NUMERICAL SIMULATION OF FLOW FIELD PAST ROAD VEHICLE WHEEL

Tamás Régert
Ph.D. Student, Department of Fluid Mechanics, Budapest University of Technology and Economics
H-1111, Budapest, Bertalan L. u. 4-6. Tel: 463-31-87; e-mail: regert@simba.ara.bme.hu

Dr. Tamás Lajos
Professor, Department of Fluid Mechanics, Budapest University of Technology and Economics
H-1111, Budapest, Bertalan L. u. 4-6. Tel: 463-40-72; e-mail: lajos@simba.ara.bme.hu

Summary

The flow past rotating road wheels is characterised by strong viscous effects and high adverse pressure gradients. The boundary layer on the surface of a rotating wheel is extremely complex, so the classical boundary layer theory assumptions don't work well. The numerical simulation using the state of the art turbulence models provides results corresponding to that of measurements.

1 INTRODUCTION

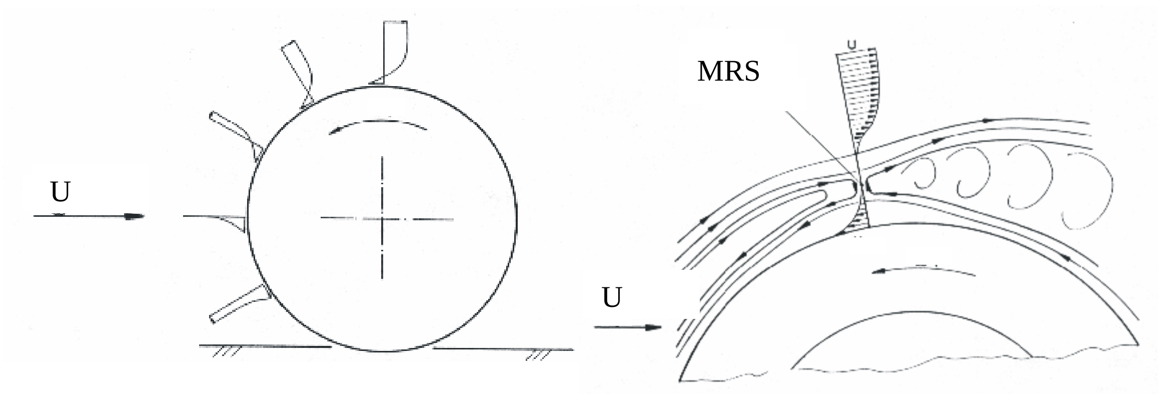
From aerodynamic point of view, the body of a car can be well optimised. The presence of wheels causes an about 30-40% increase in the aerodynamic drag on an advanced optimised car body [1], [5]. Besides this the flow past wheels influences the dispersion and transport of mud and water droplets depositing on the car body and obscuring the vision of drivers. It is therefore important to clarify the features of the flow past wheels and in the wheelhouses. This work focuses on the investigation of the flow field past wheels. From aerodynamic point of view the wheel is a bluff body: the boundary layer separates from its surface and a combination of closed and opened separation zones develop. In this case the structure of the boundary layer is unique, part of separations does not originate on the solid surface, but above that [2]. The numerical simulation of this kind of phenomenon needs a sophisticated turbulence model with special wall treatments. The commercial CFD code **FLUENT 6** has been used for simulations.

2 FLOW FIELD PAST A ROTATING ISOLATED ROAD WHEEL

We consider the air flowing past the wheel as an incompressible fluid, and observe the flow from a frame of reference travelling together with the wheel. In this frame of reference, the rotation axis of the wheel is at rest and the ground is moving downstream with a speed corresponding to that of the air. The Reynolds number based on the wheel diameter is $6{,}5\cdot10^5$, so the flow is turbulent. From theoretical point of view, the air is at rest in the absolute frame of reference, so the turbulence intensity of the approaching flow in the relative frame of reference is very small. The turbulence is generated by the wheel, so the applied turbulence model is of high importance. The diameter of the investigated wheel is 0,5m, and the width is 0,25m (aspect ratio = 0,5).

2.1 The structure of the boundary layer

SCHIEFER [2] has carried out experimental and theoretical investigation on the boundary layer on the surface of a rotating wheel. From aerodynamic point of view , the upper and lower regions of the wheel can be separated. Fig.1. [2] shows a schematic view of the velocity profiles along the circumference of the wheel in the median plane and a close-up for the upper region. The front of the wheel and the lower region is much less complicated than the upper because here the shear stresses on the wheel surface accelerate the approaching flow, so separation will not occur. In contrary there is a strong shear layer on the upper region, where the air in the boundary layer flows very fast against the approaching flow. The right hand side of Fig.1. [2] shows that the velocity profile becomes smooth in downstream direction and a free separation zone forms towards the base region. This separation can be identified as a Moore-Rott-Sears (MRS) type separation.



a) b)

Fig.1. Velocity profiles on the wheel a) and a close-up for the upper region b) [2]

As the width of a wheel is usually much smaller than its diameter, the flow past it cannot be compared to the flow about a cylinder. Due to the high resistance at the upper region of the wheel, most of the upstreaming air deflects towards the sides of the wheel and forms trailing vortices due to the interaction with the shear flow on the upper region. The lower region transports energy into the flow like a viscosity pump that moves towards the ground contact point. The approaching flow impacts into this region and stops at a stagnation point. Due to the energy produced by the wheel here the total energy is higher than that of the free stream. The pressure coefficient is therefore higher than 1 and grows until about 1,5. Due to the high pressure gradient here, a horseshoe type flow originates towards the sides and interacts with the upstreaming flow.

2.2 Experiments, quantitative results

There are only few literature that give quantitative information on the flow field, i.e. pressure distribution, velocity field, turbulence properties. The basic experimental work in this field is the measurement of the pressure distribution right on the surface of the rotating wheel by FACKRELL & HARVEY [3] and a static probe measurement by STAPELFORD & CARR [6]. There is a significant difference between the results of these authors, shown by Fig.3. The experimental arrangement was different in the two cases. At STAPELFORD's experiment the wheel did not contact the ground, but the air was deflected at the underside of the wheel to avoid high depression in the gap clearance between the ground and the wheel. This didn't influence the pressure distribution on the upper surface of the wheel. In

FACKRELL’s measurements, the pressure was measured by pressure tap, a hole in the surface of the rotating wheel, so they measured the pressure in the lowest region of the boundary layer. In this way, the overpressure also in front of the ground patch area could be measured. By STAPELFORD’s method, the static probe was placed outside the shear layer at the upper surface of the wheel, where the air flows downstream quickly, so it causes high depression on the top of the wheel, indicated by the measurement (see Fig.2.).

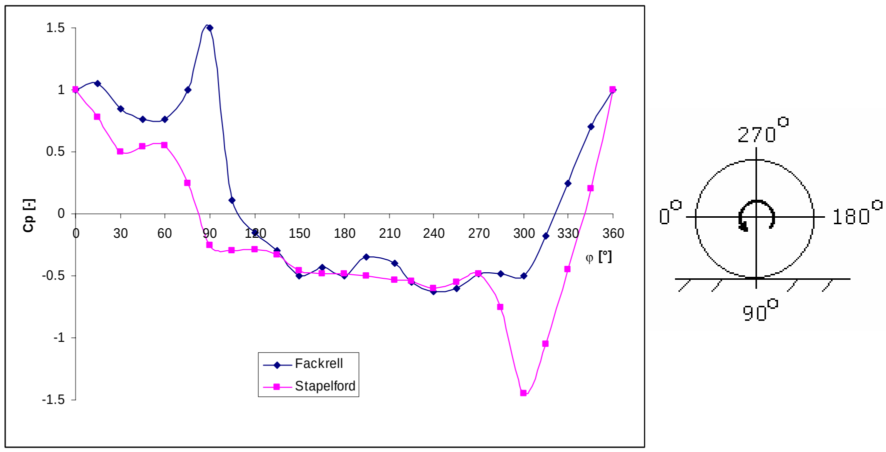


Fig.2. Pressure coefficient distribution after FACKRELL&HARVEY[3] and STAPELFORD&CARR[6]

3 NUMERICAL SIMULATION OF FLOW FIELD ABOUT ROTATING WHEEL

The numerical simulation needs a numerical mesh, in which the boundary layer is well discretized. For the turbulence models, the standard and enhanced wall functions provide the boundary conditions at walls. The dimensionless wall distance y^+ is used to characterise the fineness of the discretization. The left hand side picture of Fig.3. shows the computational domain with the surface grid, the right hand side of the figure includes a close-up for the wheel. On the right hand side of Fig.3. it can be seen that close to the surface of the wheel flat brick shaped cells, in the surrounding space tetrahedral cells are used.

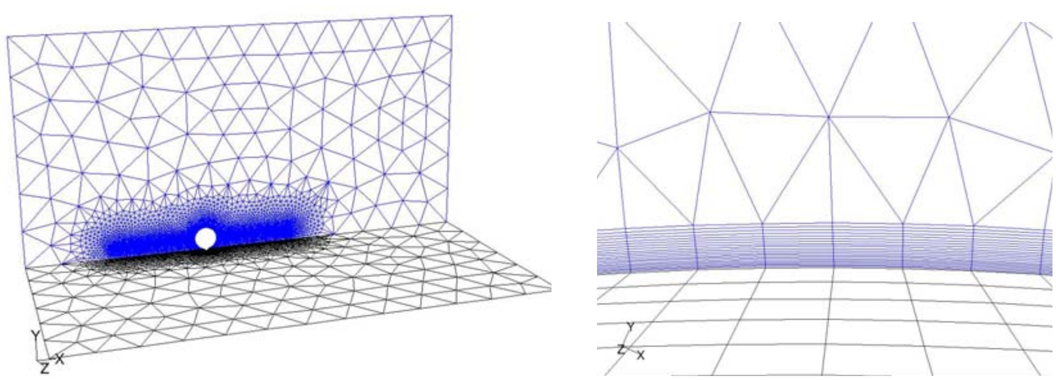


Fig.3. The computational domain and a close-up for the surface of the wheel

3.1 Boundary Conditions

For the ground and the wheel, moving wall boundary conditions are used, at the inlet the total pressure, at the outlet constant static pressure is prescribed. These boundary conditions generate an average velocity of the undisturbed flow at 20 m/s. The pressure inlet provides the possibility for the wheel to effect the flow further upstream than the inlet boundary of the domain.

3.2 The results of computation

The calculated streamlines can be seen in Fig.4. The structure of the flow field is in a good agreement with the experimental visualization.

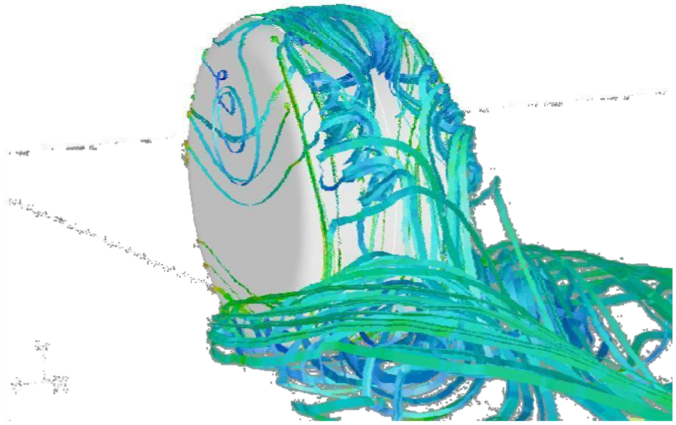


Fig.4. The streamlines about a rotating wheel

The calculated pressure coefficient distribution is compared in case of two different turbulence models. Both of them are sophisticated from the point of view of near wall treatments. The first turbulence model was a realizable k-ε model with new enhanced wall treatments, that is based on the two layer model. The pressure gradient effects were included. The second turbulence model was the shear stress transport k-ω model with the option of transitional flows. The results can be seen in Fig.5.

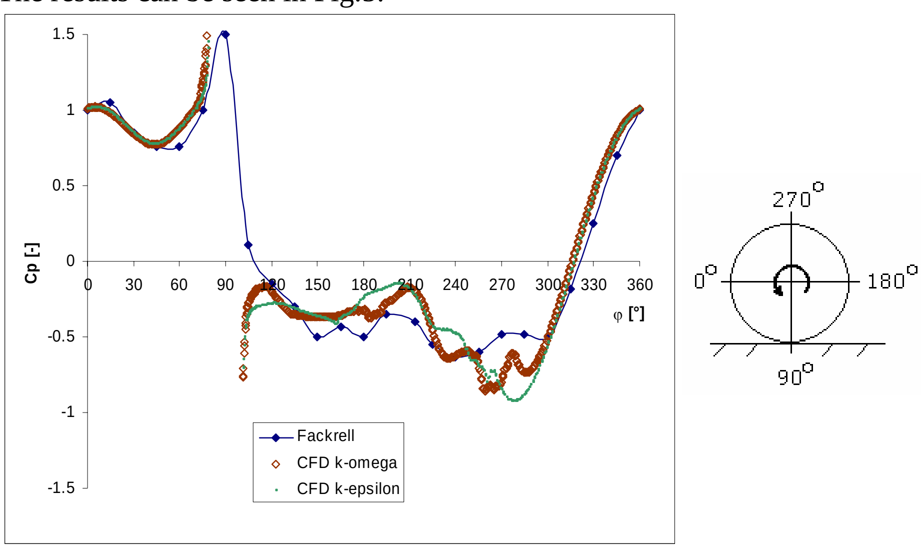


Fig.5. Calculated and measured pressure coefficient distributions

The main difference between the two turbulence models is, that in the k-ω model there is a low Reynolds number specialised (Wilson) model for the ω equation, while in case of k-ε model, the ε equation is a simple function of k and the wall functions are improved. The main difference in the result is at about 280°, where the k-ε model predicts a high depression peak.

It means, that the k-ε model predicts the details of the boundary layer less accurately than the k-ω turbulence model, however, both of the models gave quite good approximation.

CONCLUDING REMARKS

Numerical simulation of flows similar to that presented is a great challenge in respect of turbulence modelling. The structure of the flow field depends completely on the way how turbulence is generated, so the main goal is to model the near wall region properly. The SST k-ω model works very well in the present conditions. More numerical simulations can be found in references [4] [7], comparing the results of numerical simulations with standard and improved turbulence models.

Authors acknowledge the support of **FKFP 0075/2001**, **FKFP 0624/2000** and **OTKA T030116.**

REFERENCES

1. Zwicker, K., Wickern, G. et al. „Zum Einfluß von Rädern und Reifen auf den aerodynamischen Widerstand von Fahrzeugen“, *Tagung „Aerodynamik des Kraftfahrzeugs“* Haus der Technik e. V., Essen, 1995.
2. Schiefer, U., „Zur Simulation des freistehenden Fahrzeugrades im Windkanal“, *Dissertation am IVK*, Universität Stuttgart. 1993.
3. Fackrell, J. E., Harvey, J. K., „ The Flow Field and Pressure Distribution of an Isolated Road Wheel“, *Advances in Road Vehicle Aerodynamics*, Paper 10, BHRA, 1973.
4. Skea, A.F., Bullen, P. R., Qiao, J., „The use of CFD to predict the air flow around a rotating wheel“, *31. Int. Symposium on Automotive Technology and Automation*, Düsseldorf, 1998.
5. Hucho, W. H., „Aerodynamics of Road Vehicles“, *Butterworth an Co. Publishing*, Boston, 1998.
6. Stapelford, W. R., Carr, G. W., „Aerodynamic Characteristics of Exposed Rotating Wheels“, *The Motor Industry Research Association*, Rep. 1970/2, Nov.1969.
7. Axon, L., Garry, K., Howell, J., et al. „An Evaluation of CFD for Modelling the Flow Around Stationary and Rotating Isolated Wheels“, *SAE Paper 980032*, Detroit, 1998.