Reverse Engineering and Performance Enhancement of a NACA Duct for Cockpit Cooling in a GT4 Racing Car

Conference Paper · June 2024

Mohammad Harris
University of Hertfordshire

Hongwei Wu
University of Hertfordshire

Reverse Engineering and Performance Enhancement of a NACA Duct for Cockpit Cooling in a GT4 Racing Car

Mohammad Harris 1[1 2*] , Hongwei Wu[1], Williem Toet[2]
[1]*University of Hertfordshire, College Lane, Hatfield, United Kingdom*
[2]*University of Bolton, Bolton, Greater Manchester, United Kingdom*
*corresponding author: m.harris8@herts.ac.uk

This research investigates vented airflow optimisation for cockpit cooling in a GT4 car via NACA duct designs. Despite their efficiency in providing cooling with minimal aerodynamic drag, NACA duct designs are relatively overlooked in GT4 racing. Therefore, this study aims to address this gap by combining reverse engineering, additive manufacturing, numerical simulation, and wind tunnel testing. Comparative design analysis indicates that a new prototype design featuring a 5˚ ramp angle and 1:2 area ratio improves airflow by 15% and doubles suction with minimal drag. These findings offer promising insights for enhancing part designs and balancing performance enhancements whilst minimising trade-offs.

Keywords: aerodynamics cooling; numerical simulation; wind tunnel; reverse engineering; motorsports

Introduction

Currently, NACA ducts are often used for aerospace, automotive, amongst other cooling applications [1]. With proper installation, NACA ducts can enhance the vented airflow into systems with minimal disturbances (turbulence) to the free stream and surrounding boundary layers. Through effective flow area manipulations, NACA ducts create pressure differences to suck air into a system. The combination of diverging curved walls and a shallow ramp design results in causing negligible turbulence. For optimal performance, the three most important factors for NACA ducts are design, placement, and application [2-3].

Despite some research regarding the duct shape [4], nozzle angle [5], and vehicle simulations [6], ribs [7], performance enhancement [8], the literature review indicated a scarcity of available recent research on this topic. Also, it highlighted that current NACA models in the market often ignore important design features due to cost, manufacturing, amongst other restraints, especially in GT4 racing. This arguably results in inefficient cooling and reduced ergonomics of the system. Therefore, this invited the need for an investigation.

Consequently, the main aim of the paper is to reverse engineer a design that improves the vented airflow into the cockpit of a Ginetta G55 GT4 car for improved cooling, ergonomics, and driver comfort. Based on driver feedback and datalogger outputs, the initial assumption was that the vented airflow going into the cockpit was not optimal due to poor design features, and it caused turbulence and overhead pressure on the vehicle — which ultimately led to inefficient cooling, ergonomics, and unwanted drag.

Materials and Methods

The original NACA duct of the Ginetta car was preserved due to functionality purposes. As a result, the existing working part was reconstructed through different reverse engineering methods. Adopting a 'systems thinking' approach [9], the research problem was segmented into six distinct stages that contained elements of aerodynamics, 3D printing, manufacturing, and computer-aided engineering to solve the issue and produce an improved design. These stages are highlighted in the following in Figure 1. The governing equations and assumptions used for modeling were as follows:

Continuity equation: $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$ (concerns with mass conservation)

Bernoulli's equation: $\mathrm{P}_1 + \frac{1}{2}\,\rho v^2 + \rho_f g h_1 = \mathrm{P}_2 + \frac{1}{2}\,\rho v^2 + \rho_f g h_2$ (concerns with pressure changes)

Where, u, v, w – velocity components; x, y, z – directions; ρ_f – fluid density; P – pressure; h – height of fluid from a reference; g – acceleration due to gravity.

The utilisation of fundamental equations and dimensional manipulations facilitated the development of the final optimised design. Initially, CFD simulations analysed ramp angles of 5, 7, 9, and 11.5 degrees, each with varying aspect ratios [2]. The boundary conditions were set with an inlet velocity of 50 m/s (equivalent to 180 km/hr), typical speeds observed during GT4 races. The outlet was left open and exposed to atmospheric pressure (inside the cockpit), while the air temperature for both CFD and Wind Tunnel testing was taken as 298K. Additionally, it was noted that the size of the chamber leading to the outlet had negligible impact on the pressure drop or the spread of the velocity, given the open outlet to atmospheric pressure. Subsequently, the two models underwent assessments on ANSYS CFX via a k-omega model due to its superior and faster convergence than the k-epsilon model, in this case. Therefore, based on the CFD results, the 5-degree ramp angle with a 1:2 aspect ratio exhibited the highest vented airflow velocity among all designs. Consequently, the optimised model and the original model currently (affixed to the car) were fabricated and subjected to Wind Tunnel testing.

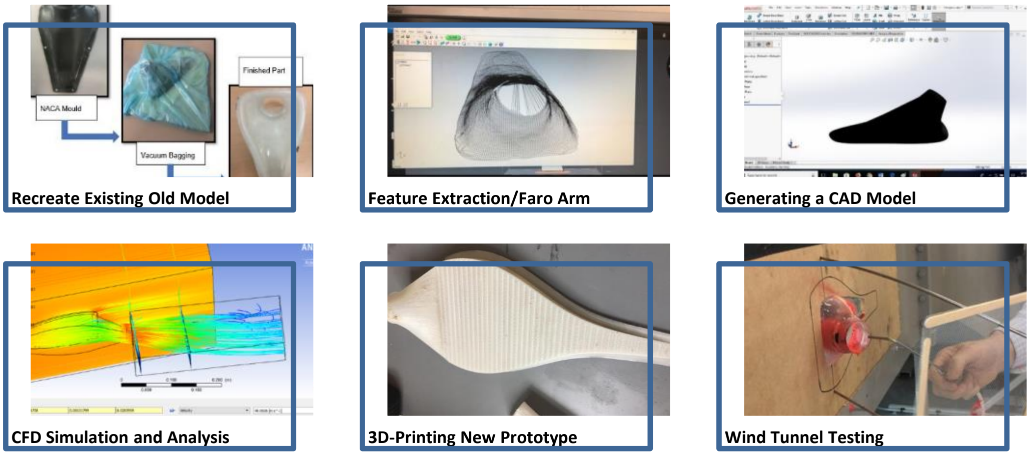

Fig. 1. All stages of the research methodology and implementation

Results and discussion

The simulations aimed to emulate wind tunnel testing; therefore, the wind tunnel, in turn, validated the numerical simulations by assessing the vented airflow of the original NACA duct, with a 12% percentage error. Due to health and safety protocols, velocities ranging from 20 - 30 m/s were employed for the testing. The simulations focused on four distinct areas: velocity streamlines, pressure differentials, turbulence wake profiles, and velocity contours for the boundary layers. A side-by-side illustrations in Figure 2 highlight key observations.

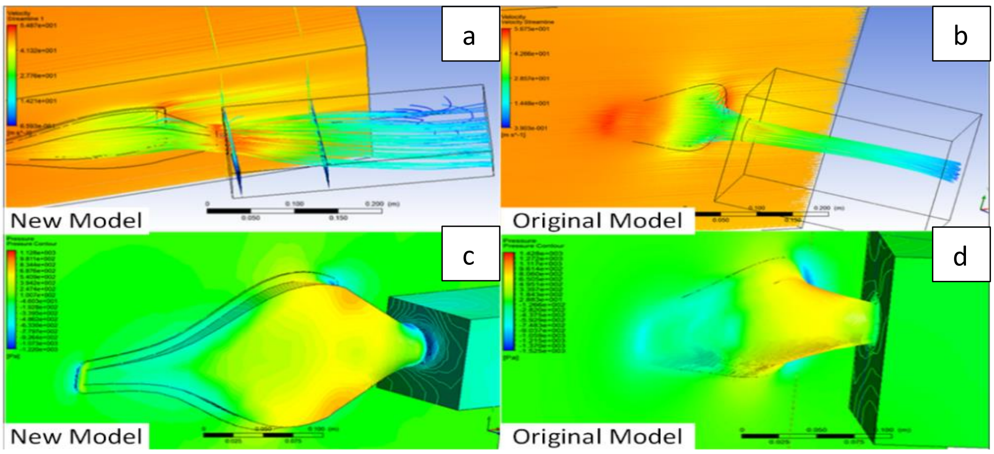

Fig. 2 (a)-(d). CFD Simulation results for velocity and pressure distribution.

Figures 2(a)-(b) demonstrated that the new model exhibited a wider and more turbulent spread of airflow inside the chamber, with minimal disturbance to the freestream, ultimately enhancing passive cooling. Furthermore, Figures 2(c)-(d) reveal high-pressure at the NACA duct edge and low-pressure zones at the inlet for the new design, indicating suction. Conversely, the original hypothesis was validated, as Figure 2(d) depicted a low-pressure zone at the NACA duct's edge and a higher-pressure zone (green) at the inlet, affirming that the old design produced undesired effects and unwanted drag.

The wind tunnel testing mainly focused on vented airflow velocities and pressure using pitot tubes and a digital Testometer, summarised in Table 1. The new model exhibited a 1.8% increase in turbulence, due to changes in geometry and airflow characteristics. This trade-off remains acceptable, as it does not necessarily imply turbulent boundary layers. The new model showed improved airflow in both CFD (12.75%) and wind tunnel testing (14.62%) results. Additionally, it rectified an undesirable negative pressure difference from the old model, potentially enhancing airflow efficiency and cooling. Furthermore, the new model introduced a beneficial 2 Kpa positive pressure difference. This positive pressure can aid in regulating the temperature inside the cockpit by promoting air circulation and heat dissipation, thereby preventing excessive heat buildup, and enhancing the driver's comfort. Future investigations can focus on heat transfer and material choices for NACA duct selections.

Table 1: Comparison of Wind Tunnel and CFD results

Parameter	New Model	Old Model	Improvement
Turbulence	13.25 joules	13.02 joules	+1.8%
Max velocity (CFD)	48.10 m/s	42.66 m/s	12.75%
Pressure difference	1.76 KPa	-0.25 KPa	2.01 KPa
Max velocity (wind tunnel)	12.39 m/s	10.81 m/s	14.62%

Conclusion

In conclusion, acknowledging any limitations of this study, the investigation successfully achieved its main aim to reverse engineer a new design that improved the vented airflow for the cockpit cooling purposes of the GT4 car by 15%, within minimal drag impacts and desired positive pressure buildup. The findings from this study can trigger new research areas or help individuals and businesses to make the best use of NACA duct technology.

References

[1] Kumar, Sanjeev, Randip Kumar Das, and Kishor Kulkarni. "Comparative study of solar air heater (SAH) roughened with transverse ribs of NACA 0020 in forward and reverse direction." Case Studies in Thermal Engineering 34 (2022): 102015.
[2] Frick, Charles W., Wallace F. Davis, Lauros M. Randall, and Emmet A. Mossman. An experimental investigation of NACA submerged-duct entrances. No. NACA-ACR-5I20. 1945.
[3] Mossman, Emmet A., and Donald E. Gault. Development of NACA Submerged Inlets and a Comparison with Wing Leading-edge Inlets for a 1/4-scale Model of a Fighter Airplane. National Advisory Committee for Aeronautics, 1947.
[4] Yilmaz, Serdar, Duygu Erdem, and Mehmet Kavsaoglu. "Effects of duct shape on a ducted propeller performance." In 51st AIAA aerospace sciences meeting including the new horizons forum and aerospace exposition, p. 803. 2013.
[5] Khan, Md Ayaz, Sanjay D. Pohekar, and Pramodkumar M. Bagade. "Hydrodynamic Analysis of NACA 4415 Hydrofoil for Marine Applications." Journal of Mines, Metals & Fuels 71, no. 11 (2023).
[6] Menon, Abhilash. "NACA Ducts in vehicle thermal simulations." (2017).
[7] Patel, Yogesh M., Sanjay V. Jain, and Vikas J. Lakhera. "Thermo-hydraulic performance analysis of a solar air heater roughened with reverse NACA profile ribs." Applied Thermal Engineering 170 (2020): 114940.
[8] Zhang, Junyao, and Baigang Mi. "Internal Aerodynamic Performance Enhancement for Aircraft with High Maneuver by Designing a Distributed Submerged Inlet." Applied Sciences 13, no. 3 (2023): 1459.
[9] Senge, Peter M. The fifth discipline: The art and practice of the learning organization. Broadway Business, 2006.