Numerical Simulations to Determine the Effect of Geometrical Windshield Parameters on the Drag Coefficient

Martin Praßler

Mechanical Engineering, Galgenbergstr. 30, 93053 Regensburg (Germany), Head: Prof. Dr.-Ing. Stephan Lämmlein

martin.prassler@stud.hs-regensburg.de
stephan.laemmlein@hs-regensburg.de
http://www.hs-regensburg.de

1. Problem Definition

From the aerodynamic point of view there are two possibilities to reduce the air resistance of modern automobile windshields. In the last few years a trend has developed that the inclination angle of the windshields more and more decreased. The reduction of the drag coefficient achieved in this way has been causing increased material costs, heating up the vehicle interior by sun and distortions of the drivers view.

The second possibility for drag-reduction is the tangential curvature of the A-pillar. Due to the more attached flow the C_D value can be reduced in this way, too.

Fig. 1: Lamborghini Murciélago using a strongly inclined windshield

Fig. 2: Design study Skoda Roomster using a tangential A-pillar transition

Appropriate simulations will be performed with systematically varied windshield parameters. So it shall be investigated, how the A-pillar curvature and the inclination angle of the windshield are effecting the C_D value.

2. Modelling

To get the desired information 20 models are generated by the CAD tool Pro/Engineer Wildfire 4.0. The variable parameters are only the inclination angle of the windshield and the A-pillar curvature.

The individual models and the relevant test section of the Regensburg Wind Tunnel are emulated digitally. The real flow characteristics are modeled as accurately as possible, too. So a comparison with the real Wind Tunnel measurements is possible.

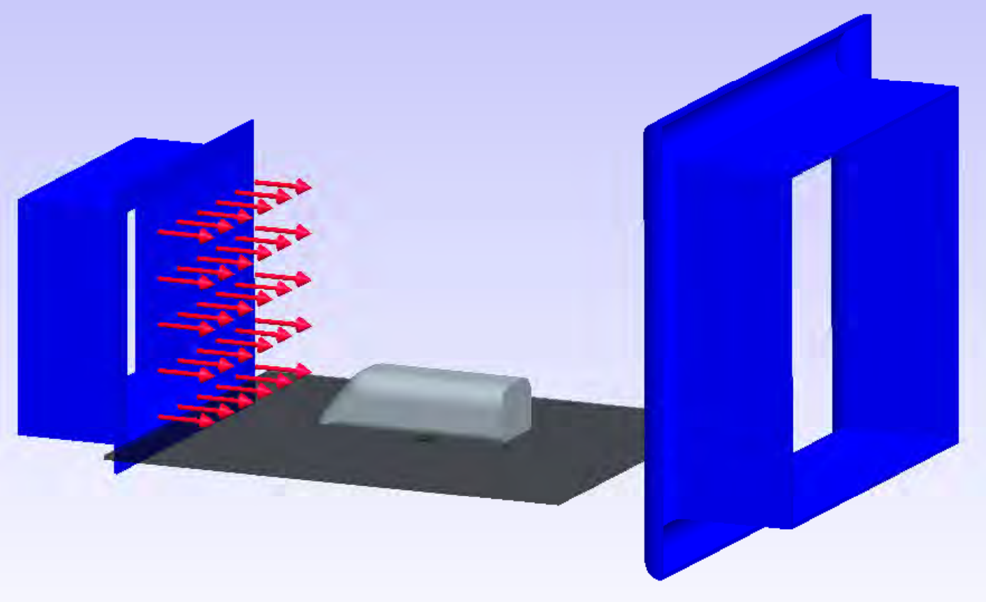
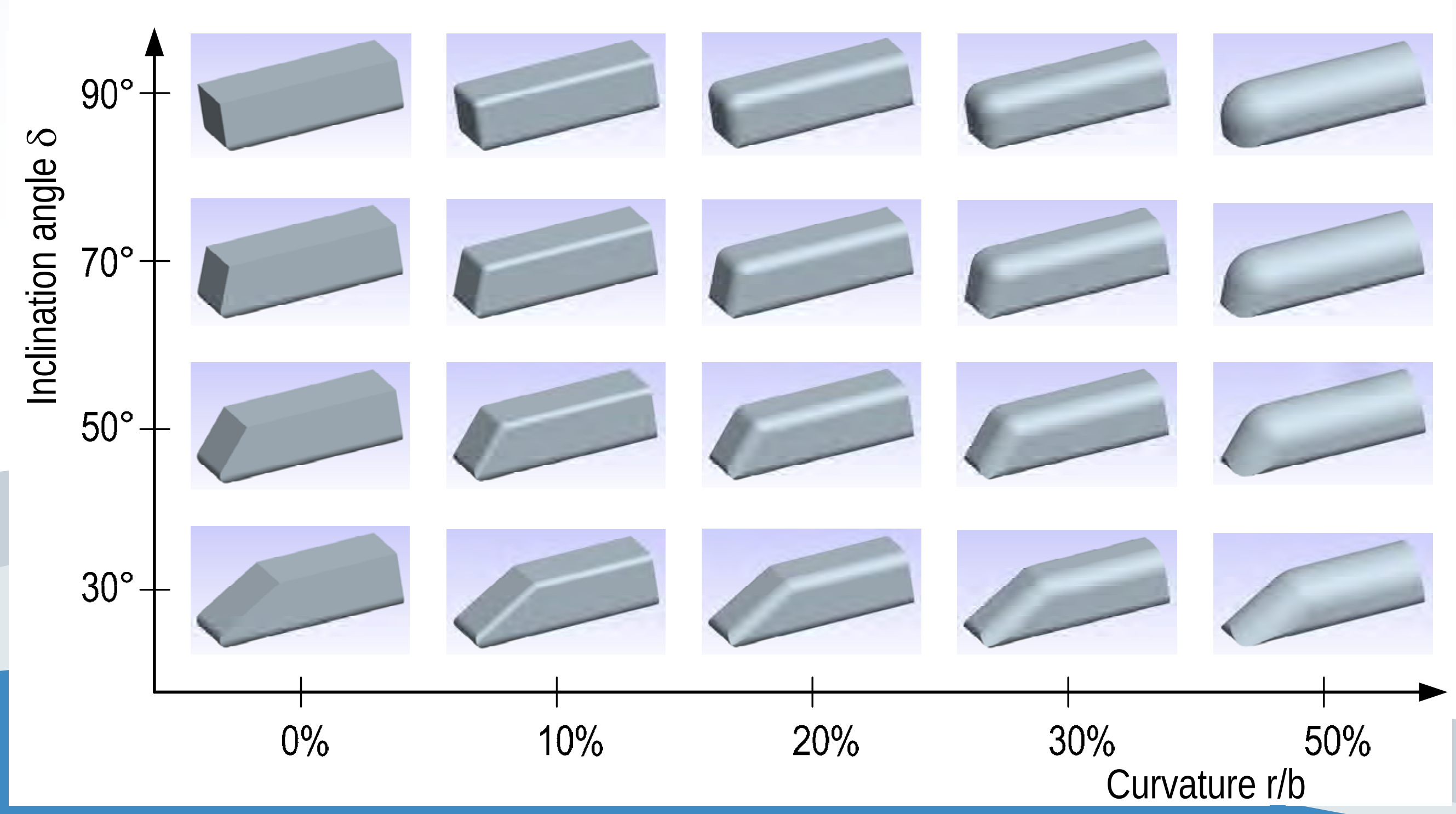


Fig. 3: Model of the relevant test section of the Regensburg Wind Tunnel with a frontally flowed body

Due to the defined variation of the windshield parameters the following 20 models are generated and compared.



3. Simulation

To perform the numerical flow calculations with EXA Powerflow first of all a case has to be set up with EXA PowerCase for each configuration. Aside from the relevant flow parameters, the characteristics of the fluid mesh have to be specified. Afterwards the simulation results are being analyzed with EXA PowerViz.

The flow at the curvature of the roof transition and the A-pillar are of special interest. If the flow is getting turbulent or vortexes occur there, this would be an indication for a increased air resistance. Hence, the resolution of the mesh has to be refined there.

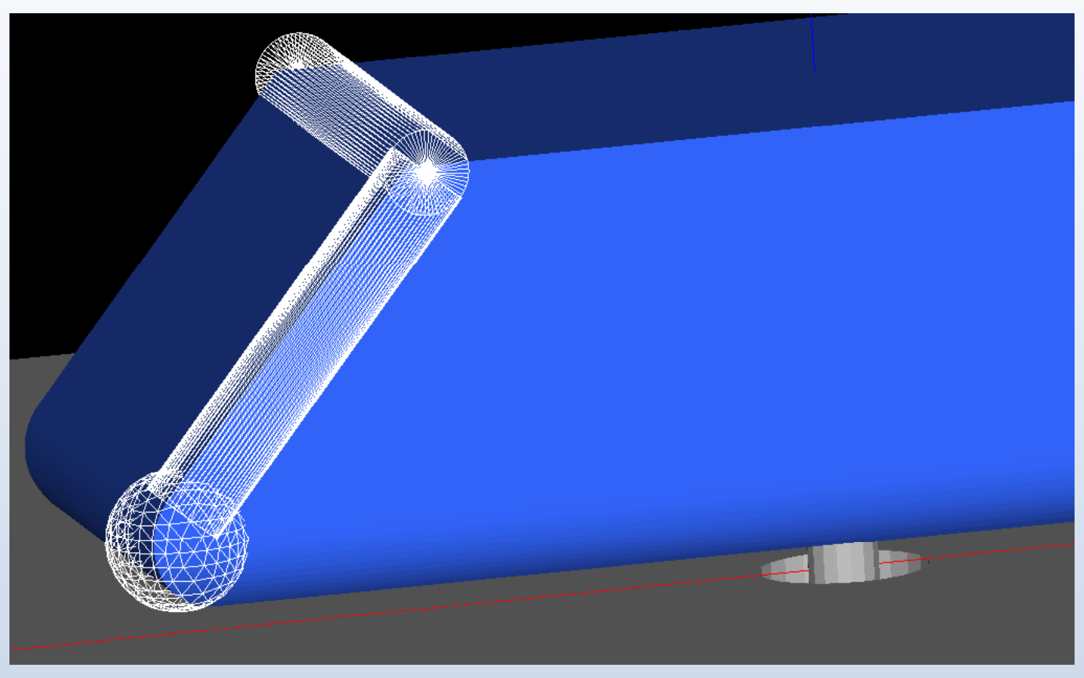
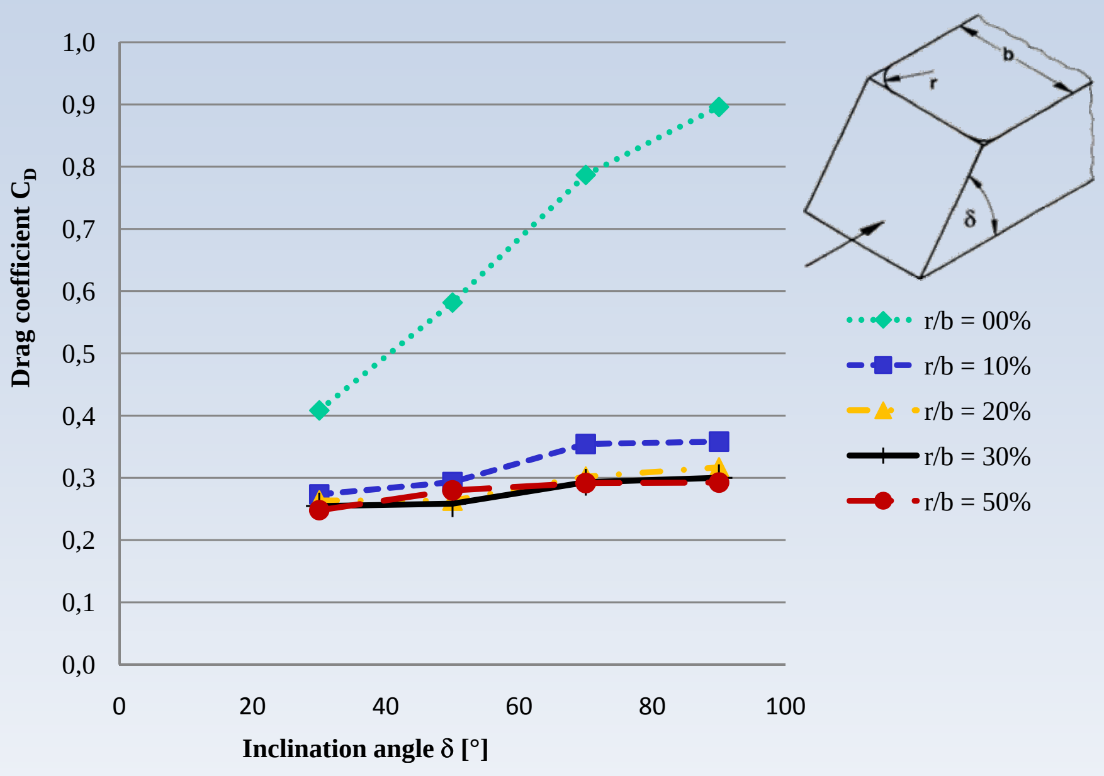


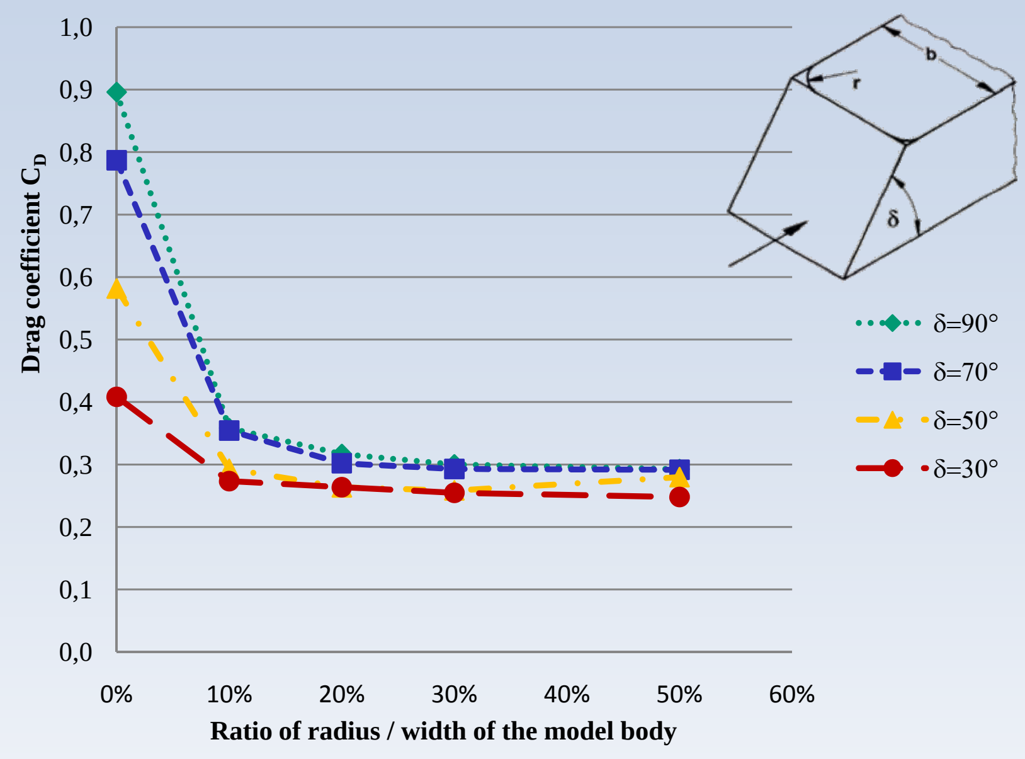
Fig. 4: Refinement of the fluid mesh around the A-pillar and the roof transition

The final setup includes the required geometries, the regions of variable resolutions and the in- and outlet of the flow field. A useful symmetry plane halves computing time and memory requirements to boot.

4. Results

The simulation results show a strongly increased drag coefficient at an A-pillar curvature below about 10 percent. Above this value the windshield inclination angle plays a minor role.





The determined drag coefficients are reflected in the pressure and velocity distributions, too. Local regions of negative pressure can be detected as well as the streamlines.

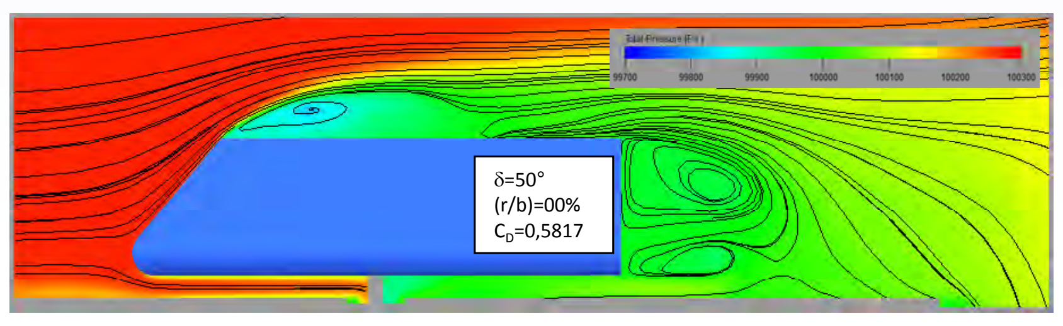


Fig. 5: Pressure distribution in the flow field

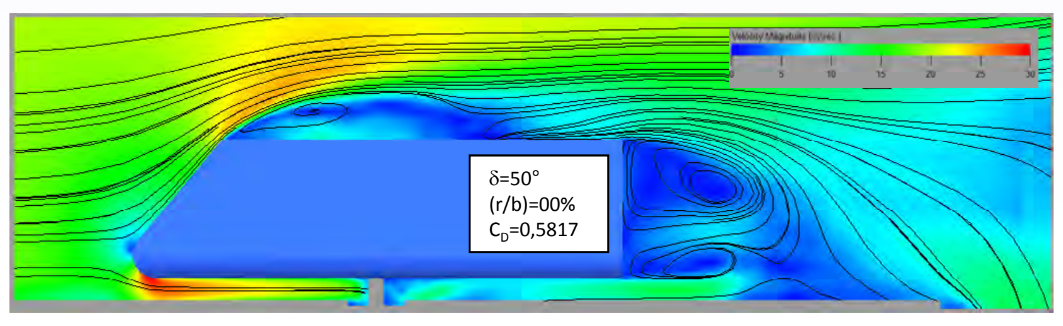


Fig. 6: Velocity distribution in the flow field

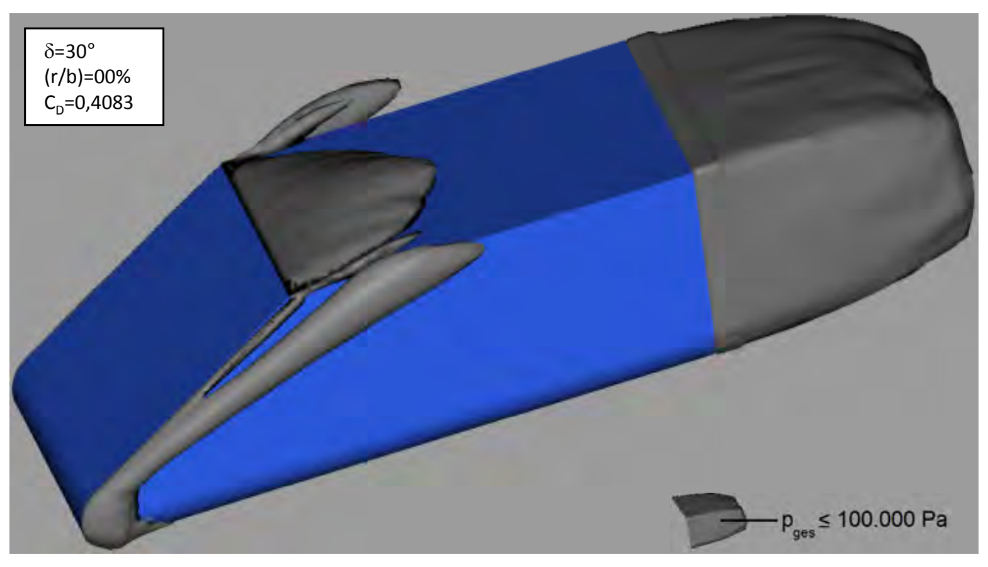


Fig. 7: Regions of negative pressure

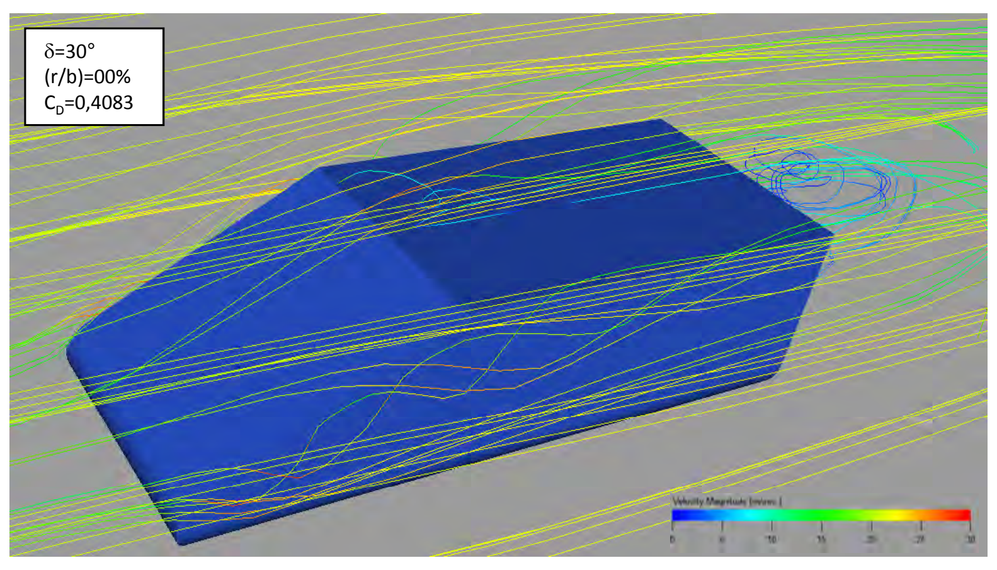


Fig. 8: Streamlines combined with velocity characteristics

Literature: HUCHO, W.-H. (Hrsg.): Aerodynamik des Automobils. Wiesbaden: Vieweg + Teubner 2008
LÄMMLEIN, S.: Vorlesungsskript Fahrzeugaerodynamik. Hochschule Regensburg 2008