Journal of Aeronautics & Aerospace Engineering

Rajesh et al., J Aeronaut Aerospace Eng 2017, 6:3
DOI: 10.4172/2168-9792.1000198

Research Article Open Access

Numerical Analysis on the Effect of Fluidic on Demand Winglet on the Aerodynamic Performance of the Wing

Rajesh A*, Badri DR and Ganesha Prasad MS

Department of Mechanical Engineering, New Horizon College of Engineering, Bengaluru, India

Abstract

A numerical study was undertaken to study the effect of the span wise injection on the performance of a 3D wing at a velocity of 15 m/s and angle of attack of 6°, 8°, and 10°. A baseline configuration along with injection at tip was studied. A study was conducted to understand the flow field and the winglet control techniques. Based on the study, a wing configuration was chosen as baseline configurations and different injection velocities were applied to this configuration. The chord wise pressure distribution is seen to change with the span wise location from the root and this distribution is affected by the wing tip vortex. The wingtip was observed to change the pressure distribution near the tip. The velocity field, stream lines and the vortices were seen to be affected by the presence of the injection. The lift and drag values were seen to decrease with the angle of attack but the l/d ratio remained nearly constant for all the injection configurations. Maximum reduction in drag of nearly 19% could be achieved with the injection. This study proved the possibility of using span wise injection as a control method to control the wing tip vortex.

Keywords: Winglet control; Injection; Vorticity; Wingtip

Introduction

The primary concern at present regarding operating aircraft is efficient usage of fuel and environmental impact. A lot of research interest is attracted over the reduction of drag of the aircraft. By reducing the drag, the fuel burnt per kg of cargo or human per nautical mile is reduced. This reduces carbon foot print of the aircraft and hence the efficient fuel usage and environmental impact can be justified.

Drag reduction for aircraft has a range of positive ramifications: reduced fuel consumption, larger operational range, greater endurance and higher achievable speeds. Winglets (Figure 1) being a small structure play an important role in reducing the induced drag in aircraft. The wingtip vortices are the source of the induced drag. Wingtip devices increase the lift generated at the wingtip and reduce the lift-induced drag caused by wingtip vortices, improving lift-to drag ratio, this increases fuel efficiency in powered aircraft. The main objective of this project is to study about the winglet design and its contribution in reducing induced drag over aircraft wing. Moreover, the role of winglet in reducing the drag of typical aircraft wing is studied and the percentage of drag reduction is calculated by a conceptual approach. Drag reduction by using winglets helps in increasing of payload and increase the range of the aircraft with same fuel consumption. In aircrafts, induce drag is created by the wing tip vortex that is generated due to the difference in pressure between top and bottom surface of the wing near the tip [1-4].

The major challenge for the designer of an aircraft is to have a highest L/D ratio possible. This can be done by reducing the drag. Since the induced drag is a major portion of the drag, by reducing the induced drag, the L/D ratio can be increased. The induced drag is usually reduced in wings by using winglets. Spheroid, flat-plate and blended winglets are some of the winglets used for reducing induced drag. The size and position of the winglet plays important role in reducing the drag. Different winglets such as blended winglets are generally a structural modification and hence form a permanent fixture which is present during the entire flight envelope. Even though this kind of winglets is found to be efficient in reducing the induced drag, they offer additional drag due to the skin friction drag added by them.

Figure 1: Typical winglet.

The aircraft cruise conditions include a angle of attack at a lower range in which the fraction induced drag to the total drag may not be significant. The induced drag, being a function of angle of attack, forms a major portion of drag during landing and takeoff. Hence it is more optimistic to use a winglet which can be present only when there is a need arises. This is not possible with a existing permanent winglet and we propose a fluidic winglet which is an on demand approach. Here fluid will be injected in the span wise direction. The flow in the span wise direction will discourage the flow from bottom to the top surface of the wing that will result in reduction in induced drag [5,6].

Objective of the Research

The objective of this research is to

1. Design and analyze the flow over aircraft wing and wing with winglet to obtain the Drag force, Lift force and Drag due to lift values for both the configurations.

2. Major loads acting on the aircraft wing are determined.

3. Induced drag will be optimized with winglet by using computational fluid dynamics techniques.

Methodology

- A detailed study about the aircraft wing and wing with winglet design parameters have been carried out.

***Corresponding author:** Rajesh A, Department of Mechanical Engineering, New Horizon College of Engineering, Bengaluru-560103, India, Tel: 098805 34935; E-mail: rajesh.inspiring@gmail.com

Received June 30, 2017; **Accepted** August 28, 2017; **Published** August 31, 2017

Citation: Rajesh A, Badri DR, Prasad MSG (2017) Numerical Analysis on the Effect of Fluidic on Demand Winglet on the Aerodynamic Performance of the Wing. J Aeronaut Aerospace Eng 6: 198. doi: 10.4172/2168-9792.1000198

- The wing and winglet design parameter values were decided based on the study. A conceptual design and modeling of the wing and wing with winglet was carried out.
- CFD analysis of the flow over wing is carried out using the Analysis packages and the co-efficient of lift vs. coefficient of drag values were plotted.
- Major Forces acting on the wing are noticed and the induced drag over wing have been optimized by using winglet.

Flow field variables analyzed

The following flow field variables are analyzed in this research.

1. Pressure field.
2. Velocity field.
3. Lift.
4. Drag.
5. Lift to drag ratio.

Parameters

The following flow parameters are considered based on the literature survey.

- Free stream velocity – 15 m/s
- Wing Configurations – Baseline wing, Wing injection
- Slot Angle of attack – 6°, 8° and 10°.

Tools used

The following tools are used in this research.

- For grid generation (unstructured) – Gambit
- For pre-processing and solution - Fluent
- For post-processing - Paraview

Model design

The aerofoil section of the wing considered is shown in Figure 2.

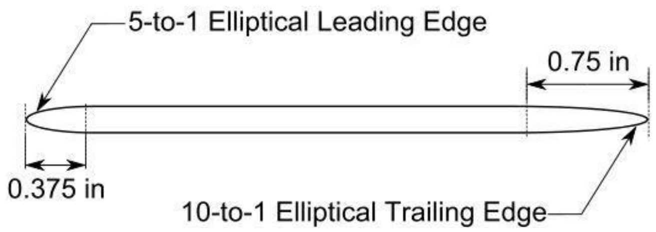

Figure 2: Aerofoil section of the wing.

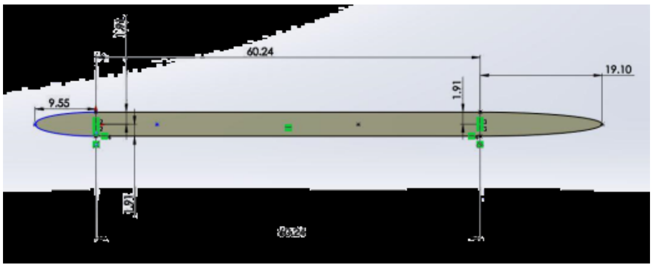

Figure 3: Dimensions of the aerofoil section of the wing.

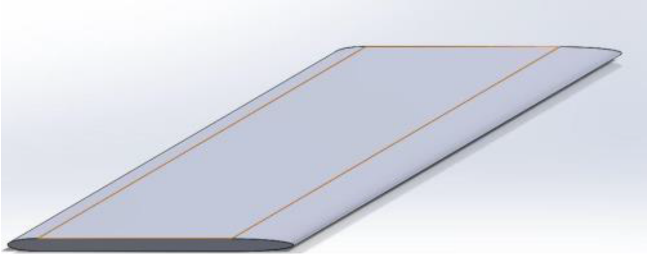

Figure 4: 3D wing model.

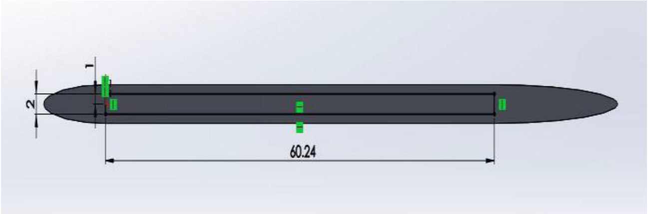

Figure 5: Dimensions of the injection slot.

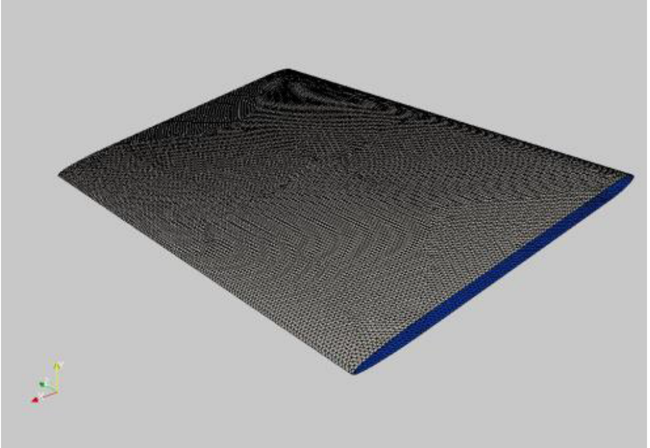

Figure 6: Mesh generated on the baseline wing.

The wing with aspect ratio 3 is considered for the present study since the induce drag increases with reduction in aspect ratio. All the parameters of the wing section is referred to the paper by Ananda et al. [7]. A 3D model of the wing with elliptical leading and trailing edge is created. The calculation of the wing parameters is given below.

- Chord = 88.9 mm
- Thickness = 4.3/100 × 88.9 = 3.82 mm
- 5 to 1 semi major axis at elliptical leading edge.
- = 3.82 / 2 × 5 = 9.55 mm = 0.375 inches.
- 10 to 1 semi major axis at elliptical trailing edge.

= 3.82 / 2 × 10 = 19.1 mm = 0.75 inches.

- Length = 88.9 mm - 9.55 mm - 19.1 mm = 60.25 mm

These dimensions are shown in the Figure 3. The screen shot of the 3D wing model is shown in Figure 4. The dimensions of the injection slot is shown in Figure 5.

Grid generation

The mesh distribution on the wing surface is shown in Figure 6.

The domain for the wing with injection slot is the same as the one for the plain wing. The major difference is the wing is having an injection slot at the side of the wing as shown in Figure 7.

A coarse mesh is used far away from the wing and a fine mesh is generated on the wing surface. This is done since the gradient in flow properties like velocity and pressure is high near the wing surface and far away from the wing surface, these gradients will usually be negligible and one can have a coarse mesh in these regions. This is shown in the Figures 8 and 9.

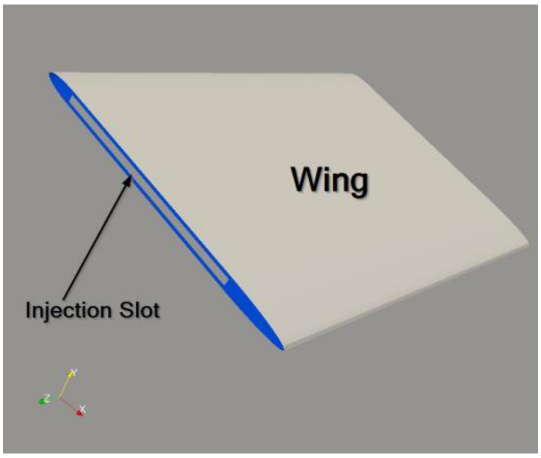

Figure 7: Wing with injection slot.

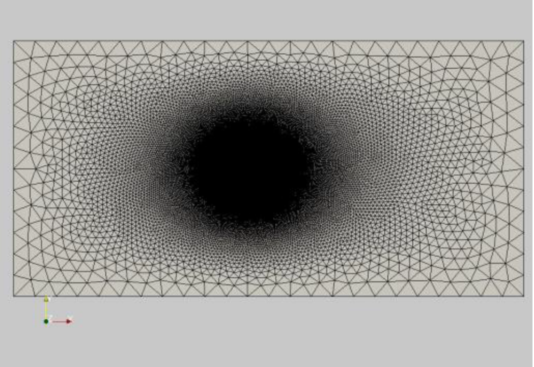

Figure 8: Coarse and fine mesh on the symmetry plane.

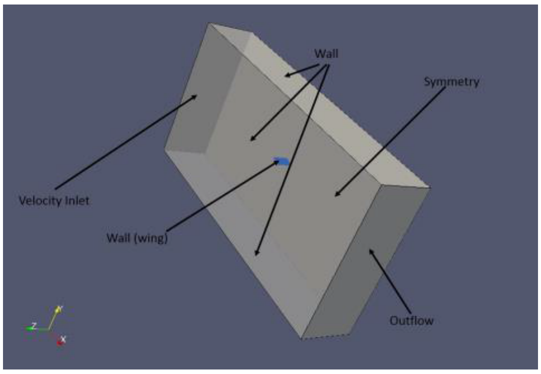

Figure 9: Boundary conditions used in the three dimensional studies.

Inlet pressure (Pa)	Inlet Velocity(m/s)
101325	14.54

Table 1: Inputs.

α	Configuration	Lift	Drag	C_L	C_D	ΔC_D (%)	L/D
6	Baseline	0.368	0.046	0.240	0.030	0	8.024
6	10 m/s	0.344	0.044	0.224	0.028	-5.01	7.885
6	15 m/s	0.335	0.042	0.218	0.027	-8.50	7.964
6	20 m/s	0.325	0.041	0.212	0.027	-11.33	7.978
8	Baseline	0.646	0.086	0.421	0.056	0	7.530
8	10 m/s	0.594	0.078	0.387	0.051	-8.74	7.581
8	15 m/s	0.575	0.076	0.375	0.049	-11.77	7.596
8	20 m/s	0.558	0.073	0.364	0.047	-15.5	7.701
10	Baseline	1.004	0.170	0.654	0.111	0	5.906
10	10 m/s	0.912	0.152	0.594	0.099	-10.76	6.009
10	15 m/s	0.867	0.146	0.565	0.095	-14.06	5.934
10	20 m/s	0.837	0.138	0.545	0.090	-18.82	6.066

Table 2: Values of lift and drag for different configurations.

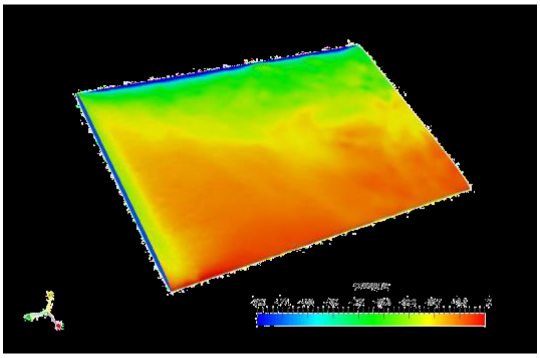

Figure 10: Pressure distribution on top surface of baseline wing at 10°.

Input and boundary conditions

The boundary condition used in the three dimensional studies are shown in Figure 9 and given here.

The wall boundary condition is applied to three surrounding walls. A symmetry boundary condition have been applied on the face at the root side of the wing since the model possesses similarity about the vertical plane at the center of the span. Outlet boundary condition was considered as an outflow Table 1.

Results and Discussion

The following Figures 10-19 show the pressure distribution, velocity distribution and streamlines near the tip of baseline wing and near the tip of wing with injection.

The values of lift and drag for different configurations are as shown in Table 2.

The variation of lift, drag, reduction in drag and l/d ratio is shown in the below Figures 20- 23.

Conclusion

A numerical study was undertaken to study the effect of span wise fluid injection on the performance of a 3D wing at a velocity of 15 m/s

Citation: Rajesh A, Badri DR, Prasad MSG (2017) Numerical Analysis on the Effect of Fluidic on Demand Winglet on the Aerodynamic Performance of the Wing. J Aeronaut Aerospace Eng 6: 198. doi: 10.4172/2168-9792.1000198

and angle of attack of 6, 8 and 10°. A baseline configuration along with injection at three different velocities was studied where a slot is used at the tip of the wing to inject the fluid. A literature survey was conducted to understand the flow field and the winglet control techniques reported in the literature. Based on the literature, a wing configuration was chosen as a baseline case and different injection velocities were applied to the baseline case. The chord wise pressure distribution was seen to change with the span wise location from the root and this distribution is affected by the wing tip vortex. The wingtip was observed to change the pressure distribution near the tip. The velocity field, stream lines and the vortices were seen to be affected by the presence of the injection. The lift and drag values were seen to decrease with the angle of attack but the l/d ratio remained nearly constant for all the injection configurations. Maximum reduction in drag of nearly 19% could be achieved with the injection. This study proves the possibility of using span wise injection as a control method to control the wing tip vortex.

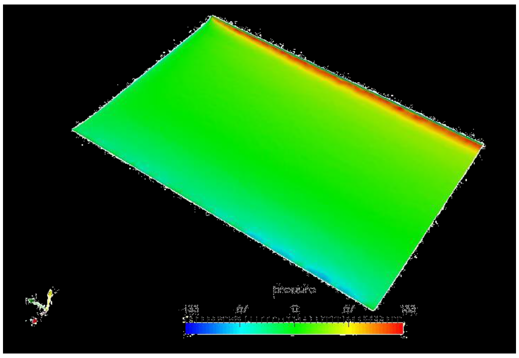

Figure 11: Pressure distribution on bottom surface of baseline wing at 10°.

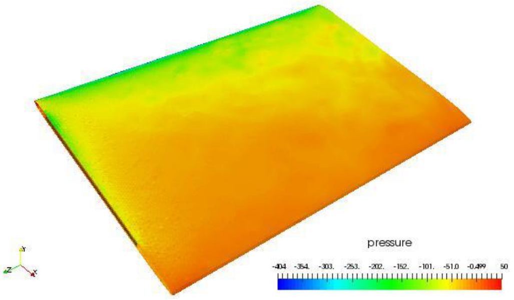

Figure 12: Pressure distribution on top surface of wing with 20 m/s injection velocity at 10°.

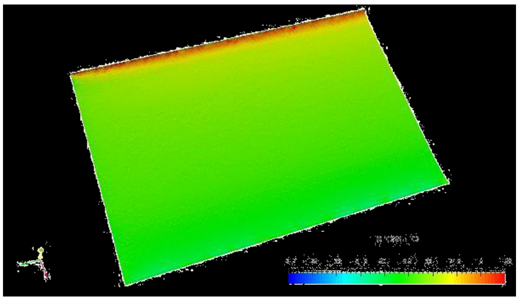
Figure 13: Pressure distribution on bottom surface of wing with 20 m/s injection velocity at 10°.

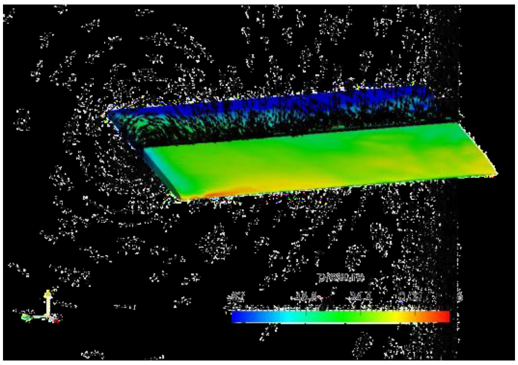
Figure 14: Velocity vectors showing the vortex for baseline case.

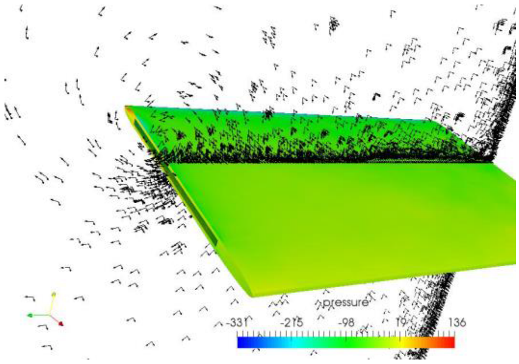

Figure 15: Velocity vectors showing the vortex for wing with injection at 20 m/s.

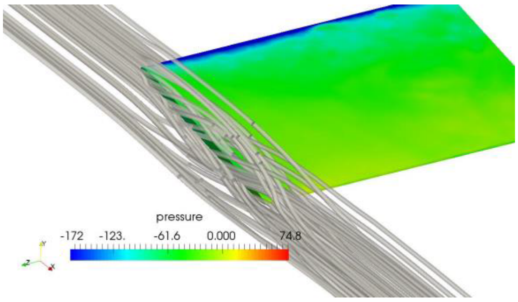

Figure 16: Streamlines near the tip of baseline wing.

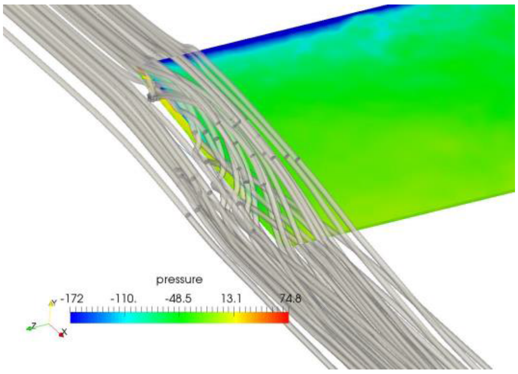

Figure 17: Streamlines near the tip of wing with injection at 20 m/s.

Future Scope

For the present study, only 15 m/s is considered as the free stream velocity. This study needs to be extended to other free stream velocities.

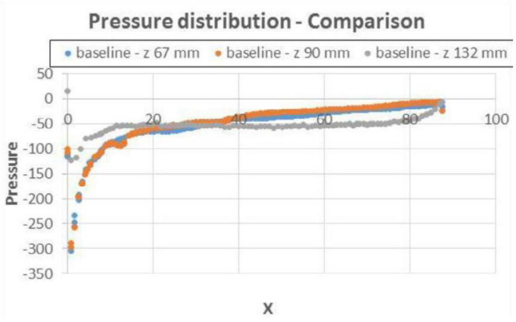

Figure 18: Comparison on Pressure distribution (xy plot) on top surface at various span location of baseline wing at 10°.

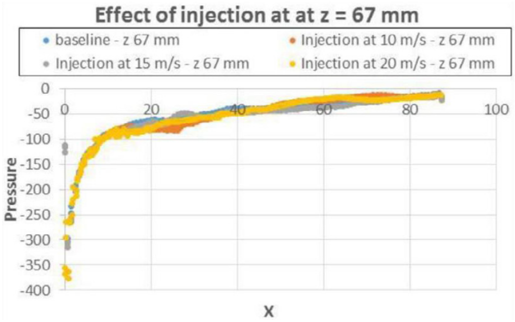

Figure 19: Effect of injection on pressure distribution at z=67mm at 10°.

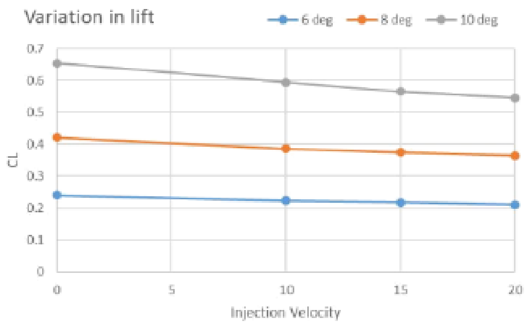

Figure 20: Variation of lift with injection velocity.

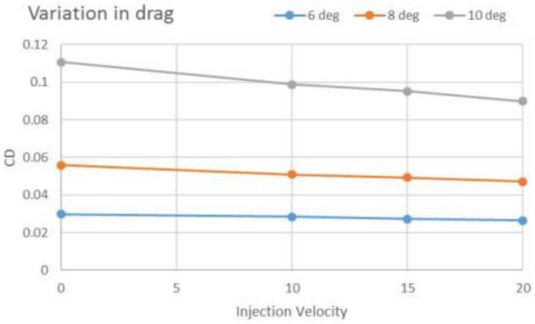

Figure 21: Variation of drag with injection velocity.

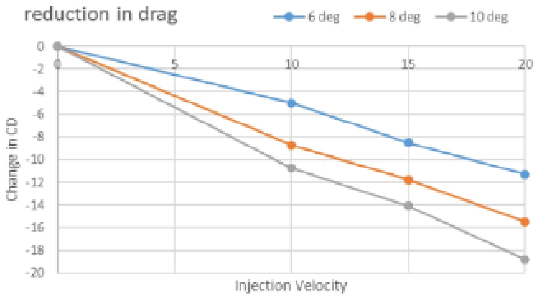

Figure 22: Reduction in drag (%) with injection velocity.

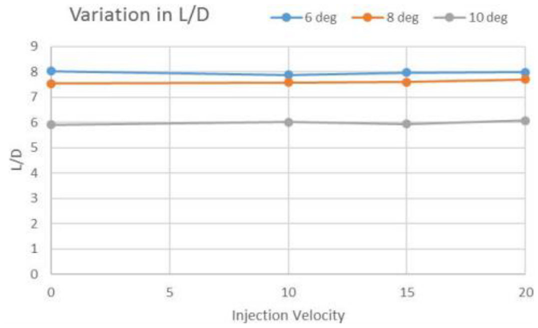

Figure 23: Variation of l/d ratio with injection velocity.

The effect of the wingtip configurations needs to be explored for the change in further, slot curvature and slot length.

References

1. Thipyopas C, Intaratep N (2011) Aerodynamics study of fixed-wing mav: Wind tunnel and flight test. In International Micro Air Vehicle conference and competitions 2011 (IMAV 2011),'t Harde, Delft University of Technology and Thales, The Netherlands.
2. Reddy SR, Dulikravich GS, Abdoli A, Sobieczky H (2015) Multi-winglets: Multi-objective optimization of aerodynamic shapes. In 53rd AIAA Aerospace Sciences Meeting.
3. Smith M, Komerath N, Ames R, Wong O, Pearson J (2001) Performance analysis of a wing with multiple winglets. AIAA.
4. Guerrero JE, Maestro D, Bottaro A (2012) Biomimetic spiroid winglets for lift and drag control. Comptes Rendus Mecanique 340: 67-80.
5. Afshar S, Shahi H (2007) Design and fabrication of a delta wing micro aerial vehicle. Int J mechanics 4: 51-58.
6. Yu Jl, Wang Ll, Ge G (2006) Using wing tip devices to improve performance of saucer-shaped aircraft. Chinese J Aeronautics 19: 309-314.
7. Ananda G, Sukumar P, Selig M (2015) Measured aerodynamic characteristics of wings at low reynolds numbers. Aerospace Science and Technology 42: 392-406.