Simulation of a passenger car cabin using a coupled GT-SUITE-TAITherm simulation model

cand. fmt. Christoph Boettcher
Matr. Nr. 2654629

Universität Stuttgart
Institut für Verbrennungsmotoren und Kraftfahrwesen
Lehrstuhl Kraftfahrwesen
Prof. Dr.-Ing. J. Wiedemann

In cooperation with

GT Technologies

November 2016
## Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Contents</td>
<td></td>
<td>III</td>
</tr>
<tr>
<td>List of abbreviations</td>
<td></td>
<td>V</td>
</tr>
<tr>
<td>Nomenclature</td>
<td></td>
<td>VI</td>
</tr>
<tr>
<td>List of figures</td>
<td></td>
<td>IX</td>
</tr>
<tr>
<td>List of tables</td>
<td></td>
<td>XI</td>
</tr>
<tr>
<td>Abstract</td>
<td></td>
<td>XII</td>
</tr>
<tr>
<td>1</td>
<td>Introduction</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>Theory of heat transfer</td>
<td>5</td>
</tr>
<tr>
<td>2.1</td>
<td>Conduction</td>
<td>5</td>
</tr>
<tr>
<td>2.2</td>
<td>Convection</td>
<td>6</td>
</tr>
<tr>
<td>2.2.1</td>
<td>Forced convection</td>
<td>7</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Natural convection</td>
<td>10</td>
</tr>
<tr>
<td>2.3</td>
<td>Radiation</td>
<td>11</td>
</tr>
<tr>
<td>3</td>
<td>Theory of the Co – Simulation</td>
<td>16</td>
</tr>
<tr>
<td>3.1</td>
<td>Flow solution in GT – SUITE</td>
<td>17</td>
</tr>
<tr>
<td>3.2</td>
<td>Energy solution in TAItherm</td>
<td>20</td>
</tr>
<tr>
<td>3.3</td>
<td>Mapping and communication</td>
<td>21</td>
</tr>
<tr>
<td>4</td>
<td>Investigation of the flow in GT – SUITE</td>
<td>25</td>
</tr>
<tr>
<td>5</td>
<td>Model building and calibration</td>
<td>33</td>
</tr>
<tr>
<td>5.1</td>
<td>Simple Box Model</td>
<td>33</td>
</tr>
<tr>
<td>5.1.1</td>
<td>Model building in TAItherm</td>
<td>33</td>
</tr>
<tr>
<td>5.1.2</td>
<td>Model building in COOL3D</td>
<td>39</td>
</tr>
<tr>
<td>5.1.3</td>
<td>Model setup in GT – SUITE</td>
<td>42</td>
</tr>
<tr>
<td>5.1.4</td>
<td>Optimization</td>
<td>43</td>
</tr>
<tr>
<td>5.1.5</td>
<td>Calibration</td>
<td>46</td>
</tr>
<tr>
<td>5.2</td>
<td>Cabin Model</td>
<td>48</td>
</tr>
<tr>
<td>5.2.1</td>
<td>Model building in TAItherm</td>
<td>48</td>
</tr>
<tr>
<td>5.2.2</td>
<td>Model building in COOL3D</td>
<td>50</td>
</tr>
<tr>
<td>5.2.3</td>
<td>Model setup in GT – SUITE</td>
<td>54</td>
</tr>
<tr>
<td>5.2.4</td>
<td>Calibration</td>
<td>57</td>
</tr>
</tbody>
</table>
### 6 Results and discussion

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1</td>
<td>Simple Box Model</td>
<td>60</td>
</tr>
<tr>
<td>6.1.1</td>
<td>Temperature sensors</td>
<td>60</td>
</tr>
<tr>
<td>6.1.2</td>
<td>Discretization study</td>
<td>68</td>
</tr>
<tr>
<td>6.2</td>
<td>Cabin Model</td>
<td>73</td>
</tr>
<tr>
<td>6.2.1</td>
<td>Temperature sensors</td>
<td>73</td>
</tr>
<tr>
<td>6.2.2</td>
<td>Velocity distribution</td>
<td>80</td>
</tr>
</tbody>
</table>

### 7 Conclusion

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.1</td>
<td>Model building</td>
<td>86</td>
</tr>
<tr>
<td>7.2</td>
<td>Simulation results</td>
<td>89</td>
</tr>
<tr>
<td>7.3</td>
<td>Future work</td>
<td>90</td>
</tr>
</tbody>
</table>

### Appendices

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Figures</td>
<td>91</td>
</tr>
<tr>
<td>B</td>
<td>Tables</td>
<td>95</td>
</tr>
</tbody>
</table>

### References

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>100</td>
</tr>
</tbody>
</table>
## List of abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CPU</td>
<td>Central Processing Unit</td>
</tr>
<tr>
<td>DB</td>
<td>DashBoard</td>
</tr>
<tr>
<td>DIN</td>
<td>Deutsche Industrie Norm</td>
</tr>
<tr>
<td>HTC</td>
<td>Heat Transfer Coefficient</td>
</tr>
<tr>
<td>HTCM</td>
<td>Heat Transfer Coefficient Multiplier</td>
</tr>
<tr>
<td>HVAC</td>
<td>Heating, Ventilation and Air-Conditioning</td>
</tr>
<tr>
<td>GT</td>
<td>Gamma Technologies</td>
</tr>
<tr>
<td>MFR</td>
<td>Mass Flow Rate</td>
</tr>
<tr>
<td>RAM</td>
<td>Random-Access Memory</td>
</tr>
</tbody>
</table>
## Nomenclature

### Latin signs

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>[m²]</td>
<td>Cross-sectional area</td>
</tr>
<tr>
<td>a</td>
<td>[m²/s]</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>c</td>
<td>[J/(kg*K)]</td>
<td>Specific heat</td>
</tr>
<tr>
<td>Cᵢ</td>
<td>[-]</td>
<td>Fanning friction factor</td>
</tr>
<tr>
<td>cₚ</td>
<td>[J/(kg*K)]</td>
<td>Specific isobaric heat</td>
</tr>
<tr>
<td>L</td>
<td>[m]</td>
<td>Characteristic length</td>
</tr>
<tr>
<td>D</td>
<td>[m]</td>
<td>Equivalent diameter</td>
</tr>
<tr>
<td>g</td>
<td>[m/s²]</td>
<td>Gravitational acceleration</td>
</tr>
<tr>
<td>Gr</td>
<td>[-]</td>
<td>Grashof number</td>
</tr>
<tr>
<td>e</td>
<td>[J/kg]</td>
<td>Total specific internal energy</td>
</tr>
<tr>
<td>h</td>
<td>[J/kg]</td>
<td>Total specific enthalpy</td>
</tr>
<tr>
<td>Kᵦ</td>
<td>[-]</td>
<td>Pressure loss coefficient</td>
</tr>
<tr>
<td>m</td>
<td>[kg]</td>
<td>Mass</td>
</tr>
<tr>
<td>ṁ</td>
<td>[kg/s]</td>
<td>Mass flow rate</td>
</tr>
<tr>
<td>Nu</td>
<td>[-]</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>p</td>
<td>[bar]</td>
<td>Pressure</td>
</tr>
<tr>
<td>Pr</td>
<td>[-]</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>Q̇</td>
<td>[W]</td>
<td>Heat transfer rate</td>
</tr>
<tr>
<td>q̇</td>
<td>[W/m²]</td>
<td>Heat flux</td>
</tr>
<tr>
<td>Re</td>
<td>[-]</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>T</td>
<td>[K]</td>
<td>Thermodynamic temperature</td>
</tr>
<tr>
<td>t</td>
<td>[s]</td>
<td>Time</td>
</tr>
</tbody>
</table>
$u \ [\text{m/s}]$ Velocity
$V \ [\text{m}^3]$ Volume
$v \ [\text{m}^3/\text{kg}]$ Specific volume
$\dot{V} \ [\text{m}^3/\text{s}]$ Volume flow rate

**Greek letters**

$\alpha \ [\text{W}/(\text{m}^2\cdot\text{K})]$ Heat transfer coefficient
$\alpha^* \ [-]$ Absorptivity
$\beta \ [1/\text{K}]$ Volume expansion coefficient
$\delta \ [\text{m}]$ Boundary layer thickness
$\eta \ [\text{kg}/(\text{m}\cdot\text{s})]$ Dynamic viscosity
$\theta \ [\degree\text{C}]$ Temperature
$\lambda \ [\text{W}/(\text{m}\cdot\text{K})]$ Thermal conductivity
$\nu \ [\text{m}^2/\text{s}]$ Kinematic viscosity
$\rho \ [\text{kg}/\text{m}^3]$ Density
$\rho^* \ [-]$ Reflectance
$\sigma \ [\text{W}/(\text{m}^2\cdot\text{K}^4)]$ Stefan-Boltzmann constant
$\tau^* \ [-]$ transmittance
$\varphi \ [-]$ View factor

**Indexes**

amb Ambient
cond Conduction
conv Convection
emit Emitted
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>max</td>
<td>Maximum</td>
</tr>
<tr>
<td>min</td>
<td>Minimum</td>
</tr>
<tr>
<td>rad</td>
<td>Radiation</td>
</tr>
<tr>
<td>S</td>
<td>Surface</td>
</tr>
</tbody>
</table>
List of figures

Figure 2-1: Simplified illustration of forced and natural convection [5] ........................................... 7
Figure 2-2: Development of velocity and thermal boundary layer [6] .................................................. 8
Figure 2-3: Distribution of radiation on a semi-transparent material .................................................... 12
Figure 2-4: View factors from a surface to itself ................................................................................... 14
Figure 3-1: Grid of sub-volumes and mesh of a car cabin ................................................................... 16
Figure 3-2: One flowsplit out of the grid of flowsplits ....................................................................... 17
Figure 3-3: Mapping of the two different discretized geometries ....................................................... 22
Figure 3-4: Communication during the co-simulation ....................................................................... 23
Figure 4-1: Flow around a cube: Model in COOL3D ................................................................. 25
Figure 4-2: Flow around a cube: Different discretization sizes ........................................................... 26
Figure 4-3: Limitation of the flow inside the grid of flowsplits ............................................................. 27
Figure 4-4: Discrete plots for the velocity in x-direction .................................................................... 28
Figure 4-5: Flow around a cube: Velocity in x-direction .................................................................. 29
Figure 4-6: Flow around a cube: Velocity in z-direction .................................................................. 30
Figure 4-7: Flow around a cube: Static pressure distribution .............................................................. 31
Figure 5-1: Box model in TAITherm .............................................................................................. 34
Figure 5-2: Solar radiation distribution on top of the box ................................................................. 37
Figure 5-3: View factor distribution on top of the box .................................................................... 37
Figure 5-4: Box model in COOL3D ............................................................................................ 40
Figure 5-5: Front and right view of the box model in COOL3D .......................................................... 40
Figure 5-6: Grid of flowsplits for the box model: Cut through the x-z-plane ................................ 41
Figure 5-7: Box model in GT-SUITE ............................................................................................ 43
Figure 5-8: Surface temperature of the back of the box model .......................................................... 44
Figure 5-9: Step profiles for the box model ....................................................................................... 46
Figure 5-10: Volume averaged temperature of the box model .......................................................... 47
Figure 5-11: Cabin model in TAITherm .......................................................................................... 49
Figure 5-12: Simplified cabin geometry in GT-Spaceclaim ................................................................. 51
Figure 5-13: Front and top view of the cabin model in COOL3D ...................................................... 52
Figure 5-14: Right view of the cabin model in COOL3D ................................................................. 52
Figure 5-15: Cut through the cabin in the x-z-plane ......................................................................... 54
Figure 5-16: Simulation model of the cabin in GT-SUITE ................................................................. 55
Figure 5-17: Overall work flow for the cabin model building ............................................................... 56
Figure 5-18: Outlet temperature of the cabin model ........................................................................... 57
Figure 5-19: Heat transfer to fluid of the cabin model ......................................................................... 58
Figure 6-1: Velocity distribution of the box model ............................................................................ 61
Figure 6-2: Temperature and pressure distribution of the box model .............................................. 63
Figure 6-3: Results of the temperature sensors of the box model (1) ....................... 65
Figure 6-4: Results of the temperature sensors of the box model (2) ....................... 66
Figure 6-5: Results of the temperature sensors of the box model (3) ....................... 67
Figure 6-6: Results of the temperature sensors of the box model (4) ....................... 68
Figure 6-7: Investigated discretization sizes of the box model ................................ 69
Figure 6-8: Results of the temperature sensors for the discretization study .......... 72
Figure 6-9: Temperature distribution inside the cabin after the hot-soak ............... 74
Figure 6-10: Temperature distribution inside the cabin after the cool-down ......... 76
Figure 6-11: Results of the temperature sensors of the cabin model (1) .......... 78
Figure 6-12: Results of the temperature sensors of the cabin model (2) .......... 79
Figure 6-13: Results of the temperature sensors of the cabin model (3) ........ 80
Figure 6-14: Velocity distribution inside the cabin (y = -0.45 m) ..................... 81
Figure 6-15: Velocity in x-direction for cuts through the cabin in the x-y-plane .... 82
Figure 6-16: Velocity in y-direction for cuts through the cabin in the x-y-plane .... 83
Figure 6-17: Velocity in z-direction for cuts through the cabin in the x-z-plane .... 84
Figure A-0-1: Results of the temperature sensors of the cabin model (4) ........ 91
Figure A-0-2: Results of the temperature sensors of the cabin model (5) ........ 92
Figure A-0-3: Results of the temperature sensors of the cabin model (6) ........ 93
Figure A-0-4: Results of the temperature sensors of the cabin model (7) ........ 94
List of tables

Table 5-1: Overview of investigated setups for the box model ........................................44
Table 6-1: Calculation times for the discretization study of the box model ..................71
Table 6-2: Final calculation times of the cabin simulation...........................................77
Table B-0-1: Layer structure of the parts of the box model........................................95
Table B-0-2: Layer structure of the parts of the cabin model....................................96
Table B-0-3: Material properties from Daimler.......................................................97
Table B-0-4: Boundary and initial conditions of the box model simulation.................98
Table B-0-5: Boundary and initial conditions of the hot-soak simulation ..................98
Table B-0-6: Boundary and initial conditions of the cool-down simulation ..............99
Abstract

Due to the higher requirements regarding the thermal comfort in passenger car cabins, the investigations of the thermal comfort have to be shifted forward in the car development process. Normally these investigations are performed experimentally in a climate wind tunnel by using prototypes. Because in the early stages of the car development process no prototypes are available, it is necessary to do basic investigation regarding the thermal comfort through cabin simulations in the early stages of the car development process. Furthermore, the designer can early minimize the energy consumption of the HVAC system while maximizing the thermal comfort inside the cabin by using a detailed cabin simulation model. ThermoAnalytics and Gamma Technologies developed a new method of cabin modeling as a co-simulation between the two simulation tools TAITherm and GT-SUITE. In this co-simulation process, TAITherm is used to model the walls of the cabin and GT-SUITE is used to solve the flow inside the cabin. To solve the flow inside the cabin, the air volume is divided into thousands of sub-volumes. This new method of cabin modeling is tested and evaluated in this work. The limitations of the flow solution in GT-SUITE are pointed out and afterwards the model building process in both simulation tools is described in detail. The cabin model is used to simulate a pull-down, where the air inside the cabin is first heated up due to solar radiation and is afterwards cooled down by using the air-conditioning system. The simulation results of the new cabin modeling method are discussed and compared to STAR-CCM+ simulation results in this elaboration.
1 Introduction

In comparison to the climatization of buildings, the climatization of a passenger car cabin is more complex. As described in [1], in a passenger car cabin the air and surface temperatures are within a much larger range and the air volume is much smaller. Hence, the air changing rate has to be much higher. Furthermore, [1] says that the temperature-, flow- and radiation field are mostly inhomogeneous in a passenger car cabin. The shape of the car cabin has a big influence, too. Due to aerodynamic optimizations, modern cars have a tilted windshield and rear window which supports the invasion of solar radiation into the car cabin. A comfortable temperature inside a passenger car cabin is not only related to comfort reason but to safety reasons as well. Especially the concentration of the driver can be increased harshly by creating a comfortable temperature in the car cabin. According to [1], a temperature increase of 25 to 35 °C leads to a decrease of 20 % regarding the ability to concentrate and combine. Thus, optimizing the comfort inside a passenger car cabin is a very important task in the automotive industry.

The requirements for the car climatization are determined in lots of standards and regulations. One very important regulation is the DIN 1946 - Part 3: Climatization of passenger cars and trucks. One test scenario out of this regulation is the so called pull-down test. The pull-down test consists of a hot-soak phase, where the car stands in the sun for 60 minutes with a constant solar radiation of 1000 W/m² acting vertically on the top of the car. As described in [2], the average air temperature in the cabin can reach very high values of around 55 to 65 °C during the hot-soak phase. The hot-soak phase is followed by a 60-minute cool-down phase, where the car drives with a constant velocity of 32 km/h. During the cool-down phase, the air-conditioning system is operating with maximum power and the lowest cabin inlet temperature. According to DIN 1946 – Part 3, the average temperature in the head region of a passenger car cabin has to be less than 25 °C after 30 minutes.

Normally climate wind tunnel tests are used to make statements about the thermal comfort inside a passenger car cabin. However, climate wind tunnel tests are very time intense and expensive [3]. Furthermore, for the tests, prototypes of the car are needed. Usually, prototypes are first available in the final stages of the car development process. At this point of time, it is already expensive and difficult to change something in the setup of the car [3].

A simulation of a passenger car cabin can already be performed in the early stages of the car development process. Thus, the optimization process is shifted forward in the
car development process. With a detailed cabin model, the designer can minimize the energy consumption of the HVAC system while maximizing the passenger comfort inside the cabin.

In respect to the resolution of the air volume inside the cabin model, the current car cabin simulation methods can be divided into the following three main categories:

- Single-Volume models,
- Multi-Volumes models and
- CFD models (millions of volumes).

For the single volume models, the car cabin is modeled as a single volume. Hence, there is only one inlet and one outlet temperature. They are mainly used in complete vehicle simulations to model loads at the HVAC system. Integrated into a complete vehicle simulation, these models are not increasing the calculation times. Further, they are easy built. Single volume models have a good accuracy regarding fuel economy calculations. Because they are only consisting of a single volume, it is impossible to make statements about the zonal thermal comfort in a car cabin with them.

The multi volumes models are consisting of two to 20 volumes. Like the single volume models, they can be used to model loads at the HVAC system in a complete vehicle simulation without increasing the calculation time of the model. The advantage of these models in comparison to a single volume models is that, the air volume inside the cabin is separated into different zones. Hence, some first statements about the temperature distribution inside the cabin can be made. Further for the different zones, different boundary conditions and initialization temperatures can be used. The accuracy of these models regarding the thermal comfort inside the cabin is still not very high but some first statements can be made.

In CFD models, the air volume inside the cabin is divided into millions of volumes. The flow and temperature distribution inside the cabin can be reproduced very detailed and accurate, hence CFD simulations have a high accuracy regarding the thermal comfort. Because the flow inside the cabin is reproduced so detailed, the complexity of the process is very high. A detailed physical background knowledge is needed to understand the flow phenomena within a car cabin. Due to the very high number of volumes, CFD models have very high calculation times. Thus, it is very difficult to use them in a complete vehicle simulation.

The new cabin modeling method, which is tested in this work, should close the big gap between the cabin models with a small number of volumes and the cabin models with millions of volumes. For this method, the air volume inside the cabin is divided into 1000 to 10000 volumes. Due to the division into many volumes, more detailed
statements about the thermal comfort inside the car cabin can be made. The goal is to optimize the calculation time of the cabin model to still be able to use it in complete vehicle simulations. The complexity of the model building process should be as low as possible.

For this new method of cabin modeling, the two simulation tools GT-SUITE and TAITherm are coupled and used as a co-simulation. GT-SUITE is a 1D flow computation program which can quickly solve the fluid domain inside the car cabin. For this calculation, GT-SUITE needs thermal wall boundary conditions. TAITherm is a 3D thermal and infrared signature modeling tool, which offers the ability to quickly solve temperatures for 3D structures including 3D-conduction, convection and multi-bounce radiation. For the solution, TAITherm needs convective boundary conditions. Coupling the two simulation tools provides a complete solution for the cabin modeling by combining the advantages of both simulation tools. GT-SUITE provides the convective boundary conditions for the energy calculation in TAITherm and TAITherm provides the thermal wall boundary conditions for the flow calculation in GT-SUITE. In this work TAITherm version 12.2a-2016-15 is used. For GT-SUITE, the version GT-SUITE v2017 build1 is used. All simulations were run on one core of a laptop with an Intel® Core™ i7 – 2720 QM CPU @ 2.20 GHz processor and a RAM of 16 GB.

The main goal of this work is to test and evaluate the new cabin modeling method. For this, the model building process in both simulation tools, TAITherm and GT-SUITE, is analyzed in detail. The limitations of the flow solution in GT-SUITE and their influence on the simulation results are investigated. Furthermore, the simulation models are optimized and calibrated by using simulation results of a STAR-CCM+ simulation. Afterwards, the simulation results of the new cabin modeling method are compared to STAR-CCM+ simulation results.

This work is divided into seven chapters. Chapter 1 includes a general introduction and the motivation of this work. The basics of the heat transfer mechanism conduction, convection and radiation are explained in chapter 2. Further, it is explained where they appear inside a car cabin and what impact they have.

Chapter 3 gives an overview of the theory of the co-simulation methodology for the cabin modeling. The basic equations of the flow solution in GT-SUITE and the basics of the energy solution in TAITherm are explained. The variables which are exchanged during the co-simulation are explained, too. Furthermore, the mapping of the two different discretized representations is described. The third chapter ends with an explanation of the communication between GT-SUITE and TAITherm during the co-simulation.
In chapter 4, the flow solution of the new cabin modeling method is analyzed and compared to the flow solution in STAR-CCM+. The limitations of the flow and the reasons for it are pointed out. Afterwards, a simple box model is used to evaluate the impact of the flow limitations on the temperature distribution.

Chapter 5 starts with a description of the model building of the simple box model in TAITherm. It continues with a description of the model building in the pre-processing program COOL3D and the model setup in GT-SUITE. Afterwards, the optimization and calibration of the simple box model is described. The new cabin modeling method is tested with a detailed cabin geometry as well. The building process of the cabin model in TAITherm and COOL3D by using the solid cabin geometry is described. Chapter 5 closes with the setup of the cabin model in GT-SUITE and the calibration of it.

The simple box model is used to simulate a cool-down phase only. With the cabin model a complete pull-down test is simulated. The results of these simulations are discussed and compared to results of a STAR-CCM+ simulation in chapter 6. The main reasons for the differences of the trend of the temperatures are pointed out and explained by taking a closer look at the velocity and pressure distribution inside the box and cabin. For the simple box model a discretization study is done as well.

Chapter 7 starts with a critical view at the model building process and shows which improvements are necessary. Furthermore, the conclusion from the simulation results is drawn. The thesis ends with a look in the future of the new cabin modeling method and a description which work should be done to further test and improve this new method.
2 Theory of heat transfer

As described in [4], the thermal analysis of a passenger car cabin not only involves geometric complexity but also strong interactions between the airflow and the three main forms of heat transfer, namely

- conduction,
- convection and
- thermal radiation.

Hence it is necessary to have a basic knowledge about the formation mechanism, the emergence in a passenger car cabin and the basic equations for the calculation of these three forms of heat transfer.

According to [5], conduction is the transfer of energy from more energetic particles of a substance to the adjacent less energetic particles as a result of interactions between the particles. Convection is the heat transfer between a solid surface and an adjacent liquid or gas in motion. Thermal radiation is the energy emitted by matter in form of electromagnetic waves as a result of changes in the electronic configurations of the atoms or molecules.

2.1 Conduction

According to [6], thermal conduction can occur in gases, liquids and solids. In gases and liquids, thermal conduction appears due to collision and diffusion of the molecules during their random motion. In solids, the formation mechanisms are a combination of energy transport of free electrons and vibrations of the molecules. Requirement for thermal conduction is a temperature gradient inside the material. If a local temperature gradient $\frac{\partial T}{\partial x}$ exists in x-direction the heat flux

$$\dot{q}_{\text{cond}} = -\lambda \frac{\partial T}{\partial x}$$

(2.01)

for heat conduction depends on the thermal conductivity $\lambda$ only [7]. This equation is known as the Fourier’s law for one-dimensional heat conduction.

Because heat is always conducted in the direction of decreasing temperature, the temperature gradient is negative when heat is conducted in the positive x-direction.
The negative sign in equation (2.01) ensures that heat transfer in positive x-direction is a positive quantity. If the thermal conductivity of a material is the same in all three directions the Fourier’s law can be written in vectorial style as

\[ \vec{Q}_{\text{cond}} = -\lambda A \text{grad} (T), \]

where

\[ \text{grad} (T) = \begin{bmatrix} \frac{\partial T}{\partial x} \\ \frac{\partial T}{\partial y} \\ \frac{\partial T}{\partial z} \end{bmatrix} \]

(2.03)

is the temperature gradient in x-, y- and z-direction [7]. The thermal conductivity mainly depends on the material but also on the temperature. A is the heat transfer area and depends on the shape of the body through which the heat is transferred.

According to [7], solids normally have high thermal conductivities

\[ 1 \text{ W/mK} \leq \lambda_{\text{solid}} \leq 450 \text{ W/mK}, \]

whereas gases have small thermal conductivities

\[ 0.015 \text{ W/mK} \leq \lambda_{\text{gas}} \leq 0.15 \text{ W/mK}. \]

For the calculation of heat conduction, it is necessary to separate between steady-state conduction and transient conduction. For transient conduction, the temperature is not only a function of the location but also a function of time. Additionally, the geometry of the material in which heat conduction occurs and the number of layers has to be considered. A more detailed description of the heat conduction and the calculation equations can be found in [5] and [7].

For the modeling of a passenger car cabin the heat conduction plays an important role for the heat transfer through the cabin body, caused by the temperature difference between the ambient air on the outside and the interior air inside the car cabin.

### 2.2 Convection

As can be read in [7], for convection a distinction between forced convection and natural (free) convection is made. If the fluid motion is caused by external forces like fans or pumps, the convection is called forced convection. For natural convection, the fluid motion is caused by natural means like differences of density caused by a
temperature gradient. According to [5], the warm fluid will rise and the colder fluid falls down (buoyancy effect). Furthermore, convection can be separated into external or internal convection, depending on whether the fluid is forced to flow over a surface or inside a channel [5]. Figure 2-1 shows an illustration of forced and natural convection on the surface of a hot plate. The arrows are showing the air motion.

![Diagram of forced and natural convection](image)

(a) Forced convection  
(b) Natural convection

Figure 2-1: Simplified illustration of forced and natural convection [5]

### 2.2.1 Forced convection

As mentioned in [7], through the movement of the fluid a transport of enthalpy and kinematic energy takes place during convective heat transfer. The heat transfer is no longer only dependent on the material properties of the fluid, but also dependent on process parameters like the velocity and turbulence of the fluid motion.

Due to friction caused by viscosity, the layer closest to the plate will have a velocity of zero and fluid particles next to the plate will be slowed down [6]. The layer where the friction affects the velocity profile is called velocity boundary layer. The formation of this velocity boundary layer for air streaming over a heated plate is illustrated in Figure 2-2 (a). The velocity of the particles directly on the surface of the plate will be zero. The blue lines in Figure 2-2 (a) are showing the velocity profile inside the boundary layer. When the fluid moves on in the x-direction, the friction has an effect further and further in y-direction. With increasing distance in y-direction the velocity approaches to the velocity of the undisturbed, free inflow.

Figure 2-2 (a) only shows the formation of the laminar velocity boundary layer. After a defined distance the laminar flow will turn into a turbulent flow. According to [3], the point where the laminar turns into a turbulent flow is called transition point. Whereas the laminar boundary layer is a layered flow without any transversal movement, inside the turbulent boundary layer, erratic transverse and longitudinal movements superpose the center flow [3]. A detailed description of the formation of the velocity boundary layer
and the differences between a laminar and turbulent boundary layer can be found in [5], [7] and [3].

In about the same way like the velocity boundary layer, a thermal boundary layer will develop when a cold fluid is streaming over a hot plate [6]. The formation of the thermal boundary layer for a cold fluid streaming over a hot plate is illustrated in Figure 2-2 (b). The fluid has a constant temperature $T_0$, the wall surface temperature of the hot plate is $T_S$. According to [5], the fluid particles in the layer adjacent to the surface will reach thermal equilibrium with the plate and assume the surface temperature $T_S$. These fluid particles then exchange energy with the particles in the adjoining fluid layer. As a result, a temperature profile like the green lines in Figure 2-2 (b) develops inside the flow. The temperature ranges from $T_S$ at the surface to $T_0$ sufficiently far from the surface outside the thermal boundary layer. Due to the described mechanism for increasing $x$, the temperature will continue to spread in $y$-direction. Hence the thickness of the thermal boundary layer increases in flow direction. A detailed description of the formation of the thermal boundary layer can be found in [5] and [7].

![Diagram of velocity and thermal boundary layers](image)

**Figure 2-2: Development of velocity and thermal boundary layer [6]**

As described in [5], the velocity and thermal boundary layers are development simultaneously inside a flow over a heated or cooled surface. The fluid velocity has a strong influence on the temperature profile, hence the development of the velocity boundary layer relative to the thermal boundary layer has a strong effect on the convective heat transfer [5]. The relative thickness of the thermal and velocity boundary layer is described by the dimensionless Prandtl number

$$Pr = \frac{\text{Molecular diffusivity of momentum}}{\text{Molecular diffusivity of heat}} = \frac{\nu}{\lambda} = \frac{\eta c_p}{\lambda},$$  

(2.04)

where $\lambda$ is the thermal diffusivity and $\nu$ the kinematic viscosity [5]. It is named after Ludwig Prandtl, who introduced the concept of boundary layer in 1904. A small Prandtl number means that heat diffuses very quickly, a high Prandtl number means heat diffuses very slowly relative to momentum. Hence for the first case the thermal
boundary layer is much thicker and for the second case much thinner relative to the velocity boundary layer [5].

As described in [5], the characteristics of the flow during forced convection are described with the dimensionless Reynolds number

\[
\text{Re} = \frac{\text{Inertia forces}}{\text{Viscous forces}} = \frac{uL}{\nu},
\]

where

- \( u \) = velocity of the fluid outside the boundary layer [m/s]
- \( L \) = characteristic length of the geometry [m]
- \( \nu \) = kinematic viscosity of the fluid [m²/s]

Small Reynolds numbers are meaning that the viscous forces are stronger than the inertia forces. An organized flow occurs, which is typical for a laminar flow. For large Reynolds numbers the inertia forces are larger than the viscous ones, thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid [5]. This is characteristic for a turbulent flow. The point where the laminar turns into turbulent flow is defined by the critical Reynolds number, which depends on the geometry. According to [6], the viscosity is greatly dependent on temperature and on the state of the flowing fluid (gas or liquid).

To calculate the amount of heat transfer through convection, Newton’s law of cooling

\[
\dot{Q}_{\text{conv}} = \alpha A (T_s - T_0)
\]

is used, where \((T_s - T_0)\) is the temperature difference, \(A\) is the heat transfer area and \(\alpha\) the heat transfer coefficient [7]. The heat transfer coefficient \(\alpha\) describes how much heat is transferred between a surface and a fluid per unit surface area per unit temperature difference. According to [5], the heat transfer coefficient depends on fluid properties like

- dynamic viscosity \(\eta\),
- thermal conductivity \(\lambda\),
- density \(\rho\) and
- specific heat \(c_p\),

but also on

- fluid velocity,
Theory of heat transfer

- geometry,
- roughness of the surface and
- the type of fluid flow (laminar or turbulent).

It is possible to calculate the heat transfer coefficient directly out of the temperature profile next to the surface. However, in most cases this temperature profile is unknown. Therefore, normally the Nusselt number

\[ \text{Nu} = \frac{\alpha L}{\lambda} \]  

is used to calculate the heat transfer coefficient [5]. Depending on the geometry, the type of heat transfer problem, the type of flow, etc. there are different correlations to calculate the Nusselt number. These correlations can be found in [5] or [7]. For forced convection the Nusselt number is a function of the Reynolds and Prandtl number. As guiding values of the heat transfer coefficient for forced convection in gases, the VDI-Heat Atlas [7] names a region from 25 to 250 (W/(m²*K)).

In the cabin modeling forced convection occurs when the air is streaming inside the cabin in order to cool it down or heat it up. When the car is driving, air is streaming over the body on the outside, which leads to forced convection, too.

### 2.2.2 Natural convection

For natural convection, also called free convection, the internal forces of the fluid are leading to the movement of the fluid, like illustrated in Figure 2-1 (b). There is no external force acting on the fluid. The velocities inside the fluid are caused by density differences - temperature differences - which are leading to buoyancy forces.

To describe the characteristics of the flow during natural convection the Grashof number is used. According to [7], it can be calculated like

\[ \text{Gr} = \frac{\text{buoyancy forces}}{\text{viscous forces}} = \frac{L^3 g \beta \Delta T}{\nu^2} \]  

where

\[ L = \text{characteristic length of the geometry [m]} \]
\[ g = \text{gravitational acceleration [m/s}^2] \]
\[ \beta = \text{coefficient of volume expansion [1/K]} \]
\[ \Delta T = (T_S - T_0) = \text{temperature difference between the surface temperature} \]

\[ T_S \text{ and the temperature of the fluid } T_0 \text{ sufficiently far from the surface [K]} \]

\[ \nu = \text{kinematic viscosity of the fluid [m}^2/\text{s]} \]

The Grashof number represents the ratio of buoyancy forces to the viscous forces acting on the fluid [5]. The coefficient of volume expansion quantifies how much the density varies due to temperature changes. According to the Reynolds number, the Grashof number is used to differentiate between laminar and turbulent flow for natural convection [6]. Therefore, the Nusselt number for natural convection is a function of the Grashof number and the Prandtl number. In comparison to forced convection, the fluid velocities for natural convection are much smaller. As a result of this, the heat transfer coefficient and thus the amount of heat transfer is much smaller, too. As guiding values for the heat transfer coefficient for natural convection in gases, the VDI-Heat Atlas [7] names a region from 2 to 25 (W/(m²*K)).

During a hot-soak, the car is standing in the sun and heats up without any forced flow inside the cabin. The solar radiation heats up the interior of the cabin, especially the parts directly behind the windows like the dashboard and the rear shelf. This leads to temperature differences between the part surfaces and the air inside the cabin. As described in [2], due to natural convection, the air inside the cabin will heat up and flow around caused by the explained buoyancy effect. If the car is standing during the simulation, there is also natural convection at the outside of the cabin.

## 2.3 Radiation

According to [5], radiation is the energy emitted by a body in form of electromagnetic waves as a result of changes in the electronic configurations of the atoms or molecules. Unlike conduction and convection, the energy transfer of radiation doesn’t need the presence of an intervening medium because electromagnetic waves can be transferred in vacuum as well. As described in [5], for the heat transfer, the thermal radiation plays an important role where bodies emit radiation because of their temperature. Each body with a temperature over zero kelvin emits radiation [7]. The maximum emitted radiation of a surface of a body with a temperature \( T \) is given by the Stefan-Boltzmann law [5] as

\[
\dot{Q}_{\text{emit,max}} = \sigma A_S T^4 ,
\]

where \( \sigma = 5.67 \times 10^{-8} \text{ (W/(m}^2/\text{K}^4)} \) is the Stefan-Boltzmann constant and \( A_S \) the surface area of the body. A body which emits the maximum radiation is called black body [7].
Figure 2-3 shows what happens to incident thermal radiation at the surface of a body. It can either be absorbed by the body or reflected by the surface. If the material is transparent, a part of the incident radiation also can be transmitted through the body. The distribution of the incident radiation for a material is defined over the three material properties reflectance $\rho^*$, absorptivity $\alpha^*$ and the transmittance $\tau^*$. According to [7], applies

$$0 \leq \alpha^* \leq 1; \; 0 \leq \rho^* \leq 1; \; 0 \leq \tau^* \leq 1$$

and

$$\alpha^* + \rho^* + \tau^* = 1.$$  \hspace{1cm} (2.10)

For a black body $\alpha^* = 1$, hence $\rho^* = \tau^* = 0$. This means for a black body all absorbed energy is emitted, hence he is a perfect absorber and emitter [5].

![Diagram of radiation distribution](image)

**Figure 2-3:** Distribution of radiation on a semi-transparent material

The name black body doesn´t refer to the color, but rather to his behavior of absorbing all incident radiation and emit it again. A black body radiates diffuse which means equal in each direction. As can be seen in equation (2.09), the energy emitted by a black body only depends on his absolute temperature.

The radiation emitted by real bodies in comparison to black bodies at the same temperature is expressed as

$$\dot{Q}_{\text{emit}} = \varepsilon^* \sigma A_S T^4,$$ \hspace{1cm} (2.11)

where $\varepsilon^*$ is the emissivity of the real body [5]. It has a range $0 \leq \varepsilon^* \leq 1$, whereas $\varepsilon^* = 1$ for a black body.
Both, the emissivity $\varepsilon^*$ and absorptivity $\alpha^*$ of a surface are depending on the temperature and the wavelength of the radiation [5]. For technical applications, the Kirchhoff’s law of radiation is used, where the emissivity and absorptivity of a surface at a given temperature and wavelength are equal [5].

In general, the calculation of the heat transfer between two parts caused by radiation is very complicated since it depends on the properties of the surface, their orientation relative to each other and the interaction of the medium between the surfaces with radiation [5]. The radiation of a part with a temperature $T_S$ and a surface area $A_S$ with the ambient surrounding can be calculated as

$$\dot{Q}_{rad} = \varepsilon^* \sigma A_S (T_S^4 - T_{amb}^4),$$

(2.12)

if the surface of the part is completely enclosed by a much larger or black surface with a temperature $T_{amb}$ and the separating gas - such as air - doesn’t intervene with radiation [5].

To consider the effects of orientation of two surfaces relative to each other on the radiation heat transfer, the view factor $\varphi$ is used. It only depends on the geometry and is independent of the surface properties or temperature. The view factor $\varphi_{ij}$ from a surface $i$ to a surface $j$ is defined as the fraction of radiation leaving surface $i$ that strikes surface $j$ directly [5]. For simple arrangements of two geometries, the view factor can be determined by analytic calculation. Otherwise a numerical calculation - like ray tracing - or experiments are needed to determine the view factor [7]. The equations to calculate the view factor can be found in [5] or [7].

There are two important relations for the view factors. First the summation rule

$$\sum_{j=1}^{N} \varphi_{ij} = 1,$$

(2.13)

which means that the sum of view factors from a surface $i$ of an enclosure to all surface of the enclosure - including to itself - must be one [5]. This rule is a result of the conservation of energy principle, which requires that the entire radiation leaving any surface of an enclosure must be intercepted by the surfaces of the enclosure.

As a result of the numerical definition of the view factor a second relation, the reciprocity relation

$$\varphi_{12} A_1 = \varphi_{21} A_2$$

(2.14)

occurs which allows the calculation of a view factor out of the knowledge of the other [5].
When \( i = j \) the view factor occurs, which determines the fraction of radiation leaving a surface and that strikes itself directly. Figure 2-4 shows how rays send out from a surface behave for different shapes of a geometry. For a plane or convex surface no rays which are sent out from the surface will fall back to itself, hence \( \phi_{11} = 0 \). For a concave surface, some rays will directly fall on the geometry itself again, hence \( \phi_{11} \neq 0 \).

![Diagram showing view factors for different shapes]

(a) Plane surface \( \phi_{11} = 0 \)  
(b) Convex surface \( \phi_{11} = 0 \)  
(c) Concave surface \( \phi_{11} \neq 0 \)

Figure 2-4: View factors from a surface to itself

One method to calculate the radiation between two parts is the net radiation method. According to [5], the net radiation heat transfer is the difference between the rates of radiation emitted by the surface and absorbed of it. The net radiation is positive if the rate of radiation absorption is greater than the rate of radiation emission. This means the surface gets energy from radiation. For negative net radiation, the surface is losing energy by radiation.

According to [7], the radiation heat transfer between two random arranged surfaces with the temperature \( T_1 \) and \( T_2 \)

\[
\dot{Q}_{\text{rad}} = \frac{\varepsilon_1 \varepsilon_2 \sigma (A_1 \phi_{12} T_1^4 - A_2 \phi_{21} T_2^4)}{[1 - (1 - \varepsilon_2)\phi_{22}][1 - (1 - \varepsilon_1)\phi_{11}]} - (1 - \varepsilon_1)(1 - \varepsilon_2)\phi_{21}\phi_{12} \tag{2.15}
\]

can be calculated with the emissivity of both surfaces \( \varepsilon_1 \) and \( \varepsilon_2 \), the surface areas \( A_1 \) and \( A_2 \) and the view factors \( \phi_{ij} \).

When both surfaces are non-concave surfaces \( (\phi_{11} = \phi_{22} = 0) \) and under the help of the reciprocity relation - equation (2.14) -, equation (2.15) can be simplified to

\[
\dot{Q}_{\text{rad}} = \frac{\varepsilon_1 \varepsilon_2 \sigma A_1}{1 - (1 - \varepsilon_1)(1 - \varepsilon_2)\phi_{21}\phi_{12}} (T_1^4 - T_2^4) \tag{2.16}
\]

For radiation between more than two surfaces complicated numerical equations are needed to calculate the view factors and the radiation heat transfer. The derivation and explanation of these equations is discussed in detail in [5] and [7].

Solar radiation is a special case of the radiation for high temperatures and is divided into direct and diffuse solar radiation. Diffuse solar radiation is caused by the fact, that
on the way from the sun to the earth a part of the solar rays is absorbed or reflected on air, water or dust molecules in the atmosphere. The intensity of the solar radiation depends on the weather - cloudy, clear sky, etc. - and the angle between the sun and earth, hence the season and daytime [8].

For the cabin modeling, the radiation heat transfer of parts inside the cabin plays a subordinate role, especially when there is a forced flow inside the cabin. Whereas the solar radiation has a big impact on the air temperature inside the cabin, especially when the car is standing in the sun and there is no forced flow inside the cabin. The surfaces and the air inside a car cabin gets really hot, when the car is standing in the sun for a while during a hot summer day with a clear sky. According to [2], the average air temperature inside a passenger car cabin can easily increase around 20 to 25 Kelvin. Thereby, values around 70 to 75 °C for the maximum air temperature can be reached in the head region of the cabin. Further, the average air temperature inside the car cabin can reach values between 60 to 65 °C [2]. For the maximum surface temperatures, temperatures around 90 °C or even higher can be reached. Especially at the surfaces directly behind the windows, like the dashboard or rear shelf.
3 Theory of the Co – Simulation

To understand the work flow of the co-simulation between TAITherm and GT-SUITE, it is necessary to know some basics of the calculation methods in both programs and which parameters are exchanged. Both, GT-SUITE and TAITherm, are containing a discretized representation of the cabin geometry. The solid outer boundaries of the cabin – walls – are represented by TAITherm, the flow volume inside the cabin is represented by GT-SUITE. Mainly GT-SUITE is solving the flow inside the cabin, TAITherm is calculating the wall temperatures. Because GT-SUITE and TAITherm are solving a different kind of problem, they are working with different discretized representations.

For the calculation in GT-SUITE, the volume inside the cabin is divided into many sub-volumes. These sub-volumes are connected at the boundaries. Hence GT-SUITE works with a grid of sub-volumes as it can be seen in the picture on the left in Figure 3-1. Each colored cube is representing one sub-volume. TAITherm works with a simulation mesh which is included in the imported cabin geometry. It mainly consists of triangle and square shaped elements. The right picture in Figure 3-1 shows a car cabin with a mesh which is used in TAITherm. Because the two discretized representations have a different kind of layout, they have to be mapped on each other before the co-simulation. This process is explained in chapter 3.3. As described in the chapter 1, the coupling is used to combine the advantages of both programs. At defined points of data exchange, GT-SUITE sends the convective boundary conditions, namely the heat transfer coefficient, the fluid temperature and the humidity (if modeled), to TAITherm. At the same time, TAITherm sends the thermal wall boundary conditions, namely the wall temperature to GT-SUITE. The heat transfer through conduction and thermal radiation is considered over the wall temperatures in TAITherm.

Figure 3-1: Grid of sub-volumes and mesh of a car cabin
3.1 Flow solution in GT – SUITE

For the discretization and conversion of 3D geometries, the pre-processing program COOL3D is used. In COOL3D it is possible to build or import 3D geometries and afterwards export them to GT-SUITE for the simulation. During the export, the 3D geometry is automatically converted into a 1D simulation model. Before the export, the 3D geometry is discretized into many sub-volumes. The size of each sub-volume is determined by the chosen discretization length along the x-, y- and z-direction. For the new cabin modeling method, it is usual to use the same discretization length in all three directions. Hence the cabin geometry is divided into cube shaped sub-volumes. During the export of the geometry to GT-SUITE, each sub-volume is converted into a flowsplit. Flowsplits are used in GT-SUITE to describe a flow volume of any shape which is connected to one or more flow components. They can have an arbitrary number of inlets and outlets at arbitrary angles. The inlets and outlets of a flowsplit are automatically created during the export. Flowsplits are connected at the boundaries. As a result, COOL3D generates a 3D staggered grid of cube shaped flowsplits during the export of a 3D geometry to GT-SUITE.

For the calculation of the flow, GT-SUITE solves the Navier-Stokes equations in one dimension. This means that all quantities are averaged across the flow direction. Figure 3-2 shows one cube shaped flowsplit. The scalar variables like the pressure, temperature, density, enthalpy, etc. are assumed to be uniform over each flowsplit. The vector variables, like the mass flow rate, velocity, etc. are calculated for each boundary of a flowsplit. The dashed surface in Figure 3-2 represents one of these boundaries. The arrows are showing the possible flow directions. Since GT-SUITE solves the 1D momentum equations in the x-, y- and z-direction, the resulting flow field is a Quasi-3D flow field. The accuracy and limitations of this Quasi-3D flow field are further investigated in chapter 4.

Figure 3-2: One flowsplit out of the grid of flowsplits
In this work the implicit solution is used as time integration method, which means that the primary solution variables in GT-SUITE are mass flow, pressure and total enthalpy. According to [9], the conservation equations solved by GT-SUITE for each flowsplit are the conservation of continuity

\[
\frac{dm}{dt} = \sum_{\text{boundaries}} \dot{m},
\]

solved at the boundaries. In this equation \(m\) is the mass of the flowsplit and

\[
\dot{m} = \rho A u
\]

the boundary mass flow rate into the flowsplit. The boundary mass flow rate can be calculated with the density \(\rho\) of the fluid, the flow area \(A\) and the velocity \(u\) of the fluid.

Besides, the conservation of momentum in \(x\)-direction

\[
\frac{d\dot{m}}{dt} = \frac{dp}{dt} A + \sum_{\text{boundaries}} (\dot{m} u) - 4C_t \frac{\rho u|u|}{2} \frac{dx A}{dx} - K_p \left( \frac{1}{2}\rho u|u| \right) A,
\]

is solved at the boundaries [9]. The most important terms in this equation are

- the discretization length \(dx\) in \(x\)-direction,
- the pressure differential \(dp\) acting across \(dx\),
- the velocity \(u\) in \(x\)-direction at the boundary,
- the time step size \(dt\) and
- the cross-sectional flow area \(A\).

For the equivalent diameter \(D\), the expansion diameter of the flowsplit in \(x\)-direction is used. For the Quasi-3D solution, the conservation of momentum equation (3.03) is solved in the \(y\)- and \(z\)-direction, too.

The third solving equation in GT-SUITE is the energy equation for the implicit solver

\[
\frac{d(\rho H)}{dt} = \sum_{\text{boundaries}} (\dot{m} h) + V \frac{dp}{dt} - \alpha A_s (T_{\text{fluid}} - T_{\text{wall}}),
\]

where \(h = e + pv\) is the total specific enthalpy, \(\alpha\) is the heat transfer coefficient and \(A_s\) is the heat transfer surface area. \(T_{\text{wall}}\) is the wall temperature which GT-SUITE receives from TAITherm, \(T_{\text{fluid}}\) is the fluid temperature which GT-SUITE sends to TAITherm at the points of data exchange during the co-simulation. The energy equation is solved ones for each flowsplit [9].
For the implicit solution, a fixed time step size $dt$ is determined before the simulation. During the simulation, the values of all flowsplits will be solved simultaneously, by iteratively solving a non-linear system of algebraic equations. Because the equations are solved iteratively, it is important that the solution reaches numerical convergence. If the solver doesn’t reach numerical convergence after a defined number of iterations, the solver will start a retry for the non-converging time step with a halved time step size. If the solver still can’t reach numerical convergence it will step forward to the next time step. For the next time step, it will automatically increase the time step size to the determined value again. The variables of the implicit solver like the time step size, the number of iterations, the number of retries, damping factors, etc. can be adjusted in the solver settings in GT-SUITE.

As described in [9], the heat transfer coefficient in GT-SUITE is calculated ones for each flowsplit at every time step with a special form of the Colburn analogy by using the fluid velocity, the thermo-physical properties and the wall surface roughness

$$\alpha = \left(\frac{1}{2}\right) C_f \rho u_{eff} c_p Pr^{-2/3},$$

(3.05)

where

- $C_f = f (Re)$ = fanning friction factor
- $\rho = f (p, T)$ = density of the fluid
- $c_p = f (p, T)$ = specific heat of the fluid
- $Pr = f (p, T)$ = Prandtl number of the fluid
- $u_{eff}$ = effective velocity

The effective velocity $u_{eff}$ is based on a weighting of all face velocities of the flowsplit. Therefore, the heat transfer coefficient is primarily a function of the local velocities at each face of a flowsplit. As this method is approximate it may require some calibration to adjust the magnitude of the heat transfer coefficient to real dimensions. More details of the flow solution in GT-SUITE can be found in the Flow Theory Manual [9].
3.2 Energy solution in TAITherm

As described in chapter 1, TAITherm offers the ability to quickly solve temperatures for 3D structures including 3D-conduction, convection and multi-bounce radiation. For this TAITherm solves the energy equations for the different heat transfer mechanisms. The equations are solved for each thermal node. Thermal nodes are automatically created at the element face centroids and for multi-layer parts between the layer interfaces.

According to [10], TAITherm automatically calculates the view factors for all elements using the ray tracing method before the simulation starts. A detailed description of the ray tracing method can be found [5]. As described in chapter 2.3, view factors are used to consider the effects of the orientation of two surfaces relative to each other on the radiation heat transfer. There are several variables to adjust the accuracy of this calculation. For example, the number of rays sent out of an element can be increased or the element can be divided into more subdivisions. A detailed description of these parameters can be found in the TAITherm User Manual [10]. However, these variables should be used carefully because they lead to a harsh increase of the calculation time.

According to [10], the radiation heat transfer in TAITherm is calculated using the net radiation method. The basics of this method is explained in chapter 2.3. Basically, the radiation heat transfer between two elements is calculated by using equation (2.15) out of chapter 2.3. Nevertheless, the radiation heat transfer equations in TAITherm are more complicated than equation (2.15), due to the fact that one element is not only sending rays to one other element of the mesh, but rather to a high number of other elements.

There are several ways to set up the calculation of the heat transfer coefficient for convective heat transfer in TAITherm. Because the automatic convection type is used at the outside of the car cabin during the co-simulation, this method will be explained in detail. For a description of the other methods please refer to the TAITherm User Manual [10]. According to [10], TAITherm is using flat plate formulas for the automatic convection type. The standard laminar to turbulent transition point is $10^5$. Depending on the conditions like

- Smoothness of the incoming flow and upstream obstructions,
- Boundary layer control,
- Surface roughness,
- Pressure gradient and

other conditions, the transition point can vary [10].
The heat transfer coefficient is computed individually for each element of a part and is a result of mixing forced convection, horizontal plate natural convection and vertical plate natural convection. With these three convection types, an averaged Nusselt number is calculated. The forced convection component of the heat transfer coefficient assumes that the flow is parallel to the element. To be able to calculate the horizontal and vertical plate natural convection, TAITherm first determines the horizontal and vertical orientation of each element using the element normal. Afterwards this orientation is used to calculate an Nusselt number for the horizontal and vertical plate natural convection. As described in [10], these three Nusselt numbers are converted to a single characteristic length base, whereas the characteristic length of the forced convection is used. Afterwards, they are mixed to obtain an overall Nusselt number. Using this overall Nusselt number, the element’s convection heat transfer coefficient in TAITherm is calculated for the automatic library option [10].

3.3 Mapping and communication

Due to the fact that the discretized representations of the geometry in GT-SUITE and TAITherm are different, the two representations have to be mapped on each other for the co-simulation. In TAITherm the mesh of the geometry normally contains elements which are much smaller than the flowsplits in GT-SUITE. Figure 3-3 (a) shows a door with the surface mesh how it is used in TAITherm. The transparent orange cubes are representing the grid of flowsplits in GT-SUITE. In reality the flowsplits are connected at the boundaries and there is no gap in the grid of flowsplits. As can be seen one flowsplit is overlapping with several elements of the mesh, caused by the difference of the resolution of the mesh in TAITherm and the resolution of the grid of flowsplits in GT-SUITE.

Due to the size and shape difference between the flowsplits and the mesh elements, it can happen that some elements are overlapping with more than one flowsplit. Hence a logic is needed which determines with which flowsplit such an element exchanges data during the co-simulation. For this the distance between the centroid of the elements and the flowsplits is used. Figure 3-3 (b) shows a simplified example of the misaligned grids in TAITherm and GT-SUITE. The red squares 1 and 2 are representing two flowsplits of the grid of flowsplits in GT-SUITE. The black triangles are representing some elements of the mesh in TAITherm. They have the wall temperatures $t_1 - t_8$. An element exchanges data with the flowsplit, where there is the smallest distance between the centroid of the element and the centroid of the flowsplit. In Figure 3-3 (b) the pointed lines are showing the distance between the centroid of
the elements and the flowsplits. The elements and flowsplit which are exchanging data have the same color.

The centroids of the elements 1 and 2 are closer to the centroid of flowsplit 1 than to the centroid of flowsplit 2. Hence these elements are exchanging data with flowsplit 1 during the co-simulation. For the wall temperature of flowsplit 1 an area averaged value of the two wall temperatures $t_1$ and $t_2$ is used. The centroids of the elements 3, 4, 5 and 6 are closest to the centroid of flowsplit 2. Hence they are exchanging data with flowsplit 2 during the co-simulation. For the wall temperature of flowsplit 2 an area averaged value of the four wall temperatures $t_3$, $t_4$, $t_5$ and $t_6$ is used. After this method, the two different discretized representations can be mapped on each other for the co-simulation.

![Intersection between the TAITherm mesh and the GT-SUITE flowsplits](image)

(a) Intersection between the TAITherm mesh and the GT-SUITE flowsplits

![Data exchange for the misaligned grids](image)

(b) Data exchange for the misaligned grids

Figure 3-3: Mapping of the two different discretized geometries

It is important to know at which points of time both programs are exchanging data during the co-simulation. Further, it is important to know which variables are used to control the data exchange. The communication interval defines the frequency of data exchange between TAITherm and GT-SUITE during the co-simulation after the time before the first data exchange. The best way to understand the communication between TAITherm and GT-SUITE during the co-simulation is to take a look at timelines of the process in both simulation tools.

Figure 3-4 shows two timelines for the first 30 seconds of the co-simulation. The timeline at the bottom shows the process in GT-SUITE, the one at the top the process in TAITherm. The unfilled squares on the top timeline are representing the time steps
in TAITherm which are determined by the definition of the time step size in TAITherm. The filled squares on the bottom timeline are representing the time steps in GT-SUITE which are determined by the definition of the time step size in GT-SUITE. In this example, the time step size in TAITherm is ten seconds, the time before the first data exchange is five seconds. Besides, the communication interval has a value of ten seconds and the time step size in GT-SUITE is one second.

After the co-simulation is started from GT-SUITE, both programs will calculate their solution for each time step independent from each other, since the point of the first data exchange is reached. This point is determined by the defined time before the first data exchange. The program that reaches this point first has to wait since the second program reaches this point, too. Then the first data between GT-SUITE and TAITherm is exchanged. Afterwards both programs are calculating their solution independent from each other again since the next point of data exchange is reached. The time interval between two points of data exchange is called communication interval and is defined before the co-simulation, too.

![Diagram of time steps and communication intervals](image)

**Figure 3-4: Communication during the co-simulation**

In the example in Figure 3-4, the time before the first data exchange is smaller than the time step size in TAITherm. This leads to the problem, that TAITherm normally starts to calculate the first solution after ten seconds but there should already be a data exchange after five seconds. The same problem appears again after 15, 25 seconds, etc. caused by the definition of the communication interval. Hence it is necessary to automatically add several time steps to TAITherm to get a solution for the exchanged data.
variables at the defined points of data exchange. In Figure 3-4 these automatically added time steps are represented by the red unfilled circles. As can be seen, it is only necessary to add time steps in TAITherm due to the bigger time step size in TAITherm in comparison to GT-SUITE.

Thus, the time before the first data exchange and the communication interval are two important variables for the co-simulation and influencing the number of time steps in both programs. Furthermore, they are influencing the calculation time in both simulation tools and it is necessary to investigate the influence of the two variables on the results before starting with the final simulations.
4 Investigation of the flow in GT – SUITE

In this chapter, the limitations of the flow solution in GT-SUITE, explained in chapter 3.1, are analyzed. For this, the flow around a cube in a defined flow space is investigated and the results are compared to STAR-CCM+ simulation results. For the creation and discretization of the model, the pre-processing tool COOL3D is used. From COOL3D the model is exported to GT-SUITE by using different discretization. Figure 4-1 shows the used model in COOL3D. The flow space has the shape of a channel with a height and width of 25 cm and a length of 50 cm. The cube inside the flow space is highlighted in green in Figure 4-1 and has a size of 5 cm x 5 cm x 5 cm. The flow inlet and outlet have the size of the whole flow cross section. As streaming fluid, dry air is used. As boundary conditions an inlet velocity of 1 m/s and an outlet pressure of 1 bar is used. The inlet temperature is set to the same value as the initial temperature, hence there is no heat transfer inside the model.

Figure 4-1: Flow around a cube: Model in COOL3D

Three different discretization sizes are investigated. Due to the fact, that for the cabin modeling the discretization size in x-, y- and z-direction should be the same, for the investigation of the flow, the same discretization size in x-, y- and z-direction is used. Hence, the model is discretized into cube shaped sub-volumes which are converted into flowsplits during the export of the model to GT-SUITE as explained in chapter 3.1. First the sub-volumes have the same size as the cube in the model, second half the size of the cube and third one fourth of it. Hence, for the first discretization the model is separated into 250, for the second into 2000 and for the third into 16000 sub-volumes. Due to the fact that the sub-volumes which are interacting with the cube...
inside the model are thrown out of the model during the export, for the first discretization one, for the second eight and for the third 32 sub-volumes are thrown out of the model. Figure 4-2 shows the division into the cube shaped sub-volumes for the three discretization sizes, starting with the smallest number of sub-volumes on the left. Each colored cube represents one sub-volume. As explained in chapter 3.1, each sub-volume is converted into a flowsplit during the export of the model.

![Figure 4-2: Flow around a cube: Different discretization sizes](image)

The flow distribution, namely the velocities, are compared to the results of a STAR-CCM+ simulation. The model is simulated in Star-CCM+ with the standard solver settings for a laminar flow using the standard temperature and pressure air dynamic viscosity and a turbulence model. The walls are modeled as isothermal walls, hence there is no heat transfer from the walls to the fluid.

Figure 4-5 shows the velocity in x-direction for the three discretization sizes and the STAR-CCM+ simulation as interpolated contour plots for a cut through the middle of the channel in the x-z-plane. The simulation mesh is included into the plots. For creating the interpolated plots in the post-processing tool GT-POST, the velocities which are calculated in GT SUITE for each boundary surface are shifted to the notes of the cube shaped flowsplits. The plots of the STAR-CCM+ results are created by using the software ParaView. The first thing that can be noticed is the difference regarding the resolution between the mesh in Star-CCM+ and the grid of cube shaped flowsplits in GT SUITE.

Out of the regular arrangement of the cube shaped flowsplits in GT SUITE, a limitation for the flow occurs. In Figure 4-3 this limitation is illustrated in a two-dimensional way. It shows an extract of the grid of cube shaped flowsplits. The black arrow is showing the direction of inflow into one flowsplit. The solid green arrows are showing the direction where the flow can stream. The red dashed arrows are showing the direction in which the flow cannot stream. The flowsplits where the flow can directly stream are
highlighted in green. The ones where it cannot stream, are highlighted in red. As can be seen, the flow cannot directly stream into the flowsplit in the upper and lower left corner. A flow which enters a flowsplit from the back can only stream into the next flowsplit above, underneath, in front, left or right of it but not into the flowsplits which are connected with it at the edges. This limitation occurs in all three directions of the flow. In comparison to this, the flow in STAR-CCM+ doesn’t have such a limitation because the elements of the mesh in STAR-CCM+ are much smaller, have no equal shape and are irregularly arranged. Furthermore, the number of mesh elements is much higher than the number of flowsplits in GT-SUITE and the mesh elements have more boundaries at which they are connected to other elements of the mesh.

![Figure 4-3: Limitation of the flow inside the grid of flowsplits](image)

When looking at Figure 4-5 it looks like that for the first discretization size there is a velocity in x-direction behind the cube, whereas for the other discretization sizes there isn’t. To check the actual calculated velocities in x-direction of each flowsplit, it is recommended to use the discrete plots out of the post-processing program GT-POST. The discrete plots are showing the actual calculated velocities in x-direction of each flowsplit as a cut through the model in the y-z-plane at a defined position along the x-axis. The discrete plots in Figure 4-4 (b), (c) and (d) are showing the actual calculated velocities of each flowsplits in the cutting plane A-A out of Figure 4-4 (a) of the three investigated discretization sizes. As can be seen in Figure 4-4 (b) for the first discretization the actual calculated velocity directly behind the cube is zero, too. Thus, it is recommended to use the discrete plots to illustrate the actual calculated velocities and verify the interpolated contour plots. The interpolated contour plots can show strange results, especially when working with a small number of sub-volumes.

For the velocity in x-direction, the main difference between GT-SUITE and STAR-CCM+ is the place where the highest velocities occur. In GT-SUITE, the highest velocities in x-direction occur at the boundaries of the flow space after the cube. This can be seen in the discrete plots in Figure 4-4 as well. In the STAR-CCM+ simulation, the highest velocities occur at the front edges of the cube, where the flow is deflected.
In the STAR-CCM+ simulation there are negative velocities in x-direction directly behind the cube, whereas in the GT-SUITE simulation no negative velocities occur.

Furthermore, in the STAR-CCM+ simulation, the velocities in x-direction in the trail of the cube are higher. In GT-SUITE it seems like there is a line with a constant velocity of zero in the trail of the cube. For the first discretization, the velocity in x-direction in the flow path of the cube is zero till the end of the flow space. For the second discretization, values of around 0.033 m/s are reached for the velocity in x-direction in the path of the cube at the end of the flow space. For the third discretization, values of around 0.1 m/s are reached for the velocity in x-direction in the path of the cube at the end of the flow space.
For the investigated discretization sizes, the magnitude of the velocity in x-direction is changing in a small range of around 0.1 m/s. For the first discretization, the magnitude of the maximum velocity was already the same like in STAR-CCM+ simulation. For the second and third discretization sizes, the maximum velocity in x-direction is around 0.1 m/s higher than in the STAR-CCM+ simulation.

(a) Discretization 1 in GT-SUITE  
(b) Discretization 2 in GT-SUITE  
(c) Discretization 3 in GT-SUITE  
(d) STAR-CCM+

![Image](image.png)

**Figure 4-5:** Flow around a cube: Velocity in x-direction

Figure 4-6 shows the velocity in z-direction of the GT-SUITE and the STAR-CCM+ simulation as interpolated contour plots for a cut through the middle of the channel in the x-z-plane. The mesh is included into the plots again. To be able to see the smaller velocities in z-direction behind the cube, the scale was adjusted to a range of -0.1 to 0.1 m/s. For all four simulations, the highest velocities in z-direction occur at the front edges of the cube where the flow is deflected. However, for the first and second discretization in GT-SUITE there are no velocities in z-direction behind the cube, whereas in the STAR-CCM+ simulation there is a velocity in z-direction behind the cube. For the third discretization, there are very small velocities in z-direction of around...
0.02 m/s behind the cube, which means the flow is combining behind the cube again. In GT-SUITE the main flow is deflected towards the boundaries of the flow space, hence in the whole vertical layer of flow splits in front of the cube, velocity in z-direction occur. In STAR-CCM+ the flow is deflected in a parabolic line towards the end of the flow space. Further the flow combines again behind the cube. Hence, the highest velocities in z-direction occur in smaller areas in front of the cube and also behind the cube.

Due to the fact that GT-SUITE solves the Navier-Stokes equations, it is a pressure driven system. Figure 4-7 shows the static pressure for the three discretization sizes in GT-SUITE and the STAR-CCM+ simulation. As can be seen, for the second and third discretization and for the STAR-CCM+ simulation, the highest pressure occurs in front of the cube. This point is called the stagnation point, thus the pressure at this point is called the stagnation pressure. For the first discretization, the highest pressure
occurs in the flowsplit directly in front of the cube, too. Like mentioned before the interpolated plots are not always showing the actual calculated values, especially not for models with a small number of sub-volumes. When taking a look at the discrete plot of the static pressure in front of the cube, it becomes clear that for the first discretization the highest pressure occurs in the flowsplit directly in front of the cube, too. The pressure distribution in GT-SUITE looks similar to the ones in STAR-CCM+ but the pressure difference between the flow inlet and outlet is higher in GT-SUITE. As can be seen, the pressure distribution in GT-SUITE behind the cube is homogeneous. Hence in GT-SUITE there is a homogeneous flow behind the cube caused by the pressure boundary condition over the whole cross section at the outlet. The flow strikes on the cube and is deflected towards the boundaries of the flow space and doesn’t combine again as it does in the STAR-CCM+ simulation.

![Discretization 1 in GT-SUITE](image1)
![Discretization 2 in GT-SUITE](image2)
![Discretization 3 in GT-SUITE](image3)
![STAR-CCM+](image4)

Figure 4-7: Flow around a cube: Static pressure distribution

Summarized it can be said that in the STAR-CCM+ simulation, the air streams smoother around the cube. Regarding the viscosity in the STAR-CCM+ simulation leads to a development of a boundary layer on the surface of the cube. Hence, two
turn in vortexes appear directly behind the cube caused by the deceleration of the flow on the surface of the cube. The flow is also deflected at the edges of the cube but not towards the boundaries of the flow space, more in a parabolic like towards the outlet. In the flow path of the cube, the flow combines again behind the vortexes.

In GT-SUITE, there is no boundary layer development. Further, there is no turbulence model, thus no vortexes (turbulent eddies) appear behind the cube. The air strikes on the cube inside the model and is deflected towards the boundaries of the flow space. For the first and second discretization, a homogeneous flow field occurs behind the cube and the flow doesn’t combine again behind the cube. For the third discretization, the flow does combine again behind the cube but not so strong like in the STAR-CCM+ simulation.

To get a good flow distribution of a model in GT-SUITE, it is recommended to work with a number of sub-volumes which is located in the area of the second discretization. The first discretization has a too small number of sub-volumes, to lead to good simulation results. The third discretization leads to a better flow distribution but the very high number of sub-volumes leads to a high calculation time and can cause convergence problems, especially when the geometries get more complex and heat transfer is modeled. To investigate which influence the limitation of the flow in GT-SUITE has on the temperature distribution inside a model, the simple box model is used.
5 Model building and calibration

Due to the complex geometry of the passenger car cabin, it is difficult to do basic investigations with it regarding the discretization, flow distribution or co-simulation setup. For a basic investigation of the co-simulation between TAITherm and GT-SUITE a simple box model is used. The geometry of the investigated passenger car cabin was provided from Daimler. It is the cabin geometry of a Mercedes-Benz C-Class. The layer structure of the parts, the material and surface properties of this passenger car cabin were provided from Daimler as well.

5.1 Simple Box Model

The total volume of the box model is nearly the same as the volume inside the later investigated passenger car cabin. In TAITherm the walls of the box are modeled, in GT-SUITE the volume inside of the box. To create a flow field which is similar to the flow field in a real passenger car cabin a simplified version of a seat was placed inside the box as a flow blockage. The simple box model is used to simulate a cool-down phase. The boundary and initial conditions for the simulation are summarized in Table B-0-4 in appendices B. The simulation results of the co-simulation are compared to STAR-CCM+ simulation results and the main reasons for the appearing differences in the results and the flow distribution are pointed out.

5.1.1 Model building in TAITherm

To create the simple box model in TAITherm, each part which is modeled with different material or surface properties is created separately as a plate with a mesh out of square elements. The size of each element of the box is 20 mm x 20 mm. Afterwards these plates are combined to a box. The total dimension of the box is 2000 mm x 1000 mm x 1000 mm. Figure 5-1 shows the structure of the box model in TAITherm. Inside of the box a simplified seat geometry is placed to have a flow blockage inside the model. The elements of the simplified seat have a size of 40 mm x 40 mm. The total number of elements of the simple box model is 26040. In Figure 5-1, the right side of the box is hidden, to be able to see the inside of the box. The left and right side of the box are modeled as doors, the ground as floor, the top is separated into a windshield and a roof. The front is modeled as dashboard and the back as rear shelf.
The parts are modeled as multi-layer parts with the material properties of the later investigated car cabin. The layer structure of the parts is summarized in Table B-0-1 in appendices B. In TAITherm materials are described with their density, conductivity and specific heat. The used material properties of the different layers are summarized in Table B-0-3 in appendices B. For the calculation of the radiation heat transfer it is necessary to define the surface conditions - emissivity and absorptivity - of each surface. As described in chapter 2.3, for technical surfaces the emissivity has the same value as the absorptivity. Because the windshield should be modeled as a transparent part, the “Glass, Conventional Automotive” is used as material. This material has a reflectance of 0.08, a transmittance of 0.76 and an absorptivity of 0.16. Thus, solar radiation can penetrate it.

The environment in TAITherm is modeled as a bounding box. This means an imaginary box is surrounding the model with a defined offset. The imaginary box has a constant wall temperature. The emissivity of the bounding box is assumed to be one, which means it behaves like a blackbody. Thus, it behaves as an infinite constant temperature heat sink.

On the outside surfaces, the box model is modeled with a constant heat transfer coefficient by using the “H to Fluid” method. When the environment is modeled as a bounding box, the ambient air has to be created as a separated part. Afterwards the created ambient air object can be selected as fluid temperature for the “H to Fluid” method. Because the ambient temperature has a constant value during the whole simulation, the ambient air object is created by using an assigned temperature part with a fixed temperature. On the inside surfaces of the box and on the surface of the
Model building and calibration

seat, the convection heat transfer is modeled by using the “co-simulation” method. Hence the heat transfer coefficient of these surfaces is calculated in GT-SUITE.

The initial temperatures for the parts of the box are set to the “Bypass SS” option, which means that the specified values are actually used as initial temperatures for the simulation. For the alternatively “Seed SS” option, TAITherm will first do a steady state simulation and afterwards use the steady state temperatures of the parts as initial temperatures for the simulation.

There are two ways to simulate solar radiation in TAITherm. First you can simulate the solar radiation with a solar lamp. Second by using a weather file. There is a third way where the incoming total solar radiation is directly defined for each part of a model. This method was also investigated but it was noticed that it doesn’t work for transparent parts. The defined incoming total solar radiation is considered as a heat source but in reality, the main part of the solar radiation is going through transparent parts. However, in a passenger car cabin there are transparent parts like the windows. Hence the third method cannot be used for modeling solar radiation in simulation of the box model.

The simulation of solar radiation with a weather file will be described here shortly. More detailed information can be found in the TAITherm User Manual [10]. Like the name already says, weather files are describing the weather for a specified position on the earth and a specific date. In addition to the solar radiation, a weather file also includes other weather phenomena like wind, clouds, etc. With weather files the change of the weather over the day time is simulated, hence the angle and intensity of the solar radiation is changing with the simulation time. A .xwa weather file can be manipulated with a text editor that the solar radiation has a constant angle and intensity over time and no other weather phenomena are modeled. For weather files the terrain is modeled, too. There are several options in TAITherm for modeling the terrain, which are described in detail in [10]. To avoid reflections from the terrain an assigned temperature for the default terrain can be used which means it is modeled as a blackbody. However, the modeling of the solar radiation with a weather file is complicated and lots of parameters have to be defined. As a result of this there are lots of possible error sources. A weather file should only be used, if it is necessary to model a natural weather which also includes other weather phenomena like wind, clouds, etc.

When only modeling solar radiation and no other weather phenomena, it is recommended to use the solar lamp part in TAITherm. It allows to model the effects of solar radiation when using a bounding box as environment. The solar lamp part in TAITherm corresponds to a solar lamp which is used in a climate wind tunnel to model solar radiation. To create a solar lamp part, a plate with one element is created. The solar lamp should have minimum the same size of the surface on which it projects the
solar radiation. For the box model a constant solar radiation with a constant angle of 90 degrees should be modeled. Hence the solar lamp is placed above the box, parallel to the top surface. It is important to make sure that the front of the solar lamp part is heading in the direction of the model. The normal direction of a part in TAITherm can be reversed by selecting the part and using the “Reverse Normals” option in the “Tools” tab. The temperature of the solar lamp is set to the same value as the ambient temperature. Because the solar lamp intensity is set to “Calculated”, the position of a pyranometer has to be defined. A pyranometer is a measurement instrument which is used in a climate wind tunnel to measure the solar radiation. It cannot be placed on transparent parts. Hence it is placed on top of the box model in the middle of the roof. In the “Pyranometer Locations” window the desired incoming solar flux is defined. The solar flux distribution will be calculated by TAITherm regarding the position of the pyranometer. The intensity distribution is set to be uniform. To understand how the solar lamp works, several setups of the solar lamp were tested. The solar distribution on top of the box was investigated with different setups for the solar lamp. A solar lamp with the size of 2 m x 3 m is created in TAITherm. First the influence of the distance was investigated. Figure 5-2 shows the distribution of the solar radiation on top of the box for three different distances between the solar lamp and the top of the box. From the left to the right the distance between the solar lamp and the top of the box is increasing from one to four meters. The top of the box consists of the windshield and the roof. The windshield is a transparent part and most of the solar radiation is going through it. In Figure 5-2 it looks like the solar radiation on top of the windshield is the same for all three distances. In reality the solar radiation on top of the windshield is changing, too. Caused by the wide range of the scale from 0 to 900 W/m² this change cannot be seen in Figure 5-2. For all three distances, the highest solar radiation appears for the element where the pyranometer is placed. For the distance of one and two meters the solar lamp creates a pattern on top of the roof in the shape of a half circle. With increasing distance, the area of high solar radiation is growing. To the lower edges of the roof, the solar radiation is decreasing. To find the reason for the changing solar radiation distribution on top of the box with increasing distance, the relation between the solar lamp and the top of the box was further investigated. Figure 5-3 shows the view factor distribution on top of the box for the three different distances. As describe in chapter 2.3 the view factor is needed to calculate the radiation heat transfer between two surfaces. Only for the blue highlighted elements in Figure 5-3 a view factor was computed by TAITherm before the simulation. Thus, the radiation heat transfer is calculated between the solar lamp and the blue highlighted elements only.
Figure 5-2: Solar radiation distribution on top of the box

Figure 5-3: View factor distribution on top of the box
To get the radiation heat transfer for all elements, TAITherm interpolates between the blue highlighted elements. With increasing distance between the solar lamp and the top of the box, the number of blue highlighted elements decreases. Thus, the number of elements for which the radiation heat transfer is actually calculated, decreases, too. Hence the difference of the solar radiation distribution can be explained with the number of elements for which the radiation is actually calculated. Increasing the size of the solar lamp with a fixed distance has the same effect on the solar distribution.

By using a solar lamp which consists out of more elements, the number of elements for which the radiation heat transfer is calculated can be increased. However, the solar radiation distribution on top of the box doesn’t change significantly, but the time to calculate the view factors and the solar lamp apparent areas increases harshly. The absolute values of the solar radiation changes but in a very small range.

By using multiple solar lamp parts and placing multiple pyranometer on top of the model, a field of solar lamps, like it is used in climate wind tunnels, can be created. For this method, the solar radiation distribution changes but the time to calculate the view factors and the solar lamp apparent areas increases harshly.

Due to the fact, that the exact values of the solar radiation are unknown and the simulation results are compared to STAR-CCM+ results and not to measured results from a climate wind tunnel, a solar lamp part with one element is used for modeling solar radiation. The distance between the solar lamp and the top of the model is set to four meters to get a nearly homogeneous distribution for the solar radiation on top of the box.

According to chapter 3.3, it is necessary to map the TAITherm mesh and the GT-SUITE flowsplit grid on each other for the co-simulation. Therefore, it is necessary to export the TAITherm mesh as a Nastran or Patran Neutral file. It is important to make sure that the unit during the export is set to meters, otherwise this can lead to problems during the co-simulation. When the solar radiation in TAITherm is modeled with a solar lamp the exported mesh file will include the solar lamp. Because the solar lamp isn’t part of the co-simulation, the exported mesh with included solar lamp leads to strange auto scale values for some plots in the post-processing program GT-POST. To avoid this problem, the mesh should be exported without the solar lamp. A very easy way to export the mesh without the solar lamp is to hide the solar lamp in TAITherm. After hiding the solar lamp, the check box “Export only visible geometry” has to be selected during the export of the mesh. With this option the mesh doesn’t include the solar lamp anymore, hence there are no more problems with the plots in GT-POST.
5.1.2 Model building in COOL3D

To model the volume inside of the box, a flow space with the same dimensions like the model in TAITherm is created in COOL3D. The seat inside the box works as a flow blockage. Hence it has to be modeled in COOL3D, too. To create the flow space with the seat inside of it, GT-Spaceclaim is used. In GT-Spaceclaim a box with the size of the flow space and a part with the geometry of the seat is created. Afterwards the seat is moved to the correct position inside the box and cut out of it. Then the geometry is imported into COOL3D and converted into a flow space. Alternatively, the geometry of the seat can be imported into COOL3D and converted into a flow blockage.

Notice, that the original position of the coordinate system in TAITherm and COOL3D don’t match. The original position of the coordinate system in TAITherm is in the right lower corner, whereas in COOL3D it is in the middle of the model. It is important to make sure that the original position of the coordinate systems in TAITherm and COOL3D are the same when creating a model for a co-simulation manually and not importing the geometry. Otherwise the different position of the coordinate systems is leading to problems when the two discretized representations are mapped on each other. To avoid this mistake, the model in COOL3D was moved, so that the original position of the coordinate system is in the right lower corner, too.

After the creation of the flow space, the air inlet and outlet are added as flow openings to the model. In the “Location” tab in the “FlowOpening” template the normal vector of the flow opening is defined. It is important to make sure that the normal vector of the flow opening is pointing in the correct direction. A normal vector of \{1,0,0\} means that the air is streaming out of the flow opening in the positive x-direction. Additionally, the boundary flow direction in the “Main” tab has to been set to “Inlet” for the flow inlet and to “Outlet” for the flow outlet. The flow inlet has a width of 50 cm, a height of 10 cm and is placed at the front of the box. The outlet has a width of 50 cm, a height of 25 cm and is placed at the back of the model.

Figure 5-4 shows a screenshot of the model in COOL3D. The flow space is transparent, that it is possible to see the seat inside. To be able to investigate the influence of the in chapter 4 explained flow limitations on the temperature, several rows of temperature sensors are added to the model. The blue points in Figure 5-4 are representing these temperature sensors.
Figure 5-4: Box model in COOL3D

Figure 5-5 shows a front and right view of the model in COOL3D with the naming logic of the temperature sensors. As can be seen in Figure 5-5 (a), there are always three temperature sensors in each row in y-direction. One column of sensors was placed in front of the seat, two columns behind it. The first part of the naming is the column name, the second part the row name out of Figure 5-5 (b). The number at the end shows the location in y-direction like in Figure 5-5 (a). The best way to understand this logic is to explain it with an example: The green highlighted temperature sensor in Figure 5-4 and Figure 5-5 is placed in the first column and in the first row. In the y-direction it is placed on the right side. Hence it has the name “Front_Top_2”.

Figure 5-5: Front and right view of the box model in COOL3D

Before the model is exported to GT-SUITE, the discretization length along the x-, y- and z-direction is defined in COOL3D. Like described in chapter 3.1, the discretization is determining the size and number of sub-volumes in which the flow space is
separated. For the box model, the same discretization length in all three directions is set to the same value. Besides of the discretization, the “Flowsplit Acceptance Ratio” is another parameter which has to be declared before exporting the model. It specifies the ratio of one flowsplit that must be contained inside the flow space, otherwise it is disregarded during the export and thrown out of the model. A value of 0.1 for the “Flowsplit Acceptance Ratio” means that 10 % of the flowsplit must be contained inside the flow space to regard it during the discretization.

For the box model, it is important to make sure that the seat inside of the box is really cut out of the grid of flowsplits and works a flow blockage. The best way to control this, is to use the “Preview” option in the export window in COOL3D. With this option, it is possible to see the grid of flowsplits before exporting the model. Because the seat is in the middle of the box, the “Model Sectioning” option is used to create a sectioning plane through the middle of the model. Figure 5-6 shows a screenshot of this preview. Each colored cube is representing one flowsplit. After moving the sectioning plane to the middle of the box it is possible to see, that there are no flowsplits placed over the seat.

![Figure 5-6: Grid of flowsplits for the box model: Cut through the x-z-plane](image)

To make sure that the seat is always cut out of the model the “Flowsplit Acceptance Ratio” is set to a value of 0.5. When working with models with more complex geometries you have to pay attention with increasing the “Flowsplit Acceptance Ratio”. Especially when the model has curved surfaces at the boundaries, it can happen that some flowsplits which you want to retain in the model are thrown out. It is recommended to use the “Preview” and “Model Sectioning” option as described above to take a look at the grid of flowsplits before exporting the model to GT-SUITE and make sure that no flowsplits are thrown out of the model which you want to retain in the model.
The boundary conditions of the simulation are added to the model in GT-SUITE, hence the model is exported as an external subassembly (.gtsub-file). When exporting the model from COOL3D to GT-SUITE, the flowsplits are connected automatically at the boundaries. One layer of flowsplits of the grid is condensed to a “MatrixFlowSplit” template. Condensing the layer of flowsplits has the advantage that models with a high number of flowsplits still remain clearly after the export to GT-SUITE. Further, the handling of the model in GT-SUITE remains good.

For the calibration of the box model, a discretization length of 100 mm in each direction is used. With this discretization, the volume is divided into 2000 flowsplits. Since through the seat 72 flowsplits are cut out of the model, the box is divided into 1928 flowsplits. After the successful calibration of the model the influence of the discretization on the results can be investigated.

5.1.3 Model setup in GT – SUITE

After the export of the model, the subassembly is integrated in the simulation model in GT-SUITE by using a “SubAssemblyExternal” template. The boundary conditions at the flow inlet are added by using a “EndFlowInlet” template. The boundary conditions at the flow outlet by using a “EndEnvironment” template. The flow medium is dry air, humidity isn’t modeled. GT-SUITE simulates the air flow inside the box, hence the following boundary conditions of the dry air need to be set up:

- Inlet mass flow rate (alternative: volume flow rate, velocity or mass flux)
- Inlet temperature
- Outlet pressure
- Outlet Temperature (only used in case of back flow)

Furthermore, the initialization temperature and pressure of the dry air for all flowsplits has to be defined. To store the signals of the temperature sensors as plots, a “SampledOutput” template is used. As co-simulation interface a “ThermalMeshCoSim” template is added to the model in GT-SUITE. In this template, the boundary conditions for the co-simulation are specified. As described in chapter 3.3, for the Co-Simulation the mesh of the model in TAITherm has to be mapped on the grid of flowsplits. In the “ThermalMeshCoSim” template the path to the exported mesh file is defined. Thus, the mesh can be mapped on the grid of flowsplits during the Co-Simulation. Figure 5-7 shows the simulation model in GT-SUITE with the flow inlet and outlet, the external subassembly, the temperature sensors and the co-simulation interface.
The box model is used to simulate a cool-down. The boundary and initial conditions for the simulation are summarized in Table B-0-4 in appendices B. In the STAR-CCM+ simulation a solar radiation of 900 W/m² and a diffusive radiation of 100 W/m² was modeled. However, it isn´t possible to model a diffusive radiation around the whole model in TAITherm when using the solar lamp template to model the solar radiation. Hence, for the solar radiation a value of 1000 W/m² is used for the simulation of the box model.

### 5.1.4 Optimization

As explained in chapter 3.3, two very important parameters of the co-simulation are the time before the first data exchange and the communication interval. They are interacting with the time step size in TAITherm and GT-SUITE and influencing the simulation behavior significantly. That’s why it was necessary to investigate the optimal setup for these parameters before starting with the final simulations. Different settings were investigated and evaluated by viewing the time course of the part temperatures in TAITherm and checking the convergence in both programs. The time step size in GT-SUITE was set to a value of 0.1 second at the beginning of the investigation. The time step size in TAITherm, the time before the first data exchange and the
communication interval were varied in size. A summary of the investigated setups can be found in Table 5-1.

Table 5-1: Overview of investigated setups for the box model

<table>
<thead>
<tr>
<th>Setup</th>
<th>Time step size GT-SUITE [s]</th>
<th>Time step size TAITherm [s]</th>
<th>Time before first data exchange [s]</th>
<th>Communication interval [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Setup 1</td>
<td>0.1</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Setup 2</td>
<td>0.1</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Setup 3</td>
<td>1</td>
<td>variable</td>
<td>1</td>
<td>variable</td>
</tr>
</tbody>
</table>

Both programs didn’t have any convergence problems during the simulations with the different setups. However, for the first and second setup an oscillation of the surface temperature of some parts of the box was detected in TAITherm. Figure 5-8 shows the surface temperature of the back of the box for the three different setups. The blue line represents the time course of the surface temperature for setup 1, the green one for setup 2 and the red one for setup 3. For the first setup, the temperature is oscillating with very high jumps between the single time steps.

![Surface temperature of the back of the box model](image)

Figure 5-8: Surface temperature of the back of the box model

The amplitude of the oscillation is decreasing with the time but after 5 minutes, it still has a value around 15 K. It is conspicuous that this oscillation of the temperature only appears for surfaces where the material has a very small conductivity, a small density...
and low thickness, thus a small mass and thermal resistance. One reason for the oscillation could be the combination of the surface material properties with a very high heat transfer coefficient at the outside of the box and a high time step size in TAITherm.

To increase the accuracy, the time step size in TAITherm, the time before the first data exchange and the communication interval were halved. As can be seen in Figure 5-8, the temperature is still oscillating at the beginning of the simulation but the amplitude of the oscillation is decreasing faster than for the first setup. At the end of the simulation the oscillation has nearly disappeared. The big drawback of the second setup is the increasing calculation time. In comparison to the first setup, the calculation time is increasing by 57.6% because the number of time steps in TAITherm was doubled.

To decrease the calculation time again but still avoid the constant oscillation of the temperature, a step profile is used for time step size in TAITherm and the communication interval. Furthermore, the time before the first data exchange is reduced to one second. Figure 5-9 (a) shows the step profile. The time step size increases from one to 60 seconds in four steps. For the first 10 seconds, the time step size is one second, from ten to 60 seconds it is five seconds, from 60 to 180 seconds it is 30 seconds and after 60 seconds till the end of the simulation.

According to the red line in Figure 5-8, the small time step size at the beginning of the simulation leads to a drastic reduction of the oscillation in comparison to the other two setups. The amplitude of the temperature oscillation already reaches values underneath one kelvin after ten seconds. Not only that the time course of the surface temperature shows the best behavior, also the calculation time decreases when using the step profile for the time step size and communication interval. In comparison to setup 1, a reduction of 30.8% is reached. This is mainly caused by the decreasing number of time steps in TAITherm in comparison to the other two setups.

To further reduce the calculation time, the time step profile is optimized. It seems like the problem with the oscillation of the temperature can already be fixed with the very small time step at the beginning. Hence the time step size in the optimized step profile is increased much faster. Figure 5-9 (b) shows the optimized step profile. For the first ten seconds, it still has a value of one second, from ten to 30 seconds the time step size is ten seconds. After 30 seconds the time step size is 60 seconds. The profile is used for the time step size in TAITherm and for the communication interval. Furthermore, the time step size in GT-SUITE is increased from 0.1 to one second. With this optimization, the calculation time could be decreased by 55.3% in comparison to the first setup and by 35.4% in comparison to the third setup. The final calculation time after the optimization was six minutes and five seconds. This means with the used
discretization, a ratio of nearly 1.0 between the calculation and the simulation time is reached. The oscillation of the surface temperature was still avoided.

![Step profile](image1)
![Optimized step profile](image2)

Figure 5-9: Step profiles for the box model

Hence, it is recommended to work with step profiles for the time step size in TAITherm and the communication interval to decrease the calculation time and avoid oscillation of surface temperatures. Smaller values should be used in areas where high temperature changes are expected. Higher values in areas where small temperature changes are expected. For the final simulations of the box model the optimized step profile out of Figure 5-9 (b) was used.

### 5.1.5 Calibration

After the optimization of the communication interval and time step sizes in both programs, the calibration of the model can start. The box model is calibrated by using results of a STAR-CCM+ simulation. In the STAR-CCM+ simulation, the same layer structure like in the TAITherm model is used to model the walls of the box model. As calibration parameter, the HTC multiplier is used.

Normally it is recommended to calibrate the model by adjusting the air outlet temperature. Unfortunately, the time course of the air outlet temperature wasn’t available for the box model. Therefore, the time course of the volume averaged temperature of the box is used for the calibration. Figure 5-10 shows the time course...
of the volume averaged temperatures for the STAR-CCM+ simulation and the co-
simulation with different HTC multipliers. The red line represents the time course of the
STAR-CCM+ simulation. The other curves are representing the volume averaged
temperatures from the co-Simulation model with different HTC multipliers.

As can be seen, with increasing HTC multiplier the volume averaged temperature is
increasing because more heat is transferred to the air inside the box. The best fit
between the STAR-CCM+ simulation and the co-simulation is achieved with a HTC
multiplier of 20. The volume averaged temperature in the co-simulation cools down
colder faster at the beginning of the simulation than in the STAR-CCM+ simulation. One
possible reason for this could be that less heat is transferred to the air in the co-
simulation than in the STAR-CCM+ at the beginning of the simulation. After 30 seconds
till the end of the simulation, the two curves have a very good fit with a maximum
deviation of 1.2 Kelvin. Hence for the final simulation of the box model a HTC multiplier
of 20 is used.
5.2 Cabin Model

With the provided Mercedes-Benz C-Class cabin geometry, a model of the cabin is created in TAITherm and COOL3D for the co-simulation. The steps of the model building are nearly the same than for the simple box model, with the difference that the geometry is imported into both programs and not manually built. Hence in this chapter mainly the differences to the model building of the simple box model are described in detail. A detailed description of the model building steps which are the same like for the simple box model can be found in the chapters 5.1.1, 5.1.2 and 5.1.3.

After the model building, the cabin model is used for a pull-down simulation which consist of a hot-soak phase followed by a cool-down phase. A description of the pull-down test can be found in chapter 1. The important heat transfer mechanism for this the two phases are explained in chapter 2. The boundary and initial conditions for the simulation of the hot-soak and cool-down phase are summarized in Table B-0-5 and Table B-0-6 in appendices B. First the hot-soak and cool-down phase are simulated separately. According to chapter 2.2, the air circulation inside the cabin during the hot-soak phase is strongly influenced by the gravity. That’s why it was necessary to model the hot-soak phase with gravity turned on. The cool-down phase was modeled without gravity because the air circulation is mainly caused by the forced flow into the cabin. The results of these simulations are compared to STAR-CCM+ results.

5.2.1 Model building in TAITherm

To create the cabin model for the co-simulation in TAITherm, a .stl-file of the cabin geometry is imported. It is important that the file already includes a mesh, because there is no opportunity to add a mesh to an imported geometry in TAITherm. After the import, the solid geometry has to be separated into the different parts of the cabin. It is recommended to separate the cabin into the parts which should be modeled with different material and surface properties, for example the doors, roof, floor, windows, seats, etc. Alternatively, the cabin model can be imported as a .nas-file in which the cabin geometry is already separated into the different parts. Figure 5-11 shows the cabin model in TAITherm after the separation. The different parts are highlighted in different colors. The windshield and the side windows on the left side are hidden, to be able to take a look inside the cabin.

The complete mesh of the cabin consists out of 83265 elements. The size and shape of the elements isn’t consistent. Especially in the areas where the air is streaming inside the cabin, like in the area of the foot room vents, dashboard vents and in the
area of the defrost vent, the elements are much smaller than in the other areas. Actually, the number of thermal nodes during the simulation is much higher because the parts are modeled as multi-layer parts. As explained in chapter 3.2, thermal nodes are not only created at the element face centroids but for multi-layer parts also between the layer interfaces. TAITherm calculates a solution for each thermal note of the model.

![Figure 5-11: Cabin model in TAITherm](image)

To be able to model the different parts of the cabin as multi-layer parts with the correct material properties for each layer, the material properties from the provided cabin geometry are added to the model. Materials are defined by their density, conductivity and specific heat in TAITherm. It is also possible to make the material properties depended on the temperature. For modeling radiation, the surface conditions like the emissivity and absorptivity for the different parts have to be defined in the model. There are a lot of predefined surface condition which can be chosen, but there is also the possibility to add new surface conditions to the model. With the correct material properties and surface conditions the different parts of the cabin are created as multi-layer parts with the properties from Daimler. The windows are modeled as transparent parts with “Glass, Conventional Automotive” as material. This material has a reflectance of 0.08, a transmittance of 0.76 and an absorptivity of 0.16. Hence solar radiation can penetrate the windows. The layer structure of the parts for the cabin model can be found in Table B-0-2, the material and surface properties in Table B-0-3 in appendices B.

The convective heat transfer for surfaces inside the cabin is calculated by using the heat transfer coefficient and fluid temperatures, calculated in GT-SUITE. For the
outside surfaces of the cabin, the convection is modeled in TAITherm dependent on the vehicle velocity using the automatic convection type out of the TAITherm library. A description of the used automatic convection type can be found in chapter 3.2.

Like for the box model, the solar radiation for the cabin model in TAITherm is modeled by using a solar lamp. Hence the environment can be modeled as bounding box with a fixed temperature. For the co-simulation, the mesh of the cabin has to be exported as well. A detailed description of the modeling of the solar radiation in TAITherm and the export of the mesh can be found in chapter 5.1.1.

### 5.2.2 Model building in COOL3D

To create the cabin model for the co-simulation in COOL3D, the solid geometry of the cabin is imported into COOL3D. Afterwards the imported shape is converted into a flow space. The geometry of the used cabin is very detailed and has lots of small tilted surfaces and edges. As a result of this, the handling of the converted shape in COOL3D was very poor. Thus, it was necessary to simplify the cabin geometry before importing it into COOL3D. The simplification of the geometry took place in GT-Spaceclaim.

The easiest way to create a simplified geometry of the car cabin is, to cut through the model in the x-z-plane at several positions along the y-axis and trace the lines of the produced cross-sections. It is recommended to do this at the points where the cross section of the cut through the cabin changes significantly. Because the upper part of the cabin has an angle, this part should be created separately. Then it is easier to incline the side of the roof. To save time, it is recommended to simplify one side of the geometry after the described procedure and afterwards mirror it at the middle x-z-plane. The differences between the two sides of the cabin can be proceed afterwards through cut out or add some areas to the model. The interior like the seats, headrest and steering wheel have to be generated separately. Figure 5-12 shows the cabin geometry after the simplification. The body is illustrated transparent, that it is possible to see the interior of the cabin.

Before the export to GT-SUITE, the volume inside the cabin is discretized into cube shaped sub-volumes. As described in chapter 3.3, the cube shaped flowsplits are normally much bigger than the elements of the mesh in TAITherm. On the one hand, it isn’t necessary to include every little curved surface or detail of the cabin geometry into the simplified model. This would not change the number of created flowsplits in COOL3D, thus the shape of the model wouldn’t change. On the other hand, it is important to pay attention, that there are no flowsplits created at the boundaries of the
model, which are not interacting with parts of the mesh in TAITherm during the co-
simulation. Further it is important to make sure that each mesh element has a flowsplit
from which it can receive data. The procedure of mapping the two different discretized
representations on each other is explained in detail in chapter 3.3.

The simplified cabin geometry is imported into COOL3D and converted into a flow
space. Afterwards, the inlet vents are added as flow openings to the model. They
should be placed with a small offset in front of the flow space. COOL3D will visualize
the flow openings through projecting the cross section of them on the flow space. During the export, they will automatically be connected to the flowsplit on which the inlet cross section is projected. At the back of the cabin, underneath the rear shelf an
outlet is added through which the air can stream out of the cabin.

In Figure 5-13 you can see a front and top view of the cabin model in COOL3D. The
flow space has the color blue and is transparent. Further you can see the inlet vents
which were added to the model. There are two vents at the sides and three in the
middle of the dashboard, two side vents and two rear vents. All inlet vents and the flow
outlet are marked in red in Figure 5-13. To be able to compare the time course of the
temperature for some positions inside the cabin to STAR-CCM+ simulation results,
temperature sensors are added to the cabin model in COOL3D at several places. In
the head area, there are three rows of temperature sensors, one at the front, one at
the middle and one at the back of the cabin. Always three sensors are placed in each
row in the head area. In the front foot rooms, three sensors are placed on each side
as well. In the rear foot rooms two temperature sensors are placed on each side.

The temperature sensors are represented by the dark blue points in Figure 5-13 and
Figure 5-14. Figure 5-14 shows a right view of the cabin model in COOL3D. The inlet
vents and the flow outlet are marked in red again. Furthermore, the arrows are showing
the flow direction of the inlet vents and the outlet. Through the dashboard vents and the rear vents, the air is streaming horizontal into the cabin. Through the two side vents, the air is streaming vertical towards the windshield into the cabin. At the back of the cabin the air is streaming out of the cabin again.

![Front view of the cabin](image1)

(a) Front view of the cabin

![Top view of the cabin](image2)

(b) Top view of the cabin

Figure 5-13: Front and top view of the cabin model in COOL3D

![Right view of the cabin](image3)

Figure 5-14: Right view of the cabin model in COOL3D

Before the model is exported to GT-SUITE, the discretization length along the x-, y- and z-direction is defined. As mentioned in chapter 3, for the modeling of a car cabin it is recommended to work with the same discretization length in all three directions. Hence the model is discretized into cube shaped flowsplits. Because the seats inside
the cabin aren’t straight like the seat in the box model, it is more complicated to make sure that the seats are actually cut out of the model and are acting as a flow blockage.

One opportunity to make sure that the seats are cut out, is to increase the “Flowsplit Acceptance Ratio”. As described in chapter 5.1.2, the “Flowsplit Acceptance Ratio” specifies the ratio of one flowsplit that must be contained inside the flow space, otherwise it is disregarded during the export and thrown out of the model. Hence by increasing the value of this parameters, flowsplits are easier cut out of the model when they are interacting with the seats. Because the “Flowsplit Acceptance Ratio” is a global parameter, it also applies on the flowsplits which are interacting with the boundaries of the cabin model. Thus, it can lead to problems at the boundaries of the model through disregarding some flowsplits which actually should stay inside the model.

To avoid this problem, discretization planes are used. A discretization plane forces COOL3D to start with the discretization at a defined position, the position of the discretization plane. If two discretization planes are placed parallel and the distance between them is smaller than the global discretization length in this direction, smaller flowsplits are created between the two discretization planes. Hence discretization planes can be used to make sure that there are no problems with “thrown out” flowsplits at the boundaries of the model. Further they can be used to create smaller flowsplits at specified places. Thus, they are a powerful help for making sure that the interior of the cabin is really cut out of the model. With a skillful use of discretization planes next to the seats and at the boundaries of the cabin, it is possible to make sure that the seats are cut out of the model without increasing the “Flowsplit Acceptance Ratio” too much and run into problems with “thrown out” flowsplits at the boundaries of the cabin geometry.

To be sure that the seats are actually cut out of the model, before exporting it to GT-SUITE, the “Preview” and “Model Sectioning” option is used again. This method was already used for the box model to make sure, that the seat inside the box is really cut out of the model. It is described in detail in chapter 5.1.2. For the start a discretization length of 100 mm in each direction is used because with this discretization good results and calculation times could be achieved for the box model. After the simulation runs stable, smaller discretization can be tested to see if they have an influence on the results. Figure 5-15 shows a cut through the cabin model in the x-z-plane in COOL3D. Each colored cube represents one flowsplit. The flowsplits which are highlighted in dark red are cut out of the model during the export of the model to GT-SUITE. As can be seen, the seat is actually cut out of the cabin model. Thus, it works as a flow blockage and the air has to stream around it. The headrests in the back are cut out of the model, too.
Model building and calibration

Figure 5-15: Cut through the cabin in the x-z-plane

After making sure that the seats and back headrests are cut out of the model and that there is a flowsplit placed in front of every flow inlet, the model is exported as a subassembly to GT-SUITE. As described in chapter 5.1.2, each layer of flowsplits is combined into a “MatrixFlowsplit” during the export, due to a better handling of the model in GT-SUITE and to remain the model clearly. With the chosen discretization, the cabin volume is divided into 2946 flowsplits.

5.2.3 Model setup in GT – SUITE

Basically, the building of the simulation model in GT-SUITE for the cabin model is the same like for the box model. The exported grid of flowsplits is integrated as an external subassembly into the GT-SUITE model. Afterwards the flow openings are added by using the “EndFlowInlet” template for the inlet vents and the “EndEnvironment” template for the flow outlet. A “SampledOutput” template is used, to store the time course of the temperature sensors. As interface for the co-simulation, the “ThermalMeshCoSim” template is added to the model. Like described in chapter 5.1.3, the parameters of the co-simulation like the time before the first data exchange and the communication interval are defined in this template. Further the link to the exported mesh file from the cabin model in TAITherm is set, that it can be mapped on the grid of flowsplits. Figure 5-16 shows the simulation model of the cabin in GT-SUITE. The main steps of the work flow from the solid CAD geometry of the cabin to the simulation model in GT-SUITE is summarized in Figure 5-17.

Before starting with the co-simulation, it is recommended to test the GT-SUITE model as a standalone model. This has the advantage that flow instabilities can be detected easier and the optimal values for the solver settings in GT-SUITE, like the time step size, can be found faster. First a time step size of one second is used in GT-SUITE for the co-simulation. With this time step size the standalone model reached numerical
convergence and good calculation times could be achieved. After the optimization of the standalone model, it can be used for the co-simulation with the TAITherm model.

![Simulation model of the cabin in GT-SUITE](image)

**Figure 5-16: Simulation model of the cabin in GT-SUITE**

As for the box model, for the communication interval and time step size in TAITherm a step profile is used for the cabin model. Because the cabin model is used to simulate a hot-soak and a cool-down phase which both have a duration of 60 minutes, the time step profile of the box model has to be adjusted for the cabin model. Further, for the simulation of the hot-soak another step profile is used than for the cool-down since during the hot-soak there is no forced flow inside the cabin. Therefore, less convergence problems and oscillation of surface temperatures are expected when using higher values for the time step sizes and the communication interval.

For the first 120 seconds the time communication interval is set to 10 seconds. After it is increasing to 30 seconds till 360 seconds. Afterwards for the hot-soak phase a constant value of 60 seconds is used till the end of the simulation. For the cool-down phase a value of 45 seconds is used from 360 seconds till the end of the simulation. The smaller value for the cool-down phase was chosen because with higher values an oscillation of some surface temperatures appeared in the model in TAITherm.

The time step size in TAITherm has the same values than the communication interval. For the hot-soak phase a time step size of one second is used in GT-SUITE. For the cool-down phase it was halved to 0.5 seconds due to convergence problems at the beginning of the simulation.
Figure 5-17: Overall work flow for the cabin model building
5.2.4 Calibration

For the calibration of the cabin model, the cabin outlet temperature of the STAR-CCM+ simulation is used. As calibration parameter, the HTC multiplier is used. Because for the simple box model a HTC multiplier of 20 was needed to calibrate the model, as a starting value for the calibration of the cabin model a HTC multiplier of 15 is chosen. One problem which appeared during the calibration was that for the “MatrixFlowSplit” parts it is only possible to initialize all flowsplits with the same temperature. Normally after a hot-soak phase a temperature stratification occurs inside the cabin. The hottest temperatures appear close to the top and the lowest temperatures close to the bottom of the cabin. For the calibration, an initial temperature of 65 °C is used.

Figure 5-18 shows the time course of the cabin outlet temperature. The solid red line is representing the STAR-CCM+ simulation results. The other lines are representing the simulation results of the co-simulation for different HTC multiplier. As can be seen, the cabin outlet temperature of the STAR-CCM+ simulation is stabilizing to a constant value of around 35 °C very quickly. Whereas in the co-simulation the temperature drops continuously and first stabilizes towards the end of the simulation. Further the temperatures of the Co-Simulation are reaching much smaller end values than the co-simulation. For a HTC multiplier of 25, the temperature starts to oscillate at the beginning of the simulation. When the HTC multiplier is further increased, the temperature starts to oscillate with a very high amplitude. The part temperatures in TAITherm also start to oscillate in a very high and unrealistic range. Because of this, the co-simulation with higher HTC multipliers was cancelled.

![Figure 5-18: Outlet temperature of the cabin model](image-url)
The question appears, what is the reason for the big difference of the temperature courses of in STAR-CCM+ and in the co-simulation. Such a big difference can only be caused by differences in the boundary conditions of both simulations. As the inlet air temperature and the inlet mass flow rate are the same for both models, the difference has to be related to the modeling of the walls or to the modeling of the solar radiation.

For an approximated calculation of the total heat transfer to the air inside the cabin, an inlet temperature of 5.43 °C, a mass flow rate of 0.1522 kg/s and a specific heat of 1000 J/(kg*K) is used. For the outlet temperature, the time courses out of Figure 5-18 are used. Figure 5-19 shows the total heat transfer to the air from the inlet to the outlet of the cabin. As can be seen, for the STAR-CCM+ simulation after around 10 minutes a heat of around 4500 W is transferred to the air constantly. Whereas for the co-simulation the amount of heat transfer to the air is smaller and also not constant over the time.

One big difference between the boundary conditions of the STAR-CCM+ and the co-simulation is the modeling of the walls. The walls of the cabin are modeled with the very detailed layer structure in the co-simulation. Unlike, in the STAR-CCM+ simulation they are modeled as walls with a constant thickness of 3 mm and averaged material properties of the detailed layer structure. For these averaged material properties, the material properties of the different layers of the detailed layer structure are weighted by regarding their impact on the layer structure over their thickness. Therefore, a different amount of heat is transferred from the walls to the air inside the cabin in the STAR-CCM+ model than in the co-simulation model.
Another possible error source could be different heat transfer coefficients in both models. To reach a very high outlet temperature like it appears for the STAR-CCM+ simulation, the heat transfer coefficients would have to be located in an unrealistic region. As the solar radiation is modeled with the same angle and magnitude in both simulation models, this shouldn’t be reason for the big difference between the results.

With the different method of the wall modeling and the possibility of different heat transfer coefficient in both simulation models, the offset between the curves can be explained but not the difference in the time course. The stabilization of the cabin outlet temperature on a constant value of 35 °C in the STAR-CCM+ simulation seems strange. It seems like there is a constant heat source in the cabin which is heating up the air. The volume averaged temperature of the air inside the cabin in the STAR-CCM+ simulation shows the same behavior as the cabin outlet temperature. It stabilizes on a constant value of around 35 °C. This is a very unrealistic and not expected behavior.

Due to the weird time courses of the air temperatures in the STAR-CCM+ simulation, the final hot-soak and cool-down simulation of the co-simulation cabin model are done with a HTC multiplier of 20. This HTC multiplier was already used for the final simulation of the box model.
6 Results and discussion

In this chapter the results of the cool-down simulation of the simple box model and the hot-soak and cool-down simulation of the cabin model discussed and evaluated. Mainly the time plots of the temperature sensors are compared to the results of a STAR-CCM+ simulation and the reasons for the differences are pointed out. For the simple box model, a discretization study was done to get a feeling for the influence of the discretization size on the results and the calculation time.

6.1 Simple Box Model

As described in chapter 5.1.5, for the final simulation of the simple box model a HTC multiplier of 20 was used. The boundary conditions and initial conditions for the simulation can be found in Table B-0-4 in appendices B. The naming logic of the temperature sensors is described in chapter 5.1.2. The volume averaged temperature for the final setup of the co-simulation and the STAR-CCM+ simulation is illustrated in Figure 6-6 (d) again. In the STAR-CCM+ simulation of the simple box the same layer structure for the walls was used as in the co-simulation.

6.1.1 Temperature sensors

All figures which are showing the results of the temperature sensors are attached at the end of this chapter. Figure 6-3 shows the results of the temperature sensors for the top and middle row in front of the backrest. The dashed red lines are showing the results of the STAR-CCM+ simulation. The solid green lines the results of the co-simulation between GT-SUITE and TAITherm. As can be seen, the temperature falls faster during the co-simulation between GT-SUITE and TAITherm for the sensors which are placed in front of the backrest. Afterwards, the temperatures of the co-simulation are smaller than the temperatures of the STAR-CCM+ simulation. For the Front_Top1 sensors the average deviation between the two curves is 4.6 Kelvin with a maximum deviation of 5.7 Kelvin. For the Front_Middle1 sensor, the average deviation - 2.6 Kelvin - and the maximum deviation – 3.8 Kelvin – is much smaller.

The best way to understand the reasons for the differences between the time courses in STAR-CCM+ and the co-simulation is to take a look at the velocity distribution of the
simple box model. Figure 6-1 shows the velocity distribution in x- and z-direction within the box model at the end of the simulation. Figure 6-2 shows the temperature distribution within the box at the end of the simulation. All plots are interpolated contour plots. When comparing the velocity distribution in x-direction for the two simulation, it is striking that the air is spreading up more after the inlet on the way to the backrest of the seat in the STAR-CCM+ simulation than in the co-simulation. In the co-simulation, the air first spreads up in front of the backrest. This difference is caused by the limitation of the flow in the co-simulation, which is explained in chapter 4.

Figure 6-1: Velocity distribution of the box model

For the right and left sensors of the top row - Front_Top2 and Front_Top3 – the co-simulation delivers results which are closer to the results of STAR-CCM+. The average deviation for the Front_Top2 sensor is 4.1 Kelvin, for the Front_Top3 sensor it is 2.4 Kelvin. In the STAR-CCM+ simulation, the temperatures of these two sensors are oscillating with an amplitude of nearly 5 Kelvin. One reason for this could be a horizontal vortex around the backrest in the STAR-CCM+ simulation. As explained in chapter 4, flow phenomena like vortexes or turbulent eddies cannot be modeled with
Results and discussion

the flow solution of the co-simulation. Hence the temperatures of the co-simulation
don’t oscillate caused by vortexes around the seat. For the right and left sensor of the
middle row – Front_Middle2 and Front_Middle3 – the average deviation – 4.1 Kelvin
and 3.7 Kelvin - is bigger than for the sensor in the middle. Like mentioned before, the
faster cool-down and smaller temperatures are mainly caused by the smaller
deceleration and spreading of the main flow after the inlet in comparison to the STAR-
CCM+ simulation. Further the elements of the mesh in the STAR-CCM+ simulation
have no equal size and shape. There are bigger elements in the regions where smaller
flow appears and smaller near the walls of the box and the surfaces of the seat. In the
cosimulation, there is the same flowsplit size everywhere in the model.

Figure 6-4 shows the results of the temperature sensor for the bottom row underneath
the seat and the top row behind the backrest. As explained in chapter 3.1, the
temperature is calculated ones per flowsplit and averaged over the volume of it. The
flowsplits in which the temperature sensors underneath the seat are placed, are
directly connected to the hot surface of the seat. Whereas in the STAR-CCM+
simulation, the volume over which the temperature is averaged isn’t directly connected
to the hot surface of the seat. Therefore, in the co-simulation, the temperatures
underneath the seat are higher than in the STAR-CCM+ simulation. This gets clearly
when taking a look at the temperature distribution at the end of the simulation in Figure
6-2. As can be seen, the temperatures directly above the floor and directly behind the
seat are higher in the co-simulation than in STAR-CCM+ simulation. To avoid the direct
connection to the surfaces, a smaller discretization size in COOL3D can be used. The
oscillation of the temperatures underneath the seat in STAR-CCM+ could be caused
by some vortexes around the seating surface of the seat. Because the frequency of
the oscillation is small, it is more realistic that they are caused by vortexes than by
numerical convergence problems during the simulation. An Oscillation caused by
numerical convergence problems during the simulation would have a much higher
frequency.

Due to the results of the basic investigations in chapter 4, it was expected that there is
less flow behind the seat in the co-simulation. Looking at the results of the temperature
sensors directly behind the seat – Back_Top1, Back_Top2 and Back_Top3 – it can be
seen that actually the opposite happens. The air behind the seat is cooling down faster
than in STAR-CCM+. The same behavior occurs in the top row of sensors –
Back2_Top1, Back2_Top2 and Back2_Top3 – which are placed a little bit farer away
from the backrest. The results of these sensor are illustrated in Figure 6-5. The average
deviation of the Back_Top1 sensors is 7.1 Kelvin and for the Back2_Top1 sensor it is
7.9 Kelvin. For the right and left sensors in this two rows the average deviation is
smaller because they are not directly placed in the main flow path of the air. The reason
for the big differences becomes clearer when taking a look at the temperature and
Results and discussion

pressure distribution in Figure 6-2. In the co-simulation, the air is sucked down after the seat due to the smallest pressure at the flow outlet. As described in chapter 5.1.2, the outlet of the box was placed nearly at the bottom of the back surface. In STAR-CCM+, the air is deflected towards the roof at the top front edge of the backrest. Afterwards it is staying attached at the roof and is streaming downwards the back of the box towards the flow outlet. The separation of the flow at the sharp corner of the backrest is a well-known flow phenomena, described in detail in [3] or other literature about aerodynamics. Directly behind the seat there is a big dead water region in STAR-CCM+. In this area, there is nearly no air flow in STAR-CCM+, whereas in the co-simulation there is.

![Figure 6-2: Temperature and pressure distribution of the box model](image)

As can be seen in Figure 6-5 and in Figure 6-6, for the two middle rows behind the backrest, the fit between the results of the co-simulation and STAR-CCM+ is better, especially for the right and left sensors.
Summarized it can be said that the offset between the results can mainly be explained by the physical differences between the Quasi-3D flow field in the co-simulation and the 3D flow field in STAR-CCM+. In the co-simulation, there is no modeling of a boundary layer building or other flow phenomena like the separation at sharp corners or vortexes. Further, there is no separation between a laminar or turbulent flow and no turbulence model. The position of the outlet has a big influence on the flow distribution in the co-simulation, when using a pressure boundary condition at the outlet. The co-simulation model is more sensitive regarding the positioning of the temperature sensors because of the bigger sub-volumes. Hence, the positions of the sensors have to be chosen carefully, especially for sensors which are lying very close to a wall.

Another reason for the differences between the temperatures could be the different material properties of the window of the box. To be able to model the window as a transparent part in TAITherm, the predefined material “Glass, Conventional Automotive” was used. Otherwise it wasn’t possible to model the window as a transparent part. Hence the material properties are different in comparison to the window in STAR-CCM+ and it could be possible that another amount of heat is transferred through it. Due to a different transparency, another amount of solar radiation can pass it. Further, in the STAR-CCM+ model, a solar radiation of 900 W/m² and a diffusive radiation of 100 W/m² was modeled. As previously mentioned in chapter 5.1.3, it isn’t possible to model a diffusive radiation around the model with a solar lamp. Hence the solar radiation was increased to 1000 W/m² in the TAITherm model. However, the solar radiation only affects the top of the box model and not the side walls or the ground. In a test model the diffusive radiation was modeled as heat source of 100 W/m² for the surfaces, except for the top of the box model. The influence on the results of the temperature sensors is moving in a range between 0.3 and 1.2 Kelvin.

Further in the STAR-CCM+ simulation the gravitational acceleration is modeled, whereas in the co-simulation model it isn’t. However, the gravitational force shouldn’t have a big influence on the flow distribution inside a model when the forced velocity of the air is high. The gravitational force mainly has an influence on the flow distribution inside a model where there is no forced flow.

To make sure that the same amount of heat is transferred from the walls to the fluid, it is recommended to compare the heat transfer from the walls to the fluid of both models for each single part separately. Further it is recommended to compare the heat transfer coefficient distribution and the surface part temperatures. Unfortunately, the time course of the air outlet temperature of the box model wasn’t available. Otherwise it would be possible to compare the total heat transfer from the walls to the fluid.
Results and discussion

Figure 6-3: Results of the temperature sensors of the box model (1)
Results and discussion

Figure 6-4: Results of the temperature sensors of the box model (2)
Results and discussion

Figure 6-5: Results of the temperature sensors of the box model (3)

(a) Back_Middle1
(b) Back2_Top1
(c) Back_Middle2
(d) Back2_Top2
(e) Back_Middle3
(f) Back2_Top3

Figure 6-5: Results of the temperature sensors of the box model (3)
6.1.2 Discretization study

To investigate the influence of the discretization size on the simulation results of the simple box model, the box model is rerun with the following discretization sizes:

- Discretization 1: 100 mm x 100 mm x 100 mm
- Discretization 2: 80 mm x 80 mm x 80 mm
- Discretization 3: 60 mm x 60 mm x 60 mm
- Discretization 4: 50 mm x 50 mm x 50 mm
Results and discussion

Figure 6-7 shows a cut through the middle of the box model in the x-z-plane for the four investigated discretization sizes. As can be seen in Figure 6-7 (b) and Figure 6-7 (c), for the second and third discretization the seat is cut out differently than for the first and fourth discretization due to the positioning of the flow splits inside the model.

![Figure 6-7: Investigated discretization sizes of the box model](image)

Figure 6-8 shows the results of some selected temperature sensors for the four different discretization sizes and the STAR-CCM+ simulation. The red dashed lines are showing the results of the STAR-CCM+ simulation, the solid green lines the simulation results of the first discretization, the dotted blue lines the simulation results of the second discretization, the dash dotted orange lines the simulation results of the third discretization and the purple dash dot dotted lines are showing the simulation results of the fourth discretization.

As can be seen in Figure 6-8 (a) and (b), for the Front_Top1 and Front_Middle1 temperature sensors, the temperature is falling faster when the discretization size is getting smaller. Further, the end values of the temperatures are getting smaller with a smaller discretization size. A similar behavior can be observed for the other temperature sensors in the top and middle row in front of the backrest. Hence the previous used discretization is already delivering the best results in this area.

A complete different behavior can be observed for the temperature sensors underneath the seat. Figure 6-8 (c) shows the results for one of the temperature
sensors underneath the seat. Notice that in Figure 6-8 (c) the range of the y-axis had to be adjusted that it is possible to see the results of the third discretization in the plot. For the second discretization, the temperature is falling more slowly than for the first discretization. For the third discretization, the temperature is falling very slowly and reaches a very high end value. The reason for this is that, in the third discretization the temperature sensors is placed directly in a flowsplit next to the wall of the seat. Due to the smaller flowsplits, the temperature is averaged over a smaller volume and increases in comparison to the first and second discretization. The best match for the temperature sensors underneath the seat occurs for the fourth discretization. The reason for this is that for the fourth discretization the temperature sensors underneath the seat are placed in a flowsplit which has no direct connection to the seat or floor surface. Hence less heat from the walls is transferred to it and the air cools down faster.

For the two top rows of temperature sensors behind the backrest of the seat, the time course of the temperature only changes in a very small range. As can be seen in Figure 6-8 (d) and (f), the results for the first and second discretization are nearly the same. For the third and fourth discretization, the temperature falls faster and reaches smaller end values.

Figure 6-8 (e) shows the results of one temperature sensor out of the middle row behind the backrest. For the other temperature sensors in the middle row, the same behavior can be observed. For the middle rows of temperature sensors behind the backrest of the seat, the second discretization delivers the results with the best match in comparison to the STAR-CCM+ results. Furthermore, the third and second discretization are showing a better behavior than the first one. The reason for this is that in the box model with the first discretization, the temperature sensors is placed in a flowsplit directly behind the backrest whereas for the other discretization sizes, the flowsplits are not directly connected to the backrest.

Another important influence variable for the choice of the discretization size is the calculation time. In Table 6-1 the calculation time for the different discretization sizes are summarized. Further the table includes the number of flow volumes and the real-time factor. The real-time factor is defined as the ratio between the calculation and the simulation time. For all four discretization sizes, the step profile out of Figure 5-9 (b) was used for the time step size in TAITherm and the communication interval.

For the first discretization, a time step size of one second was used in GT-SUITE. Due to convergence problems, for the second discretization, the time step size in GT-SUITE had to be reduced to 0.2 seconds. For the third discretization size, the box model had to be run with a time step size of 0.1 seconds in GT-SUITE to reach numerical convergence. For the fourth discretization size, further adjustments at the solver settings had to be made that the model reached numerical convergence during the
simulation. The flow rate damping factor was increased from 0.1 to 1. The time step size in GT-SUITE remained 0.1 seconds for the fourth discretization.

Table 6-1: Calculation times for the discretization study of the box model

<table>
<thead>
<tr>
<th>Discretization</th>
<th>Flow Volumes</th>
<th>Calculation Time [min]</th>
<th>Real-time factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>100 x 100 x 100 mm$^3$</td>
<td>1932</td>
<td>09.03</td>
<td>1.5</td>
</tr>
<tr>
<td>80 x 80 x 80 mm$^3$</td>
<td>3482</td>
<td>28.27</td>
<td>4.7</td>
</tr>
<tr>
<td>60 x 60 x 60 mm$^3$</td>
<td>9255</td>
<td>264.49</td>
<td>44.1</td>
</tr>
<tr>
<td>50 x 50 x 50 mm$^3$</td>
<td>15428</td>
<td>366.44</td>
<td>61.1</td>
</tr>
</tbody>
</table>

As can be seen, the calculation time increases harshly with an increasing number of flow volumes. For the first and second discretization, TAITherm is the limiting factor to reach a better calculation time, whereas for the third and fourth discretization the calculation of the flow solution in GT-SUITE is the limiting factor. It can be said that a smaller discretization size not automatically leads to better simulation results. Rather the positioning of the flowsplits inside the model and in front of the flow openings plays an important role. Hence it is recommended to check the positioning of sub-volumes inside the model before exporting it to GT-SUITE.

Due to the very high calculation times, it isn’t recommended to use a discretization with a higher number than around 5000 flow volumes. Rather, it is recommended to pay attention with the positioning of sensors inside the model. As the results of the temperatures sensors are showing, the position of the temperature sensors can cause big differences in the results, especially when they are placed close to a wall of the model. Thus, it is recommended to check the positions of the sensors which are placed inside the model before exporting it to GT-SUITE.
Results and discussion

Figure 6-8: Results of the temperature sensors for the discretization study

(a) Front_Top1
(b) Front_Middle1
(c) Front_Bottom1
(d) Back_Top1
(e) Back_Middle1
(f) Back2_Top1
6.2 Cabin Model

As explained in chapter 5.2.4, the cabin model couldn’t be successfully calibrated to the STAR-CCM+ simulation results. Hence, the final simulations of the cabin model were run with a HTC multiplier of 20. During the hot-soak phase, it was necessary to model gravity in the flowsplits. The boundary and initial conditions of the final simulations of the cabin model are summarized in Table B-0-5 and Table B-0-6 in appendices B.

6.2.1 Temperature sensors

The temperature sensors inside the cabin model are named after the following logic: The first part indicates the position in the cabin – Head Area or Foot Room – the second part the position in x-direction – Front, Middle or Rear – and the last part, the position in the row – Right, Middle or Left. For the foot rooms the position in y-direction – Left or Right – is additionally added at the third position of the name. All plots of the temperature sensors which are placed inside the cabin can be found at the end of this chapter and in the appendices A.

Figure 6-11 shows the results of the middle temperature sensors of each row in the head area for the STAR-CCM+ simulation and the co-simulation between GT-SUITE and TAItTherm. The dashed red lines are representing the simulation results of the STAR-CCM+ simulation. The solid green lines are representing the simulation results of the co-simulation. The first 60 minutes are showing the results for the hot-soak phase, followed by the 60-minute cool-down phase.

As can be seen, the trend of the results of both simulations is very different. In the hot-soak phase, the air is heating up faster in the head area in the STAR-CCM+ simulation. In the co-simulation, the air is heating up more slowly. In the front, middle and rear region of the head area, different end values are reached at the end of the hot-soak phase in the co-simulation. In comparison to that, the trend of the temperature in the STAR-CCM+ simulation is nearly the same everywhere in the head area of the cabin. In the co-simulation, a temperature around 57 °C is reached in the front head area, a temperature around 60 °C in the middle head area and a temperature around 64 °C in the rear head area. The higher temperatures in the rear are caused by the stronger heat up of the rear shelf in comparison to the dashboard due to the incident solar radiation.
As mentioned before, for the hot-soak phase it was necessary to model gravity in the flowsplits. Figure 6-9 (a) shows the temperature distribution inside the cabin after the hot-soak phase without gravitational acceleration modeled for a cut through the model in the x-z-plane (y-direction = -0.45 m). Figure 6-9 (b) shows the temperature distribution inside the cabin after the hot-soak phase with gravitational acceleration modeled. As can been seen, without the gravitational acceleration only the air directly above the parts, which are directly irradiated by the solar radiation, is heating up a lot. The hot air isn’t moving around inside the cabin like it would in reality caused by natural convection explained in chapter 2.2.2. When the gravitational acceleration is modeled, the air is moving around and a temperature stratification occurs inside the cabin like it was expected. The lowest temperatures occur at the bottom of the cabin underneath the seat and in the front foot room. The highest temperatures occur in the rear area of the cabin above the rear shelf due to the bigger surface of the rear shelf in comparison to the dashboard. Hence, the air in the rear head area is heated up more than in the front head area.

As can be seen in Figure 6-11, during the cool-down phase, the air in the head area is cooling down stronger in the co-simulation than in the STAR-CCM+ simulation. As discussed in chapter 5.2.4, it is a very unrealistic behavior that the air only cools down for the first 10 minutes of the cool-down phase and afterwards nearly doesn’t change till the end of the simulation like it does in the STAR-CCM+ simulation. As illustrated in Figure 6-11, the air in the head area cools down fast at the beginning of the cool-down phase of the co-simulation, too. Afterwards it doesn’t reach a constant temperature, rather it continues cooling down slowly. For the co-simulation, temperatures around 25 °C are reached at the end of the cool-down phase.
As explained in chapter 1, according to DIN 1946 – Part 3, the average temperature in the head region of a passenger car cabin has to be less than 25 °C after 30 minutes. Hence, maybe the HTC multiplier was set to too high values in the co-simulation. The results of the other temperature sensors in the head area can be found in Figure A-0-2 and Figure A-0-3 in appendices A. The different trend of the temperature in the co-simulation and in the STAR-CCM+ simulation during the hot-soak and cool-down phase is mainly caused by a different amount of heat transfer to the air. As described in chapter 5.2.4, in the STAR-CCM+ simulation more heat is transferred to the air. Thus, the air is heating up faster during the hot-soak phase and cooling down less during the cool-down phase.

Figure 6-12 shows the time course of the temperature sensors in the front foot room on the left side of the cabin. The results of the temperature sensors in the front foot room on the right are nearly the same. They can be found in Figure A-0-4 in appendices A. As can be seen, for all sensors in the front foot room, the air is heating up more slowly during the hot-soak phase in the co-simulation than in the STAR-CCM+ simulation. Further, the temperatures of the co-simulation are reaching smaller values at the end of the hot-soak phase. In the cool-down phase, the end value of the temperature of the co-simulation and the STAR-CCM+ simulation in the front foot rooms is nearly the same. Due to the small air movement and the fact that no direct solar radiation is falling into the front foot room areas, there is only a small temperature change in this area during the whole 120 minutes of the pull-down simulation.

Figure 6-10 shows the temperature distribution inside the cabin at the end of the cool-down phase of the co-simulation. As can be seen, the highest temperatures occur in front foot rooms, whereas the lowest temperatures occur in front of the backrest of the rear seat. The temperature in the front foot rooms is nearly the same as for the end of the hot-soak phase out of Figure 6-9 (b). The reason for the lowest temperatures in front of the backrest of the rear seat is that a part of the cold air out of the dashboard vents is streaming towards the backrest of the rear seats. Furthermore, cold air out of the two rear vents is streaming directly towards the rear seats, too. Hence there is a high air flow in this area.

The trend of the temperatures in the rear foot rooms is showing a different behavior. Figure 6-13 shows the results for the temperature sensors in the rear foot room on the right side. The results of the temperature sensors in the rear foot room on the left side are illustrated in Figure A-0-1 in appendices A. As can been seen, the trend of the temperature during the hot-soak phase in both simulations is different again, but the temperatures at the end of the hot-soak phase are nearly the same. In the STAR-CCM+ simulation, the air is cooling down at the beginning of the cool-down phase but reaches a steady state with a temperature of around 35 °C very quickly.
In comparison to this, in the co-simulation, the temperature is constantly falling during the cool-down phase. It was expected that the temperature will only change in a very small range and not cooling down so much in the rear foot rooms. Normally there is no active vent during the cool-down out of which cold air is streaming directly into the rear foot rooms. After taking a look at the velocity distribution in the rear area of the cabin, it was determined that the air, which is streaming out of the rear vents, is deflected at the front edge of the rear seat. Hence a part of the cold air is directly streaming into the rear foot rooms. Normally this should not happen. The reason that the air is deflected at the front edge of the rear seat is that during the simplification of the cabin geometry in GT-Spaceclaim it wasn’t worked accurate enough in this area of the cabin. Hence a part of the air streams downwards in the rear foot rooms instead of upwards in the direction of the passengers in the rear seats. With a more accurate simplification of the cabin geometry in the area of the rear seats and foot rooms this error could be avoided. The low temperatures in the rear foot rooms can also be seen in Figure 6-10.

As described in chapter 5.2.3, for the communication interval and the time step size in TAITherm a step profile was used for the co-simulation. During the hot-soak phase, a time step size of one second and during the cool-down phase of 0.5 seconds was used in GT-SUITE. All simulations were run on one core of a laptop. The specifications of the laptop are described in chapter 1.

At the beginning of the first simulation, TAITherm has to calculate the view factor file and the solar lamp apparent areas. Afterwards these files are stored and can be used for other simulations again. The calculation of the view factor file of the cabin model in TAITherm took 51.45 minutes, the calculation of the solar lamp apparent areas took
Results and discussion

27.53 minutes. The simulation of the hot-soak phase took 91.23 minutes, whereas the simulation of the cool-down phase took 129.29 minutes. This leads to a real-time factor – ration between the calculation and the simulation time – of 1.52 for the hot-soak and of 2.15 for the cool-down phase. Hence a whole pull-down simulation took 220.52 minutes, which leads to a real-time factor of 1.84. The calculation times and real-time factors of the final cabin simulations are summarized in Table 6-2.

For all simulations of the cabin model, TAITherm was the limiting factor of the calculation time. GT-SUITE needs around 3 to 4 iterations per time step, whereas TAITherm needs around 40 to 50 iterations per time step. Furthermore, it wasn’t possible to further decrease the time step size in TAITherm otherwise the surface temperatures of some parts are starting to oscillate. One possible reason for the higher calculation times in TAITherm could be the big difference of the shape and size of the mesh elements of the cabin. In the areas of flow openings, like the foot room or defrost vent, the mesh contains very small mesh elements with a strange shape. Another possible reason could be the very detailed layer structure of the parts. Due to this, the surfaces of some parts have a very small thickness. In combination with the material properties, this can lead to an oscillation of the surface temperature, hence TAITherm needs more iterations per time step to reach the tolerance limit of the temperature.

Table 6-2: Final calculation times of the cabin simulation

<table>
<thead>
<tr>
<th>Calculation time [min]</th>
<th>Real-time factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hot-Soak phase</td>
<td>91.23</td>
</tr>
<tr>
<td>Cool-Down phase</td>
<td>129.29</td>
</tr>
<tr>
<td>Pull-Down</td>
<td>220.52</td>
</tr>
</tbody>
</table>
Figure 6-11: Results of the temperature sensors of the cabin model (1)
Results and discussion

Figure 6-12: Results of the temperature sensors of the cabin model (2)
6.2.2 Velocity distribution

For a better understanding of the temperature distribution inside the cabin, it is recommended to take a look at the velocity distribution inside the cabin. Figure 6-14 shows the velocity distribution in x- and z-direction inside the cabin of the STAR-CCM+ simulation and the co-simulation. All plots in Figure 6-14 are showing a cut through the cabin in the x-z-plane in the area of the driver seat (y-direction = -0.45 m). All plots are interpolated contour plots.

In the STAR-CCM+ simulation, the highest velocities occur next to the roof, especially in the rear of the cabin. In the co-simulation, the highest velocities occur next to the
flow inlets and also in the rear of the cabin, where the air is streaming towards the outlet. One main difference between both models is the flow direction at the backrest of the back seat. In the STAR-CCM+ simulation, the air streams downwards, whereas in the co-simulation it is streaming upwards. In the STAR-CCM+ simulation a part of the air is sucked back underneath the driver seat to the front of the cabin.

For the co-simulation, it is hard to make statements about the flow distribution inside the cabin when only looking at the velocity distribution in one plane. Hence, it is recommended to take a look at the velocity distribution in the cabin for several cuts through the model, especially in the areas of the flow inlets. In this way, it is easier to get a good impression of the velocity distribution inside the model. Unfortunately, there are currently no plots in GT-POST which are showing the combined velocity. Hence, the velocity in x-, y- and z-direction must be viewed separately. It is recommended to look at all velocity plots at the same time and combine them to an overall velocity distribution. All plots in Figure 6-15, Figure 6-16 and Figure 6-17 are interpolated contour plots created in GT-POST.

Figure 6-15 shows the velocity in x-direction of the co-simulation for three cuts through the cabin in the x-y-plane (top view). For the three cuts, the z-direction is changing. The first cut – Figure 6-15 (a) – is in the area of the side vents, the second one – Figure...
6-15 (b) – is in the area of the dashboard vents and the third one – Figure 6-15 (c) – in the area of the rear vents. Thus, it is possible to see the velocity in \( x \)-direction in the areas of the active inlet vents.

Figure 6-15: Velocity in \( x \)-direction for cuts through the cabin in the \( x-y \)-plane

Figure 6-16 shows the velocity in \( y \)-direction of the co-simulation for three cuts through the cabin in the \( x-y \)-plane (top view). The area of the cuts in \( z \)-direction is the same like for the velocity in \( x \)-direction in Figure 6-15.
Figure 6-17 shows the velocity in z-direction for a cut through the cabin in the x-z-plane for two different y-positions (side view). The first cut is in the area of the driver seat, the second one is going through the middle of the cabin.

Figure 6-16: Velocity in y-direction for cuts through the cabin in the x-y-plane

When taking a look at the velocity plots in the area of the side vents – Figure 6-15 (a), Figure 6-16 (a) and Figure 6-17 (a) – it becomes clear that the air which is streaming vertically inside the cabin towards the windshield, is separated into two flows when striking at the windshield. Some air is deflected to the middle of the cabin and from there it streams to the rear of the cabin. The rest of the air is streaming to the back of the cabin along the side of the cabin. The deflection of the air at the windshield explains...
the high velocities in y-direction in front area of the cabin in Figure 6-16 (a). In the middle area of the cabin, the air has to stream around the headrests of the front seats. In the rear of the cabin, the air has to stream around the headrests of the backseat. 

Due to the fact that only the three middle dashboard vents are active during the pull-down, the main flow in the area of the dashboard vents occurs in the middle of the cabin. The flow distribution in the area of the dashboard vents can be seen in Figure 6-15 (b), Figure 6-16 (b) and Figure 6-17 (b). As can be seen, the highest velocities in x-direction occur in the middle of the cabin. According to Figure 6-17 (a), some air has to stream towards the front seats because a velocity in z-direction occurs at the backrest of the front seats. According to Figure 6-16 (b) and Figure 6-17 (b), the air is deflected upwards and to the sides of the cabin at the backrest of the rear seat.

The air which is streaming into the cabin through the rear vents is showing a similar behavior than the air out of the dashboard vents. As can be seen in Figure 6-15 (c), the velocities in x-direction close to the back seat are higher, due to the smaller distance between the rear vents and the backrest of the rear seat. Hence in the area of the rear vents, higher velocities in y-direction occur. This can be seen in Figure 6-16 (c). Due to the fact that the air out of the rear vents and the dashboard vents is streaming towards the middle of the backrest of the rear seat, the highest velocities in z-direction in the middle of the cabin occur at the backrest of the rear seat. Figure 6-17 (b) illustrates this situation. Because the air out of the dashboard vents and out of the rear vents are both streaming towards the backrest of the rear seat, a very high air flow is streaming up at the backrest of the rear seat.

Due to the explained velocity distribution inside the cabin model of the co-simulation, the temperature distribution out of Figure 6-10 occurs inside the cabin at the end of the cool-down phase. The air in front of the backrest of the rear seat is cooling down.
the most because of the high air flow in this area. The high temperatures occur in the front foot rooms at the end of the cool-down phase can be explained by the fact that there is no direct air flow in this region of the cabin. Hence, the air in this area is cooling down slowly.

Because of the limitation of the flow in the grid of cube shaped flowsplits explained in chapter 4, the velocity distribution inside the cabin is looking different in the co-simulation than in the STAR-CCM+ simulation. Unfortunately, the impact of the different velocity distribution inside the cabin model on the temperature distribution couldn´t be further investigated because the STAR-CCM+ simulation didn’t deliver good results for a comparison.
7 Conclusion

For the conclusion of this work, the model building process in TAITherm and COOL3D is evaluated. The cabin modeling will be the main application area of the new co-simulation methodology between TAITherm and GT-SUITE. Due to this and the fact that the model building process for the simple box model and for the cabin model are nearly the same, in this chapter the model building process of the cabin is used to evaluate the model building process. Afterwards, the simulation results of the simple box model and the cabin model are evaluated. This chapter ends with a look into the future of the new cabin modeling method.

7.1 Model building

For the creation of the cabin model in TAITherm, a solid geometry of the cabin was imported in TAITherm. One drawback of the model building process of the cabin model in TAITherm is that, the geometry already has to include a simulation mesh when importing it into TAITherm. There is no opportunity to add a mesh to imported geometries. An opportunity to mesh imported parts would be nice. Further, an opportunity to simplify an imported mesh through combining smaller elements would be useful. The mesh of the used cabin geometry had some very small mesh elements which could be combined for the usage of the geometry in TAITherm because a high resolution of the mesh is not required for the co-simulation process.

After the geometry was imported in TAITherm, it was separated into the parts of the cabin which were modeled with different material and surface properties. A detailed layer structure was used for the modeling of the separate parts in TAITherm.

For the modeling of the solar radiation in TAITherm, the solar lamp option is the best opportunity to create a field of solar lamps like in a climate wind tunnel. One drawback of the solar lamp part is that, a position of a pyranometer has to be defined. Hence, it requires the results of a climate wind tunnel test or a good knowledge about the expected solar radiation distribution. Otherwise, a weather file can be used to model the solar radiation and also include other weather phenomena in the simulation. However, the modeling of a solar radiation only with a weather file is complicated and time intense because the weather file has to be manipulated and lots of other variables have to be defined like the global position and the background. Further, the start and end time of the simulation has to be defined with a day and daytime and cannot just
simply be set to a simulation duration. In a pull-down simulation, it is mainly necessary to model a constant solar radiation with a constant zenith angle over the whole simulation time. Sometimes a diffusive solar radiation should be included in the model, too. One possible way to model the solar radiation during a pull-down simulation could be, to define the magnitude of direct solar radiation and the zenith angle. Further, a diffusive radiation could be defined which acts around the whole cabin model. All those variables could be made dependent on time if it is needed. This would only require the definition of three variables, hence it would spare a lot of time in comparison to manipulate a weather file and the process would be easier.

To be able to import the cabin geometry into the pre-processing program COOL3D, it was necessary to simplify the geometry in GT-Spaceclaim before. Otherwise, it was difficult to handle the very detailed cabin geometry in COOL3D. The simplification of the cabin geometry in GT-Spaceclaim was difficult and time intense. An automated process is needed for this step of the model building. One possible way could be to use the volume extract option in GT-Spaceclaim and extract the volume inside the cabin geometry. For this it would be necessary to add something like an acceptance ratio for tilted surfaces as process variable, otherwise this method would extract exactly the same geometry again. As explained in chapter 5.2.2, it is not necessary to include every detail of the cabin geometry into the COOL3D model because the model will be discretized into cube shaped sub-volumes. The goal of this automated process has to be, to work as detailed as possible but only as detailed as required. Hence, the high number of tilted surfaces of the geometry should be erased during this automated process but the outer boundaries of the cabin geometry should remain the same.

With the manually created simplification of the cabin geometry, the cabin model in COOL3D was generated. Afterwards, the flow openings and temperature sensors were added to the model and the discretization size was defined. Before the model was exported to GT-SUITE, it was necessary to make sure that the interior of the cabin, like the seats, are actually regarded as flow blockages by cut out some flowsplits of the grid of flowsplits. This process is described in detail in chapter 5.2.2. It can be said that the whole process would be much easier with a local “Flowsplit Acceptance Ratio”. The seats and the other interior could be imported separately in COOL3D and converted to flow blockages. Afterwards a different local “Flowsplit Acceptance Ratio” could be used for each part, to make sure that the flowsplits which are interacting with them are thrown out of the model during the export. Such a method would have the advantage that the problems with thrown out flowsplits at the outer boundaries of the cabin model would be avoided.

After the model building in COOL3D was finished, the model was exported to GT-SUITE. In GT-SUITE, the output signals of the temperature sensors had to be manually
connected to a “SampledOutput” template. An automated storage of the sensor output signals as time plots could be a good improvement. Then it would no longer be necessary to manually connect all output signals of the sensors to store the time plots of them.

Before the co-simulation between GT-SUITE and TAITherm could be started, the co-simulation variables like the time before the first data exchange, the communication interval and the time step size in both programs had to be defined. In this work these variables were manually optimized regarding the numerical convergence in both programs and the calculation time. Due to the fact that this co-simulation method should also be used for transient simulations as well, it would be nice to be able to use a dynamic profile for the communication interval and time step size. In transient simulations, the boundary conditions, like the inlet temperature, are changing with the simulation time and the size and point of change is unknown before the simulation. The size of the communication interval and the time step size in both programs could automatically change depending on the trend of another simulation parameter, for example the temperature gradient between the inlet and outlet temperature. However, this is isn’t so easy to implement because it requires the ability to estimate the temperature changes in the simulation before they actually happen.

At the moment, it is only possible to initialize all flowsplits of the cabin model with the same air temperature. For the simulation of a cool-down phase without modeling the hot-soak phase before, it is necessary to initialize the air inside the car cabin by using different temperatures for the horizontal layers of flowsplits. One opportunity to regard the temperature stratification inside the cabin before the cool-down phase could be, to define a vertical temperature gradient for the initialization temperature of the air. Each horizontal layer would be initialized with a different air temperature. The vertical temperature gradient would define the temperature difference between the top and bottom horizontal layer of flowsplits in the cabin model. Alternatively, the initial temperature of each horizontal layer could be defined manually. Another opportunity to regard the temperature stratification inside the cabin before the cool-down phase could be, to map a temperature field from the end of a hot-soak simulation of a CFD simulation on the grid of flowsplits. Afterwards, the mapped temperature field would be used as initialization temperatures for the flowsplits.

Summarized it can be said that the model building process in TAITherm and COOL3D is already having a high degree of automation and it was possible to create a cabin model for the co-simulation out of a solid cabin geometry in both programs. As mentioned above, some improvements have to be made to optimize and speed up the model building process in both programs.
7.2 Simulation results

As described in chapter 6.1.1, for the simple box model, the differences between the co-simulation and STAR-CCM+ simulation results are mainly related to the different kind of flow field in both programs. In the co-simulation, there is a Quasi-3D flow field with the flow limitations explained in chapter 4, whereas in the STAR-CCM+ simulation there is a 3D flow field. Furthermore, the discretization study in chapter 6.1.2 showed that the positioning of the temperature sensors inside the model and the positioning of the sub-volumes in front of the flow openings can have a big influence on the simulation results. Furthermore, the position of the flow outlet influences the flow distribution inside the simple box model of the co-simulation. With the first discretization size, acceptable results and calculation times could be achieved for the simple box model.

The analysis of the simulations results of the cabin model was more complicated due to the weird STAR-CCM+ simulation results. As described in chapter 6.2.1, for the hot-soak phase it was necessary to include the gravitational acceleration in the co-simulation otherwise there was no temperature stratification inside the cabin at the end of the hot-soak phase. The pull-down simulation results of the co-simulation and the STAR-CCM+ simulation were looking very different. One difference between both simulations was the modeling of the walls. As explained in chapter 5.2.4, in the co-simulation the walls of the cabin were modeled by using the very detailed layer structure. Unlike, in the STAR-CCM+ simulation, the walls were modeled as walls with a constant thickness of 3 mm and averaged material properties of the detailed layer structure. Further, for the STAR-CMM+ simulation it seemed like there was a constant heat source inside the cabin because a high amount of heat was constantly transferred to the air inside the cabin. As mentioned in chapter 6.2.1, due to the big difference between the results of the two simulations, it was hard to compare the simulation results and make statements about the accuracy of the co-simulation results.

Furthermore, with the used positions of the temperature sensors it was hard to make statements about the thermal comfort inside the cabin. The in the cabin model used temperature sensors are normally used to make statements about the trend of the temperature inside the cabin during the pull-down simulation to comply with the regulations out of DIN 1946 - Part 3. To be able to make precise statements about the thermal comfort of a human body inside a passenger car cabin, the temperature sensors have to be placed at other locations inside the cabin. For example, in the area of the head, chest (seat belt) or feet of the passenger.

The velocity distribution inside the cabin was analyzed for the co-simulation and compared to the velocity distribution in the STAR-CCM+ simulation as well. It can be said that the differences between the velocity distribution in both simulation models are
mainly caused by the different flow fields, which also applies to the simple box model. For a better and easier comparison of the flow fields in both simulation models, it would be nice to have the opportunity to plot the velocity results of the co-simulation as vector plots with an overall velocity for each flowsplits. Hence it would no longer be required to analyze three different velocity plots and combine them in your head.

7.3 Future work

One main task for the future is to test the new cabin modeling method by using measurement data. First the cabin model has to be calibrated by using measurement data. Afterwards, the simulation results should be compared to measurement results as well. Further it is recommended to place temperature sensors inside the cabin at locations where statements about the thermal comfort can be made.

In this work the co-simulation cabin model was only used to simulate a pull-down. Another important test scenario is the heat-up. In this test scenario, the air inside the cabin is initialized with a temperature of –20 °C. The car is driving with a constant velocity and the HVAC system is operating with maximum power. Hot air is blown into the cabin, to heat up the air inside of it. After 30 minutes, the average air temperature inside the cabin should reach a value of 20 °C. The regulation DIN 1946 - Part 3 includes a detailed description of the heat-up test scenario. To further investigate and test the new cabin modeling method, it is recommended to perform heat-up simulations as well.

For a better fit of the flow distribution inside the cabin between the co-simulation and a CFD simulation, a new method was developed from Gamma Technologies. This method is called “CFD Flow Field Mapping”. In this method, a steady flow field of a CFD simulation is mapped on the grid of flowsplits of the co-simulation model. For this, the velocities of the steady CFD flow field are transferred to the flowsplits. For one simulation, several steady flow fields of a CFD simulation can be used dependent on the simulation time. This seems to be a promising method to get a better fit of the flow distribution of the co-simulation and a CFD simulation. In the future, the effects of this method on the flow and temperature distribution inside the cabin should be investigated and evaluated.
A Figures

Figure A-0-1: Results of the temperature sensors of the cabin model (4)
Figure A-0-2: Results of the temperature sensors of the cabin model (5)
Figure A-0-3: Results of the temperature sensors of the cabin model (6)
Figure A-0-4: Results of the temperature sensors of the cabin model (7)
# Tables

Table B-0-1: Layer structure of the parts of the box model

<table>
<thead>
<tr>
<th>Part Name</th>
<th>Layers</th>
<th>Component</th>
<th>Material</th>
<th>Thickness [mm]</th>
<th>Surface emissivity&amp; absorptivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Floor</td>
<td>5</td>
<td>Front Layer</td>
<td>Teppich</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 3</td>
<td>PU-Hartschaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 4</td>
<td>PE-Weichschaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Window</td>
<td>1</td>
<td>Front Layer</td>
<td>Glass, Conventional Automotive</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roof</td>
<td>3</td>
<td>Front Layer</td>
<td>Verbund Dach</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>Blech</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Door</td>
<td>8</td>
<td>Front Layer</td>
<td>KST-Leder</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 3</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 4</td>
<td>Naturfaserpressteil</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 5</td>
<td>Vlies</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 6</td>
<td>Aluminiumblech</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 7</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Aluminiumblech mit Luftspalt</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Front</td>
<td>2</td>
<td>Front Layer</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Back</td>
<td>3</td>
<td>Front Layer</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Seat</td>
<td>2</td>
<td>Front Layer</td>
<td>Echtleder</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>PU-Weichschaum</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: Some values in the table are blacked out due to a non-disclosure agreement between Daimler and Gamma Technologies
# Table B-0-2: Layer structure of the parts of the cabin model

<table>
<thead>
<tr>
<th>Part Name</th>
<th>Layers</th>
<th>Component</th>
<th>Material</th>
<th>Thickness [mm]</th>
<th>Surface emissivity &amp; absorptivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windshield</td>
<td>1</td>
<td>Front Layer</td>
<td>Glass, Conventional Automotive</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Side Windows</td>
<td>1</td>
<td>Front Layer</td>
<td>Glass, Conventional Automotive</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rear Window</td>
<td>1</td>
<td>Front Layer</td>
<td>Glass, Conventional Automotive</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Roof</td>
<td>3</td>
<td>Front Layer</td>
<td>Verbund Dach</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>Blech</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Doors</td>
<td>8</td>
<td>Front Layer</td>
<td>KST-Leder</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 3</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 4</td>
<td>Naturfaserpresseteil</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 5</td>
<td>Vlies</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 6</td>
<td>Aluminiumblech</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 7</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Aluminiumblech mit Luftspalt</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rear Shelf</td>
<td>3</td>
<td>Front Layer</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 2</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Seats</td>
<td>2</td>
<td>Front Layer</td>
<td>Echtleder</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>PU-Weichschaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Floor, Firewall</td>
<td>5</td>
<td>Front Layer</td>
<td>Teppich</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Middle Console</td>
<td></td>
<td>Layer 2</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 3</td>
<td>PU-Hartschaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Layer 4</td>
<td>PE-Weichschaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Center Console,</td>
<td>2</td>
<td>Front Layer</td>
<td>PU-Schaum</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Glovebox, Dashboard, Steering Wheel</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Back Layer</td>
<td>PP-Traeger</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: Some values in the table are blacked out due to a non-disclosure agreement between Daimler and Gamma Technologies.
Table B-0-3: Material properties from Daimler

<table>
<thead>
<tr>
<th>Material Name</th>
<th>Density [kg/m³]</th>
<th>Specific Heat [J/(kgK)]</th>
<th>Conductivity [W/(mK)]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Glass, Conventional Automotive</td>
<td>2529.6</td>
<td>754.04</td>
<td>1.1717</td>
</tr>
<tr>
<td>PP-Traeger</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Blech</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>KST-Leder</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Naturfaserpressteil</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Vlies</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aluminiumblech</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Echtleder</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Teppich</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PU-Hartschaum</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PE-Weichschaum</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aluminiumblech mit Luftspalt</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Verbund Dach</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PU-Weichschaum</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PU-Schaum</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Luftspalt/Steuergeraete</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Note: Some values in the table are blacked out due to a non-disclosure agreement between Daimler and Gamma Technologies.
### Table B-0-4: Boundary and initial conditions of the box model simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air mass flow rate</td>
<td>0.15 kg/s</td>
</tr>
<tr>
<td>Inlet air temperature</td>
<td>8 °C</td>
</tr>
<tr>
<td>Outlet pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Ambient temperature</td>
<td>40 °C</td>
</tr>
<tr>
<td>HTC outside surfaces</td>
<td>50 W/(m²*K)</td>
</tr>
<tr>
<td>Solar radiation</td>
<td>1000 W/m²</td>
</tr>
<tr>
<td><strong>Initial air temperature</strong></td>
<td>80 °C</td>
</tr>
<tr>
<td><strong>Initial air pressure</strong></td>
<td>1 bar</td>
</tr>
<tr>
<td><strong>Initial part temperatures</strong></td>
<td>80 °C</td>
</tr>
<tr>
<td><strong>Initial HTC coefficient inside surfaces</strong></td>
<td>5 W/(m²*K)</td>
</tr>
<tr>
<td><strong>Simulation time</strong></td>
<td>6 min</td>
</tr>
</tbody>
</table>

### Table B-0-5: Boundary and initial conditions of the hot-soak simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air mass flow rate</td>
<td>0 kg/s</td>
</tr>
<tr>
<td>Outlet pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Ambient temperature</td>
<td>40 °C</td>
</tr>
<tr>
<td>HTC outside surfaces</td>
<td>Dependent on vehicle velocity</td>
</tr>
<tr>
<td>Solar radiation</td>
<td>1000 W/m²</td>
</tr>
<tr>
<td>Vehicle velocity</td>
<td>0 km/h</td>
</tr>
<tr>
<td>Initial air temperature</td>
<td>40 °C</td>
</tr>
<tr>
<td>Initial air pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Initial part temperatures</td>
<td>40 °C</td>
</tr>
<tr>
<td>Initial HTC coefficient inside surfaces</td>
<td>5 W/(m²*K)</td>
</tr>
<tr>
<td><strong>Simulation time</strong></td>
<td>60 min</td>
</tr>
</tbody>
</table>
Table B-0-6: Boundary and initial conditions of the cool-down simulation

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet air MFR side vents (2x)</td>
<td>0.03805 kg/s</td>
</tr>
<tr>
<td>Inlet air MFR dashboard vents (3x)</td>
<td>0.01522 kg/s</td>
</tr>
<tr>
<td>Inlet air MFR rear vents (2x)</td>
<td>0.01522 kg/s</td>
</tr>
<tr>
<td>Inlet air MFR total</td>
<td>0.1522 kg/s</td>
</tr>
<tr>
<td>Inlet air temperature all vents</td>
<td>5.43 °C</td>
</tr>
<tr>
<td>Outlet pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Ambient temperature</td>
<td>40 °C</td>
</tr>
<tr>
<td>HTC outside surfaces</td>
<td>Dependent on vehicle velocity</td>
</tr>
<tr>
<td>Solar radiation</td>
<td>1000 W/m²</td>
</tr>
<tr>
<td>Vehicle velocity</td>
<td>32 km/h</td>
</tr>
<tr>
<td>Initial air temperature</td>
<td>From hot-soak / 65°C</td>
</tr>
<tr>
<td>Initial air pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Initial part temperatures</td>
<td>From hot-soak / 80 °C</td>
</tr>
<tr>
<td>Initial HTC coefficient inside surfaces</td>
<td>5 W/(m²*K)</td>
</tr>
<tr>
<td>Simulation time</td>
<td>60 min</td>
</tr>
</tbody>
</table>
References


