Improvement of Porsche Cayenne Aerodynamics using Reverse Engineering, CAE and Rapid Prototyping

Haricharan Kannan ,Ibrahim Shehadeh and Junling Hu
Department of Mechanical Engineering, University of Bridgeport

Abstract

The external vehicle aerodynamics have a strong impact on the vehicle's fuel efficiency and handling behavior. External aerodynamic simulations using computational fluid dynamics (CFD) have been used in the automotive industry to develop modern cars. This project aims to study the external aerodynamics of a Porshe Cayene model. The car model is scanned by a 3D laser scanner and the scanned data is used to reconstruct the 3D CAD model by reverse engineering for CFD simulation. The CFD simulation results will be used to develop a new car model shape to improve the aerodynamic performance. The new car model will be prototyped by a 3D printer

Introduction

Studying automotive Aerodynamic has paramount of importance to reduce the undesired drag force and lift forces at high speeds and thus reducing fuel consumptions and environment pollution and improve vehicle safety. The automotive industry has relied on wind tunnel experiments and computational fluid dynamics (CFD) simulations to study vehicle aerodynamics. In recent year, CFD simulations have seen increasing applications in the development of new vehicles due to the improvement of computing power and CFD software capabilities and fast development of reverse engineering and rapid protyping.

Currently, Reverse Engineering is employed widely in different engineering fields in order to capture the geometry of an existing product. Rapid prototyping methods quickly fabricate a scale model of a physical part or assembly using three-dimensional computer aided design (CAD) data. Combining these technologies with computer aided engineering (CAE) can create new ways of improving the current product. It can break down the complicated problems and enhance researches with valuable capabilities that help to save time, experiments expenses, and other advantages of that. But with each new method there are new challenges that could affect on the process simplicity, and lead us to new ways of analyzing, employing and estimating each parameter.

Studying the turbulent structure around car model by getting the geometry through 3D scanning and transfer the geometry to CAE packages carries a lot difficulties. In this project, we address these obstacles through multiple stages. Firstly defining the 3D scanned geometry problems that related to the Resolution, speed, accuracy, reconstructing the surface and mesh, the impact of work piece surface and color on laser beam, the limitation of software in editing the raw data, mesh and reconstructing the surface and transferring the data to CAD software like Pro/Engineer or Solidworks, detecting the points that could fail prototyping the work piece. Secondly, the Exporting geometry to different software in order to edit the data, modifying and improving different types of geometry to study each one separately in ANSYS. Thirdly computational fluid dynamic is used as effective tool to study the case. The process goes through selecting the suitable parameters and functions that impact on mesh quality, result visualization, solver setups. In this can the research can detect the external shape problem, improve it and repeat the simulation until getting the optimized one. Fourthly, printing the improved model using a 3D printer. In this stage we will try to choose the suitable orientation of creation inside the printer, select the best setting that improve the surface quality and minimize the possible creation damage.

Reverse Engineering

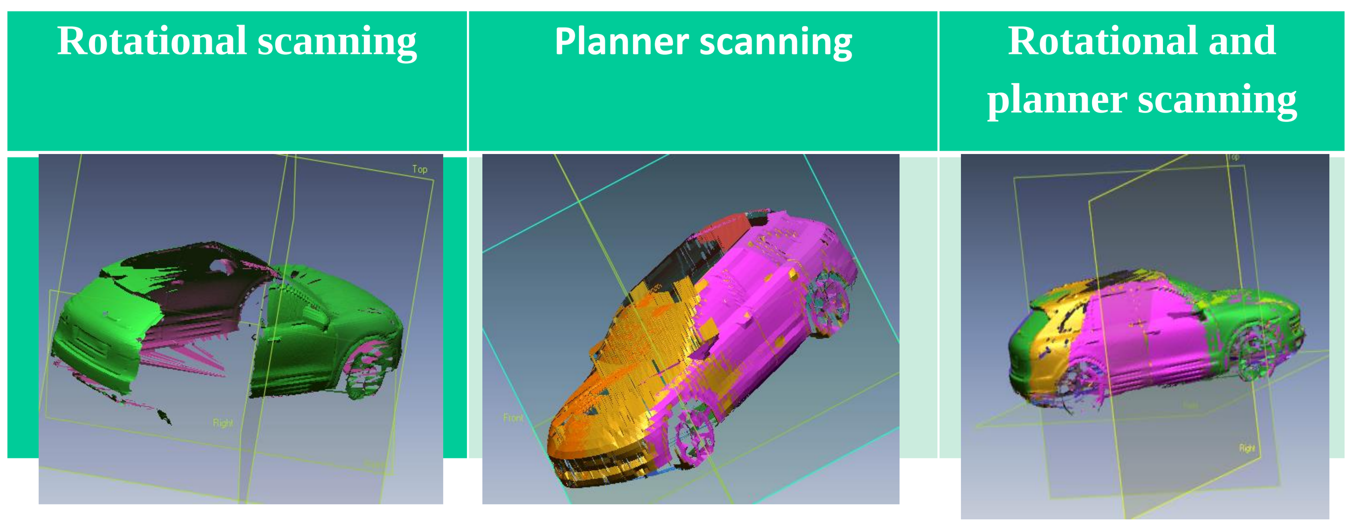
The scanner that used to capture the geometry was PICZA 3D laser scanner LPX-600 and the software was Pixform pro2.We did the scanning in multiple ways, Firstly, the rotational scanning that gave us good original shape with some deformations and missed parts, because the laser beam's path changed at the corners. For that, we repeated the scanning with planner scanning that corrected what was wrong with the rotational scanning.

The Rotational scanning parameters:

Work piece Height	Height - direction Pitch	Circumferential Pitch	Cutoff angel	Estimated scan time
71.4 mm	0.2 mm	0.18 mm	90 deg	41 min

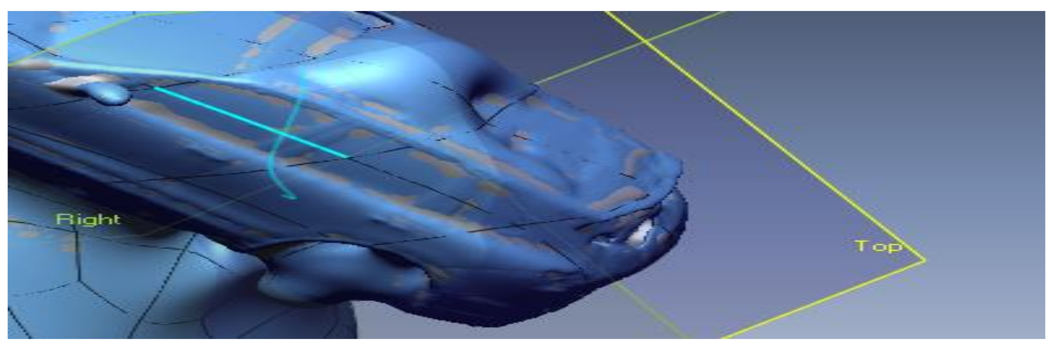
The planner scanning parameters:

Surface to scan	Height - direction Pitch	Circumferential Pitch	Cutoff angel	Estimated scan time
6	0.2 mm	0.2 mm	90 deg	161 min



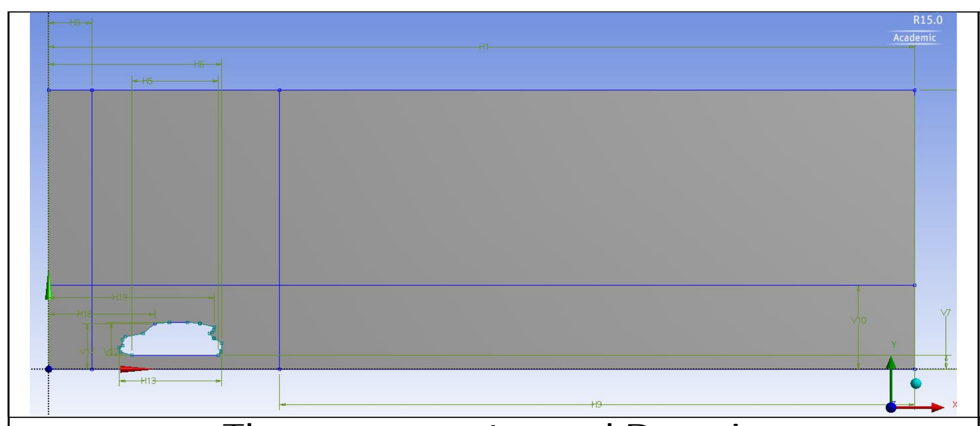
Scanning results

Reconstructing the surface:



CAE using ANSYS Fluent

The project will develop a numerical model to simulate the complex turbulent flow structures around a car in ANSYS Fluent. A 2D CFD model will first be developed to simulate the turbulent flow around a representative of the car cross-section. The computational domain of the 2D car model shown below includes an inlet, nose, road, top, back and outlet which are separated into various surfaces. The 2D car model simulation shares the same workflow of a 3D simulation as model preparation, mesh generation, CFD case setup and solution run, and post processing of simulation results. A 3D CFD simulation model will be further developed to include the 3D reconstructed geometry from reverse engineering for studying the external car aerodynamics, including complex turbulent structure, vortices formation, drag and lift. The simulations can enhance understanding and provide guidance for designing an car with smaller drag and higher fuel mileage.



The car geometry and Domain

Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 13	
H1	30 m
H13	3.544 m
H16	3.6781 m
H19	5.7363 m
H5	3 m
H6	6 m
H8	1.5 m
H9	22 m
V10	3 m
V12	1.1803 m
V17	1.6353 m
V2	10 m
V7	0.5 m

Dimension Table

Rapid prototyping

A 3D printer will be used to print the reconstructed surface by reverse engineering and compare with the original car model. It will also be used to print the improved car model obtained through CFD simulations. In this stage we will try to choose the suitable orientation of creation inside the printer, select the best setting that improve the surface quality and minimize the possible creation damage.

Conclusion

In this study Porsche Cayenne is scanned and a 3D CAD model will be reconstructed using reverse engineering for computer aided engineering. ANSYS Fluent will be used to simulate the aerodynamics and improve the car body shape for better aerodynamic performance. The optimized car model will then e printed using rapid prototyping.