

FORMULA FOR RACING SUCCESS

Students apply robust design principles to develop a highly competitive race car.

By **Philipp Epple**, Professor, **Stefan Gast**, Professor, and **Peter Neugebauer**, University of Applied Sciences Coburg, Faculty of Mechanical Engineering and Automotive Technology, Coburg, Germany

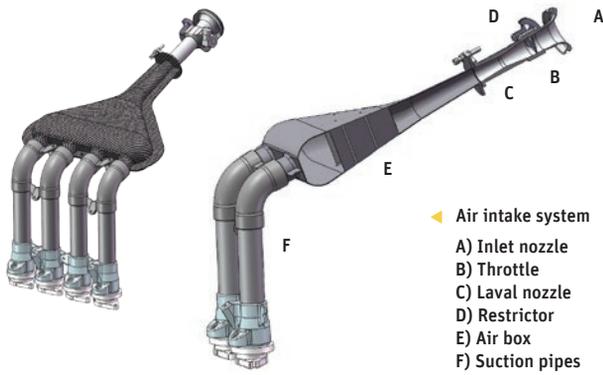


▲ C-12 Puma from CAT Racing

Formula Student Germany (FSG) is an international race car design competition for students at universities of applied sciences and technical universities. The winning team is not the one that produces the fastest racing car, but the group that achieves the highest overall score in design, racing performance, business planning and marketing.

Since 2007, students of the University of Applied Sciences Coburg (UAS Coburg) have participated in this competition as part of the Coburg Automobile Team (CAT). Members of different faculties, including mechanical engineering and automotive technology, business administration, and civil engineering and design, take part. During the four races of the 2012 championship, CAT Racing twice achieved second place. Group members accomplished this success with hard work, discipline and outstanding technical equipment.





◀ Air intake system
 A) Inlet nozzle
 B) Throttle
 C) Laval nozzle
 D) Restrictor
 E) Air box
 F) Suction pipes



Race car aerodynamics, and aerodynamics in general, takes two forms: external and internal flows. External aerodynamics applies to the external shape of the race car; engineers study it to determine the down force on the chassis that will deliver ideal driving dynamics by minimizing aerodynamic drag. The aerodynamics of the car's internal components is just as important. For example, predicting aerodynamics of the air intake system is crucial to optimizing car performance as speed changes. The air intake system consists of an inlet nozzle, throttle, Laval nozzle-shaped restrictor, air box and cylinder suction pipes.

AIR INTAKE DESIGN

The team from UAS Coburg improved the car's air intake system using ANSYS CFD within the parametric ANSYS Workbench environment. FSG regulations limit the minimum diameter of the restrictor to 20 mm, which regulates the maximum intake mass flow rate. The air box, downstream of the restrictor, directly influences the amount of fresh air reaching the cylinders. An air box that is too large causes the motor to react too slowly to the accelerator and, in combination with short suction pipes, triggers the engine to develop sufficient torque only at high rotation speeds. An air box that is too small behaves in the opposite



▲ Formula Student Germany CAT Racing team from UAS Coburg

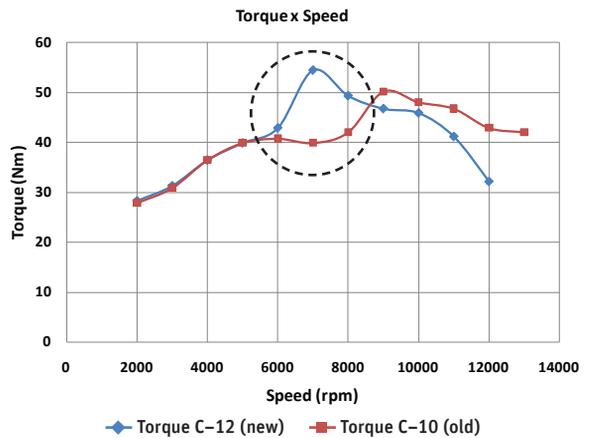


manner. Therefore, the team needs to carefully design the air box and match it with the suction pipe lengths to optimize torque over the entire range of operating speeds. In earlier CAT Racing cars, the air box was designed mainly based on ease of construction; in 2012, ANSYS Workbench was employed to develop a true aerodynamic design. As a result, the 2012 race car model, the C-12 Puma, delivers the correct torque to the driver at the right time (that is, at the proper speed). This optimization allowed the team to win two second-place finishes.

The faculty of Mechanical Engineering and Automotive Technology at UAS Coburg has access to modern technical equipment, including an engine test rig, which the team used to test the ANSYS Workbench design before road testing. Facilities also include an advanced computer lab, where students employ ANSYS software to obtain the optimal design.

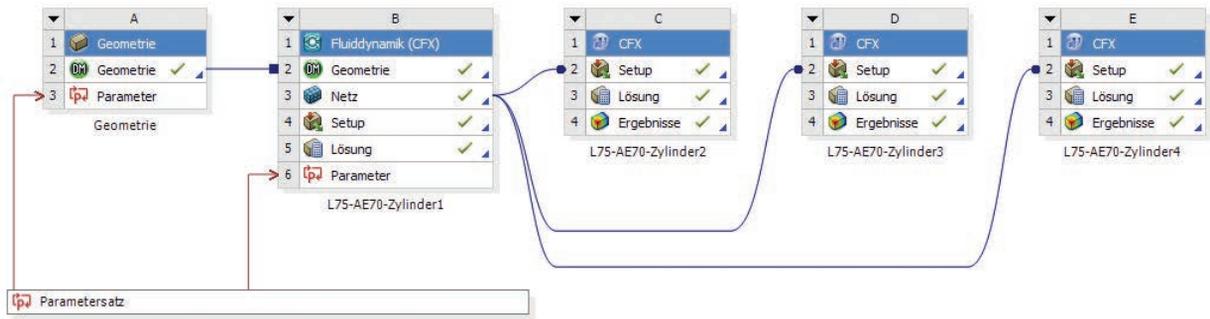
The team developed the air intake system in three steps:

- Students computed and analytically dimensioned the system based on equations of theoretical gas dynamics using an Excel® design tool.
- The team implemented the design in CAD using SolidWorks® to determine the flow domains for CFD simulation.
- UAS Coburg performed optimization within the ANSYS Workbench environment. The team generated a parameterized CAD model and transferred the model to ANSYS Workbench using the CAD interface for SolidWorks. CAT Racing then generated a table of design points within Workbench. At each new design point, the data was transferred through the ANSYS CAD interface to SolidWorks, where a new geometry was generated. This new geometry was then returned automatically to Workbench, where a new grid was generated, and the CFD solver started automatically. This process was repeated until all design points were processed.



▲ The C-12 Puma generated the correct amount of torque at the right speed.





▲ ANSYS Workbench workflow

ANSYS WORKBENCH PARAMETERIZED WORKFLOW

The optimization procedure in ANSYS Workbench is very effective and user friendly, employing a simple workflow. Geometry is easily imported into ANSYS DesignModeler within Workbench using the ANSYS CAD interface for SolidWorks. Still within Workbench, the mesh is generated from this geometry using ANSYS meshing. The grid parameters are set once, and then the grid can be generated automatically in the background for all configurations of the geometry without reopening the meshing tool. For the air intake, the team included four configurations (corresponding to each of the four cylinders) in the workflow, and each design was run for all of these configurations. Workbench auto-detected these configurations and ran them sequentially.

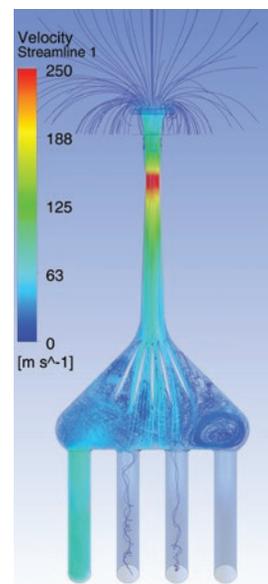
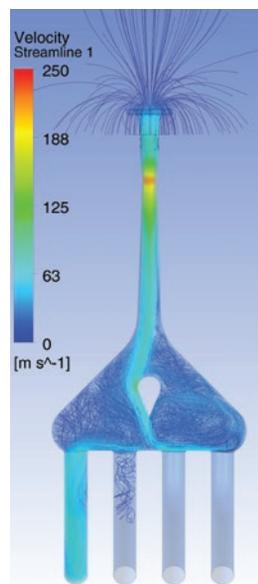
Once the CFD solver (ANSYS CFX) finished running all four configurations, the next parameter in the table of design points was sent back to SolidWorks. Using this parameter, SolidWorks generated the next geometry set, which was then passed to Workbench, and the solution process was reinitiated.

Using this method, the team investigated two- and three-dimensional models of the air intake system. The advantage of this procedure is that 2-D models are computed rapidly so that fundamental alternatives can be explored and the design quickly altered – for example, to maximize mass flow rate through the system. CAT Racing then developed 3-D models for the 2-D geometries that gave the best performance, and conducted 3-D CFD simulations. Again, the CAD interface and table of design points generated CAD models and grids, and provided parametric

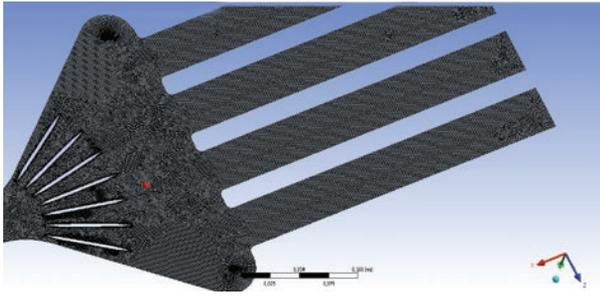
solutions within the Workbench environment. High-performance computing was conducted on some of the 3-D cases to increase solution speed.

In the original air box, the CFD simulation showed that the flow failed to reach the cylinder in a direct path. The central separation caused by the design of this original box based on structural requirements actually obstructed the flow. To solve this problem, the UAS Coburg team added guide vanes to the air box. With an automatic optimization loop implemented in ANSYS Workbench, the team optimized the number of guide vanes, their thicknesses, the opening angle of

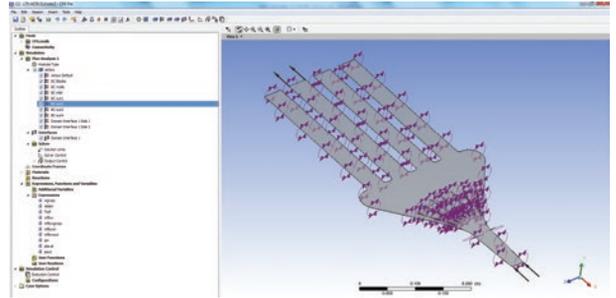
The optimization procedure in ANSYS Workbench is very effective and user friendly, employing a simple workflow. ▶



▲ Old C-10 (left) and new C-12 (right) air intake system. In the original air box, the flow does not reach the cylinder in a direct path. The addition of guide vanes in the new system provides better intake and more uniform distribution of air flow.



▲ Grid generation with ANSYS meshing



▲ One of the four setups in ANSYS Workbench, with the second suction pipe in operation

the air box and other parameters. This system of guide vanes in the air box diverted the flow into the corresponding open cylinder with negligible flow separation. Losses in the system were reduced, and the mass flow rate through the restrictor was increased. The structural stability of the air box also was improved. In the new system, the air box flow distribution is more uniform.

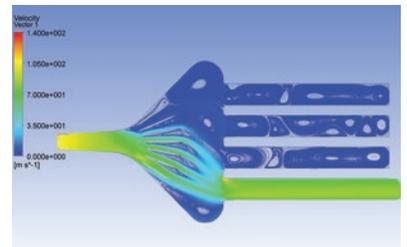
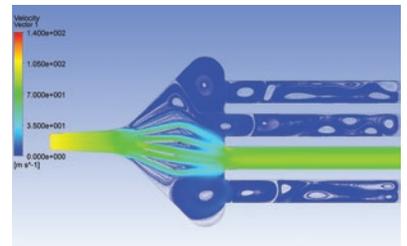
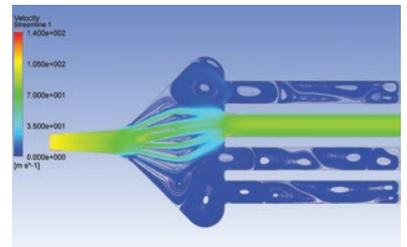
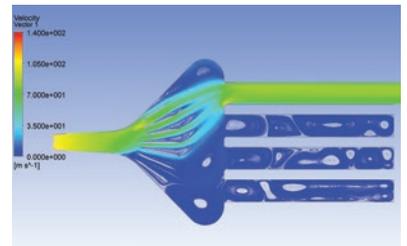
ENGINEERING BY THE NUMBERS

The requirements for simulation post-processing go beyond producing a series of pretty pictures. The team needed to extract quantitative information, so the CFD result files were further post-processed using power syntax within CFD-Post. This feature can be integrated with the well known and powerful programming language PERL to access complex post-processing functionality.

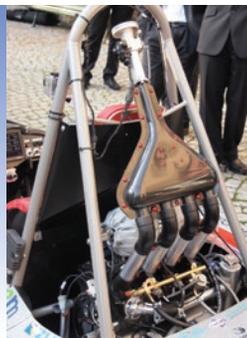
Using PERL and CFD-Post power syntax, the faculty of the Mechanical Engineering and Automotive Technology

department wrote complex post-processing scripts to precisely extract relevant data from the CFD result files and write the calculations to an Excel file. These results then were displayed and analyzed in charts. Using this method, it was possible to analyze and compile a huge amount of simulation data in a clear and concise way. The team analyzed total pressure, static pressure, Mach number, entropy, enthalpy and other data throughout the system. Based on this information, CAT Racing was able to analyze the system objectively and gain the knowledge required to effectively improve the design of the air intake system.

Additional aspects of internal and external aerodynamics of the Formula student race car are currently being analyzed at UAS Coburg. Combined with a sound knowledge of theoretical aerodynamics and a modern test facility, CFD simulation using ANSYS Workbench continues to be a key technology in the car's aerodynamic development. ▲



▲ Two-D model of the new air box showing flow through cylinders



▲ Air box CAD model (left), 3-D CFD simulation (center), air box installed in racing car (right)