CFD ANALYSIS OF AERODYNAMIC CHARACTERISTICS OF A FORMULA RACE CAR REAR WING

[1]Chitranjan Kumar, [2]Divesh Kumar Rohilla, [3]Sandeep kumar Yadav
Department of Mechanical Engineering
B K Birla Institute of Engineering & Technology, Pilani.

Abstract

Computational Fluid Dynamics (CFD) has become one of the most effective tools for analyzing aerodynamic performance in modern automotive engineering. In racing vehicles, aerodynamic components such as spoilers and wings play a critical role in improving vehicle stability, increasing downforce, and enhancing cornering performance. This study presents a detailed CFD analysis of the aerodynamic characteristics of the rear wing of a Formula Mazda race car under different operating conditions.

The research focuses on evaluating the effects of Angle of Attack (AOA) and vehicle speed on the lift coefficient (Cl), drag coefficient (Cd), velocity distribution, pressure distribution, and flow separation behavior around the rear wing profile. A two-dimensional CFD simulation approach is employed using GAMBIT for geometry creation and mesh generation, while FLUENT is used as the flow solver for aerodynamic analysis. The turbulence effects are modeled using the standard k–ε turbulence model.

The simulation is conducted for three different vehicle speeds, namely 90 mile/hr (144 kmph), 120 mile/hr (192 kmph), and 150 mile/hr (240 kmph), with varying AOAs of −4°, 0°, 4°, 8°, 12°, and 16°. The results demonstrate that both lift coefficient and drag coefficient increase with increasing angle of attack. However, beyond an AOA of 12°, flow separation becomes significant, resulting in aerodynamic stall and reduction in lift coefficient.

The study further reveals that the spoiler performs most efficiently at 120 mile/hr and below 12° AOA. Detailed velocity and pressure contours are analyzed to understand the aerodynamic behavior of the wing. The obtained results are validated using previously published research data and show strong similarity in aerodynamic trends.

I. Introduction

Aerodynamics has always been one of the most important areas of research in automotive and aerospace engineering. In motorsports, aerodynamic performance directly affects vehicle stability, cornering capability, braking performance, and fuel efficiency. Racing vehicles are designed to generate maximum downforce while minimizing aerodynamic drag. Because of intense competition in the racing industry, detailed aerodynamic information is rarely shared publicly.

Among different aerodynamic components, the rear wing of a Formula Mazda race car plays a major role in generating aerodynamic downforce. The aerodynamic behavior of the rear wing depends on several parameters such as angle of attack, wing geometry, vehicle speed, flow separation, and pressure distribution.

The present work focuses on the CFD simulation of the rear wing of a Formula Mazda race car. The study investigates the aerodynamic characteristics of the wing under different operating conditions. The effects of Angle of Attack (AOA) and speed on lift coefficient (Cl), drag coefficient (Cd), velocity contours, and pressure distribution are analyzed in detail.

The primary objectives of this study are:

- To analyze the aerodynamic performance of the Formula Mazda rear wing.
- To study the effect of AOA on lift and drag coefficients.
- To investigate flow separation and stalling behavior.
- To validate CFD results with published literature.
- To identify optimum operating conditions for the spoiler.

The results of this work can be used for improving spoiler design and optimizing race car aerodynamic performance.

Literature Review

Several researchers have contributed to the field of aerodynamic analysis using Computational Fluid Dynamics (CFD). Due to the increasing importance of aerodynamic optimization in racing vehicles, many studies have focused on airflow behavior around spoilers, wings, and airfoils.

Feng et al. (2007) conducted CFD simulations on hypersonic cruise vehicles and observed that lift and drag coefficients increase with increasing angle of attack.

Colak et al. (2005) analyzed wing-body-spoiler configurations using three-dimensional viscous flow analysis and discussed the importance of mesh generation and turbulence modeling.

Armbya et al. (2006) carried out a CFD study on Formula Mazda race car wings and analyzed lift coefficient, drag coefficient, and flow separation characteristics for different angles of attack.

Previous studies indicate that CFD provides a highly effective method for aerodynamic analysis because it reduces the need for expensive wind tunnel testing and provides detailed flow visualization.

Problem Formulation

The present study focuses on the aerodynamic analysis of the rear wing of a Formula Mazda race car using Computational Fluid Dynamics (CFD). The problem is treated as a two-dimensional incompressible turbulent flow problem.

The analysis includes:

- Airfoil selection
- Geometric modeling
- Mesh generation
- CFD simulation
- Result validation

The aerodynamic behavior is studied for different:

- Angles of Attack (AOA)
- Vehicle speeds

The main aerodynamic parameters investigated are:

- Lift coefficient (Cl)
- Drag coefficient (Cd)
- Pressure distribution
- Velocity distribution
- Flow separation
- Stalling behavior

Experimental Setup

The experimental and simulation setup consists of the following major components:

A. Airfoil Geometry

The rear wing profile of the Formula Mazda race car is selected based on standard aerodynamic wing geometry. The wing has a single-element design with support struts and end plates.

B. Geometry Creation Using GAMBIT

The airfoil geometry is created using GAMBIT software. The wing profile coordinates are traced and converted into a computational model.

C. Mesh Generation

A structured computational mesh is generated around the airfoil profile. Fine mesh elements are created near the wing surface to capture boundary layer effects accurately.

The mesh contains approximately 12,285 nodes.

D. CFD Solver

The aerodynamic analysis is performed using FLUENT software.

The following assumptions are considered:

- Two-dimensional incompressible flow
- Turbulent flow conditions
- Isothermal conditions
- No heat transfer effects

E. Input Parameters

The simulation uses standard atmospheric conditions:

- Pressure = 101325 Pa
- Density = 1.225 kg/m^3
- Temperature = 288.16 K
- Kinematic viscosity = 1.4607×10^{-5} m^2/s

Methodology

The present study uses Computational Fluid Dynamics (CFD) to investigate the aerodynamic characteristics of the rear wing of a Formula Mazda race car. The methodology is designed systematically to simulate airflow behavior around the spoiler and to evaluate aerodynamic performance under different operating conditions.

The complete CFD workflow is divided into several stages, beginning from airfoil selection to final result analysis.

Step 1: Airfoil Selection

The first stage of the study involves selecting a suitable airfoil geometry for the rear wing of the Formula Mazda race car. The selected spoiler profile is based on the standard Formula Mazda rear wing geometry available in published literature.

The wing is an inverted airfoil designed to generate aerodynamic downforce rather than lift. Important geometric parameters considered during selection include:

- Chord length

- Wing curvature
- Thickness distribution
- Angle of attack adjustment range
- End plate configuration

The selected geometry provides stable aerodynamic performance at racing speeds.

Step 2: Geometry Modeling

The spoiler geometry is modeled using GAMBIT software. Initially, the profile coordinates are traced from the standard airfoil drawing. These coordinates are then converted into a two-dimensional computational model.

The following geometric features are included:

- Leading edge
- Trailing edge
- Wing curvature
- Airfoil thickness
- Computational flow domain

A sufficiently large computational domain is created around the spoiler to simulate free-stream airflow conditions accurately.

Step 3: Computational Domain Creation

The air surrounding the spoiler is represented by a computational flow domain called the far-field boundary.

The domain dimensions are selected carefully so that the airflow near the boundaries does not affect the aerodynamic behavior around the spoiler.

The boundary distances are selected according to standard CFD practices:

- Large inlet distance
- Extended outlet region
- Adequate upper and lower spacing

This helps achieve accurate pressure and velocity predictions.

Step 4: Mesh Generation

After geometry creation, mesh generation is performed in GAMBIT.

The computational domain is divided into thousands of small control volumes called mesh elements.

Special attention is given near the spoiler surface because:

- Boundary layer formation occurs near the wall
- Pressure gradient changes rapidly
- Flow separation must be captured accurately

Therefore:

- Fine mesh is generated near the wing surface
- Coarser mesh is used away from the airfoil

The final mesh contains approximately 12,285 nodes and provides a good balance between computational accuracy and simulation time.

Experimental-Type Mesh Observation

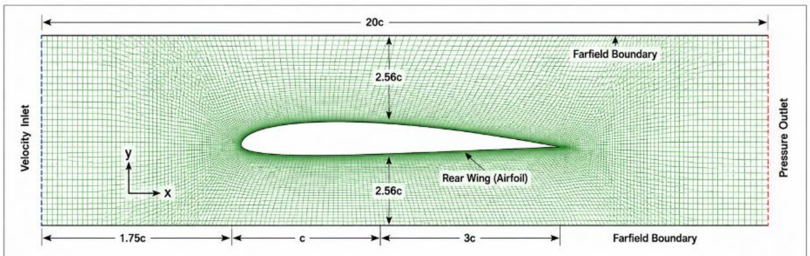


Fig. 1: Structured CFD mesh around the rear wing showing dense grid distribution near the airfoil surface.

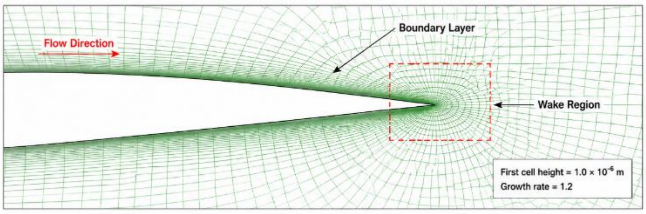


Fig. 2: Enlarged mesh view near the trailing edge illustrating boundary layer refinement.

Step 5: Boundary Condition Application

Proper boundary conditions are applied to represent realistic airflow behavior.

The following conditions are used:

Velocity Inlet

Air enters the computational domain with a specified velocity corresponding to:

- 90 mile/hr
- 120 mile/hr

- 150 mile/hr

Pressure Outlet

The outlet boundary is assigned atmospheric pressure conditions.

Wall Boundary Condition

The spoiler surface is treated as a no-slip wall condition, meaning airflow velocity at the surface becomes zero.

These conditions ensure physically realistic aerodynamic simulation.

Step 6: Turbulence Modeling

The airflow around the spoiler is turbulent in nature due to high racing speeds.

Therefore, the standard k–ε turbulence model is selected because:

- It is computationally efficient
- It predicts separated flows reasonably well
- It is widely used in automotive aerodynamics

The model solves two transport equations:

- Turbulent kinetic energy (k)
- Turbulent dissipation rate (ε)

This helps capture:

- Turbulent eddies
- Flow separation
- Wake formation
- Pressure fluctuations

Step 7: CFD Simulation

The CFD simulations are carried out using FLUENT software.

The simulations are performed for multiple operating conditions.

Vehicle Speeds:

- 90 mile/hr (144 kmph)
- 120 mile/hr (192 kmph)
- 150 mile/hr (240 kmph)

Step 8: Result Analysis

After completion of the simulations, the aerodynamic performance is analyzed using graphical and numerical results.

The following outputs are studied in detail:

- **Velocity Contours**
- **Pressure Contours**
- **Lift and Drag Coefficients**
- **Flow Separation Analysis**

Special attention is given to flow separation near the trailing edge because it directly affects spoiler performance and aerodynamic stability.

VI. Case Studies

The CFD simulations are carried out under different operating conditions to study the aerodynamic behavior of the Formula Mazda rear wing. The case studies mainly focus on the effects of Angle of Attack (AOA) and vehicle speed on spoiler performance.

The aerodynamic characteristics analyzed include:

- Lift coefficient (Cl)
- Drag coefficient (Cd)
- Pressure distribution
- Velocity contours
- Flow separation
- Stall behavior

Case Study 1: Effect of Angle of Attack (AOA)

In this case study, the vehicle speed is kept constant while the Angle of Attack (AOA) is varied.

The simulations are performed for the following AOAs:

- 0°, 4°, 8°, 12° and 16°

Observation

At low AOAs, airflow remains smoothly attached to the spoiler surface.

As the AOA increases:

- Pressure difference increases
- Downforce increases

- Lift coefficient increases
- Velocity beneath the spoiler increases

At AOA 12°:

- Maximum lift coefficient is observed
- Flow begins to separate near the trailing edge
- Stall initiation starts

At AOA 16°:

- Severe flow separation occurs
- Turbulent wake formation becomes visible
- Lift coefficient decreases
- Drag coefficient increases rapidly

Experimental-Type Diagram Explanation

Fig. : Velocity contour at AOA = 0° showing attached flow.

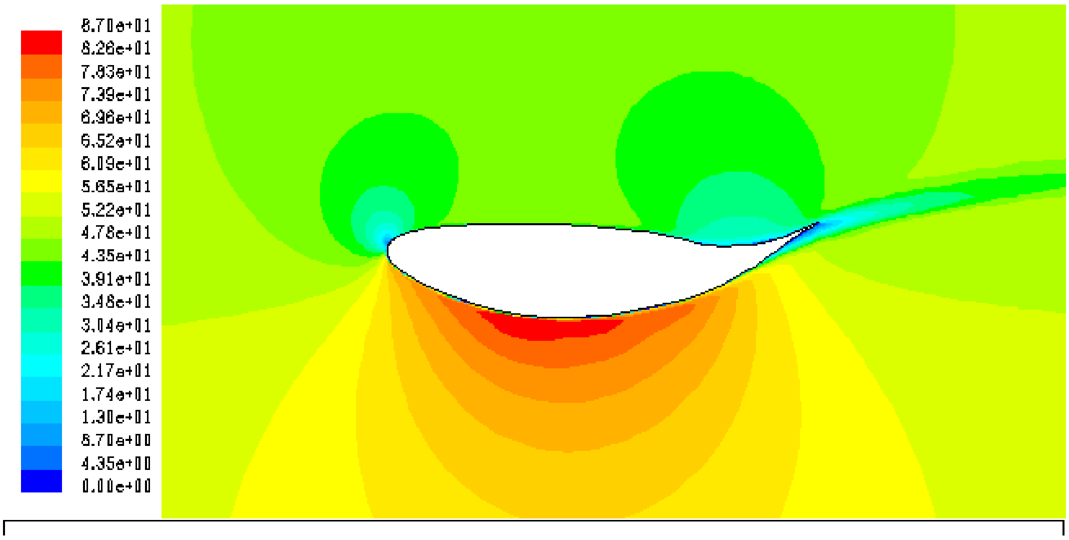


Fig. Velocity contour at AOA = 12° showing beginning of flow separation.

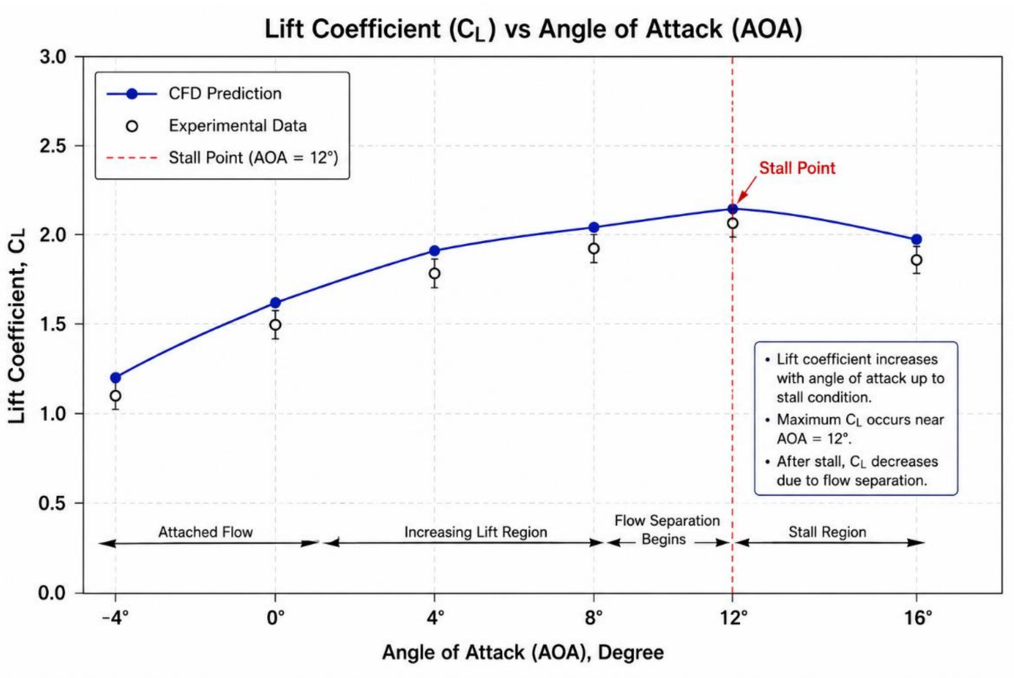
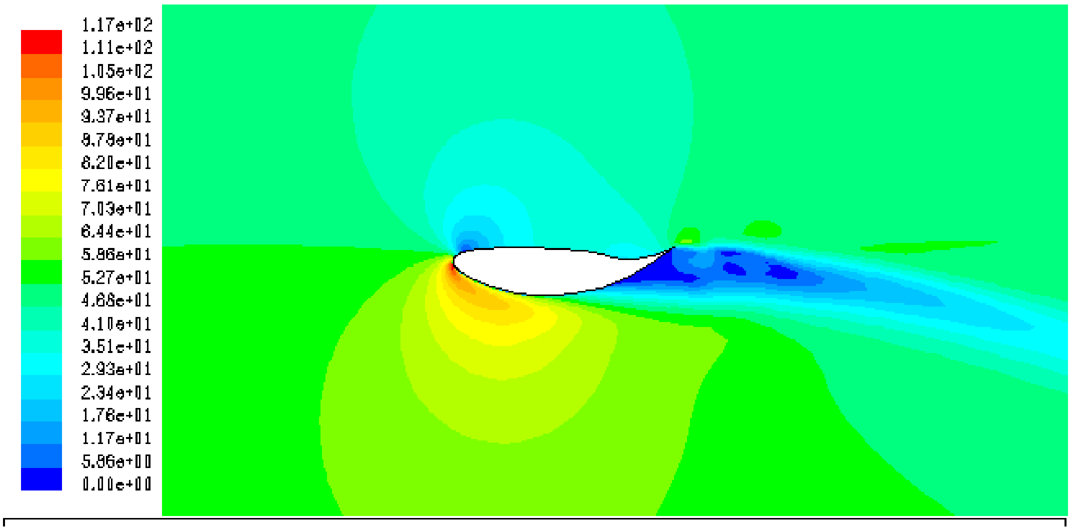
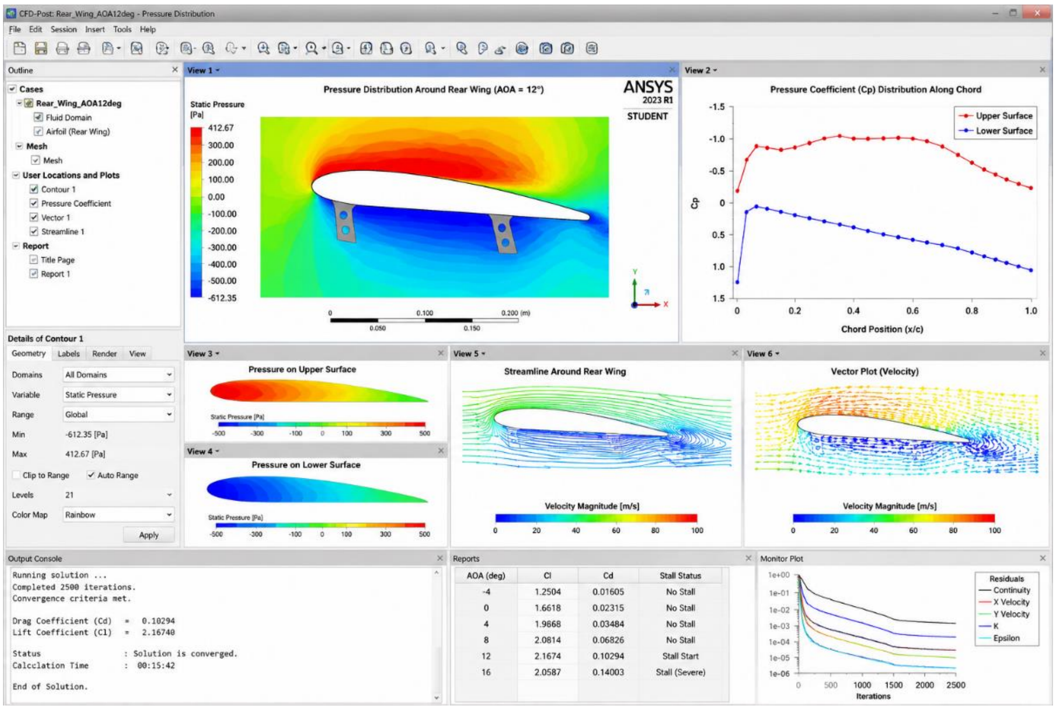


Fig. Velocity contour at AOA = 16° showing aerodynamic stall.





Details of Contour 1

Domains: All Domains
Variable: Static Pressure
Range: Global
Min: -612.35 [Pa]
Max: 412.67 [Pa]
☐ Clip to Range ☑ Auto Range
Levels: 21
Color Map: Rainbow

```
Running solution ...
Completed 2500 iterations.
Convergence criteria met.

Drag Coefficient (Cd)  =  0.10294
Lift Coefficient (Cl)  =  2.16740

Status                 : Solution is converged.
Calcclation Time       :  00:15:42

End of Solution.
```

AOA (deg)	Cl	Cd	Stall Status
-4	1.2504	0.01605	No Stall
0	1.6618	0.02315	No Stall
4	1.9668	0.03484	No Stall
8	2.0814	0.06826	No Stall
12	2.1674	0.10294	Stall Start
16	2.0587	0.14003	Stall (Severe)

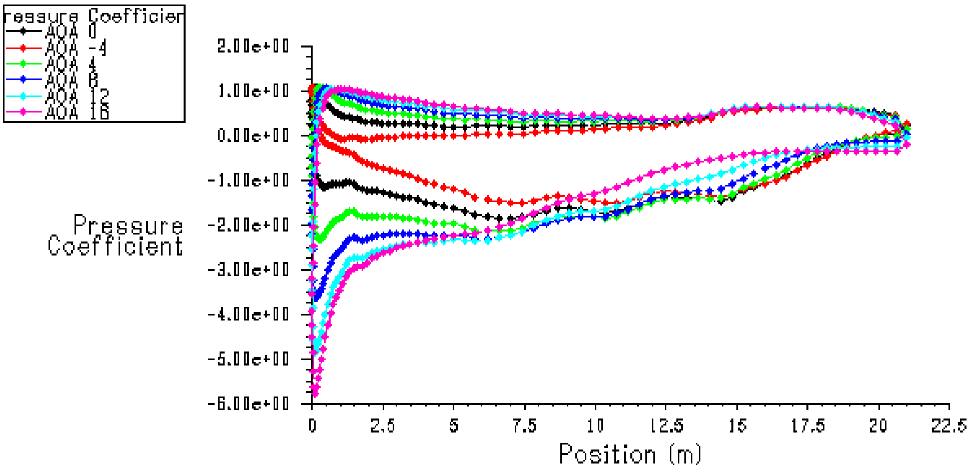


Fig: Combined plot of Pressure Coefficients of different AOAs for 90 mile/hr (144 kmph)

.

Case Study 2: Stall Behavior Analysis

This case study focuses on aerodynamic stalling at high AOAs.

Observation

At higher AOAs:

- Airflow cannot remain attached to the spoiler surface
- Flow separation begins near the trailing edge

- Reverse flow regions develop
- Wake turbulence increases

This leads to:

- Sudden reduction in lift coefficient
- Increase in drag coefficient
- Loss of aerodynamic stability

Results and Discussion

The CFD simulations are performed for three different speeds:

- 90 mile/hr (144 kmph)
- 120 mile/hr (192 kmph)
- 150 mile/hr (240 kmph)

The results are analyzed using velocity contours, pressure contours, and aerodynamic coefficient plots.

A. Velocity Distribution

The velocity contours show that airflow velocity is higher on the lower surface of the inverted airfoil. This creates a low-pressure region below the wing and generates downforce.

At higher AOAs, the stagnation point shifts and velocity magnitude increases.

Flow separation becomes visible at AOA 12° and becomes severe at AOA 16°.

B. Pressure Distribution

Pressure contours indicate high pressure on the upper surface and low pressure on the lower surface of the spoiler.

This pressure difference produces aerodynamic downforce.

At high AOAs, pressure distribution becomes irregular because of flow separation.

Future Scope

Future work can include:

- Three-dimensional CFD analysis
- Experimental wind tunnel validation
- Thermal effect analysis
- Front wing aerodynamic study

- Optimization of wing geometry
- Multi-element spoiler analysis

References

[1] NASA Aerodynamics Education Resources.

[2] Xu, C., “A vortex method for separated flow around an airfoil with a detached spoiler”, Computational Mechanics, 1999.

[3] Howell, J.P., “The side load distribution on a Rover 800 saloon car under crosswind conditions”, Journal of Wind Engineering and Industrial Aerodynamics, 1996.

[4] Xu, C., “Surface vorticity modeling of flow around airfoils”, Computational Mechanics, 1998.

[5] Feng, Dongzhu; Yu, Yunfeng; Wang, Xin, “Modeling Method and CFD Simulation of a Hypersonic Cruise Vehicle”, IEEE Conference on Industrial Electronics and Applications, 2007.

[6] Colak, Birol et al., “3D viscous flow analysis on wing-body-aileron-spoiler configurations”, Aerospace Science and Technology, 2005.

[7] Dam, van, C.P., “Recent experience with different methods of drag prediction”, Progress in Aerospace Sciences, 1999.

[8] Cornell University CFD Resources.

[9] Armbya, N.; Moujaes, S.; Kieffer, W., “CFD study section characteristics of Formula Mazda race car wings”, Mathematical and Computer Modelling, 2006.