DESIGN OF AIRCRAFT WINGLET TO REDUCED THE INDUCED DRAG FORMATION IN SWEPT WING

YOKESVARAN K., ABISHEK V.R and A. AGIL

Department of Aeronautical Engineering, Madha Engineering College, Kundrathur, Chennai – 69.

ABSTRACT

An attempt has been made to reduce the induced drag formation in thewing tip of swept wing by creating a new winglet structure. The winglet is a tool used to improve the efficiency and performance of an aircraft by preventing fluid flow jump from the lower surface to the upper surface at the wingtip. The addition of this winglet resulted in improved lift and reduction indrag force from the aircraft wing. This project describes Aerodynamic characteristics of the winglet for the reduction in induced drag without increasing or decreasing the span of the aircraft wing. A swept wing isdesigned with a NACA 4415 series airfoil. The wing, winglet, and wing tip tank are designed using CATIA V5 and analyzed using Ansys CFD 18.1. Overall result is compared between the wing without a winglet, a wing with a split winglet, and a wing with a combination of a split winglet and a wingtip tank. Compared data provide a reduced turbulence and fluid flow jump in the edges of the wing.

Keywords: winglet, Ansys and Aerodynamics.

INTRODUCTION

The aircraft design process is a loosely defined method used to balance many competing and demanding requirements to produce an aircraft that is strong, light weight, economical and can carry an adequate payload while being sufficiently reliable to safely fly for the design type of the aircraft. The aircraft design simply relies on the design of every structure of the aircraft. The design of aircraft winglet is a simple structure that plays a role in reducing the induced drag formation. This winglet design is an innovative design that is comprised of two existing winglet structures (Hart *et al.,* 1994).

Wing: The wing is a frame work made up of spars and ribs and covered with metal. Spars are the main structural members of the wing, and it extends from the fuselage to the tip ofthe wing. It carries flight loads and the weight of the wings while on the ground. Other structural and forming member such as ribs may be attached to the spars, with stressed skin construction also sharing the loads where it is used. They may be more than one sparin a wing or none at all. It can be considered as the skeleton or the frame of the aviation wing. The ribs give a wing its cambered shape, and they transmit loads from the skin and stringers to the spars.

Swept Wing: A swept wing is a wing that angles either backward or occasionally forward from its root rather than in a straight sideway direction. Swept wing is the most common platform for a high speed (transonic and supersonic) jet aircraft. Swept wings have been flown since the pioneer days of aviation. Adolf Busemann invented the swept wing (Fuller *et al.,* 1998). Swept wings are designed to reduce turbulence by slowing down the air as it moves the surface of the wing.

Winglet: Winglets are vertical extensions of wingtips that improve an aircrafts fuel efficient and cruising range. Designed as small airfoil winglet reduce the aerodynamic drag associated with vortices that develop at wingtips as the airplane moves through the air. Airplane winglet are the “baby wings” placed at an angle at the end of the airplane. Most modern passenger jets have winglets. They become a stable of the aircraft industry the 1980’s when concern about oil prices pushed increased research into their feasibility. Winglets increase an aircrafts operating efficiency by reducing what is called induced drag at the tip of the wing. An aircrafts wing is shaped to generate negative pressure on the upper surface and positive pressure on the lower surface as the aircraft moves forward.

Split Winglet: Split scimitar winglets are named after a Middle Eastern sward with a distinct curved blades ending with a sharp point. Split scimitar winglets were developed by aviation partners boeing and are available for the 737-800 and 737-900ER after entering service in 2014.Winglet are added to the end of planes wings to reduce drag and ultimately enable more efficient flight. They work by reducing a process known as vortex drag, caused by different air pressure converging at the tips of each wing and slowing the aircraft down. By producing a vortex or mini-tornado effect at the end of the wing, their drag reduced (Im *et al.,* 1998). This is what winglets are designed to do.

Wingtip Tank: Wing tip tanks are cylindrical structure located in the tip of the aircraft. Tip tank enhance overall stability improve spin characteristics and lower landing and stallingspeeds at the aircrafts original gross weight. It distributes the weight more evenly across the wing spar. On fighter aircraft, they may also be fitted with hard points for mounting drop tanks and weapons systems, such as missiles and electronic counter measures. It is intended to improve the efficiency of fixed wing aircraft by reducing drag.

CATIA: Catia is a multi-platform software suite for computer-aided design, computer-aided manufacturing, computer aided engineering 3D modeling and product lifecycle management developed by the French company dassault system.

Initial release – 1982
Stable release- P3, V5, 6R

CATIA V5 - Flagship of Dassault systems, it is the leading solution for product development for many years and remain a reference in the industry. V5 is most reliable which is used for 3D modeling of mechanical part design and product manufacturing.

ANSYS: ANSYS is a general-purpose, finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis, heat transfer and fluid problems, as well as acoustic and electromagnetic problems. There are two methods to use ANSYS. It is industry-leading fluid simulation software known for its advanced physics modeling capabilities andindustry leading accuracy. Ansys fluent generates accurate and reliable fluid simulation results. Ansys fluent unlocks new potentials for CFD analysis. A fluid simulation software with fast pre-processing and faster solve times to help you be the fastest to break into the market. The Ansys CFX solver uses finite elements (cell vertex numeric), similar to those used in mechanical analysis, to discretize the domain (Riggins *et al.,* 1995). In contrast, the Ansys fluent solver uses finite volumes. Fluent meshing is a powerful meshing tool for generating easier, faster, and more high quality meshes. Mosaic meshing technology allows disparate elements to be connected using polyhedral cells allowing for high quality optimal meshing in each region. This paper creates a effective design which provide reduced induced drag formation in the tips ofthe winglet.

METHODOLOGY

Study area is the field of study which can be concentrated to get knowledge about the project. The purpose is to make clear why you study was needed for the specific contribution you research made to furthering academic knowledge in you field. In this project the core of Aerodynamics is selected as the base for the field of project.

The initial idea for this new winglet design is proposed from the vintage business jet Learjet-23. As the aircraft variant is different from the current commercial airlinersbecause it have a unique winglet. This Learjet-23 consists of a wingtip tank in it, which act as a winglet as well as a storage tank for the fuel (Ben-Yakar and Hanson, 1998). Nowadays this structure winglet is not merely noted in commercial airlines as the wingtip tank reduces the aerodynamic stability while the fuel decreases. Likewise the cylindrical missile shaped structure is used as a winglet in the Euro-fighter Typhoon fighter aircraft (Fig 1-2).

Figure 1. Learjet-23 aircraft

Figure 2. Eurofighter Typhoon

The airfoil NACA 4415 is selected by conducting a survey and comparison. As per the survey this series is effective and provide high lift as compared to this relative airfoil because of its higher camber thickness (Fig 3-4). Hence the airfoil is generated as two size for the both ends of the sweptwing. These dimensions are calculated for the test section capability of the wind tunnel.

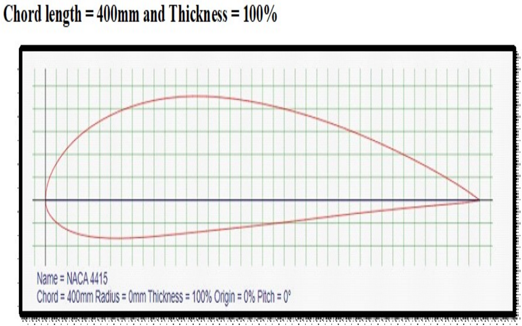


Figure 3. NACA 4415 Airfoil for first profile of swept wing

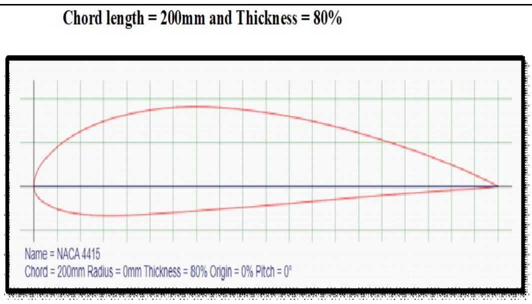


Figure 4. NACA 4415 Airfoil for Second profile of swept wing

3D DESIGN

The above plotted airfoils are imported as co-ordinates in the Excel work sheet. These coordinates are arranged arithmetically as per the geometrical need in the Catia 3D workbench. Numeric value are plotted as XYZ coordinates. Using the Generative shape design file the XYZ coordinates are directly plotted in Catia workbench by running the macros on the background.

The structure for the three design are developed in Catia. This developed design are further analysed to find the aerodynamic property of the structure (fig 5).

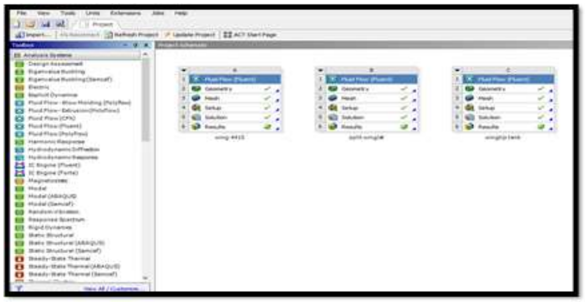


Figure 5. Ansys CFD fluent workbench imported with three designs

The Ansys workbench is opened with the CFD fluent analysis there three project schematic is opened and named individually as Wing 4415, Split winglet and Wingtip tank. These project are saved without importing any geometry to make a work interference ready in Ansys (Fig 6).

Meshing is the process of turning irregular shapes into more recognizable volumes called elements. By meshing the part domain is break up into pieces, each piece representing an element. One of the purposes of meshing is to actually make the problem solvable using Finite Element. The geometry of all the three design are imported and generated in the project schematic workbench. This geometry is further proceeded to create mesh over the structure of the design. The Ansys Meshing(AM) workbench is opened in the background, the created geometry for specific design is auto generated in the module. As basic the units should be first set for the mesh development of the generated design. The mesh should be generated not only for the Wing section but also created for the boundary layer, is the enclosure, which act as the boundary layer for the flow field over the surface of the design (Fig 7).

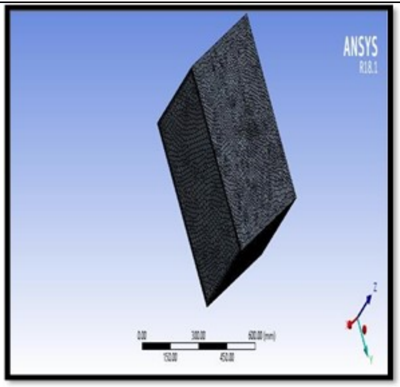


Figure 6. Mesh for wall

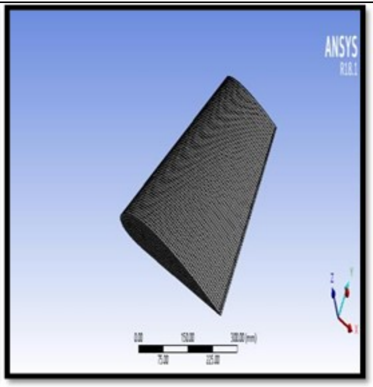


Figure 7. Mesh for Swept wing

After the mesh is developed, names selection for the faces of the wall body is updated. By selecting the front area where the airflow is begun is considered as the inlet and it is named as the velocity inlet of the design. Then the exact opposite side of the inlet is considered as the outlet where as it is names as the velocity outlet. Then the whole wall body is considered as the air medium and it is named as the air. This face selection is used to create a body setup for the design in further steps.

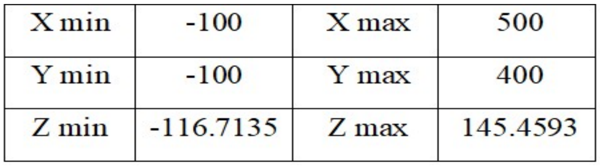
SETUP

The SETUP is the procedure followed for every Ansys fluent design as this the preprocessing of the design to load the result for the particular design that is created. In setup the required value that is followed to ensure the environmental study that is given to the design structure. In setup the setting up domain is updated as per the requirement of the user. The physics for the design is determined by using the setting up physics option the setup section. User defined readings and solving of the design is updated. In this section the exact values of velocity pressure inputs are given for the design for the calculations. The graph plotting between two co-ordinates setting are updated in this setup work bench for the Designs.

RESULTS

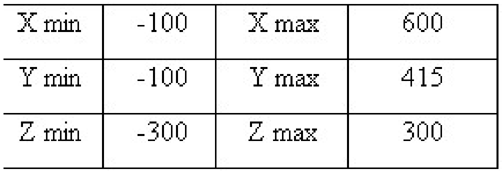
No function hooks selected and the regular compiled calculation symbols were used. As the flow is linear and AOA is constantly maintained as zero as cruising condition there is no fan blade or rotating angle of wind is given Table 1-4. These setting are commonly used for all the three design. This must not be changed to create a identical atmospheric condition for all the three designs. The same process is followed for other two designs before running the solution. After completing the setup for the all the three design using FL workbench, it is cross checked for the data mismatch as it will affect the accuracy of the result comparison.

Table 1. Domain extent for Swept wing



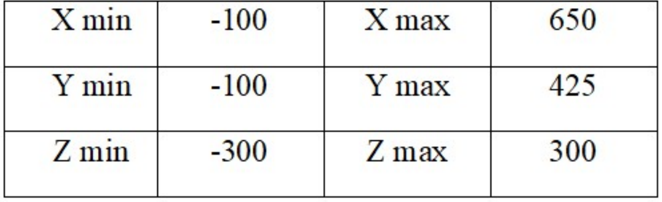
X min	-100	X max	500
Y min	-100	Y max	400
Z min	-116.7135	Z max	145.4593

Table 2. Domain extent for split winglet



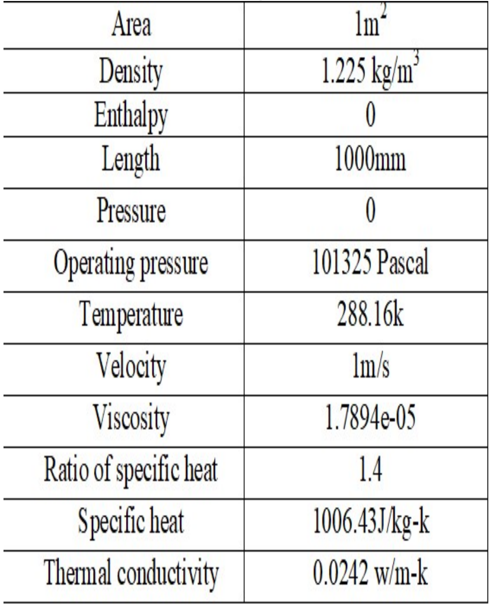
X min	-100	X max	600
Y min	-100	Y max	415
Z min	-300	Z max	300

Table 3. Domain extent for Wing with split and wingtip tank



X min	-100	X max	650
Y min	-100	Y max	425
Z min	-300	Z max	300

Table 4. Operating condition for all the three designs



Area	1m^2
Density	1.225 kg/m^3
Enthalpy	0
Length	1000mm
Pressure	0
Operating pressure	101325 Pascal
Temperature	288.16k
Velocity	1m/s
Viscosity	1.7894e-05
Ratio of specific heat	1.4
Specific heat	1006.43J/kg-k
Thermal conductivity	0.0242 w/m-k

The default setting is set for all the three designs, after completing this the mesh quality is again checked for errors this checking provides different reading for three design.

For the Swept wing:

Minimum orthogonal Quality = 1.79520e-01

Maximum Aspect Ratio = 1.92533e+01

For The wing with split winglet:

Minimum orthogonal Quality = 7.46949e-03

Maximum Aspect Ratio = 2.90452e+01

For the wing with both split and wingtip tank winglet:

Minimum orthogonal Quality = 4.81672e-03

Maximum Aspect Ratio = 1.02624e+02

After completing the setup the Ansys project schematic workbench is updated with the list of data for all the three diagram and saved. This setup is known as the pre processing of the solution where as the post processing is important for individual plotting of graph and isometric projection of the design in the console for comparison. The zones in settingup physics section is selected as which shows three option where boundary is selected and it is arranged by zone type which show primarily the inlet and outlet faces, followed by that it arrange interface, internal, outlet and wall region. Here the inlet is selected and the inlet velocity magnitude is given as 50m/s. And the constant temperature gradient is used as 300k. This magnitude and temperature is also common for all the three design. Hence the air flow with 50m/s is applied to run over the surface of the Wings and winglets in various aspects. This values obtained with reference of the journal paper which is commonly used for many CFD analysis, because it is considered as atmospheric constant and also the Wind tunnel show used in experiment have the capacity to run at 50m/s. This help to compare the flow visualization in both theoretical and practical method.

Like the setup solution also run in the FL interface as it is interlinked but the process is different. After common setup data input, the solution is carried out. Solution isnothing but it is a details and method of calculation for the result generation. In physics of FL workbench the Energy equation for viscous flow is selected. Where the k-epsilon two equation is selected as the formula for running the calculation. The k-epsilon turbulence model is the most common method used in CFD to stimulate mean flow characteristics for turbulent flow conditions. It is a two equation model that gives a general description of turbulence by means of two transport equations (PDE). Where k stand for the transport of turbulent kinetic energy and the rate of dissipation of the turbulent kinetic energy is ε. The behavior of turbulent flow is given in terms of these two properties in this model. The standard method of observation is used for k-epsilon model. Near wall treatment is setup to Enhanced wall treatment without mentioning any gradient value. The curvature correction and production limiter is not selected as the default setting not use these kindof formation setting over the surface layer of the mesh (Table 5).

Table 5. Modal constant for all three design using k-epsilon equation

CMU	0.09
C1-Epsilon	1.44
C2-Epsilon	1.92
TKE Prandtl Number	1
TDR Prandtl Number	1.3
Energy Prandtl Number	0.85
Wall Prandtl Number	0.85

In solution solver preference the method of calculation is set as simple scheme is used as default for all the design. For spatial discretization the function method is least square cell based is used. Second order function is used for the pressure and momentum of the flow. The turbulent kinetic energy(k) and the turbulent dissipation energy(ε) section will use the first order upward wind function for all these designs (Table 6).

Table 6. Solution limits for all the three designs

Minimum absolute pressure (pascal)	1
Maximum absolute pressure (pascal)	5e+10
Minimum static temperature (k)	1
Maximum static temperature (k)	5000
Minimum turbulent kinetic energy (m^2/s^2)	1e-14
Minimum dissipation rate (m^2/s^3)	1e-20
Maximum turbulent viscosity ratio	100000

The residuals are prepared to plot the XYZ velocity and pressure over the surface of these designs. At basic the residuals are plotted for the k-epsilon equation as these section is divided into seven category like continuity, x-velocity, y-velocity, z-velocity, energy, kand epsilon. And the iteration limits for this basic residuals are set to thousand numbers and the graph will plot the residuals at every iteration for the flow field identification.

The report definitions are given for these design as it is common for all the design to plot the required amount of data that is needed for the computation of the data. In definition there we need seven data to get plotted in the graph. They are drag coefficient, lift coefficient, maximum pressure and velocity and minimum pressure and velocity and turbulence formation in the designs. These all are plotted for each three design. The surface report is opened newly and the fact maximum is opened, this facet is first plotted for pressure with static pressure settings. File setting is set for report and plot the data during the calculation running. The graph is plotted between every three iterations. Every surface region which is considered for the calculation hence all the region of mesh is selected and no new surface is created and highlighting the surface is not used as every layer is selected as boundary.

Turbulence monitoring is one of important phase of this experiment as it will decide the efficiency of the design while comparing with other designs. The computation provide data that the turbulence is higher at the trailing surface of the wing which pointed 19.4639 Kg/s. Whereas the leading edge of the wing exhibits the lower turbulence as it is recorded as 1.8648 Kg/s.

Wing with split winglet

This structure is simple aerodynamic construction, which have a split winglet structure in the tip of the Swept wing. This design is modified and used in modern B747 Max aircraft. The simplified version of the Split wing is used in this design. The calculation is plotted are same as the calculation which is used for swept wing for the comparison.The pressure acting on this design is higher at the impact point which is known as the leading edge of the wing. The split winglet also exerts certain amount pressure similar to the leading edge of the wing. The middle surface of the wing exhibit low pressure and the trailing edge of the also exhibits certain pressure which is higher than the pressure in the middle surface and lower than the pressure acting in the leading edge of the wing. The maximum pressure obtained for this design of wing with split winglet by the computation is 1280.15 Pascal. And the minimum pressure acting on the wing is -1604.84 Pascal.

Wing With Split and Wingtip Tank

Similar method of calculation is used for this design of Wing with Split and wingtip tank in it. The computation provide various particle tracking over the period of iterations and the pressure velocity graph are plotted for every three iteration. This computation provides a positive sign of convergence in the end of the calculation.

The maximum velocity is flow over the upper surface of the design as it is drastically decrease the pressure flow in the upper surface and generates the lift. The maximum velocity computed is 64.1227 m/s and minimum velocity obtained for this design is 12.5929 m/s. The turbulence computation for this design found the hard airflow between the tank and the split winglet. But in turn it is reduced after it passes through the rear section of the wingtip tank. The maximum turbulence monitored is 19.2905 Kg/s and the minimum turbulence obtained is 1.38945 Kg/s.

The streamline flow for the design of wing with split and wingtip tank is given above. This graphics perfectly show the aerodynamic flow over the surface of the wing. As atthe initial stage the vortex is formed at the area where the split and wingtip tank is joined and this is further normalized in the rear section of the wingtip tank. The wingtip tank construction in this design widely help the airflow to pass smoothly over the surface ofthe winglet and reduces the vortex formation. This further help this structure to produce increased lift production and make this design efficient. The lift produced for this design of wing is computed as 3.5 N and the drag produced for this design is -15 N.

CONCLUSION

In this paper the aerodynamic behavior of the wing analysed with threedifferent cases of wing. This cases depends on the structure in which the first case only consist of Swept wing, followed by this the second case consist of wing with split wing attached to it. And the third case consists of a new design comprising of a winglet with split and wingtip tank in it. These are constructed and subjected to a velocity of 50m/s providing different Lift value according to the aerodynamic structure. The design with split wing provides 8.7% higher lift when compared to the swept wing without any winglets. And the design which consists of wing with split and wingtip tank in it provides12.98% lift when compared to the wing with split winglet. The further decrease the induced drag formation by 2.4% when compared to the wing without winglet. This shows pure optimization of the wing with the addition of winglet by increasing its dynamic performance.

REFERENCE

HartŽ eld, R. J., Hollo, S. D., and McDaniel, J. C., “Experimental Investigation of a Supersonic Swept Ramp Injector Using Laser-Induced Iodine Fluorescence,” Journal ofPropulsion and Power, Vol. 10, No. 1, 1994, pp. 129–135.

Riggins, D. W., McClinton, C. R., Rogers, R. C., and Bittner, R. D., “Investigation of Scramjet Injection Strategies for High Mach Number Flows,” Journal of Propulsion andPower, Vol. 11, No. 3, 1995, pp. 409–418.

Riggins, D. W., and Vitt, P. H., “Vortex Generation and Mixing in ThreeDimensional Supersonic Combustors,” Journal of Propulsion and Power, Vol. 11, No. 3, 1995, pp. 419–426.

Fuller, R. P., Wu, P., Nejad, A. S., and Schetz, J. A., “Comparison of Physical and Aerodynamic

Ramps as Fuel Injectors in Supersonic Flow," Journal of Propulsion and Power, Vol. 14, No. 2, 1998, pp. 135–145.

Im, H. G., Chen, J. H., and Law, C. K., "Ignition of Hydrogen-Air Mixing Layer in Turbulent Flows," Proceedings of the Twenty-Seventh International Symposium on Combustion, Combustion Inst., Pittsburgh, PA, 1998, pp. 1047–1056.

Ben-Yakar, A., and Hanson, R. K., "Experimental Investigation of Flame-Holding Capability of a Transverse Hydrogen Jet in Supersonic Cross-Flow," Proceedings of the Twenty-Seventh International Symposium on Combustion, Combustion Inst., Pittsburgh, PA, 1998, pp. 2173–2180.

McDaniel, J. C., and Graves, J., Jr., "Laser-Induced-Fluorescence Visualization of Transverse Gaseous Injection in a Nonreacting Supersonic Combustor,"Journal of Propulsion and Power, Vol. 4, No. 6, 1988, pp. 591– 597.