Effect Of Cavity on Dynamic Stall Over an Airfoil in Martian Atmospheric Conditions

K. Bharath [1], Pasam Saranya [2] ,D. Anitha[3]

[1] Undergraduate Student, Department of Aeronautical Engineering
[2] Undergraduate Student, Department of Aeronautical Engineering
[3] Assistant Professor, Department of Aeronautical Engineering Institute of Aeronautical Engineering, Dundigal, Hyderabad-500043

1bharathkoppunoor@gmail.com, 2saranyapasam1511@gmail.com, 3d.anitha@iare.ac.in

***Abstract*— This study investigates the effect of surface cavities on the dynamic stall behavior of a NACA 0018 airfoil in Martian atmospheric conditions with low pressure, low density, and high CO₂ concentration. Based on ANSYS Fluent simulations, computational fluid dynamics (CFD) was performed by increasing the angle of attack gradually from 0° to 28° and investigating both the baseline and cavity-modified geometries for their aerodynamic performance. Cavities were inserted at 10%, 45%, and 70% chord lengths along a 165 mm chord to be positioned in areas susceptible to flow separation. Findings indicated that cavity-altered airfoils exhibited better lift performance and postponed stall initiation—especially within 8° to 18°—on account of improved flow attachment and decreased separation. At increased angles, minimal drag increase was noted due to localized vortex activity. In total, the research shows that appropriately located cavities are good passive flow control devices that improve the aerodynamic efficiency and stability in the rarefied Martian environment with promising implications for future aerial vehicle design on Mars.**

***Index Terms*—**
Dynamic Stall, NACA 0018, Martian Atmosphere, Airfoil Cavity, CFD, ANSYS, Lift Enhancement, Angle of Attack.

I. INTRODUCTION

The design of Martian aerial explorers poses special challenges because of the thin Martian atmosphere, holding about 1% of the sea level pressure on Earth and mostly consisting of carbon dioxide. With low-density conditions, producing adequate lift and ensuring stability of flight become much more complicated, particularly under high angles of attack under which dynamic stall is imminent. Dynamic stall is accompanied by delayed flow separation and sudden modifications in aerodynamic loads, and it contributes to control loss and degradation in performance. Passive flow control devices like surface cavity integration provide an attractive solution through the modification of boundary layer behaviour without active energy injection requirements. They are particularly suitable for Martian missions where power efficiency and mechanical simplicity are important considerations. The impact of surface cavities strategically positioned on the dynamic stall characteristic of a NACA 0018 airfoil under Martian ambient conditions is examined in this investigation by performing CFD simulations in ANSYS Fluent. The cavities, according to geometries taken from prior experimental and numerical studies are semi-circular shaped with uniform depth, and at 10%, 45%, and 75% of the 165 mm chord length. These are selected so that they could affect various stages of boundary layer growth and flow separation over the airfoil surface.

Table 1 Properties of Earth and Mars

Gas	Mars	Earth
CO_2	95.30%	0.03%
N_2	2.70%	78.08%
Ar	1.60%	0.93%
H_2O	0.13%	~2%
CO_2	0.07%	0.12ppm
Ne	2.5ppm	18ppm
Kr	0.3ppm	40ppm
Xe	0.08ppm	0.09ppm
O_3	0.03ppm	40ppm
Other	-	<7ppm
Dust	~10ppm	-

The PhD. Research of Olsman, W. F. J. ".“Effect Of Cavity on Dynamic Stall Over an Airfoil in Martian Atmospheric Conditions”.

which showed that rectangular surface cavities have the ability to change pressure distribution and aeroacoustic behavior of airfoils significantly. The report indicated changes in frequency response and decreases in amplitude fluctuations of pressure, indicating the cavity's function in suppressing unsteady aerodynamic effects.

Figure 1 Vorticity contour plots for the airfoil without and with cavity *[1]*

Specifically, the cavity-affected NACA 0018 airfoil had increased lift-to-drag ratio in some ranges of angles, though higher drag was observed at high angles as a result of vortex formation [1]. CFD simulations of a Martian airfoil of triangular shape with three different turbulence models: SA, SST k-ω, and WA. The WA model showed the best accuracy and computational efficiency to predict lift, drag, and pressure distribution along angles of attack ranging from 0° to 14°. Flow separation first emerged at angles above 6°, becoming more intensified with rising angle of attack, particularly towards the trailing edge. The WA model successfully predicted vortex generation and nonlinear aerodynamic responses, positioning it for Martian low-Reynolds-number, compressible flow conditions of interest to future Mars rotorcraft and aircraft design [2]. Simulations were performed at angles of attack from 0° to 20°, and lift coefficient, drag coefficient, and onset behaviour of stall were evaluated as the most significant aerodynamic parameters. The aim is to evaluate the potential of these cavities in enhancing aerodynamic performance and stability in thin Martian air and ultimately contribute to designing future aerial systems for Mars exploration.

II. METHODOLOGY

This research utilizes a computational method to study the effect of the cavity on dynamic stall behavior of NACA 0018 airfoil in Martian atmospheric conditions. The research aims at the variations in aerodynamic performance parameters like lift, drag and dynamic stall angles, utilizing validated CFD methods.

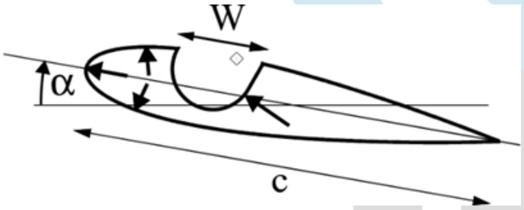

Figure 2 Placement of cavity on airfoil [1]

Mars atmospheric conditions i.e. low surface pressure of about 610 Pa, mean temperatures of approximately 210K, and mostly carbon dioxide composition is applied to simulate the flow environment. Simulations are performed on a typical airfoil without cavity and with cavity position at 10%, 45% and 70% chord length of the airfoil respectively. A computational domain that is C-configured extends to a sufficient extent to reduce the effect of boundaries, and a fine mesh with high resolution close to the wall is used to resolve boundary layer effects. Simulations are carried out at varying angles of attack.

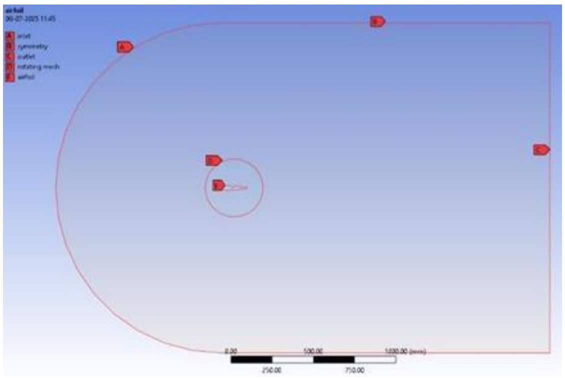

Figure 3 Computational C-domain around the airfoil

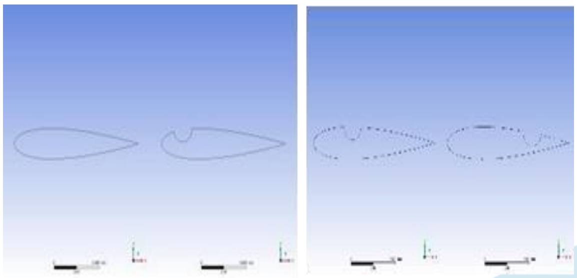

Figure 4 Airfoils without and with cavity placed at 10%,45% and 70% of chord-length of airfoil.

Boundary conditions are modified for Martian flight by specifying suitable inflow velocities and static pressures. The solver runs until residuals are below a chosen threshold and force coefficients become stable

1. Meshing

A regular mesh was employed, with a focus on particular refinement within the airfoil surface to capture the boundary layer appropriately. Inflation layers were created with a growth rate of 1.2–1.3 from the wall extending outward to capture the high velocity gradients within the boundary layer. The thickness of the first layer was set such that $y^+ < 1$, which is needed to capture near-wall physics, particularly when using turbulence models such as SST k-ω or WA.

Figure 5 Mesh over the C-Domain around the airfoil

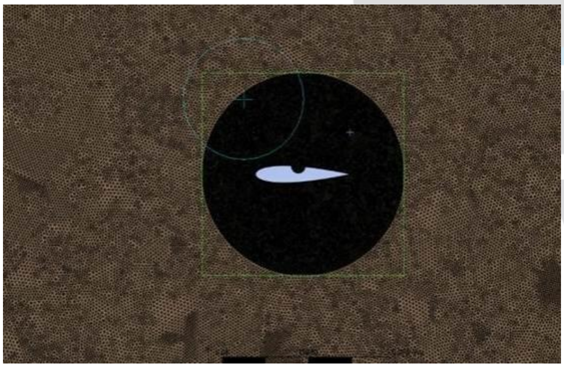

Figure 6 mesh around the airfoil

The grid was also finer closer to the leading and trailing edges, where high-pressure gradient and flow separation occur under dynamic stall conditions. The grid gradually builds up to a coarser mesh away from the airfoil to minimize computational expenses without compromising accuracy. Grid independence test was conducted with coarse, medium, and fine meshes to verify that the simulation results were gridresolution-independent. The medium resolution mesh gave a better balance between accuracy and computational cost and was thus utilized for the final simulations.

2. Boundary conditions

Following are the input parameters:

- Fluid: Air
- k-ω SST viscous model
- Temperature:210K

- Density: 0.015 kg/m3

- Velocity: 60 m/s2
- Viscosity: 1.3e-5

III. RESULTS

CFD simulations were conducted using ANSYS fluent to evaluate the aerodynamic performance of a NACA 0018 airfoil with and without cavities under Martian atmospheric conditions (pressure=610Pa, CO2 environment and low Reynolds number regime). Cavities were introduced at 10%, 45% and 70% of the chord length and simulations were carried out for at varying angles of attack from 0 o to 28o .

Case -I: In this case airfoil without cavity as performed in Martian atmospheric conditions at varying angle of attacks. Pressure contours of the NACA 0018 airfoil in dynamic stall conditions were determined through the use of ANSYS Fluent and represent the unsteady nature of aerodynamics while undergoing a pitching motion. At a shallow angle of attack (0°), the flow is still fully attached, with a symmetric pressure distribution and low lift generation. As the angle increases (10°–12°), there is a stronger low-pressure area over the top surface with more lift while maintaining largely attached flow.

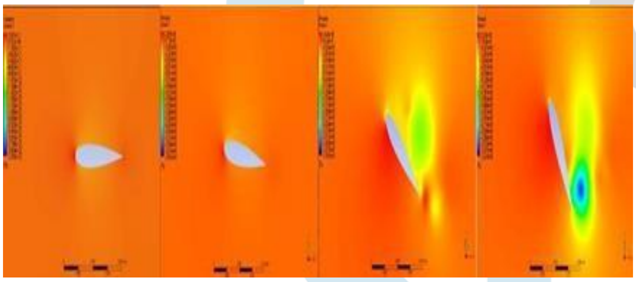

Figure 7 Pressure contours at varying angle of attacks without cavity on airfoil

Close to the critical angle (18°–20°), dynamic stall is initiated with a strong lowpressure vortex and some separation of the flow. At high angles of attack (more than 22°), full stall takes place. A detached vortex controls the upper surface, and the pressure distribution is distorted and asymmetric, leading to a serious loss of lift. These contours clearly illustrate the evolution from attached to fully separated flow, underlining the significance of preserving transient effects in dynamic stall analysis

Case –II: In this case the airfoil as introduced to cavity at 10% of its chord length and analyzed at varying angles of attack. Pressure contours depicted below are the NACA 0018 airfoil with 10% chord-length cavity solved under dynamic stall conditions by ANSYS Fluent.

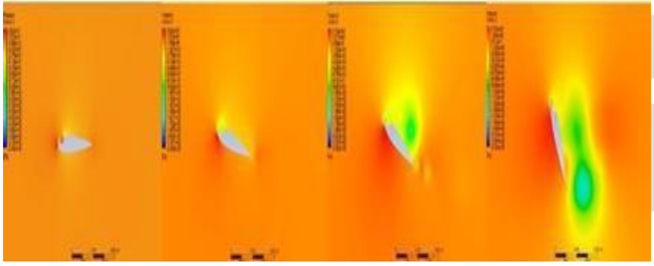

Figure 8 Pressure contours at varying angle of attacks with cavity at 10% chord length of airfoil

As one goes down to lower angles of attack, the pressure distribution is relatively smooth, and there is a high-pressure area under the leading edge and low pressure over it, which suggests attached flow. With increasing angle, the cavity causes local flow disturbances, which give rise to premature separation near the region of the cavity. At mid-cycle (about 15°–18°), a dynamic stall vortex is initiated above the top surface, indicated by the low-pressure green and blue areas. At higher angles (above 22°), the vortex is fully developed and separated, controlling the flow field and resulting in large asymmetry and flow separation. The cavity enhances the onset of stall by serving as a trigger point for vortex development. These findings show that introduction of a cavity modifies the pressure field and increases unsteady aerodynamic response.

Case- III: In this case airfoil as introduced to cavity at 45% of its chord length. The pressure contours of the NACA 0018 airfoil having a 45% chord-length cavity, simulated under dynamic stall conditions by ANSYS Fluent, exhibit substantial changes in flow behavior compared to a smooth airfoil.

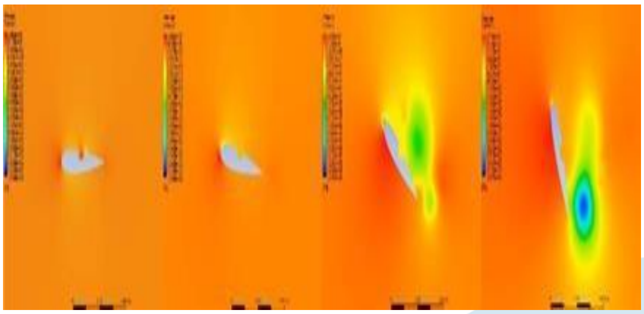

Figure 9 Pressure contours at varying angle of attacks with cavity at 45% chord length of airfoil

At low angles of attack, the flow is largely attached, with symmetric pressure regions and minor disturbances in the vicinity of the cavity region. As the angle rises, the cavity located mid-chord serves as a secondary separation point and ruptures the smooth pressure gradient on top. A significant lowpressure area starts developing above the cavity during mid-cycle, promoting early vortex formation and flow detachment partially. At steeper angles (20°–28°), the stall is more intense, with the cavity enhancing flow separation and expanding the detached vortex area, as seen by the larger green and blue regions. Such findings mean that a mid-chord cavity enhances unsteady effects and promotes dynamic stall ahead of time, resulting in premature lift loss and more turbulent flow behavior.

Case- IV: The pressure contours of the NACA 0018 airfoil with a cavity placed at 70% chord length, as computed in ANSYS Fluent, illustrate the effect of aft-camber surface modification on dynamic stall behavior. At small angles of attack, the flow is relatively attached with the cavity having a minimal effect of disturbance because of its downstream position.

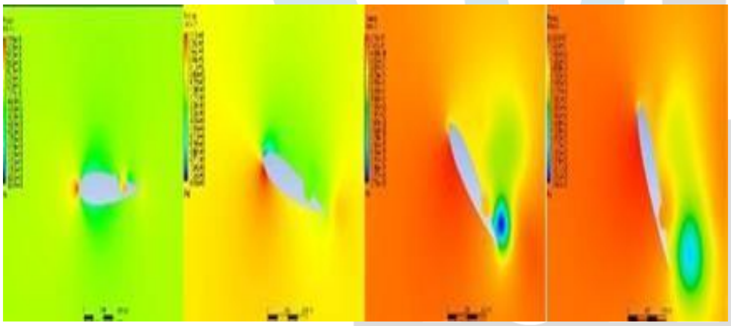

Figure 10 Pressure contours at varying angle of attacks with cavity at 70% chord length of airfoil

As the angle of attack rises, however, a gentle low-pressure area forms over the cavity, modifying trailing-edge flow behavior slightly. At mid to high angles (18°–28°), the dominant dynamic stall vortex develops, as evidenced by the increasing blue and green areas above the top surface. The downstream cavity postpones initial separation but accelerates the eventual intensification of vortex generation close to the trailing edge. With respect to mid-and leading-chord cavities, this configuration yields late stall onset and more focused pressure drop towards the back. The results indicate that a trailingedge cavity has an impact on vortex strength and location and alters unsteady aerodynamic loads.

IV. CONCLUSION

This research examined the impact of the introduction of a cavity at various chordwise locations—10%, 45%, and 70%—on the NACA 0018 airfoil during dynamic stall conditions through ANSYS Fluent. Pressure contour analysis showed dramatic changes in unsteady aerodynamic performance based on the location of the cavity. These changes affected the maximum lift coefficient (CL_max) directly, which is an important performance parameter in measuring airfoil performance. The baseline, no-cavity case displayed a CL_max of 0.177. Adding a cavity at 10% chord improved the aerodynamic performance considerably, with the peak CL_max being 0.292. This can be traced back to beneficial vortex interaction and retarded flow separation in

Table 2 CLmax values of airfoil at various conditions:

C_{Lmax} values of airfoil with cavity at			
No cavity	10%	45%	70%
0.177	0.292	0.179	0.078

The initial part of the airfoil that enhanced lift production during the pitching cycle. On the other hand, the 45% cavity gave a

CL_max of 0.179, almost the same as the nocavity case, showing a zero effect. Yet, the 70% chord length cavity dramatically decreased lift, with a CL_max of merely 0.078. This is caused by early vortex shedding and destabilization of trailingedge flow, which hastens stall and decreases lift. As a whole, the findings indicate that cavity location is an important factor in determining dynamic stall behavior. A forward-placed cavity (10% chord) can improve lift and postpone stall, whereas a cavity close to the trailing edge (70%) negatively impacts aerodynamic performance. These findings are significant for rotorcraft blades and MAVs, where control of unsteady flow events is vital to efficiency and stability.

V. REFERENCES

[1] E. A. R. Witold J. F. Koning, "Withdrawn: Study of Aerodynamic Characteristics and Turbulence Effects on Low Reynolds Number and HighLift Airfoils in Martian Atmosphere," On Improved Understanding of Airfoil Performance Evaluation Methods at Low Reynolds Numbers.

[2] H.-Y. X. C.-L. Qiao, "Dynamic Stall Control on the Wind Turbine Airfoil via a Co-Flow Jet," Energies , vol. 9, no. 6, 2016.

[3] W. Olsman, "Influence of a cavity on the dynamical on the dynamical behaviour of an airfoil," eindhoven university of technology, 210.

[4] R. K. A. Han Yang, "CFD simulations of a Triangular Airfoil for martrian atmosphere in low-Reynolds Number compressible flow," American institute of Aeronautical and astronautics , 2019.

[5] T. C.Corke, "Dynamic stall in Pitching airfoils: Aerodynamic damping and coompressibility effects," Annual review of fluid mechanics.

[6] R. Koning Witold J.K, "Low Reynolds Number airfoil evaluation for the Mars Helicopter Rotor," On Improved Understanding of Airfoil Performance Evaluation Methods at Low Reynolds Numbers.

[7] S. J. M.Prakash, "Numerical study of natural convention loss from open cavities," International Journal of Thermal Sciences, vol. 51, pp. 23-30, 2012.