

AERODYNAMIC DEVICES TO REDUCE THE BASE- AND UNDERBODY DRAG OF SEMITRAILER UNIT

Matěj Sulitka, Ing.,
Czech Technical University in Prague,
Institute of Fluid Dynamics and Power
Engineering, Division of Fluid Dynamics
and Thermodynamics,
Technická 4, 166 07 Praha 6,
Czech Republic;
tel.: ++420 224 352 661
fax: ++420 224 310 292
matej.sulitka@fs.cvut.cz

Jiří Nožička, Prof., Ing., Ph.D.,
Czech Technical University in Prague,
Institute of Fluid Dynamics and Power
Engineering, Division of Fluid Dynamics
and Thermodynamics,
Technická 4, 166 07 Praha 6,
Czech Republic;
tel.: ++420 224 352 580
fax: ++420 224 310 292
jiri.nozicka@fs.cvut.cz

ABSTRACT

An experimental study of aerodynamic optimization of semitrailer unit was performed. Attention has been paid to the reduction of base-drag by means of rear-end tapering panels, reduction of water spray behind the vehicle and reduction of semitrailer-under-body drag using side fairings of various forms. Experiments have been performed with a scaled 1:15 model in a wind-tunnel. Comparison of some results with measurements taken with the vehicle in natural size in 1:1 wind-tunnel is mentioned.

Next to force measurements, PIV technique has been employed for flow-field analysis in the wake downstream the semitrailer. Results of PIV-measurements are discussed in comparison with numerical simulation of flow around the semitrailer in the case of semitrailer rear-end tapering.

KEYWORDS

Heavy truck aerodynamics, aerodynamic drag reduction, semitrailer boat-tailing, semitrailer side skirts, wind tunnel testing, PIV, CFD

INTRODUCTION

Tractor-semitrailer units play nowadays a decisive role in goods transportation, especially thanks to the flexibility and quickness which are the virtues railroad transportation is not able to offer. On the other hand, road delivery of one ton to the distance of one km is in comparison with railroad more energy demanding, though. The reduction of fuel consumption of heavy road vehicles is therefore in connection with still increasing number of cars getting to be a very important issue. At the end of 20th century it has been estimated, that keeping the annual growth of oil consumption the world's known resources of oil would last for some more 70 years only. Aerodynamic drag of heavy road vehicles significantly participates on the overall energy efficiency of vehicles and thus represents a challenge for its optimization.

If we consider a tractor-semitrailer unit of 40t weight and a drag coefficient c_D of 0,6 (coefficient of rolling resistance $f_r = 0,0055$), the aerodynamic drag-part of the overall driving resistance exceeds 50% at the speed of about 90 km/h. Considerable effort since the 80s of the last century has already been invested to the aerodynamic optimization of tractor units. Compared to this, semitrailers are even nowadays having aerodynamically still very ineffective form. One of the biggest European producers of semitrailers, German Firm Kögel Fahrzeugwerke in Ulm, is quite strongly interested in the possibility of aerodynamic optimization of their units and promotion of such improvements on the market. With this intention the Firm Kögel entered into cooperation with the Institute of Fluid Mechanics and Power Engineering at CTU in Prague.

Based on literature ([1], [2]) it may be estimated that the contribution of semitrailer to the aerodynamic drag of a whole vehicle in the present form of tractor-semitrailer unit runs to some 15% in the case of 0° yaw angle. This number significantly increases with yaw angle, at the yaw of $\beta=5^\circ$ being some 40% already. Main part of semitrailer drag originates from its uncovered undercarriage. Some 5-6% of the overall vehicle drag is caused by friction drag of the semitrailer body, the estimation being based on some simplified calculation. Overall drag of tractor-semitrailer unit is also co-determined by base pressure drag of the semitrailer, the magnitude of which reaches appr. 15-20% of the overall aerodynamic drag of the vehicle.

The potential for fuel consumption reduction for "typical driving conditions" is shown in the Fig. 1, which has been overtaken from Ref [3]. Theoretically, for flat road it would rather be estimated, that fuel consumption reduction should equal the aerodynamic drag reduction times its part on the overall driving resistance. Average

fuel consumption of a 40t tractor-semitrailer unit for middle-heavy highway at the speeds of about 80 km/h runs between 30-35 l/100km. A typical tractor may log between 150 and 200 000 km per year.

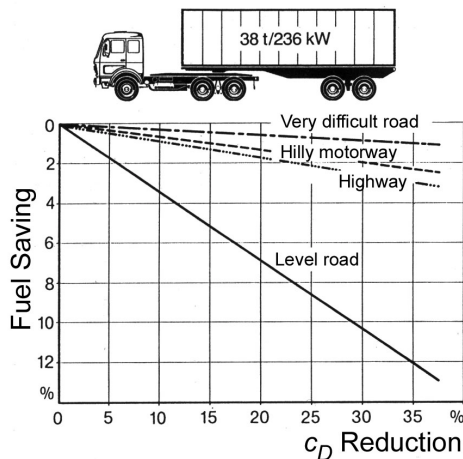


Fig. 1 Fuel savings. From Ref. [3].

The aim of this study is to investigate possible aerodynamic drag reduction of a typical Kögel semitrailer. The work has been done experimentally with a model build in the scale of 1:15 in the wind tunnel of the Institute of Fluid Mechanics and Power Engineering at the Czech Technical University in Prague. Next to force measurements, PIV system for flow visualization of wake downstream the semitrailer has also been employed. Based on the drag components analysis described above, the attention has been paid to the under-body and base pressure drag reduction as the main sources of semitrailer aerodynamic drag.

CFD simulations (Fluent SW) of flow field and correspondent forces acting on the semitrailer are also being performed. In this paper, the flow pattern found in CFD simulations is compared with the PIV measurements.

OPTIMIZATION OF TRACTOR-SEMITRAILER UNIT

Under-body drag of semitrailer unit can be reduced by side fairings of various form, or additionally by completely closing of the under-body space by a second floor. Base pressure drag can be influenced by a number of techniques. The target is to reduce the under-pressure acting on the back area of the semitrailer. The first group of such devices is in principal of mechanical nature, such as rounded edges, rolling cylinders or body length extension by means of end panels, either straight ones or with some tapering towards the end. The next group is represented by pneumatic devices, working on Coanda-effect, surface blowing out or boundary layer suction. This group of modifications may also be called as active flow control. With respect to possibly the simplest construction of some device on a real semitrailer, the function of end-tapering panels has been selected for investigation in this study. Examined is effectiveness of either only vertical panels, or panels enclosing the whole back. Survey of all measured variants is given in Fig. 2.

WIND TUNNEL, MODEL AND EXPERIMENTAL SETUP

Tests have been performed in close-circuit wind tunnel with open test section of the laboratory of the Institute of Fluid Mechanics and Power Engineering at the Czech

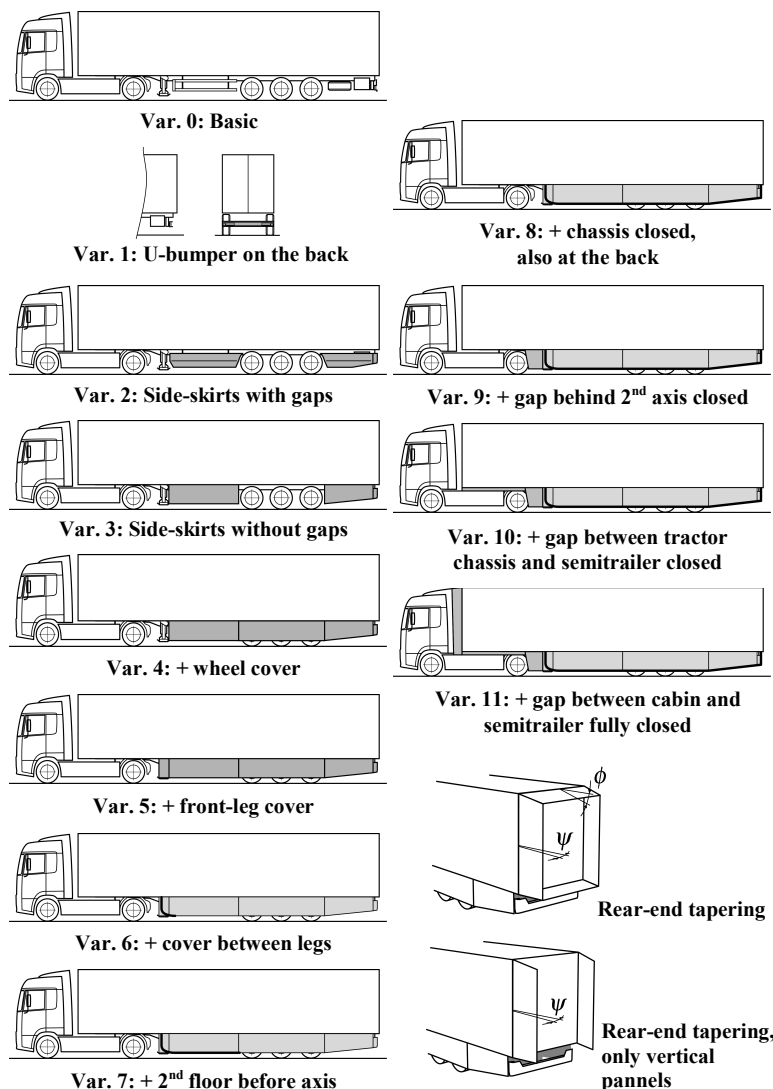


Fig. 2 Modifications investigated.

Technical University in Prague. Cross-section of the wind tunnel is rectangular, with the nozzle outflow size of 0,55 x 0,75 m. Overall length of test section is $L_{TS} = 1,5$ m. Air in the wind tunnel is put into motion by a radial fan driven by a 32kW asynchrony electromotor. Maximum speed in the test section is appr. 20 m/s. Especially due to the radial fan, the turbulence level in the test section is quite high, being appr. 1,3% at the air velocity of 7,5 m/s, resp. 3,9% at 16,5 m/s ([4]). Test section is equipped with a turntable allowing for simulations of side-wind effects. Road is simulated by a fixed plate elevated by 2,5 cm from the lower edge of the nozzle exit to eliminate boundary layer effects. Forces are at the time being measured by one-component strain-gauge aerodynamic weight, the resolution of which is appr. 0,02N. Velocity in the test section is evaluated by so called “plenum technique”, in which the pressure before the nozzle entrance is measured relative to the plenum pressure in the test section.

A scaled 1:15 model of a typical tractor-semitrailer unit has been manufactured for the measurements. Geometrical *similitude* of the model is kept up to details in the order of 3-4 mm in the model scale. Main dimensions of the model are given in Fig. 3. Blockage of test section with this model reaches 11,2%. This number would already call for application of blockage and test section corrections, especially in non-zero yaw conditions. For open test section such corrections need to be evaluated experimentally, which will be next step of our research. In this study, the results of relative improvements were expected and therefore the corrections for this purpose were not needed.

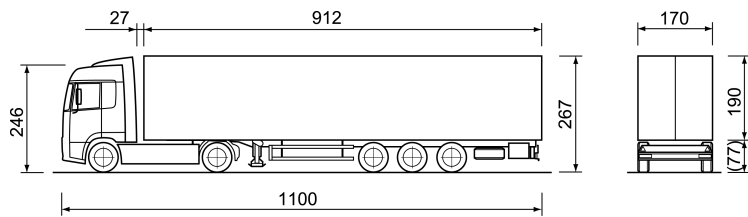


Fig. 3 Model dimensions.

Measurements were conducted at the Reynolds number of $Re = 1,2E+06$, which is related to the overall length of the model (for a real vehicle the Re number would be at the speed of 80 km/h $Re = 2,35E+07$). Assuming the enhanced turbulence level, effective Re number Re_{eff} will be appr. $Re_{eff} = 5,0E+06$. For box-objects this almost represents the level, from which drag coefficient already does not change with increasing Re -number.

Standard measurement uncertainty using Gauss quadratic law has been evaluated for the test rig used. At the velocities and aerodynamic forces acting during the measurements on the model the relative uncertainty of tangential drag force coefficient is found to be 1,4%. Considering the relative change of drag coefficient expressed in % relative to the original state as the final quantity, uncertainty of this variable is in absolute measure 1,7%.

EXPERIMENTAL PROCEDURE AND DATA TREATMENT

Data measured in various sets were corrected to fit the level of first measurement. Zero-value reading was performed before the start of each measurement. Results are processed into the tangent drag coefficient c_T , resp. its reduction Δc_T . Drag force in the longitudinal axis-direction of the vehicle is given by $F_T = c_T \cdot 0,5 \cdot \rho \cdot U^2 \cdot A$, ρ being the density of oncoming flow and A the reference area of the vehicle. U is resulting velocity speed, as can be seen on Fig. 4, where U_w denotes the wind speed of arbitrary direction.

Drag force has been measured in the range of yaw angle $\beta = \pm 10^\circ$ with the step of $2,5^\circ$. Representative evaluation of drag reduction should be done taking into account the influence of side wind. In central Europe it is found, that the wind velocity U_w of cca 3 m/s is most probable. Mean tangent drag coefficient $\overline{c_T}$ has been calculated as

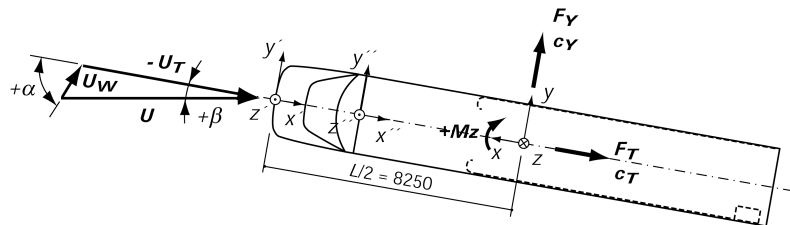


Fig. 4 Unsymmetrical flow conditions.

$$\overline{c_T} = \int_0^{2\pi} \int_0^{U_w^{\max}} c_T(\beta) \cdot \left[1 + \left(\frac{U_w}{U_T} \right)^2 + 2 \cdot \left(\frac{U_w}{U_T} \right) \cdot \cos \beta \right] \cdot p(U_w, \alpha) d\alpha dU_w, \quad (1)$$

whereby according to Ref. [5] the probability distribution of side-wind blow-direction may be taken uniform in all directions. Side wind velocity U_w has been set to be 3 m/s and cruising velocity U_T of the vehicle 88 km/h in those calculations.

EXPERIMENTAL RESULTS AND THEIR DISCUSSION

Drag coefficient of basic Var.0 at 0° yaw angle, which has been corrected according to procedure derived by Wuest [6], was $c_D = 0,6477$ (uncorrected value $c_D = 0,6175$). Further results are shown already as a relative improvement of c_D , resp. c_T coefficient.

Function of semitrailer side fairings and under-body covering shows increasing efficiency with yaw angle. Var. 1, which was to show possible influence of replacing the typical Kögel high back bumper, which almost encloses the space between the back-lamps and undercarriage, with low U-form bumper and keeping the rest of space open, proved, there is no change in this measure. Such result was due to the fact that the velocities close to the undercarriage are very small. On the other hand, it is known, that the rubber skirt mounted on the back and touching the road can increase the drag even up to 24% (by busses, see [3]). Simple variant of side skirts, Var. 2, with a big gap between the side skirt and undercarriage (and road as well) showed that such kind of side skirts still allows for considerable flow through the undercarriage resulting in high drag. Just starting from the Var. 3 with fully enclosed sides, the gains are becoming noticeable. Lower edge of side skirts of this kind is designed to be appr. 280 mm above the ground in the case of a real vehicle. In the Var. 5, additional covering of front legs on the semitrailer brought another 1% improvement. Var. 6,7,8 were designed with second floor undercarriage closing. Surprisingly, there was no drag change found in those variants. Further closing of all remaining gaps on the sides realized in the variants 9, 10, 11 and 12 showed significant drag reductions (Var. 12 is the same as Var. 11 plus the opening in the cabin-deflector is closed). In the last variant 13, mirrors on the cabin have been fully removed. Fig. 5 a), b), c), d) show the c_T reduction results, finally in the Fig. 6 the $\overline{c_T}$ reduction is calculated.

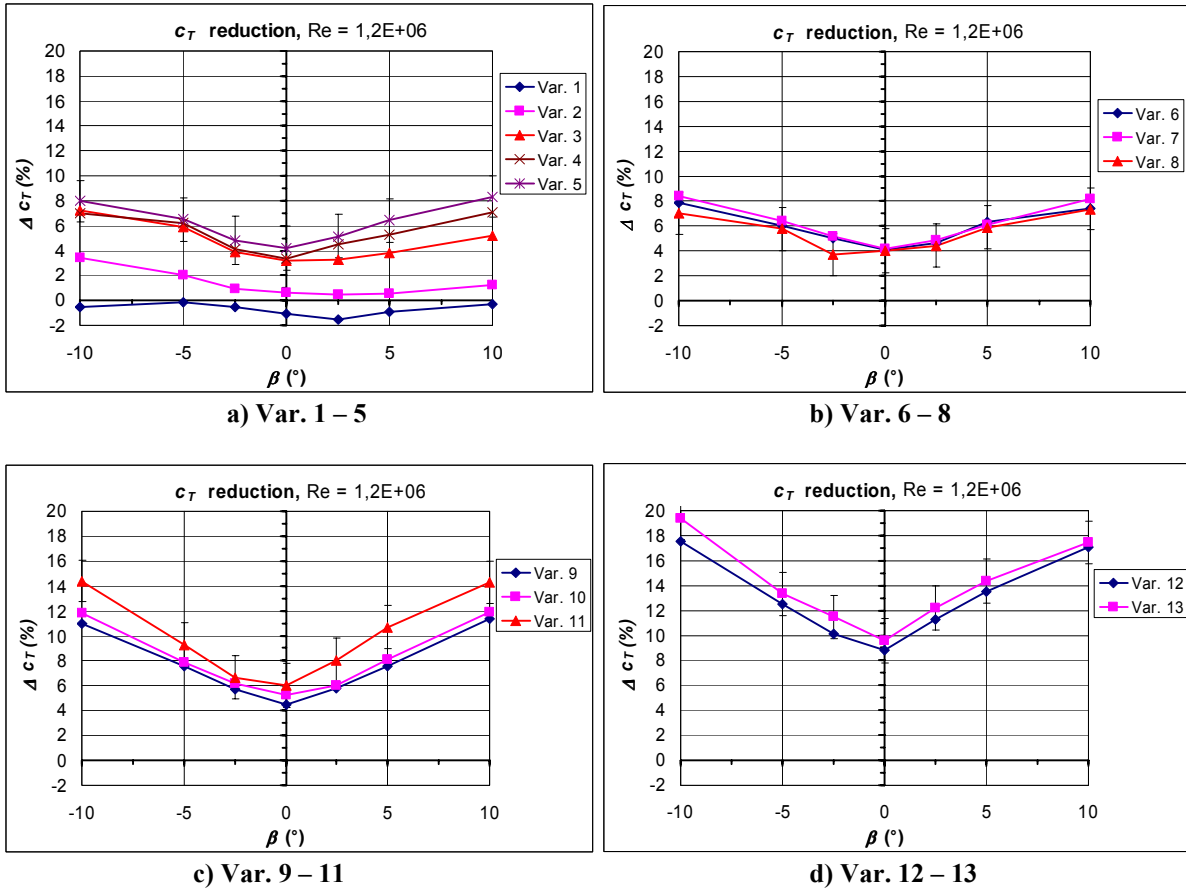


Fig. 5 c_T reduction.

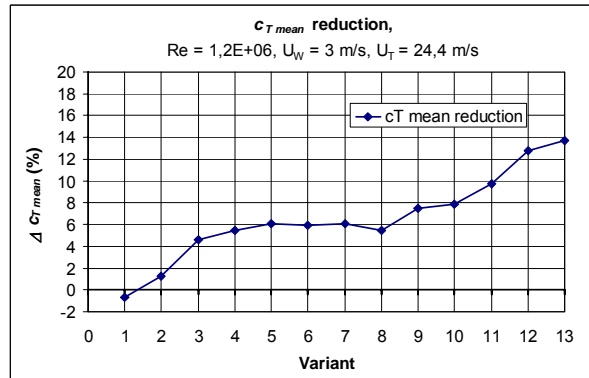


Fig. 6 Reduction of c_T mean.

Drag reduction of semitrailer unit using side-skirts was measured also on a vehicle in real size in the DNW Windtunnel. The configuration of side-skirts was the same as Var. 5 in model case presented above. Comparison of drag reduction as measured in model-scale with that of real vehicle is given in Fig. 7. Maximum efficiency in the case of real vehicle measurements was found at 5° yaw. At 10° yaw, contrary to model-scale measurement, the reduction of c_T already stagnated, resp. was lower. This could have been caused by closed test section of DNW Windtunnel and high blockage ratio (10,6% at 0° yaw), thanks to which at higher yaw angles the flow was considerably constrained.

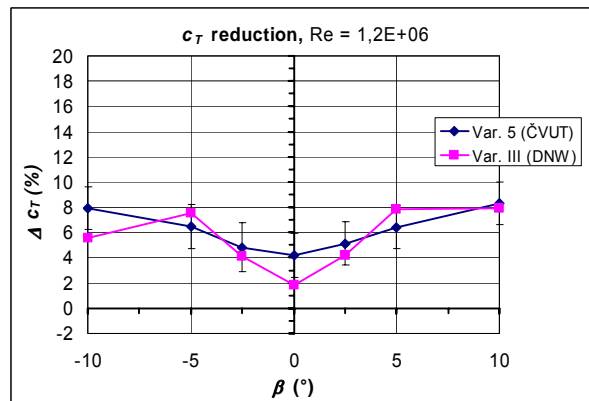


Fig. 7 Comparison of c_T mean reduction from 1:15 (ČVUT) and 1:1 (DNW) measurement. Var. III (DNW) corresponds to Var. 5 (ČVUT).

Rear-end tapering was tested in two lengths, $l_p = 30$ and 57 mm, which represents cca 17,5%, resp. 33,5% of the semitrailer width. Flat panels with sharp edges on the roof and sides have been chosen to fit best their simple application on a real semitrailer and simplicity of its geometrical description. The angles ϕ and ψ of the rear-end tapering were kept equivalent, whereby their range of $4, 8, 12$ and 16° was investigated. Reference case for this investigation was the variant 5. Results revealed, that the drag-reducing function of rear-end tapering is not so strongly dependant on the yaw angle as it was with the side skirts. Since up to 10° yaw the flow on the leeward side of semitrailer is still attached, the reduction of base under-pressure does not change dramatically with yaw angle. Fig. 8 a), b) depict the reduction of c_T coefficient for overall rear-end contraction, Fig. 9 a) summarizes c_T reduction (compared with $c_T(0)$ reduction at 0° yaw) as a function of contraction angle. It may be seen, that longer panels installed around all end-edges give by 1% better efficiency than the shorter ones. With only vertical panels, no drag reduction was measured (values of Δc_T reduction fall within the measurement uncertainty range; see Fig. 9 b)).

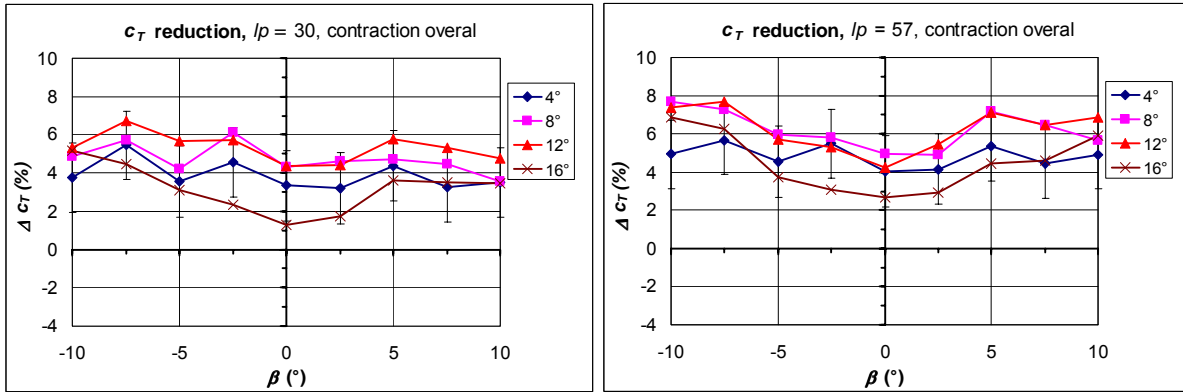


Fig. 8 a) c_T reduction of rear-end tapering, $l_p = 30$ mm

b) $l_p = 57$ mm

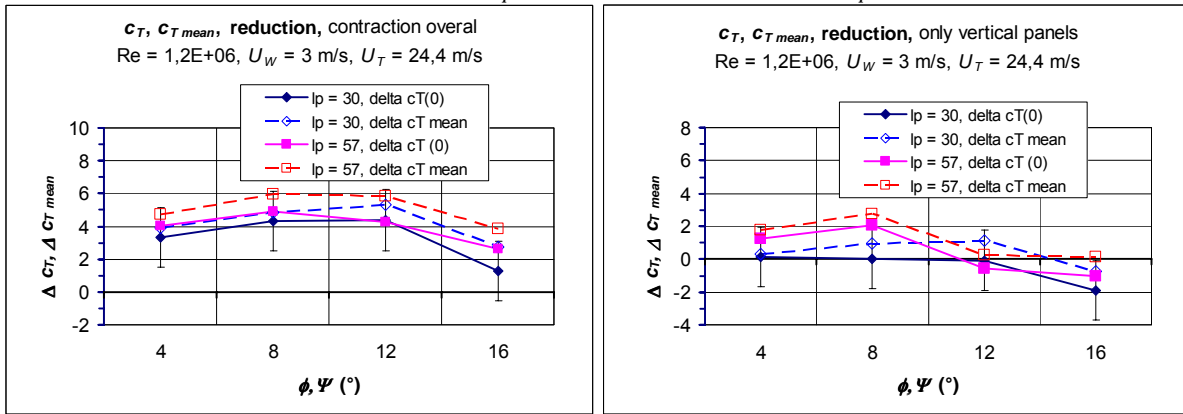


Fig. 9 a) $c_{T(0)}$ and c_T mean reduction of rear-end tapering

b) with vertical panels only.

PIV MEASUREMENTS

The aim of 2D PIV measurements was to visualise flow pattern in the wake, especially in the case of 16° rear-end tapering. For measurements, vertical cut in the longitudinal symmetry plane and one horizontal plane in the half of the semitrailer-body were selected. Measurements have been done with 2D PIV system from DANTEC. Reynolds number of those experiments was $Re = 1,2E+06$.

Visualisation of flow field in the wake of 16° rear-end tapering 57 mm long revealed attached flow over the panels both in the vertical symmetry plane, as well as in the horizontal plane (see Fig. 9 a), b)). Flow around the contraction panels remained attached also in the whole range of investigated yaw angles of $\pm 10^\circ$.

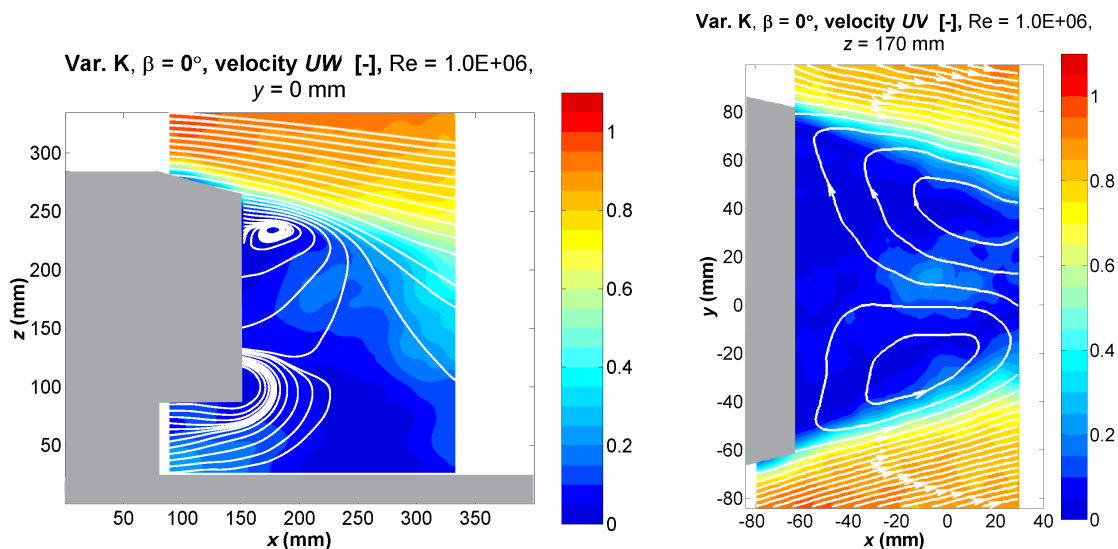


Fig. 9 a), b) PIV flow visualisation downstream the back of semitrailer. Contraction 16° .

a) Vertical symmetry plane

b) horizontal plane in the middle of the semitrailer-body height.

CFD FLOW SIMULATIONS

Long term aim of CFD flow simulations using Fluent is to optimize rear-end tapering of the semitrailer. In the step presented here the flow over simplified truck-semitrailer geometry was simulated. Control volume which extended 5 lengths upstream, 5 lengths downstream, 10 widths to the side and 10 heights above the model was meshed with unstructured mesh and prismatic elements in the wall vicinity. Calculations are performed at the Re number of $Re = 1,1E+06$ and $2,54E+07$ and employing Spalart-Almaras, k-eps RNG or k-w SST model of turbulence. Segregated solver was employed, where all variables on the cell-faces of the mesh were calculated using second order discretization scheme. Model geometry and surface mesh may be seen on Fig. 10.

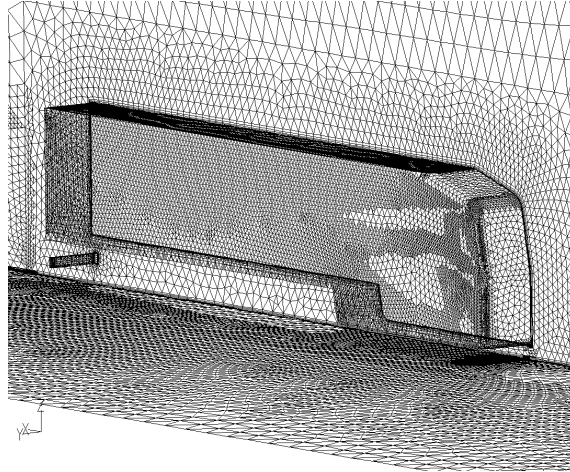


Fig. 10 CFD model and its surface mesh.

Rear-end tapering of the model was 16° . With smaller Re number both k-eps RNG and k-w SST showed probably too big production of turbulence kinetic energy resulting in the case of tapered back in boundary layer separation on the roof and its elevation to the free stream. For $Re = 1,1E+06$ only Spalart-Almars model showed reasonable results, despite again with separation on the roof before the tapered end. The same flow structure was also found for all of the turbulence models mentioned in the case of $Re = 2,54E+07$. Flow structure of 16° rear-end tapering at the $Re = 1,1E+06$ and Spalart-Almaras model of turbulence may be seen on the Fig 11.

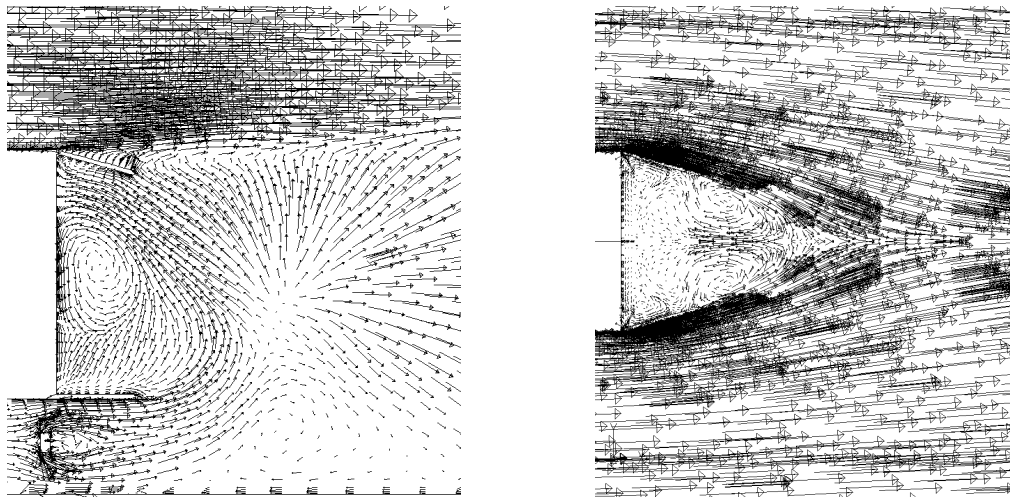


Fig 11 CFD flow simulation, contraction 16° . $Re = 1,1E+06$.

a) Flow patten in the vertical symmetry plane

b) horizontal plane.

CONCLUSIONS

Semitrailer drag optimization of its under-body part showed its first optimum in the case of sides fully closed, opened at the bottom and on the back to allow the air on the back for flowing out (Var. 5). Reduction of $\overline{c_T}$ in this case is 6%. Ideal closing of all other gaps between the tractor and semitrailer can bring reductions in $\overline{c_T}$ of up to 13% in the case of Var. 12. The optimum of rear-end tapering was found for the tapering angles ϕ , ψ of 12° . Rear-end panels, which were long cca 33,5% of the semitrailer width revealed maximum $\overline{c_T}$ reduction of 6%. When the rear-end of the semitrailer is equipped with vertical panels only, there is almost no reduction of $\overline{c_T}$. If the semitrailer was equipped with side skirts according to Var. 5 and full 12° rear-end tapering, the overall $\overline{c_T}$ reduction would be nearly 12%. Such reduction of $\overline{c_T}$ would correspond to some 4,5% fuel savings according to Fig. 1. On the level at the speed of 90 km/h the fuel consumption reduction would be expected of up to 6%.

Some discrepancy has been found between flow pattern downstream the 16° tapered rear-end visualised by means of PIV and CFD flow simulations using Fluent. In the next step it is intended to make use of Fluent for rear-end tapering angle optimization.

ACKNOWLEDGMENT

This study has been supported by means of Research Centre of J. Božek.

REFERENCES

- [1] Gilhaus, A., Hau, E., Künstner, R., Potthoff, J.: *Über den Luftwiderstand von Fernlastzügen, Ergebnisse aus Modellmessungen im Windkanal – Teil I.* Automobil-Industrie 9/79, 1979, pp. 125-137
- [2] Gilhaus, A., Hau, E., Künstner, R., Potthoff, J.: *Über den Luftwiderstand von Fernlastzügen, Ergebnisse aus Modellmessungen im Windkanal – Teil II.* Automobil-Industrie 3/80, 1980, pp. 45-62
- [3] Hucho, W. H.: *Road Vehicle Aerodynamics.* 4th Edition, SAE, Warrendale, 1998
- [4] Antoš, P., Sulitka, M.: *Modifications of close circuit windtunnel of Department of Fluid Mechanics and Thermodynamics of CTU in Prague (in Czech).* In *Proceedings of XIII. International conference on experimental and numerical techniques in Fluid Mechanics.* University of Žilina, Žilina (Slovakia), 2002, p. 126-131. ISBN 80-7100-955-5
- [5] *SAE Information Report: Aerodynamic Testing of Road Vehicles - J2071.* SAE, Warrendale, March 1990
- [6] Wuest, W.: *Strömungsmesstechnik.* Braunschweig, Verlag Vieweg & Sohn GmbH, 1969
- [7] Sulitka, M., Nožička, J.: *Zusammenfassung der Ergebnisse aus DNW-Windkanalmessungen.* Research Report. Prague, Czech Technical University in Prague, Faculty of Mechanical Engineering, 2002. 177/02, 16 p.