
</tbody>
</table>

Table A.2: ASMO: Virtual wind tunnel dimensions.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tunnel length $L_{WT,A}$</td>
<td>9 $[m]$</td>
</tr>
<tr>
<td>Clearance Inlet → ASMO $I_{WT,A}$</td>
<td>4 $[m]$</td>
</tr>
<tr>
<td>Clearance ASMO → Outlet $O_{WT,A}$</td>
<td>4.19 $[m]$</td>
</tr>
<tr>
<td>Width $W_{WT,A}$</td>
<td>1.916 $[m]$</td>
</tr>
<tr>
<td>Height $H_{WT,A}$</td>
<td>1.916 $[m]$</td>
</tr>
<tr>
<td>Cross section $A_{WT,A}$</td>
<td>3.67 $[m^2]$</td>
</tr>
</tbody>
</table>

The surface mesh of the wind tunnel and the ASMO model, including the wheels, consists of 286486 first order triangle elements during all simulations, including the moving ground case. Most of the wind tunnel triangles are equilateral with an edge length of 0.067 $m$. 
A.2 ASMO analysis - Further Plots

Figure A.1: Virtual Wind Tunnel Tool.
A.3 Sedan Analysis - Model Properties

Figure A.2: ASMO: Pressure distribution paths in symmetry plane

Figure A.3: ASMO: Vorticity Field at different time steps
A.3 Sedan Analysis - Model Properties

The core zone is defined equivalent to the refinement zone in the ASMO analysis, only the dimensions are changed. The core zone extends to 2 m in vertical- and 4 m in spanwise direction[1]. In flow direction, the zone length will vary with $L_{W,L,S}$ as in the ASMO analysis. In the front the core zone starts at a distance of 0.9 m to the first

---

1 In the sedan model, the y- and z-direction are interchanged. This was done in order to evaluate, which code segments of the batch-postprocessing scripts must be adapted to yield correct views automatically.
Table A.4: Sedan: Virtual wind tunnel dimensions.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tunnel length $L_{WT,S}$</td>
<td>45 [m]</td>
</tr>
<tr>
<td>Clearance Inlet $I_{WT,S}$ → Sedan</td>
<td>15 [m]</td>
</tr>
<tr>
<td>Clearance Sedan $I_{WT,S}$ → Outlet $O_{WT,S}$</td>
<td>25.81 [m]</td>
</tr>
<tr>
<td>Width $W_{WT,S}$</td>
<td>11.48 [m]</td>
</tr>
<tr>
<td>Height $H_{WT,S}$</td>
<td>6 [m]</td>
</tr>
<tr>
<td>Cross section $A_{WT,S}$</td>
<td>68.88 [m$^2$]</td>
</tr>
<tr>
<td>Blockage ratio $\zeta_{WT,S}$</td>
<td>3.3 [%]</td>
</tr>
</tbody>
</table>

car surface mesh point, i.e. a stagnation point.

The surface mesh of the wind tunnel, the vehicle, and the wheels consists of 329438 respective 490868 first order triangle elements for the simple sedan or complex sedan model, respectively. The equilateral triangles have an edge length of 0.3 m.
A.4 Sedan Analysis - Mesh Parameters

Table A.5: Simple sedan: Investigated mesh parameters.

<table>
<thead>
<tr>
<th>Mesh parameter</th>
<th>Parameter</th>
<th>Value $[m]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$n_{flh}$</td>
<td>16</td>
<td>$1.22 \times 10^{-4}$</td>
</tr>
<tr>
<td></td>
<td>17</td>
<td>$6.10 \times 10^{-5}$</td>
</tr>
<tr>
<td>$n_z$</td>
<td>7</td>
<td>0.125</td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.06125</td>
</tr>
<tr>
<td></td>
<td>9</td>
<td>0.030625</td>
</tr>
<tr>
<td>$n_{bot}$</td>
<td>13</td>
<td>$9.76 \times 10^{-4}$</td>
</tr>
<tr>
<td></td>
<td>15</td>
<td>$2.44 \times 10^{-4}$</td>
</tr>
<tr>
<td>$L_{WL}$</td>
<td>0.25</td>
<td>1.2275</td>
</tr>
<tr>
<td></td>
<td>0.75</td>
<td>3.6825</td>
</tr>
<tr>
<td></td>
<td>1.25</td>
<td>6.1375</td>
</tr>
<tr>
<td>BL constant</td>
<td>Growth rate (MOL)</td>
<td>Num${BL}$: —</td>
</tr>
<tr>
<td></td>
<td>Number of layers (NBL)</td>
<td>Num${BL,vehicle}$: 15 ; Num${BL,bot}$: 6</td>
</tr>
</tbody>
</table>

Table A.6: Complex sedan: Improved mesh control parameters.

<table>
<thead>
<tr>
<th>Mesh parameter</th>
<th>Parameter</th>
<th>Value $[m]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$n_{flh}$</td>
<td>17</td>
<td>$6.10e-05$</td>
</tr>
<tr>
<td>$n_{z1}$</td>
<td>9</td>
<td>0.030625</td>
</tr>
<tr>
<td>$n_{z2}$</td>
<td>10</td>
<td>0.0153125</td>
</tr>
<tr>
<td>$n_{bot}$</td>
<td>15</td>
<td>$2.44e-04$</td>
</tr>
<tr>
<td>$L_{WL}$</td>
<td>1.25</td>
<td>6.1375</td>
</tr>
<tr>
<td>BL type</td>
<td>MOL</td>
<td>Num${BL}$: —</td>
</tr>
</tbody>
</table>
A.5 Sedan Analysis - Further Plots

Figure A.5: Complex sedan: DDES Transition

(a) Time step 1000

Figure A.6: Simple sedan: Moving ground

(a) Time step 1000
A.6 AcuSolve - Solver Settings

The convergence checks in AcuSolve\textsuperscript{TM} are performed using the residual ratio defined by:

\[
\frac{\left\| \mathbf{R}_m \right\|}{\left\| \mathbf{M}(\mathbf{R}_m) \right\|_1}, \quad (A.1)
\]

where \( \mathbf{M}(\mathbf{R}_m) \) determine the out-of-balance forces at a node which make up this nodal residual. The norm of the solution increment is normalized with respect to the norm of the solution field prior to updating the solution:

\[
\frac{\left\| \Delta \mathbf{u}^{(i-1)}_{n+1} \right\|}{\left\| \mathbf{u}^{(i-1)}_{n+1} \right\|_2}, \quad (A.2)
\]

Further utilized solver settings are:

- Two stagger iterations (flow, turbulence, flow, turbulence)
- Number of Krylov vectors for GMRES: 10 (velocity); 40 (eddy viscosity)
- Time-stepping convergence tolerance: 0.001
- Stagger convergence tolerance (convergence tolerance for non-linear iterations of the stagger): 0.1
- Convergence tolerance of Conjugated Gradient Method: 0.01
- Convergence tolerance of GMRES Method: 0.01
- Minimum number of linear iterations: 10
- Maximum number of linear iterations: 1000