Plenum Design for Circulation Control Wings

Konstantinos Kanistras
Pranith Chander Saka
Kimon P. Valavanis
Nikolaos I. Vitzilaios
Matthew J. Rutherford

Thursday 20th November, 2014

Contents

1

Introduction

The main goal of this report is to describe the methodology that is followed to achieve a plenum design capable of distributing the flow equally at the slot of a Circulation Control Wing. The report presents the SolidWorks designs and the ANSYS CFX results. At the end concludes with the design configuration that achieved flow uniformity.

Plenum designs for CCWs have been investigated in the past, and different techniques to achieve flow uniformity at the slot-exit along the span have been tested. To achieve flow uniformity at the slot across the span of the wing for Circulation Control (CC) at small scale fixed wing UAVs is an issue, which is still under investigation. One of the major considerations influencing the performance of CC is the non-uniformity across the slot of the wing. To tackle this problem several plenum designs able to distribute evenly the air at the slot exit of the wing are investigated and the results are presented. ANSYS CFX, which is a commercial CFD program used to simulate fluid flow in a variety of applications, is used next to simulate the flow inside the plenum and calculate the velocities at the slot-exit in a given inlet velocity.

2

Plenum Designs

Details of the CAD design geometry and the Computational Fluid Dynamics (CFD) analysis along with the results for all tested designs are presented in this chapter.

A NACA 0015 airfoil shape CCW with zero leading and trailing edge sweep and no winglets is chosen for the outer shape of the plenum design. The NACA 0015 airfoil, which is a symmetrical airfoil with a 15% thickness, is selected not only because it is well-studied airfoil, but also because it showed good CCW characteristics for upper slot blowing cases. The plenum designs that are tested will be part of a NACA 0015 CCW for future wind tunnel testing. The inlet and outlet geometry characteristics are kept constant so as the comparison between the different plenum design is valid. For the inlet, the inner diameter tube is kept to 10mm and the slot-exit is 1mm height. The span of the plenum is eqaul to 150mm.

2.0.1 Design - I

SolidWorks is used for the geometry design of all the plenums before the cfd analysis and Figure 2.1 shows the first design.

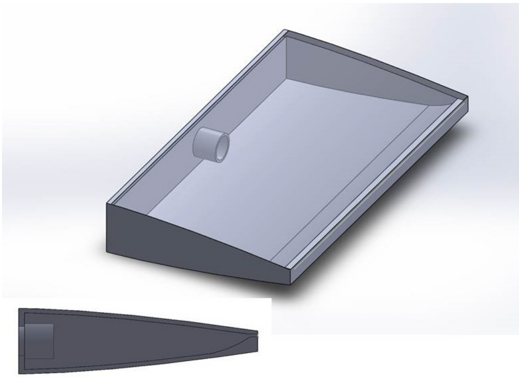

Figure 2.1: Trimetric and side vied of the CAD Design - I.

Figure 2.2 shows the results of the first plenum design that is tested. The plenum is simulated for four different inlet velocities. Those different velocities correspond to four different RPM values that an Air Supply Unit (ASU) performs. The ASU, which is a centrifugal compressor, is built to supply air to the plenum and for validating the configuration is decided to use inlet velocities based on the performance of the ASU. The inlet velocities that correspond to 7000, 12500, 17500 and 21500 RPMs are 17.2m/s, 31.1m/s, 41.7m/s and 53.6m/s respectively.

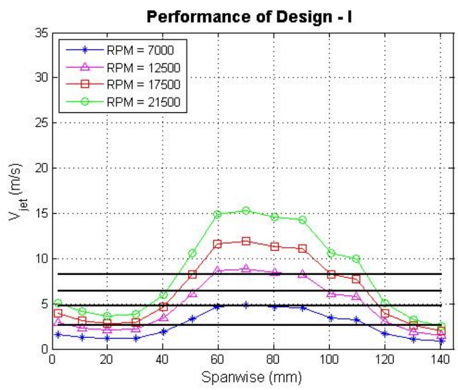

Figure 2.2: Results of Design - I.

The lines in black in the plot, show the average velocities for each inlet velocity. As it is expected, since only a tube is used in this plenum design, the flow is concentrated is higher at the middle of the slot since the tube is located at the middle of the plenum.

2.0.2 Design - II

The second design spits the inlet tube in to two parts and brings the flow closer to the slot. The CAD design is shown in Figure 2.3 and Figure 2.4 show the plenum's performance.

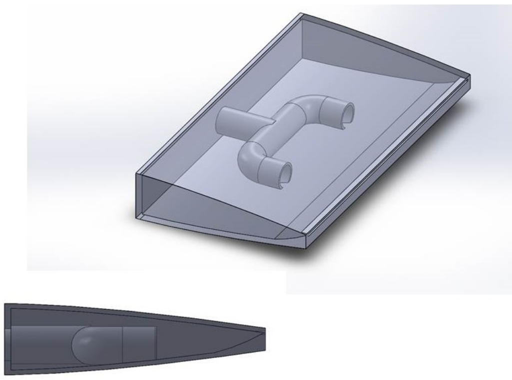

Figure 2.3: Trimetric and side view of the CAD Design - II.

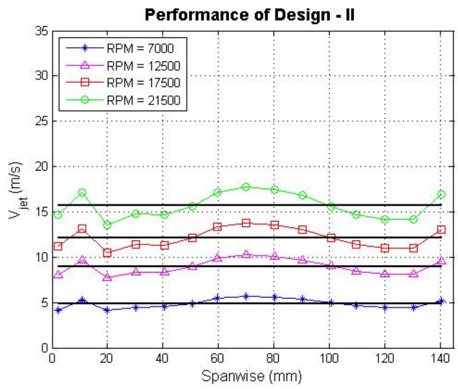

Figure 2.4: Results of Design - II.

A better distribution is achieved but still the jet velocities are low and flow uniformity is not accomplished.

2.0.3 Design - III

The diffuser design is desided to be tested next (Figure 2.5) but it is shown (Figure 2.6) that the velocities are decelerated before they reach the slot and flow uniformity is not achieved.

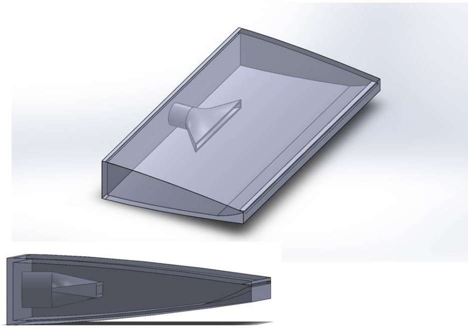

Figure 2.5: Trimetric and side view of the CAD Design - III.

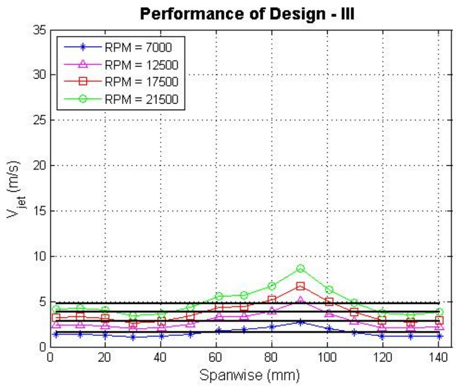

Figure 2.6: Results of Design - III.

2.0.4 Design - IV

To avoid velocity losses due to the diffuser it is decided to use walls and direct the flow to the slot as Figure 2.7) shows.

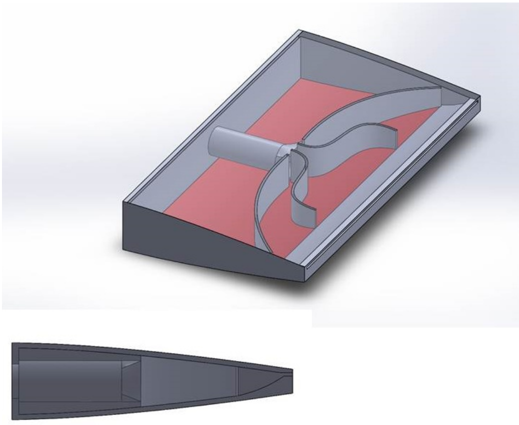

Figure 2.7: Trimetric and side view of the CAD Design - IV.

The plot in Figure 2.8shows that the velocities are higher by using this geometry but still uniformity with high velocities is not achieved. However across the span the velocities do not variate a lot away from the average velocity.

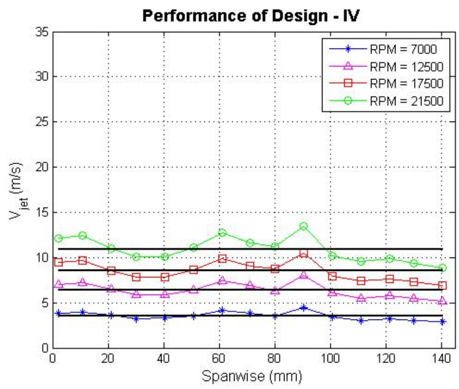

Figure 2.8: Results of Design - IV.

2.0.5 Design - V

The next design that is tested (Figure 2.9) showed higher velocities at the slot but due to the curved walls and the Coanda effect, the concentration of flow was close to the walls (Figure 2.10). It was decided to replace the curved wall separators with straight designs.

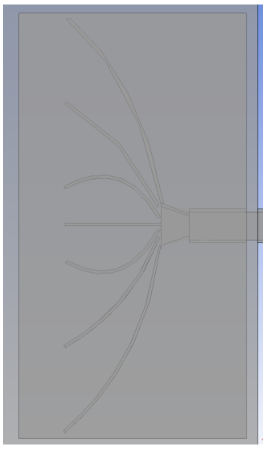

Figure 2.9: Trimetric and side view of the CAD Design - V.

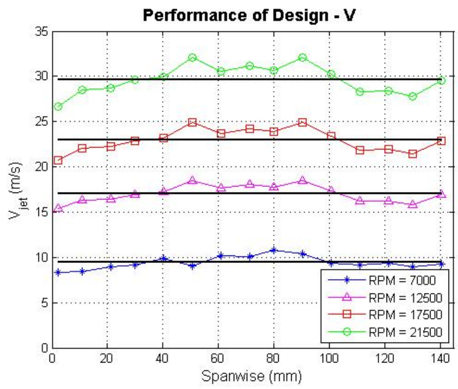

Figure 2.10: Results of Design - V.

2.0.6 Design - VI

The last two designs are shown similar behavior but flow uniformity is better at the last design since the left half is totally symmetric with the right part as Figure 2.11 shows.

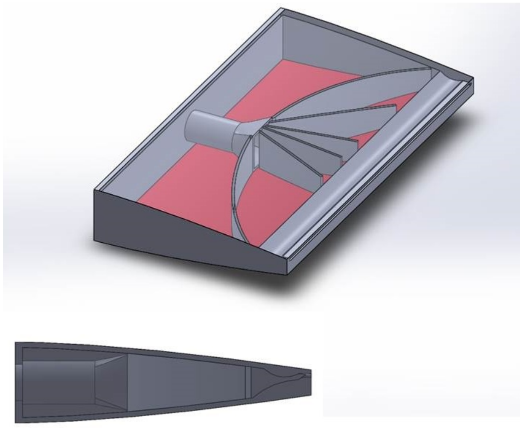

Figure 2.11: Trimetric and side view of the CAD Design - VI.

The results (Figure 2.12) show high flow uniformity and good velocity values at the slot.

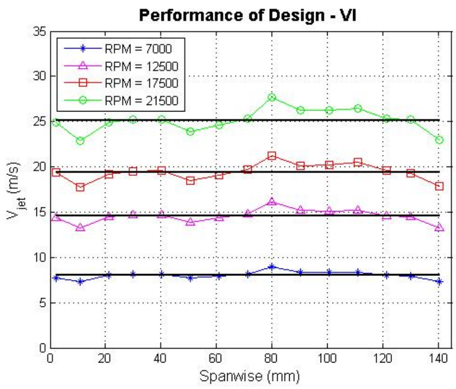

Figure 2.12: Results of Design - VI.

2.0.7 Design - VII

Here (Figure 2.13) the design is symmetrical and the walls (splitters) are all straight so no coanda effect is introduced.

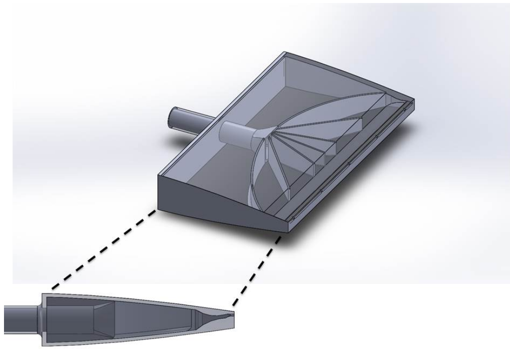

Figure 2.13: Trimetric and side view of the CAD Design - VII.

The results (Figure 2.14) indicate high flow uniformity along the slot for three out of the four inlet velocity values. As the velocity increases, it is observed that flow uniformity is influenced due to vortices that are created in the plenum. The outlet velocity values are also influenced due to those vortices.

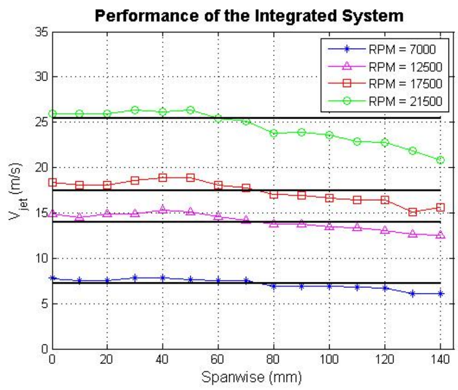

Figure 2.14: Results of Design - VII.

3

Conclusions and Future Work

Future work needs to be focused on the reduction of the vortices and the reduction of the velocity losses at the slot. A design that accelerates the velocities at the jet is the ideal outcome. However a design that gives high flow uniformity is designed and a CFX analysis is carried out successfully.