A Coupled Panel and Vortex Particle Method for Simulation of a Rotor in Hover

Timothy Harlow

A capstone report submitted to the faculty of
Brigham Young University
in partial fulfillment of the requirements for the degree of

Bachelor of Science

Andrew Ning, Advisor

Department of Physics and Astronomy

Brigham Young University

ABSTRACT

A Coupled Panel and Vortex Particle Method for
Simulation of a Rotor in Hover

Timothy Harlow
Department of Physics and Astronomy, BYU
Bachelor of Science

Panel methods are a powerful tool for computing the influence of solid bodies such as aircraft in computational fluid dynamics (CFD). A panel method works by discretizing the object into geometric regions (panels), assigning each panel a source, dipole, or vortex singularity distribution, and solving for their strengths to satisfy the governing fluids equations [1]. This can be used in conjunction with a Vortex Particle Method (VPM), which gives more accurate wake predictions. However, the method of using panels with vortex particles remains untested for a variety of interesting scenarios. This project will test the code's accuracy for the case of a rotor in hover [2]. The code was unsuccessful in predicting thrust for this case, but the concept is valuable and will continue to be discussed in upcoming publications.

Keywords: Vortex particle method, panel method, rotor, unsteady, hover

ACKNOWLEDGMENTS

Many thanks to Ryan Anderson, a Ph.D. student in the FLOW Laboratory for providing regular support and developing the code discussed in this paper. I am also grateful to Dr. Andrew Ning and the Mechanical Engineering Department at BYU for providing Research Assistant funding throughout this project.

Contents

Table of Contents iv

List of Figures v

List of Tables v

1 Introduction 1

2 Methods 3
2.1 Numerical Methods . . . 3
2.2 VPM Convergence Study . . . 5

3 Results 9
3.1 Future Work: Ground Effect . . . 12

4 Discussion 13

Bibliography 15

Index 17

List of Figures

2.1 Spanwise convergence 6

2.2 Azimuthal convergence 6

2.3 Temporal convergence 7

2.4 C_T as a function of σ. 8

3.1 Rotor in hover 10

3.2 Rotor mesh 11

List of Tables

2.1 Numerical parameters 7

3.1 Physical parameters (rotor in hover) 9

3.2 Physical parameters (ground effect) 12

Chapter 1

Introduction

With the increasing complexity and broadening use of aircraft in diverse scenarios, fast and accurate simulation is a vital part of the design process. With accurate simulations to use in design optimization, aircraft from drones to commercial airlines will have better performance. As Alonso et al. mention, full aircraft optimization can be prohibitively expensive [3]. Thus, for such a simulation code to be useful, it must be fast, allowing designers to try many different design parameters in quick succession. This paper will validate code developed in BYU's Flight, Optimization, and Wind (FLOW) Laboratory. The code will be validated against an experimental case of a drone in hover as provided by Zawodny et al. [2]. The code uses state of the art computational fluid dynamics (CFD) techniques such as a panel method and a Vortex Particle Method (VPM). A brief description of these methods follows.

Panel methods are numerical schemes for solving potential flow fields for arbitrary geometries representing real aircraft at relatively low cost. Panel methods all solve a *potential flow* problem, in which the bulk fluid is assumed to be inviscid and irrotational. With these assumptions, the general Navier-Stokes equations can be simplified and expressed in terms of a velocity potential which satisfies Laplace's equation [4]. The velocity potential obeys the principle of superposition, meaning the total potential field is the sum of all individual sources. Thus, we can discretize the geometry into panels and assign each a source, dipole, or vortex singularity distribution [1]. A linear system is solved that gives strength of each panel satisfying appropriate boundary conditions (specifically, that there is no local flow normal to each panel). These strengths are then used to evaluate the fluid velocity field, which is used to predict a pressure distribution using Bernoulli's equation. The process is repeated as the system evolves in time.

Panel methods work well to describe flow behavior near a solid boundary, but a Vortex Particle Method (VPM) is better for capturing turbulence and viscosity in the wake. In a VPM, a set of point vortices are shed from the surface of the aircraft, their strengths are used when calculating the velocity field, and each vortex is convected with that field when the time step is advanced [5]. The advantages of this include limited numerical dissipation, low computational cost, and excellent accuracy for unsteady and turbulent wakes, such as those produced by rotors [5].

Recently, Alvarez et al. have improved on the VPM, reformulating it to be significantly more stable than the classical VPM. This reformulated VPM (rVPM) converges numerically and is extensively validated [6]. A full mathematical description of the method can be found in their paper [6]. The rVPM is what the formulation we will use in this report.

This paper will validate the use of panel methods with a VPM for a rotor in hover, with experimental data provided by [2]. For this case, a coefficient of thrust (C_T) of 0.0037 was predicted, compared to the experimental value of $C_T = 0.072$. Significant work is being done to uncover the source of the error, so the results discussed in this paper should be thought of as preliminary.

Chapter 2

Methods

This chapter will outline some of the theory behind panel methods and the Vortex Particle Method (VPM). The numerical methods each take a different region of the flow (i.e. the rotor, ground, or wake) and apply an appropriate technique to determine the velocity field in that region. The velocity field can be used to determine forces on areas of interest.

To ensure the validity of the numerical parameters used in the VPM, a convergence study was also performed and will be detailed in this section.

2.1 Numerical Methods

The rotor and ground were modeled using a panel method. Panel methods utilize potential flow theory, which expresses the velocity $\vec{V}$ as the gradient of a scalar function ϕ [4]. That is,

$$\vec{V} = -\nabla\phi. \tag{2.1}$$

If we say that $\vec{V}$ is incompressible, then ϕ will satisfy Laplace's equation,

$$\nabla^2\phi = 0. \tag{2.2}$$

This equation can be solved when appropriate boundary conditions are satisfied. One such condition is that the velocity at infinity must approach the freestream velocity $\vec{V}_\infty$. Another is the no flow through condition, which is that there can be no velocity normal to the surface of a body. This condition can be mathematically expressed as

$$\vec{V} \cdot \hat{n} = 0. \tag{2.3}$$

In practice, the body is discretized into panels, and the velocity at each panel can't be normal to that surface. The $\vec{V}$ in equation 2.3 refers to the total fluid velocity at each panel, which can be broken into the the velocity from external sources $\vec{V}_\infty + \vec{V}_{wake}$ and velocity induced by all panels $\vec{V}_{panels}$. Thus, the no flow condition can be written as $(\vec{V}_\infty + \vec{V}_{wake} + \vec{V}_{panels}) \cdot \hat{n} = 0$, or

$$\vec{V}_{panels} \cdot \hat{n} = -(\vec{V}_\infty + \vec{V}_{wake}) \cdot \hat{n}. \tag{2.4}$$

To get $\vec{V}_{panels}$, we multiply the circulation of each panel Γ by an influence coefficient matrix A. The component A_{ij} gives the influence of panel j on panel i. The influence matrix can be constructed in different ways depending on whether the panel singularities are sources, dipoles, or vortices, as outlined by Erickson [1]. Once the A matrix is constructed, a simple linear solve is performed to find Γ,

$$A\Gamma = b, \tag{2.5}$$

where b is the external velocity term $-(\vec{V}_\infty + \vec{V}_{wake}) \cdot \hat{n}$.

With the panel method, the velocity induced by all surfaces is computed. What is left is to compute the velocity induced by the rotor's wake. The wake was simulated using a Vortex Particle Method (VPM). This method solves the Navier-Stokes equations in a Lagrangian formulation, where fluid packets (in this case vortices) are followed and evolved in time, rather than evolving a fixed point of space in time.

At each time step, the rotors will "shed" vortex particles at specified places along the span of the rotor. This is done by creating a vortex particle with vorticity equal to the local vorticity given by the panel method at the trailing edge of the aircraft. These particles will then be convected downstream, their trajectory governed by the effect of the panels, the freestream, and other vortex particles. At each time step, $\vec{V}_{wake}$ is used to update the right-hand side of equation 2.5.

For the convergence study in section 2.2, we were primarily interested in how our discretization parameters affect the VPM, so we used a cheaper surface model for the rotor: a Vortex Lattice Method (VLM) rather than panels. A VLM is similar to a panel method in that it's a potential flow problem and boundary conditions like the no-flow-through condition are used to determine appropriate strengths for source elements [7]. The main difference is the geometry of the sources. While panel methods cover the entire solid body, the VLM simply places thin tubes (vortex filaments) along the camber line. The engineer specifies how may of these lines are placed along the spanwise direction of the wing. Once vortex filament strengths are solved for and the fluid velocity field is determined, forces can be determined at each filament location using lift and drag polars. The decreased number of computational artifacts in the VLM improves the cost, at the expense of accuracy for complex flow scenarios.

2.2 VPM Convergence Study

The VPM has several parameters that are defined by the user, such as where and how often particles are shed. Thus, to ensure that our results are not an artifact of our chosen spatial and temporal resolution parameters, we performed a convergence study. In this study, we increased resolution in one dimension (while holding all others constant) until our answer no longer changed. The result metric in this case is the coefficient of thrust C_T. The goal of this study is not to get a specific C_T, but rather to have C_T remain constant as resolution is increased. This would indicate our parameters

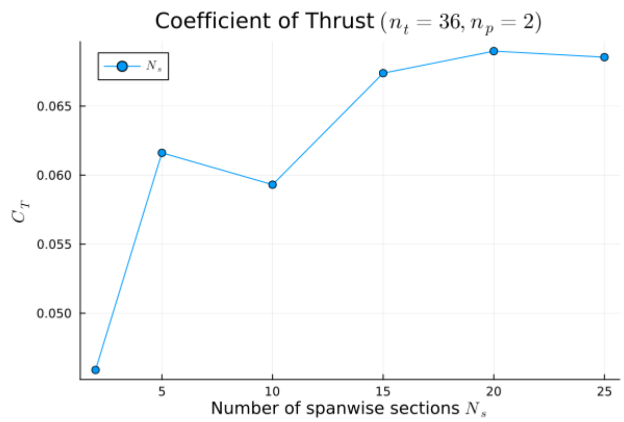

Figure 2.1 Coefficient of Thrust (C_T) vs. number of spanwise sections

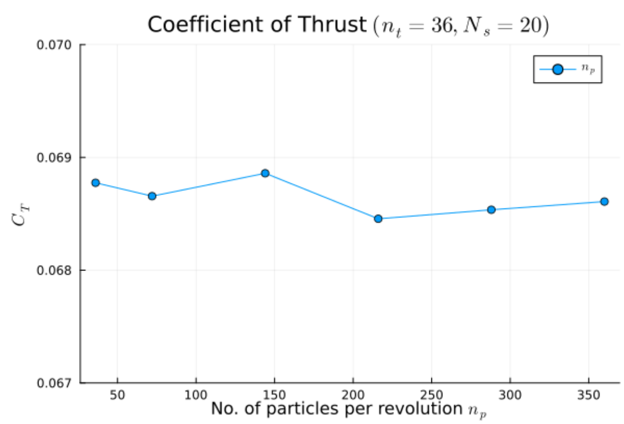

Figure 2.2 Coefficient of Thrust (C_T) vs. number of particles shed in a single revolution

no longer have an arbitrary effect on the result. The case used for this study is a single rotor in hover out of ground effect. A VLM was used to model the rotor.

We refined both spatial and temporal parameters. For the spatial parameters, two orthogonal dimensions were discretized: the radial and azimuthal. The radial discretization is defined by the number of spanwise sections N_s, and the azimuthal discretization is determined by the number of particles shed per revolution N_p from each spanwise section. Results of the spatial convergence studies can be found in figures 2.1 and 2.2.

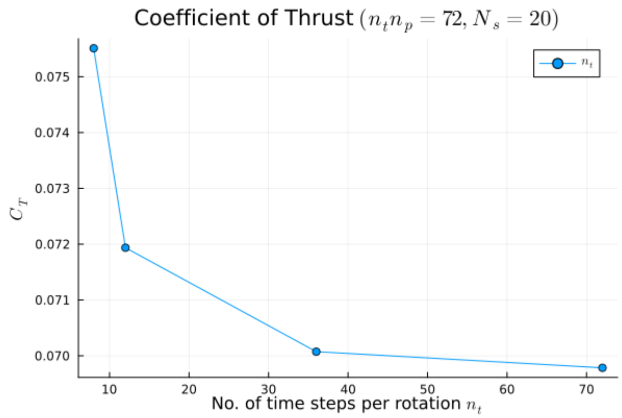

Figure 2.3 Coefficient of Thrust (C_T) vs. number of time steps per revolution (n_t)

Parameter	Value
VPM sheds per time step (n_p)	2
No. of time steps per rotation (n_t)	36
Particle smoothing radius (λ)	$0.093R$

Table 2.1 Numerical parameters for all simulations.

For the temporal discretization, both the number of time steps per revolution n_t and the number of particles shed per time step n_p had to be changed simultaneously in order to keep the same number of particles per revolution N_p. Specifically, the product $N_p = n_t n_p$ must remain constant.

The results of the temporal convergence study are shown in figure 2.3. We concluded that a time step of $n_t = 36$ balanced accuracy and speed the best. Similarly, we found that using a spanwise spacing $N_s = 20$ and azimuthal spacing $N_p = 72$ (or $n_p = 2$) gave the best results. These and additional parameters used in the final simulations are listed in table 2.1.

One additional parameter of importance is the particle smoothing radius σ. We found that the smoothing radius must be in the range $0.01R < \sigma < 0.25R$, or else the results blow up and are not physical. This behavior can be seen in figure 2.4, which shows C_T as a function of σ. For the simulations in this paper, we chose the value recommended by Alvarez of $\sigma = 0.093R$ [8].

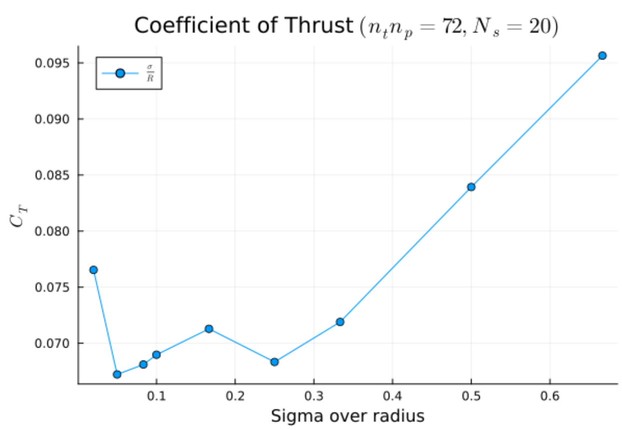

Figure 2.4 C_T as a function of σ.

Chapter 3

Results

Using the methods described in Chapter 2, we simulated a rotor in hover, similar to the validation by Alvarez and Ning [9]. Important physical parameters for this case are shown in table 3.1.

This case compares the thrust coefficient C_T to an experimental case by Zawodny and Boyd [2]. C_T is defined in this case using the propeller convention as

$$C_T = \frac{T}{\rho n^2 d^4}$$

where T is the thrust and n is angular speed in rotations per second. Zawodny and Boyd define C_T differently, but we will use the definition given by Alvarez and Ning [9], giving $C_T = 0.072$. Figure

Parameter	Value
Rotor radius, R	9.4 in
Rotor rotation speed, Ω	5400 rpm
Ambient air density, ρ	1.225 kg/m^3
Advance ratio, J	0.0001

Table 3.1 Physical parameters for Zawodny case. Advance ratio J is listed as 0.0001 instead of 0 (which would correspond to hover) for numerical stability.

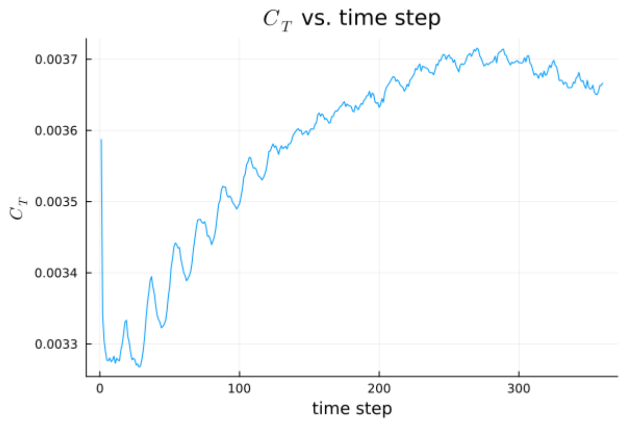

Figure 3.1 Coefficient of thrust C_T vs. time step for a single rotor in hover.

3.1 shows C_T as a function of number of time steps. These values were averaged over the last two rotations (72 time steps) to give a final C_T value. The C_T calculated by the VPM has significant relative error of 94.9%. This is a preliminary result, and future research will determine the cause of the error.

One potential source of error is the mesh that was used. The mesh was downloaded as an STL file from the website GrabCAD, modified using SolidWorks to have a sharp trailing edge, and meshed in Gmsh. The final mesh is pictured in figure 3.2.

Two issues with this mesh are apparent. One is that the force is inordinately high near the tips. Another is that the strength in the near wake doesn't vary smoothly, and sometimes has sharp jumps in magnitude. The reasons for this are unclear, but the cell size refinement is a potential cause. When creating the mesh, the cell size was refined in proximity to the leading and trailing edges. It's possible that the cell sizes are too small near these edges, or that the difference in cell sizes in the center vs. edges is too great. More research will need to be done to determine the exact source of the error.

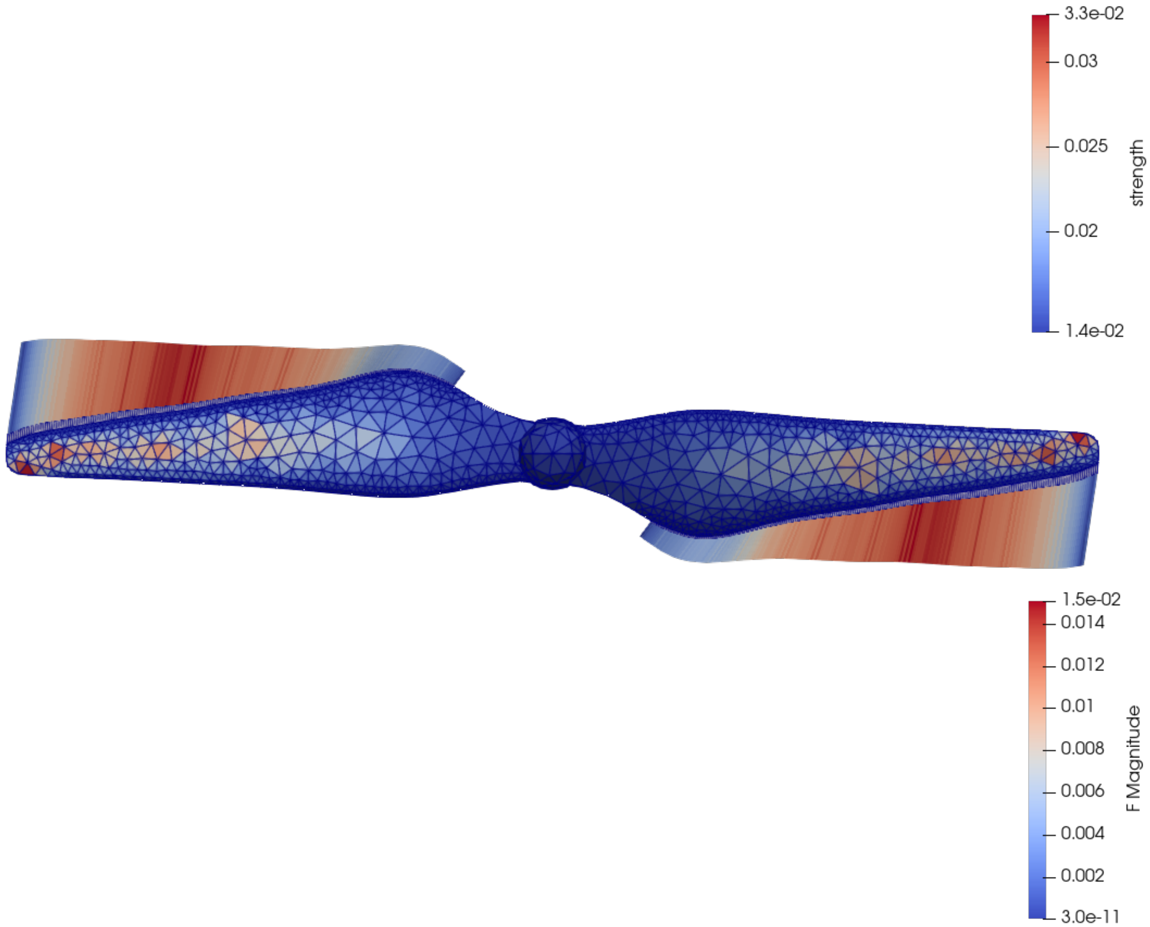

Figure 3.2 Mesh used to model the rotor. Cell sizes are smaller toward the leading and trailing edges. Colors indicate total force magnitude F (rotor), and strength Γ (wake).

Parameter	Value
Rotor radius, R	119.5 mm
Rotor rotation speed, Ω	6000 rpm
Ambient air density, ρ	1.16 kg/m^3
Advance ratio, J	0.0001
Height above ground, z	$0.5R$

Table 3.2 Physical parameters for Otsuka case. Advance ratio J is listed as 0.0001 instead of 0 (which would correspond to hover) for numerical stability.

3.1 Future Work: Ground Effect

We plan to run a simulation that compares a rotor both in ground effect (IGE) and out of ground effect (OGE) to demonstrate the panel method and VPM's ability to predict complex flows. For this simulation, IGE will mean the rotor distance from the ground is $z = 0.5R$. The experimental data that will be used was collected by Otsuka, Kohno, and Nagatani at Tohoku University and the University of Tokyo, Japan. More details can be found in their original paper [10]. Important parameters for our future study are listed in table 3.2.

Chapter 4

Discussion

We have been unable to demonstrate accurate simulation using panel methods coupled with VPM in this paper. This paper should be thought of as an introduction to methods and a first crack at using these methods combined. Future work with improved accuracy is forthcoming from researchers at the FLOW Lab.

While these sorts of computational analyses are great for preliminary design and optimization, final aircraft performance must be evaluated experimentally. This is because of the simplifying assumptions that must be made, both about the flight environment and the governing fluids equations. In this case, that means there is no incoming turbulence, no interfering wake structures from other aircraft, purely incompressible flow, and a host of other idealizations. Also, there are always numerical instabilities associated with any scientific computation. To fully capture the complex interplay between the many variables associated with flight, a prototype must be made and tested in the field. However, these considerations aren't to discount the value of simulation, which allows the engineer to iterate through designs much more quickly.

There is ongoing work in this domain within the FLOW Lab and the aerospace community at large. More validation of the VPM without panel methods can be found in articles [6] and [9] by Alvarez et al. We plan to expand on the cases provided by Alvarez et al. by exploring ground effect. We will also be examining further the shortcomings of this paper, looking at mesh generation techniques in particular. We will also be implementing viscous corrections for the panel method in the near future. This is an exciting field of study, and will be instrumental in bringing about a future of effective and efficient commercial, military, and urban air travel.

Bibliography

[1] L. L. Erickson, “Panel Methods - An Introduction,” Technical Report No. 2995, National Aeronautics and Space Administration (1990) .

[2] N. S. Zawodny, J. Boyd, D. Douglas, and C. L. Burley, *American Helicopter Society (AHS) Annual Forum* (2016).

[3] J. Alonso, P. LeGresley, and V. Pereyra, “Aircraft design optimization,” Mathematics and Computers in Simulation **79,** 1948–1958 (2009).

[4] J. L. Hess and A. O. Smith, “Calculation of potential flow about arbitrary bodies,” Progress in Aerospace Sciences **8,** 1–138 (1967).

[5] H. Lee, B. Sengupta, M. S. Araghizadeh, and R. S. Myong, “Review of vortex methods for rotor aerodynamics and wake dynamics,” Advances in Aerodynamics **4,** 20 (2022).

[6] E. J. Alvarez, J. Mehr, and A. Ning, “FLOWUnsteady: An Interactional Aerodynamics Solver for Multirotor Aircraft and Wind Energy,” In *AIAA Aviation Forum*, (Chicago, IL, 2022).

[7] H. Joshi and P. Thomas, “Review of vortex lattice method for supersonic aircraft design,” The Aeronautical Journal **127,** 1–35 (2023).

[8] E. J. Alvarez, *Reformulated vortex particle method and meshless large eddy simulation of multirotor aircraft* (Brigham Young University, 2022).

[9] E. J. Alvarez and A. Ning, "Stable Vortex Particle Method Formulation for Meshless Large-Eddy Simulation," AIAA Journal **62,** 637–656 (2023).

[10] H. Otsuka, M. Kohno, and K. Nagatani, "Fountain Flow Visualization in Quadrotor Wake Decreasing Rotor Thrust In-Ground Effect," Journal of Aircraft **61,** 761–773 (2024).

Index

convergence, 5

ground effect, 12

panel, 3

VPM, 4