

Experimental and Numerical Investigation of the Under Hood Flow with Heat Transfer for a Scaled Tractor-Trailer

Beat Heinzelmann
Daimler AG

Thomas Indinger and Nikolaus Adams
Technical University of Munich

Reinhard Blanke
Daimler AG

ABSTRACT

Aerodynamic design and thermal management are some of the most important tasks when developing new concepts for the flow around tractor-trailers. Today, both experimental and numerical studies are an integral part of the aerodynamic and thermal design processes. A variety of studies have been conducted how the aerodynamic design reduces the drag coefficient for fuel efficiency as well as for the construction of radiators to provide cooling on tractor-trailers. However, only a few studies cover the combined effect of the aerodynamic and thermal design on the air temperature of the under hood flow [8, 13, 16, 17, 20].

The objective of this study is to analyze the heat transfer through forced convection for a scaled Cab-over-Engine (CoE) tractor-trailer model with under hood flow. Different design concepts are compared to provide low under hood air temperature and efficient cooling of the sub components. The measured data have been used to validate the computational simulations performed with STAR-CCM+.

At first, the experimental and numerical investigation of a transitionally rough surface on a flat plate with zero pressure gradient at high Reynolds number is presented to understand more about the boundary layer profile and skin friction coefficient. Furthermore, a generic study examines the temperature field of the near-wake flow for a heated surface-mounted block.

In the main experimental and numerical part of the work, some first results of the heat transfer for the scaled tractor-trailer model with under hood flow are given. In the experiments, hot air has been generated by flowing through a heated scaled radiator to investigate the effect of the under hood mass flow by covering the inlet. For the standard configuration the measured air temperature distribution after the heat exchanger and under hood side wall inside the engine compartment as well as the drag coefficient has been compared to the numerical results.

CITATION: Heinzelmann, B., Indinger, T., Adams, N. and Blanke, R., "Experimental and Numerical Investigation of the Under Hood Flow with Heat Transfer for a Scaled Tractor-Trailer," *SAE Int. J. Commer. Veh.* 5(1):2012, doi: 10.4271/2012-01-0107.

INTRODUCTION

WALL ROUGHNESS

The nature of the wall in technical applications is more or less rough. The roughness of the wall is considerably important for technical applications because roughness has a huge influence on the behavior of the flow. There is an infinite amount of textures of a wall which depend on the manufacturing process (milling, turning, honing, grid

blasting). This leads to the point that every surface has its own technical roughness k_{tech} .

There are many investigations for sand grain roughness, single roughness elements or two-dimensional shaped roughness elements distributed on the surface [1, 11, 14, 15, 18]. To compare these results and specify the effect of the roughness on the boundary layer, Schlichting [19] recommended assigning every technical roughness to an equivalent sand grain roughness. The equivalent sand grain



Documents

Document Sharing Hub

roughness is defined as spherical sand uniformly distributed on the surface. The height of these spherical elements is called the sand grain roughness height k_s . It is common to link the roughness height to the boundary layer by using the following dimensionless roughness characteristic number

$$k_s^+ = \frac{k_s}{\delta_v} = \frac{k_s \cdot u_\tau}{\nu} \quad (1)$$

Where δ_v is the characteristic height of the viscous sublayer and $u_\tau = \sqrt{\tau_w / \rho}$ is the friction velocity of the boundary layer. Sometimes k_s^+ is called the roughness Reynolds number.

The turbulence characteristic in the viscous sublayer of the boundary profile is affected by the roughness characteristic number k_s^+ . A surface is aerodynamically smooth when the roughness height is so small that the roughness is buried in the viscous sublayer. When the roughness elements are higher than the sublayer, the sublayer is completely eliminated and the flow can be considered to be independent of the molecular viscosity. The roughness can therefore be divided into the following three regions by Cebeci and Bradshaw [6]. Aerodynamically smooth flow occurs for $k_s^+ \leq 2.25$. Schlichting found that $k_s^+ < 5$. Negligible effects of the roughness are for $2.25 < k_s^+ < 90$. Fully rough flow typically occurs for $k_s^+ \geq 90$.

THE LAW OF THE WALL FOR SMOOTH AND ROUGH SURFACE

Cebeci and Bradshaw [6] assume to split the turbulent boundary layer for the two-dimensional flow over a smooth flat plate in steady conditions in three regions consisting of the viscous sublayer including the buffer zone, the log-law region and the outer layer. The mean velocity distribution is given by the well known law of the wall. For the viscous sublayer,

$$u^+ = y^+ \quad (2)$$

where $y^+ = y \cdot u_\tau / \nu$ is called the dimensionless distance from the wall and $u^+ = u / u_\tau$. In this region the viscous plus turbulent shear stress varies only slowly with the distance from the wall and the mean velocity distribution is completely determined by the friction velocity, the viscosity and the distance from the wall. The thickness of the viscous sublayer is about $y^+ \leq 5$.

When the dimensionless distance from the wall is between $5 < y^+ < 50$ the region is called buffer zone. For y^+ larger than about 50 the mean velocity departs from [equation \(2\)](#) and is given by the famous logarithmic law,

$$u^+ = \frac{1}{\kappa} \ln y^+ + C_s^+ \quad (3)$$

where κ is a constant and experimentally determined to be about 0.41. Other authors found κ to be between 0.40 and 0.42. C_s^+ generally depends on the wall roughness. For a smooth surface without a pressure gradient C_s^+ is constant and experimentally determined to be about 5.0- 5.2. Other authors found C_s^+ to be between 4.8 and 5.3, see E.-S. Zanoun [21].

On rough surface the mean velocity distribution only affects the inner region of the boundary layer. Cebeci and Bradshaw assume that the law of the wall for rough surface with uniform roughness elements is a function of the dimensionless distance from the wall and the roughness Reynolds number $u^+ = f(y^+, k_s^+)$. For a given sand grain roughness the law of the wall can be written as

$$u^+ = \frac{1}{\kappa} \ln y^+ + C_r^+ \quad (4)$$

It is reasonably assume κ to be the same as on smooth surfaces, because κ is not affected by the sublayer conditions. C_r^+ is now a function of the roughness Reynolds number $C_r^+ = f(k_s^+)$ for a given shape of roughness elements and must be determined from experiments. The function of C_r^+ is given by Cebeci and Bradshaw [6] for smooth, transitionally rough and fully rough surface, which can be seen for the three regions in [table 1](#).

Table 1. Integration constant C_r^+ as a function of k_s^+ by Cebeci and Bradshaw [6].

k_s^+	$C_r^+ = f(k_s^+)$
smooth: $k_s^+ \leq 2.25$	$C_r^+ = 5.2$
transitionally rough: $2.25 < k_s^+ < 90$	$C_r^+ = C_s^+ + [8.5 - C_s^+ - \frac{1}{\kappa} \ln k_s^+] \sin 0.4258 \cdot (\ln k_s^+ - 0.811)$
fully rough: $k_s^+ \geq 90$	$C_r^+ = 8.5 - \frac{1}{\kappa} \ln k_s^+$

For $k_s^+ \leq 2.25$ the roughness elements are smaller than the viscous sublayer and therefore C_r^+ is a constant value. When the roughness Reynolds number is between $2.25 < k_s^+ < 90$ the roughness elements are larger than the viscous sublayer and have a significant effect on the boundary layer. C_r^+ is then a

function of the roughness height. For $k_s^+ \geq 90$ the roughness elements take up the entire wall layer so that the viscosity is of no further importance and C_r^+ is nearly constant. In general the velocity profile of the rough surface follows the same logarithmic profile as the smooth surface, except that there is a linear shift of the Profile Uu^+ in negative direction with increasing wall roughness.

STAR-CCM+ uses a slightly expanded version of the expression given by Cebeci and Bradshaw. The integration constant C_r^+ is defined as:

$$C_r^+ = \frac{1}{\kappa} \ln \left(\frac{E}{f} \right), \quad (5)$$

where $\kappa = 0.42$ and $E = 9.0$. The roughness function f that modifies the log-law coefficient E is a function of the roughness Reynolds number and different to the formulation given by Cebeci and Bradshaw. The Roughness function is given in table 2 as follows:

Table 2. Roughness function in STARCCM+ [5].

k_s^+	$f = f(k_s^+)$
smooth: $k_s^+ \leq 2.25$	$f = 1$
transitionally rough: $2.25 < k_s^+ < 90$	$f = (0.253 \cdot k_s^+)^a$
fully rough: $k_s^+ \geq 90$	$f = 0.253 \cdot k_s^+$

The exponent is expressed as:

$$a = \sin \left[\frac{\pi}{2} \frac{\log \left(\frac{k_s^+}{2.25} \right)}{\log \left(\frac{90}{2.25} \right)} \right]. \quad (6)$$

There is a shift of the integration constant for the transitionally rough region between both formulations from Table 1 and Table 2. This leads to a larger integration constant in STAR-CCM+ and a smaller shift of the velocity profile compared to the experimental results of Cebeci and Bradshaw.

BOUNDARY LAYER PARAMETERS

The boundary layer thickness in laminar flow δ_l over a smooth flat plate is given by the point where the velocity reaches a certain percentage of the free stream velocity, e.g. 99%. The dimensionless boundary layer thickness related to the local length x of the plate is defined as

$$\frac{\delta_l}{x} = \frac{5}{\text{Re}_x^{1/2}}. \quad (7)$$

For clarity an index l is used for laminar flow. The Reynolds number Re_x is defined as $\text{Re}_x = \frac{U_\infty \cdot x}{\nu}$ and built with the local length, starting from the leading edge of the flat plate. The boundary layer thickness over a smooth flat plate in turbulent flow by Albring [2] is:

$$\frac{\delta_t}{x} = \frac{0.370}{\text{Re}_x^{1/5}}. \quad (8)$$

The index t is used for turbulent flow and x is measured from the leading edge of the plate.

When using the power-law assumption for the velocity distribution the local skin friction coefficient is a function of the Reynolds number and given by Schlichting [19] $c_{f_s} = 0.059 / \text{Re}_x^{1/5}$. A more accurately empirical formula is based on the momentum thickness δ_2 and would then be expected to follow the equation given by Kays [12]:

$$c_{f_s} = 0.025 \left(\frac{\delta_2 \cdot U_\infty}{\nu} \right)^{-0.25}. \quad (9)$$

For the rough surface the skin friction coefficient is a function of the Reynolds number and the roughness height $c_{f_r} = f(\text{Re}_x, k_s)$. From measurement of P. M Ligrani et. al [15] the skin friction coefficient of a rough surface can be written as

$$c_{f_r} = 2a \left(\frac{2\delta_2}{k_s} \right)^{-b} \quad (10)$$

where k_s is the sand grain roughness. At a given free stream velocity of $U_\infty = 10.1 \text{ m/s}$ the constant $a = 0.00381$ and the constant $b = 0.332$. Both coefficients are a function of the free stream velocity.

SIMILARITY LAW

When using a scaled model the question arises whatever or not the flow field of the scaled model is physically similar to the actual object. This can only be answered, when looking at the similarity parameters. The knowledge of the right similarity parameters is of fundamental importance for the analysis of the flow field for scaled models. Therefore, a lot of authors, e.g. [4, 19], explain the method to find the right similarity parameters by writing the momentum and energy equation with dimensionless parameters. Some detailed scaling relations for heat exchangers can be found in [10].

The mechanical similarity of two flows is satisfied when the Reynolds number

$$\text{Re} = \frac{U_{\infty} \cdot l}{\nu} \quad (11)$$

for the scaled model and the actual object is the same. To ensure a full similarity between the heat and momentum transfer the turbulent Prandtl number

$$\text{Pr} = \frac{\mu \cdot c_p}{k} = \frac{\nu}{\alpha} \quad (12)$$

has to be equal between the scaled model and the actual object. When replacing the dynamic viscosity $\mu = \rho \cdot \nu$ the Prandtl number is the ratio of momentum diffusivity ν to thermal diffusivity α . For full similarity, the Eckert number also has to be similar. The Eckert number is the relationship between the kinetic energy and the enthalpy. The Eckert number has to be observed, when the friction of the fluid creates a rise of the fluid temperature. In this experimental investigation, the free stream velocity causes no significant temperature rise and therefore the effect of the Eckert number can be neglected.

MOMENTUM TRANSFER IN TURBULENT BOUNDARY LAYER ON A FLAT PLATE FOR SMOOTH AND ROUGH SURFACE

PRESENT APPROACH

The turbulent boundary layer on a smooth and transitionally rough surface with zero pressure gradient is investigated. A single hot wire probe is used to measure the mean velocity profile over the flat plate at different downstream positions. The free stream velocity is set to $U_{\infty} = 10 \text{ m/s}$ which corresponds to a Reynolds number of $\text{Re}_l = 632000$. The smooth and rough surface is made of aluminum sheet and can be replaced. The nature of the first surface is $k_s^+ = 0$ to generate an aerodynamically smooth environment. The nature of the second surface is transitionally rough with a roughness parameter of $k_s^+ = 40$. To calculate the skin friction coefficient from the mean velocity profile the well known Clauser method has been used, see Francis H. Clauser [7]. The accuracy of this method depends on the accuracy of the measured velocity points near the wall.

The experimental data of the velocity profile and the skin friction coefficient are compared to numerical data taken. Therefore, two different numerical approaches were set up containing the vertical mounted flat plate in a three-dimensional flow. For the grid generation and the numerical simulation the commercial CFD code STAR-CCM+ Version 6.04 has been used. For the smooth surface a low Reynolds

mesh and for the transitionally rough surface a high Reynolds mesh is generated. With the wide experimental basis a validation of the boundary layer profile and the skin friction coefficient for the smooth and rough surface is achieved.

GEOMETRY AND SETUP

Geometry of the flat plate

The geometry of the flat plate shown in [figure 1](#) is chosen to investigate the turbulent boundary layer and the skin friction coefficient.

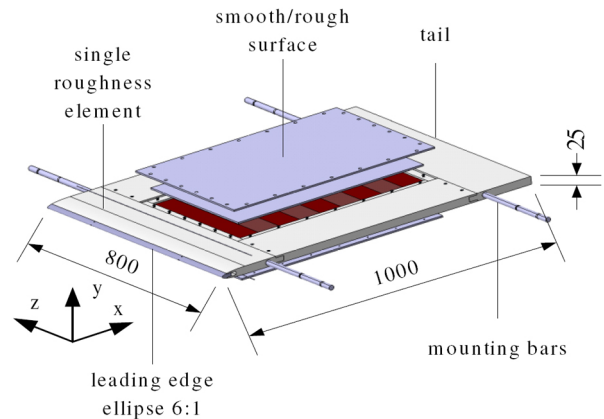


Figure 1. Three dimensional geometry of flat plate with leading edge.

The dimension of the flat plate is $l = 1000 \text{ mm}$, $b = 800 \text{ mm}$ and $d = 25 \text{ mm}$. For a small increase of the local pressure at the leading edge of the flat plate a 6:1 ellipse is used. The plate is mounted vertically with four bars to the upper and lower wall of the wind tunnel. The middle part of the flat plate has a size of $x = 600 \text{ mm}$, $y = 400 \text{ mm}$ and can be replaced to provide different surface roughness. The distance of the leading edge from the nozzle is 600 mm . The detail of the development of the turbulent boundary and the coordinate system used for the present study is shown in [figure 2](#).

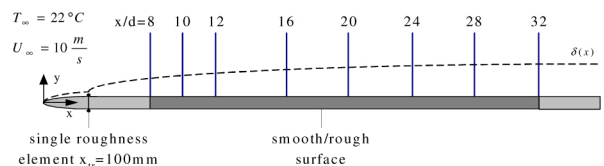


Figure 2. Coordinate system and location of the boundary layer profiles at $x/d=8, 10, 12, 16, 20, 24, 28, 32$.

The laminar boundary layer starts at the leading edge of the flat plate. After a length of $x_{tr} = 100 \text{ mm}$ a single roughness element with a diameter of $d_w = 1 \text{ mm}$ is placed on the surface to generate a well-defined transition point of the flow. The design of the single roughness element is corresponding to the formula given by Dryden [9]. Related to

this guideline the height of the single roughness element is not higher than the viscous sublayer.

Mean velocity profiles are taken for the positions $x/d = 8, 10, 12, 16, 20, 24, 28, 32$ in the symmetry plane of the plate. The roughness of the smooth surface is $Ra = 1.2\mu m$, measured by a surface roughness analyzer at three different positions on the surface. With $k_s = 4.2 \cdot Ra$, recommended by Schlichting, the roughness parameter is calculated. The dimensionless roughness characteristic number from equation 1 is $k_s^+ \approx 0$. For the rough surface sand grains are scattered all-over the aluminum sheet to generate the roughness. To get sharp-edged sand grains with a diameter between $1mm \leq d_s \leq 2mm$ the sand was filtered with two different meshes. The mean diameter of the sand grains is equivalent to a sand grain roughness parameter of $k_s = 2.5mm$. The dimensionless roughness characteristic number is then $k_s^+ = 40$. The flow from the leading edge to the rough surface at $x = 200mm$ can be considered as aerodynamically smooth.

Experimental Apparatus and Procedure

The wind tunnel C of the Technical University Munich is a subsonic closed circuit wind tunnel and generally used for studies on buildings, structures and the near wake flow of airplanes. Therefore, the test section of the wind tunnel is 21m long. The cross section of the wind tunnel is 1.8m by 2.7m. The top wall can be adjusted to avoid a static pressure gradient over the test section. Honeycomb flow straightener and turbulence reducing screens are located in front of the contraction section. The contraction ratio is 2.11:1. The fan is located behind the collector after the turning vanes to suck the air. The turbulence intensity at the beginning of the test section is $Tu_x \leq 0.5\%$. More details of the wind tunnel can be found in [3].

To measure the mean velocity distribution, a single hot wire probe type 55P15 from Dantec Dynamics with cranked probe is used. The wire is a wolfram wire with a diameter of $d_p = 5\mu m$ and the probe permits measurements close to a solid wall without disturbance from the probe body. The hot wire probe is operated by an anemometer system AN-1005 from A.A.Lab Systems Ltd. The hot wire probe is mounted to the traverse system together with a thermocouple and prandtl tube to measure the temperature, total and static pressure at free stream position. The configuration is shown in figure 3.

The traverse system is an Isel Germany AG linear unit placed at the bottom of the flat plate to avoid flow interaction between the surface and the traverse system. The mounting of the hot wire probe has a diameter of $d_m = 4mm$. The angle between the mounting and the surface is approximately $\beta \approx 25^\circ$ to avoid a local increase of the pressure due to the probe body. The accuracy of the traverse system is $\Delta s = \pm 0.02mm$. The software LabVIEW is used to operate the traverse and acquire the analog signal from the anemometer.

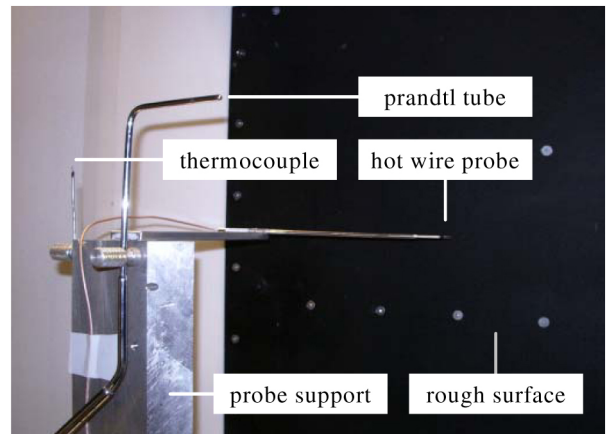


Figure 3. Experimental setup of the flat plate with hot wire probe and traverse system.

Numerical Setup

The numerical domain for the flat plate is a three dimensional volume mesh. The flat plate consisting of the leading edge and the surface itself together with the mounting bars is placed in the middle of a huge domain $x = 7m, y = 2m$ and $z = 4m$. The velocity inlet is located $x_i = -2000mm$ from the leading edge. A cut through the finite volume mesh for the smooth and rough surface is presented in figure 4.

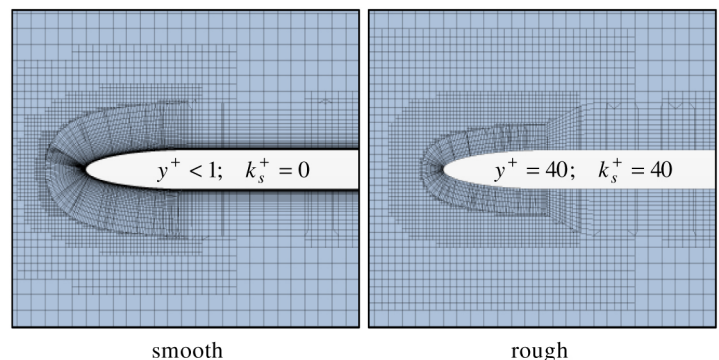


Figure 4. Mid plane of the finite volume mesh for the smooth and rough surface.

For the experiment, a single roughness element was placed after the ellipse at $x_{tr} = 100mm$ to require a well defined transition point and to start the turbulent boundary layer. For the simulation the single roughness element is not used. Hence, the boundary condition from the leading edge to the transition point x_{tr} is set to a laminar behavior. The thickness of the sublayer near to the wall is 30mm and reduced to 8mm from the leading edge to the transition point for the rough surface.

The viscous sublayer of the smooth surface is resolved by the mesh to obtain reliable results for the skin friction coefficient. Thus the dimensional distance from the wall for the low Reynolds mesh is $y^+ < 1$. In STAR-CCM+ roughness