Structural Analysis of Airplane Wing UsingComposite and Natural Fiber Materials

Dhananjay Kumar[1], Pankaj Pandey[2], Dr. Lakhan Patidar[3]
[1]*Research Scholar, Oriental Institute of Science and Technology, Bhopal M.P (INDIA)*
[2,3]*Assistant Professor, Oriental Institute of Science and Technology, Bhopal M.P (INDIA)*

Abstract-The primary goal of this research is to analyse of fly wings of Airplane with different material such as EGR Glass, R Glass and S Glass with the help of Finite Element Analysis (FEA) and superior replacement materials for the existing material that is used in aeroplane wings in order to reduce the amount supply energy. This research is being conducted in order to fulfil this objective. This is the fundamental aim of the investigation that is being proposed. As a direct result of this, the present inquiry tackled the design process by using simulation tools such as Catia and Ansys workbench 20. This was done in order to get a better understanding of the system. The aviation industry is placing a significant amount of emphasis on weight reduction measures, which may be achieved by implementing innovative designs, more efficient manufacturing processes, or the use of alternative materials. It has been found out that the kind of materials that are most suitable for aeroplanes are ones that are able to withstand high pressure yet are relatively light in weight overall. Static structural analysis was done for the Aerofoil wing structure and performance was measured using different materials. The finding reveals the best material for maximum deformation under the pressure of 0.005MPa was S Glass which gives 0.049 mm deformation. S Glass may one of the best alternative materials among all others as it has better performance on deformation and this can help to minimize the deformation under high pressure as well. All three materials selected for this analysis performed identically for this particular condition, and it could be different for different working conditions. S Glass performs overall well as it gives a maximum von- Mises stress 0.049MPa and maximum deformation of only 0.049 mm.

Keywords: FEM, Aerofoil structure, Wing, Distortion, Optimum Condition, Composite and Natural Fiber.

I.INTRODUCTION

The most cutting-edge design for a wing incorporates a large wing box that now not bestgives the shape the pressure and power it wishes to be able to face up to the excessive aerodynamic hundreds, but additionally stores the majority of the fuel that is carried through the aircraft. This makes the maximum advanced layout for a wing the one that isused. This design is regarded as being at the cutting edge of contemporary layout practices. If this rigid form were to be replaced with a more malleable one, it may cause many people to have headaches, and it might be needed to arrange an alternate manner ofstoring the fuel in the event that this shape became more malleable. However, if morphing structures are utilized to replace most effective the moveable high raise gadgets, then it isachievable to get a better laminar glide without sacrificing the traditional wing box idea.Combining the two concepts will be used to accomplish this goal. In addition to the laminarization of the glide, clever morphing excessive elevating deviceswill bring an incredible number of other advantages. Because the requirements for the wing performance change all through the flight profile, the conventional layout of an Airplane is optimized for a single operating scenario, such as takeoff, cruise, or touchdown. This is because the necessities for the wing performance change all through the flight profile. This is made possible as a result of the fact that the requirements for the wing performance shift over the whole of the flight profile. Becauseof this, the overall performance of the aircraft throughout the off-layout flying phases is much below what would be considered to be the best possible. However, the shape of thewing may be altered to optimize the aircraft's aerodynamic overall performance during any phase of the assignment profile if morphing high lift devices are used. This will let the jet to fly more efficiently.

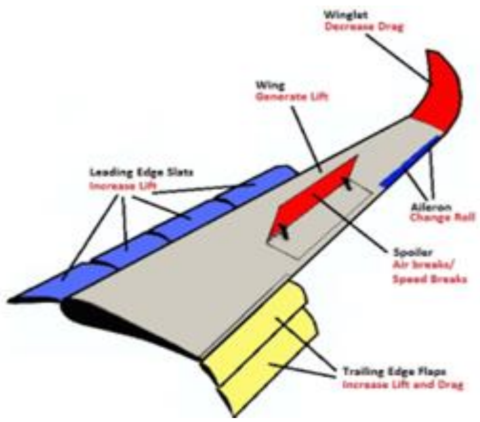


Figure 1 Typical commercial aircraft wing

II.LITERATURE REVIEW

Mohamad et al., (2022) The authors have proposed the doing of a research assignment with the working title " Buckling Analysis of a Thin-Walled Structure Using Finite Element Method and Design of Experiments." In this issue of the newsletter, an attempt was made to create a composite building with thin walls and the form of a C-move segment. The results of this endeavour were analysed. The structures have been modelled and analysed with the assistance of an industrial tool called ABAQUS. The approach that was used was a finite detail technique. The essential buckling loads for various kinds of laminate have been determined after extensive research and testing. Because composite substances and systems are made up of a variety of elements, the results of analysis might likewise vary based on the particular parameters under consideration. As a consequence of this, a strategy known as the layout of experiments is used, and the tools MINITAB 20 and design-expert thirteen are utilised, in order to maximise the effectiveness of the cutting-edge mode. After then, the results of the investigation were compared to the information that had been obtained earlier. In conclusion, the investigation's study revealed that the reaction surface procedures generated the best conceivable results. The line graph illustrates, almost conclusively, that the regression equation yielded the highest quality outcomes that were attainable. It explains how the components influence the critical buckling load and also displays the best aggregate for a bigger essential buckling load. Nasir et al., (2022) The Authors have proposed a research study entitled “Performance Evaluation of Unmanned Aerial Vehicle Wing Made From Sterculiasetigeradelile Fiber And Pterocarpus Erinaceous Wood Dust Epoxy Composite Using Finite Element Method Abaqus And Structural Testing” In this research a 3 dimensional model of Throughout the process of designing, assembling, and simulating the UAV wing, the Abaqus software was used. According to the results of the simulation, the capability of the UAV wing to sustain a wing loading of 167.75 to 895 Newtons (corresponding to a weight range of three to sixteen kilogrammes for the UAV) was determined. This value corresponds to a closing load aspect of 20.27. This indicates that it has the capability of withstanding a closing load thing of 20.27 for an unmanned aerial vehicle (UAV) that has a mass n of 4.5 kg. Because of its location, heavier components, such as weapons and cameras of a contemporary and high-quality standard, may be attached to an unmanned aerial vehicle (UAV). The results of the simulation had been effectively confirmed via the fabrication of the UAV wing in an efficient manner and thorough inspections to ensure that its structural integrity was maintained. During the course of the testing, the weight on the wing of the UAV ranged anywhere from 167.75N to 335.50N. (3kg to 6kg UAV mass). The pressure that changed into applied with the aid of the utilisation of sandbags was no more an excessive amount of for the wing of the unmanned aerial vehicle.

Arijeet Nath et al., (2021) The Authors have proposed a research study entitled “Modelling and Finite Element Analysis of an Aircraft Wing using Composite Laminates” This test's objective is to ascertain which of the aforementioned categories of chemicals has the most favourable ratio of power to mass. It became observed that, in preference to version 2, a weight reduction of 2.37% changed into observed, the deformation became decreased by means of 51%, and the von-Mises pressure changed into decreased via 85% for a given uniform load on the bottom of the aircraft's skin. All of those improvements took place for a given uniform load on the lowest of the aircraft's skin. These findings were derived based on a predetermined consistent weight that was placed on the underside of the plane's pores and skin. The orientations of the plies are a significant factor in determining the total power of the composite stack up. This is due to the fact that the orientations of the plies play a key role. In order to test different exclusive stacking orientations, iterative techniques are used, and the results are analysed and compared. According to the findings of the study, model 2.B is the best alternative for composite layups in a number of the laminate orientations that were evaluated. This is due to the fact that, in comparison to the other models, it had a greater energy-to-weight ratio while also exhibiting lower values for deformation and stress. Another reason for this is that it demonstrated lower stress and deformation values. More research should presumably be aimed on developing materials that have lower manufacturing costs and optimising the design of the wing to obtain the best possible balance of power and weight.

Diaa Emad et al (2021) The writers of the study advocated a research test with the operating identify " Low-Computational-Cost Technique for Modelling Macro Fiber Composite Piezoelectric Actuators Using Finite Element Method" This is the name of the research check out. An approach that takes use of the finite element method is presented as a powerful tool for modelling MFC in this newsletter. The strategy is shown as an example. Because of the application of the cautious approach, the MFC actuator has been replaced with an equivalent simple monolithic piezoceramic actuator that uses only two electrodes. This has resulted in a significant reduction in the amount of computational work that is needed. The proposed approach has been shown to be genuine from a theoretical standpoint due to the fact that it generates the same electric powered subject, strain, and displacement as the real MFC. In addition, the results of the studies effectively demonstrated the method that ultimately turned out to be advised while maintaining a high degree of consistency. The suggested method was put to the test via the simulation of a morphing wing that was nearly completely shielded by MFCs at a minimal computational cost. The outcome of the simulation validated the viability of the suggested method. The effectiveness of the method was able to be shown thanks to this. Can Kandemgr (2020) The Authors have proposed a research study entitled “Weight Optimization of An Aircraft Wing Composite Rib Using Finite Element Method” In the course of this investigation, step one became the procedure for the design of an aeroplane wing. The goal of this endeavor was to figure out how the load should be distributed over the composite wing ribs. The first stages in the process of designing an aircraft wing include making a choice on the kind of airfoil to be used and modelling the outer geometry of the wing. After this process, decisions are made on the several types of spars and ribs, as well as their positions on the wing, the initial thickness values, and the stacking sequences. In order to investigate the effects of the optimization method on the structure of the wing field, a further spherical of finite detail study of the optimised design is carried out.

Sarah David et al (2020) The authors have recommended doing a research study that will be named " Application of the Finite Element Method in the Analysis of Composite Materials: A Review" This article from the 21st century provides a list of the key commercial industries that make use of composite material simulation, as well as the advantages that these sectors get from the use of the technology. Aeronautics, aircraft, automobiles, navy, power, civil, sports, manufacturing, and even electronics are all included in these industries. The book also discusses the many failure criteria that were developed and utilised for the modelling of these substances, much as it discusses the various kinds of factors—solids, peel, plate, and cohesive—that are most often employed to mimic composites. Solids, peel, plates, and cohesive materials are all included in this category. In addition, this newsletter provides a list of the key businesses that make use of composite cloth simulation as well as a list of the benefits that these industries gain from the benefits that these companies derive from the usability of composite cloth simulation. These businesses may be in the fields of aeronautics, aerospace, automobiles, naval engineering, electrical engineering, civil engineering, manufacturing, or even electronics. As a direct consequence of this, the finite detail approach has been used as a tool for the purpose of analysing composite materials when they are being subjected to the most specified circumstance. J. N. Reddy (2019) The Authors have proposed a research study entitled “Introduction to the Finite Element Method” in the accompanying present item In addition to doing numerical simulations of physical approaches on a computer, we make use of a numerical method to investigate the mathematical model of the method while at the same time running numerical models of the methods. The term "numerical" refers to this kind of simulation. The finite element method is a powerful numerical methodology that was developed to analyse more complicated aspects of physical systems. Its name comes from the fact that it uses elements that are infinitely small. Equations of varying degrees of complexity, including algebraic, differential, and fundamental, might be solved with its help. The following three fundamental aspects contribute to the method's prominence: 1. The influence of the problem is modelled as a collection of key subdomains, which may also be referred to as finite elements. The term "finite element mesh" refers to the mesh that may be created with the assistance of the collecting of several finite factors. The essential idea that underpins the finite detail technique is discussed in more than thirty-five distinct textbooks. It is not necessary for a novice to examine any additional publications on the finite element approach since the existing book offers entire information of the technology as it's been adapted to linear area issues. The examples presented in this book are from a variety of fields, including engineering and applied sciences, such as fluid mechanics, heat transfer, solid and structural mechanics, and others. When approaching a problem with limited details, it is not always necessary to study additional literature when dealing with a beginner.

An academic study with the working title " Optimization of a Hybrid Carbon/Glass Composites Afterbody of the Amphibious Plane with Finite Element Analysis" has been cautioned with the assistance of the authors by Mongkol (2019) (2019). The major objective of this study is to identify the most suitable configuration for the afterbody of the amphibious aircraft in terms of its capacity to support weight in accordance with ASTM F2245 (general Specification for layout and overall performance of a light game plane). The finite element analyses of hybrid carbon/glass composites are carried out with the assistance of ANSYS ACP. This is done under the presumption that hybrid carbon/glass fibre composites have the potential to combine the beneficial properties of carbon and glass fibre strengthened polymer to strike a healthy balance between power, weight, and cost in order to fulfil the requirements of each and every design of the aircraft. However, before the first flight can be carried out, the physical prototype has to be tested in accordance with the ASTM F2245 recommended standard. This is necessary to ensure that the aircraft is airworthy and that its operation is safe. O.Nurihan et al.,(2019) Authors have proposed a research study entitled “The Impact Behaviour of Carbon Fiber-Epoxy Composite Leading Edge using Finite Element Method” This study is primarily focused on examining the impact behaviour of composite panels that approach the leading edges of wings when such panels are exposed to a rigid spherical projectile. The leading edge of a wing is the part of the wing that is closest to the centre of the aircraft. [Further citation is required] The most common types of panel designs are flat panels, semi-round panels, and semi-elliptical panels. The thickness of a panel may be adjusted by utilising either 2, 4, or 8 layers of material. The panels are made of laminated composites with woven carbon fibres, and the angle of orientations are [0/90] n, [0/45]n, and [45/-45]n, where n is the number of layers contained inside the composite and will give a range of orientations. The panels have a thickness of [0/90] n. In situations in which the failure criteria of Hashin are applied, the Mat-58 material type that is appropriate for woven-type fibre is the one that is employed. In conclusion, with regard to the influence of the orientation angle, it is feasible to kingdom that the most desirable stacking sequence is discovered to be for the aggregate of the 0 degree and 45 degree stacking sequences. This conclusion can be reached by referencing the have an impact on of the orientation angle. In the context of this book, J. Naveen et al. (2019) has recommended carrying out a research project with the working title " Finite element analysis of natural fiber-reinforced polymer composites" The Authors of this text have advised carrying out this research project. Performing an analysis on any natural fibre composite fabric, even if it has a complicated geometry and loading, is made very easy using the RVE programme. Numerous sectors, such as aerospace, automotive, civil and mechanical engineering, as well as commercial gadgets, electronic products, and so on, make extensive use of the finite detail approach. Among these industries are those that deal with mechanical and civil engineering. In a nutshell, natural fiber-reinforced composites have been gaining a lot of interest in a variety of applications as of late due to the fact that they are biodegradable, have an inexpensive material price, are readily available, and can be recycled. These are just some of the benefits that natural fiber-reinforced composites offer. In the field of finite element modelling, the consultant volume element method is the most common homogenization-primarily based multiscale constitutive technique used to study the impact of microstructures on the mechanical and thermal properties of NFRPC. This technique is used to determine how the microstructures influence the properties. Salu Kumar & Sandipan (2018) The Authors have proposed a research study entitled "Finite Element Analysis of Aircraft Wing Using Carbon Fiber Reinforced Polymer and Glass Fiber Reinforced Polymer." The purpose of this cutting-edge test is to examine the wing of an aeroplane that has been created out of carbon fibre reinforced polymer (CRFP), glass fibre reinforced polymer (GRFP), and aluminium alloy in order to find a material that is suitable for usage within the process of making wings. The research was conducted with the help of ANSYS, and the wing was designed with the help of the solid modelling tool CATIA V5 R20. The assessment was carried out using the use of the finite element approach. In a potential development for the future, the number of major load-bearing contributors can be modified, in addition to an expansion of different substances, with a variety of boundary conditions. This will allow for the discovery of substances that are more suitable with regard to their structural and aerodynamic properties. in addition, one may conduct an investigation. Yuchen Wen et al., (2018) The Authors have proposed a research study entitled "Feasibility Analysis of Composite Fuselage Shape Control Via Finite Element Analysis" An investigation into the possibility of carrying out the recommended management strategies for the form of this research has resulted in the completion of a feasibility study. In order to conduct a feasibility study, it is necessary to first construct an accurate finite detail model that can mimic the production of a composite fuselage. Only then can the feasibility study be carried out. This version ought to reflect the appropriate parameters for the material, ply patterns, geometric combinations, and fixture systems. We conduct a feasibility test on the fresh new form manipulation apparatus in order to determine whether or not it is possible to use it. To begin, we develop a finite element model that accounts for an adequate thickness of the material, the arrangement of the ply, the form of the fixture, and the location of the actuators. This model also takes into consideration the geometry of the fixture. The subsequent stage is to validate and adjust the finite detail model based on the findings of the physical trials. The feasibility analysis that was carried out with the use of FEA covered not only the single-aircraft dimensional manipulation capability research but also the double-aircraft scheme assessment, the stress/stress analysis, and the failure test. We have arrived at the conclusion that the design with a single aircraft and ten actuators is feasible for form management, and that the actuators no longer do any damage to the fuselage. This was reached when we came to the realisation that the actuators do not cause any harm to the fuselage. Galla & Dr. B Venkata (2018) "Modeling and Finite Element Analysis of Delaminated Composite Beams" In this check, the aeroplane spar wing beam with four extraordinary locations of delamination (X-zero, X-zero. Three, X-0.five, and X-0.7) changed into advanced the use of the parametric software programme CREO, and the use of the software programme ANSYS changed into examined for its performance. Static structural analysis, fatigue assessment, and modal evaluation are all performed on a rectangular cantilever composite beam with unmarried-part distortion. These evaluations are carried out on the beam. Static structural, fatigue, and modal studies are carried out in order to investigate the pressure, safety problem, and natural frequency of a range of substances with varied delamination length ratios. It has been determined that composite fabric has provided substantially greater overall performance when compared to conventional fabric. This conclusion was reached after comparing the two. Yogita U. & Poonam S (2017) The Authors have proposed a research study entitled "Comparative Analysis of Composite Materials by Using Finite Element Method" In the course of these investigations, testing and analysis of the free vibration of a cantilever beam formed of composite and metallic materials are carried out. The conclusions drawn from these investigations are detailed below. By making use of ANSYS 17.1, we were able to determine not only the mode shape but also the natural frequency of the plates. These materials are put through an investigation into the influence of a modal in order to have a better understanding of vibration assessment. In order to examine the properties of a cantilever beam after it has been subjected to unfastened vibration, an analytical approach is used. In this comparison, the outcomes obtained using the numerical approach, which is also known as the finite element method (FEM), are contrasted with the conclusions obtained through the analytical method. In addition, we are able to make predictions about how that specific fabric could act when it is subjected to a certain natural frequency. The inference that can be derived from this is that the discoveries produced using FEM have a totally unbelievable settlement with the outcomes obtained analytically. This is the conclusion that can be drawn from this. Opukuro S et el., (2017) The Authors have proposed a research study entitled "The Effect Of Ply Orientation on The Vibration Characteristics of 'T ' Stiffen Composites Panel: A Finite Element Study" According to the results of this study, the vibration overall performance of the shape is significantly influenced by the floor ply orientation of carbon fibre reinforced composite panels as well as the stacking sequence. This has a significant effect on the overall performance of the shape. The Lanczos apparatus was used in order to extract the dynamic functions of the panels, and the results were that the natural frequencies of vibration as well as the mode shapes were received. This was possible because of the fact that the panels were able to be broken down into their component parts. The findings of this study have implications for the selection of composite laminate lay up for the best combinations of stiffness, vibration, compression, and shear behaviour. These implications have been drawn as a result of the findings. Aditya Milind (2017) The Authors have proposed a research study entitled "Finite Element Analysis of Composite Aircraft Fuselage Frame" According to the results, there is a high-quality resource that may serve as a standard against which to evaluate modern corporate resources. Both E-glass tape and carbon material are capable of producing better results, or at the very least, outcomes that are somewhat equivalent to those that can be achieved with the assistance of aluminium 7075. The level of safety provided by materials derived from carbon fibre is increased, although these materials are not successful in the Z direction. Fabric made from carbon fibre experiences less strain when subjected to the same load conditions. It is feasible to use carbon cloth as a composite material in the frame building process of an aeroplane; however, this will need more testing in the stacking technique. The discovery raises the prospect of more work that may be accomplished in the future in relation to the investigation of composite frames. Either the move-phase or the thickness of the pass-section may be optimised, depending on the situation. Mohamed Mahran et al., (2016) The Authors have proposed a research study entitled "Aero-Elastic Analysis of Composite Plate Swept Wings Using the Finite Element Method" For the purpose of this particular piece of study, an aeroelastic version of composite plate wings was built making use of the FE technique. With the assistance of the FE form functions matrix, the relationship between the aerodynamic version and the FE model can now be established. An example of the finite detail and aerodynamic mesh that is being analysed for the swept front wing can be seen on page 17, which discusses the influence of a variety of laminate configurations on the aeroelastic characteristics of composite wings. Validity of the version is increased by examination and analysis in comparison to other works presented within the context of the relevant study.

The present model is in a position to be used in an inquiry of the manner in which the divergence and flutter rates are affected by the wing sweep viewpoint. Ashawesh et al., (2016) Authors have proposed a research study entitled "Experimental and Finite Element Analysis For Composite Aircraft Structures." This article, which is based entirely on two wonderful types of research (experimental and analytical), demonstrates that when working with complex composite systems, a wide variety of outcomes is possible due to the presence of unexpected variables. This is demonstrated by the fact that this article is based entirely on two wonderful types of research. Despite this, one must continue to proceed with great care before depending on these analytical processes on their own, particularly in light of the knowledge gained via those studies. just the intrinsic frequencies and mode shapes of the wing field, which are considered to constitute its primary dynamic characteristics, are presented here. These are the only dynamic properties that are being shown here. This study provides a synopsis that is both clear and brief of the experimental studies that have been conducted on the wing container. Xuan Wang, et al., (2015) The Authors have proposed a research study entitled "Iso-Geometric Finite Element Method For Buckling Analysis of Generally Laminated Composite Beams With Different Boundary Conditions" An application of the is geometric finite detail method (IGA), which is primarily based at the Non-Uniform Rational B-splines (NURBS) basis function, is made for the buckling evaluation of a typically laminated composite beam with a diffusion of boundary conditions in this look at. The beam in question was subjected to a variety of boundary conditions. The display of numerical results of crucial buckling masses and mode shapes demonstrates both the efficacy and accuracy of the currently used IGA approach. The results of this investigation are then compared to the many records that have previously been compiled. In a similar fashion, the affects of the modulus ratios, slenderness ratios, stacking sequence, and the fibre angle, particularly the Poisson impact, are clearly depicted on the critical buckling hundreds of the composite beam. In conclusion, the benchmark responses that were revealed over the course of this research have the potential to serve as a reference for the work that will be carried out in the years to come.

III.MATERIALS AND METHOD

Imported the geometry that was already built-in to Catia V5 into the Design Workbench of ANSYS's window for static structural analysis. Defined the material parameters for the modelled aeroplane wing in the Engineering Data, including the density and modulus of elasticity of the material. The geometry was loaded into the Ansys modeller window so that the remaining tasks, including meshing and defining boundary conditions, could be carried out. It was decided to use a meshing size of 1 mm for this particular meshing job, which was carried out on the connecting rod. Boundary condition was used, and in this case, pressure of 0.005 MPa was applied at the tangent of the wind blowing from the rear in an upward direction, while the fixed support was regarded at the fixed end of the wind blowing to the aeroplane magazine for the study. The problem was solved with the help of the Ansys solver once the boundary condition and the load had been implemented.

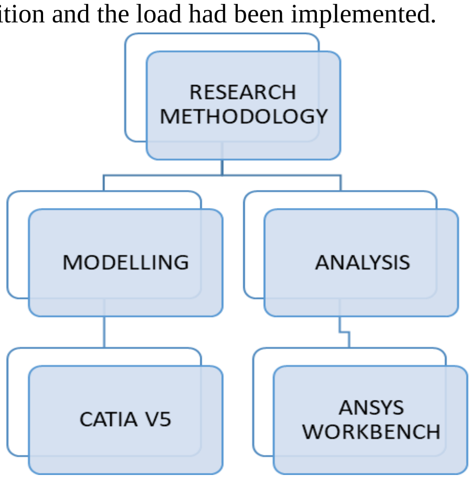


Figure 2 An overview of the design approach for modelling and analysis

- STEP TAKEN FOR FEA ANALYSIS

Ansys is a computer aided engineering tool, which is used for the structural analysis tool and is based on Finite Element Method, Finite Element Method (FEM), firstly breaks the domain, which is object here into several smaller element, which is called Discretization and then equations for all the smaller element are done and get it calculated, it is done for the accurate result.

The following steps were taken to execute the static structural analysis using Ansys Workbench.

a) Imported the geometry that built-in Catia V5 into the Design Workbench of ANSYS static structural analysis window.
b) Defined material properties for the modelled aero plane wing model in the Engineering Data, Such as modulus of elasticity and density.
c) The geometry was opened in the Ansys modeller window to perform the rest of the activities, such as meshing, and boundary conditions.
d) Meshing of the Connecting rod was done, and a 1mm meshing size was selected for thismeshing work.
e) Boundary condition was applied, and here, Pressure (0.005MPa) was applied at the tangent of the wind of the backside in an upward direction while the fixed support was considered at the fixed end of the wind to the aeroplane magazine for the analysis.
f) After the boundary condition and the load were applied, it was solved using the Ansys solver.
g) The next step was to select the output variables, so here, total deformation, von Mises stress and the von-Mises strain were tested.

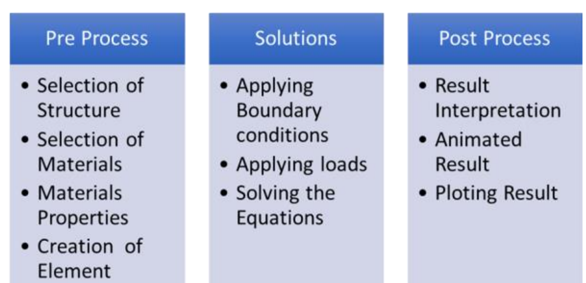


Figure 3 An overview of FEA Analysis

The results were demonstrated using the output display unit of the Ansys workbench, and min-max Prob was selected to show the maximum and minimum value. Transferred geometry from Catia V5's native representation to ANSYS's Design Workbench for static structural analysis. The wing's material specifications have been set in the

Engineering Data section of the model. Density and modulus of elasticity are two of the material parameters considered here. To make it easier to do the remaining tasks, including meshing and defining the boundary conditions, the geometry was imported into the Ansys modeller window.

- MODELLING AND MESHING OF AN AIRPLANE WING

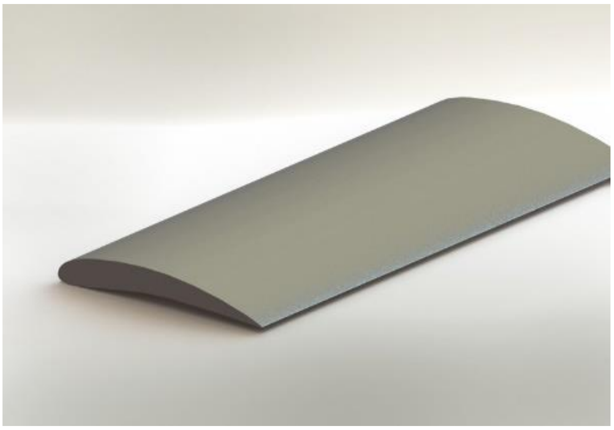


Figure 4 Modeling view

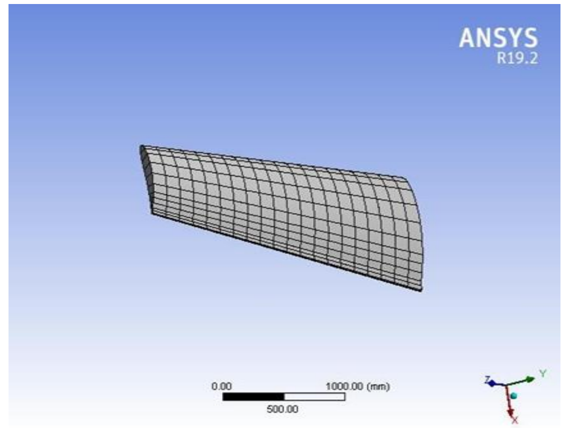


Figure 5 Meshing view

It was decided that a meshing size of 1 mm would be optimal for this operation, which was performed on the connecting rod. A pressure of 0.005 MPa was applied at the tangent of the upward-blowing wind from behind as the boundary condition employed here. Meanwhile, the research assumed that the stationary support was located in the windiest part of the aircraft's magazine. The problem was solved with the aid of the Ansys solver once the boundary condition and the load were included into the model.

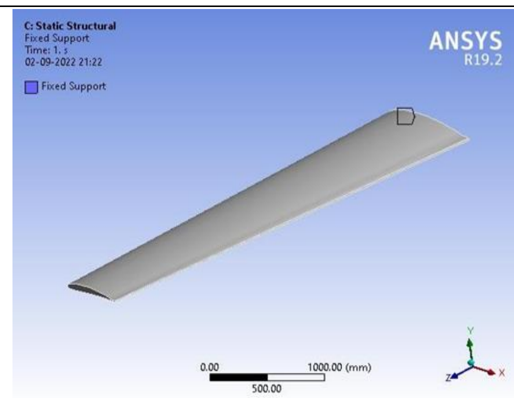
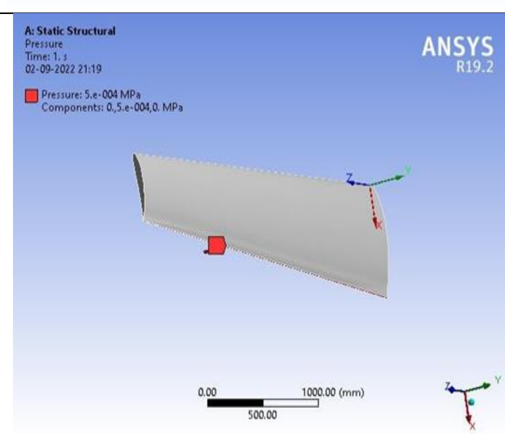
<table>
<tr><td rowspan="3">Fixed at the bigger end of the Airplane</td><td>EGR Glass</td><td rowspan="3">
</td></tr>
<tr><td>R Glass</td></tr>
<tr><td>S Glass</td></tr>
<tr><td rowspan="3">Pressure applied at wind in an upward direction</td><td>EGR Glass</td><td rowspan="3">
</td></tr>
<tr><td>R Glass</td></tr>
<tr><td>S Glass</td></tr>
</table>

Figure 6 Boundary conditions of material

IV.RESULT AND DISCUSSION

According to the findings, the material that allowed for the greatest amount of deformation when subjected to a pressure of 0.005MPa was S Glass, which resulted in a deformation of 0.049 mm. Because of its superior performance on deformation, S Glass may be one of the best alternative materials among all the others. This can help to limit the deformation that occurs under high pressure as well. The results of this investigation showed that all three of the materials had the same performance under these specific parameters; however, the results may have been different under other types of working situations. The maximum von Mises stress that can be achieved with S Glass is 0.049MPa, and the maximum deformation that can be achieved with it is just 0.049 mm.

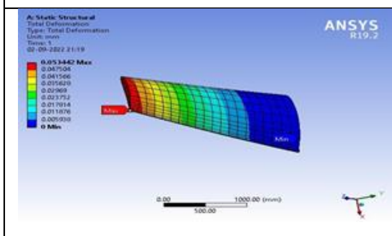
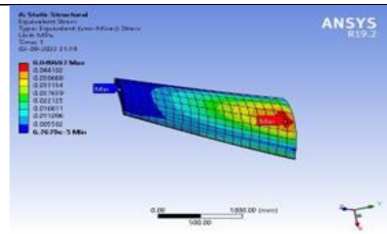
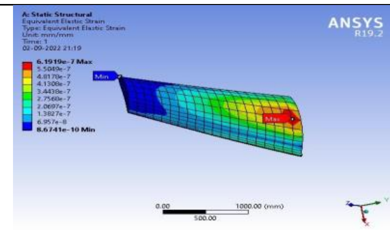
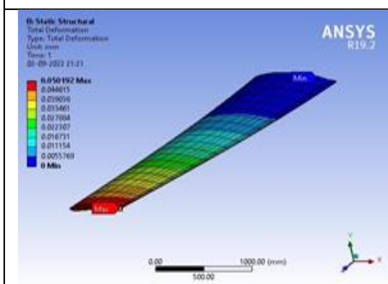
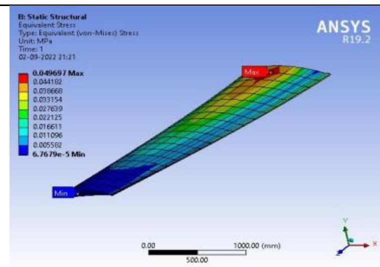
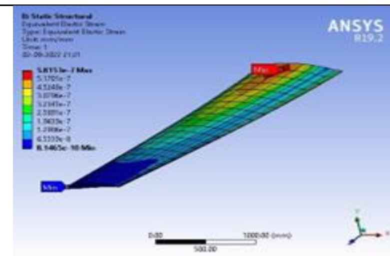
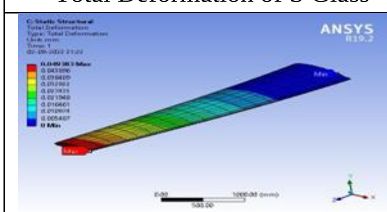
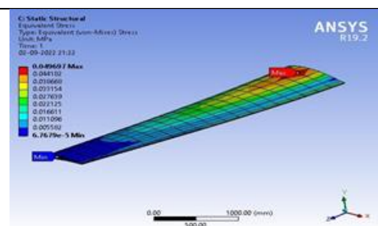
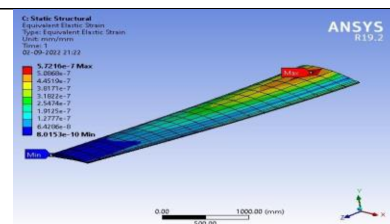
Total Deformation of EGR Glass	VON MISES STRESS of EGR Glass	VON MISES STRAIN of EGR Glass
		
Total Deformation of R Glass	VON MISES STRESS of R Glasses	VON MISES STRAIN of R Glass
		
Total Deformation of S Glass	VON MISES STRESS of S Glasses	VON MISES STRAIN of S Glass
		

Figure 7 Analysis of material

There was a static structural analysis of the aerofoil wing's construction, and the wing's performance was measured with a number of different materials. It was discovered that at 0.005MPa pressure, S Glass deformed by 0.049 mm, making it the material that allowed for the most degree of deformation. S Glass, among all the alternative materials, may be one of the best due to its greater performance on deformation. Also, this may help restrict the amount of distortion brought on by extreme pressure. It was determined from this study that all three materials performed similarly within these limits; nevertheless, it is possible that the outcomes might have varied in other contexts. Maximum von Mises stress in S Glass is 0.049MPa, while maximum deformation is just 0.049 mm.

Table 1 Total Deformation produced in different materials

S NO	PROPERTIES	EGRGLASS	RGLASS	SGLASS
1	TOTAL DEFORMATION (mm)	0.053	0.050	0. 049

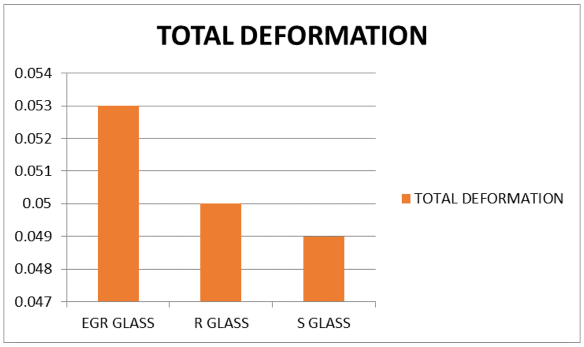


Figure 8 Total Deformation produced in different materials

In Figure 4.9 shown clearly that minimum deformation shown in S glass show we can say that S Glass is more suitable for manufacturing of Airplane Wings because less deformation means more rigid more rigid means act like link or structure which means whatever shape we want we can make it without any loose of applied forces. We can also used here R Glass and EGR Glass because the deformation among all material is in very less in different so according to Strength of Material the major principle stress and minor principle stress are same or approximate.

Table 2 Stress produced in different materials

S.NO	PROPERTIES	EGRGLASS	RGLASS	SGLASS
1	VON-MISESSTRESS(MPa)	0.049	0.049	0. 049

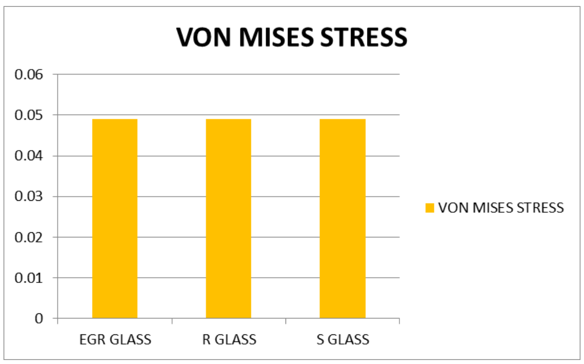


Figure 9 Stress produced in different materials

Figure clearly show that von-Misses stress ($\ddot{\sigma}$) is same among all materials it means all the parameter depends on the stress such as force and cross section area is same.

General formula of stress is

$$Stress(\sigma) = \frac{force}{crossestion\ area}$$

According to theory of failure or machine design (safe design)

$$\ddot{\sigma} = \frac{3}{\sqrt{2}}\sigma_{act}$$

Table 3 Strain produced in different materials

S.NO	PROPERTIES	EGRGLASS	RGLASS	SGLASS
1	VON-MISESSTRAIN	$6.19e^{-7}$	$5.18e^{-7}$	$5.77e^{-7}$

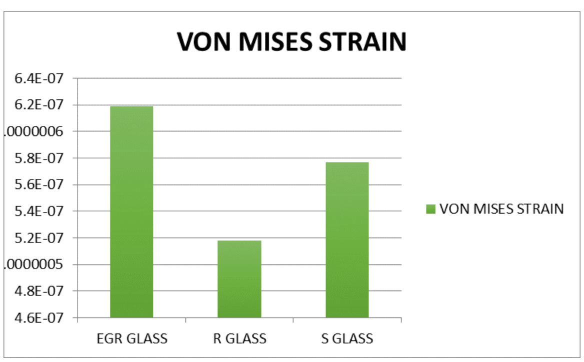


Figure 10 Strain produced in different materials

V.CONCLUSION

The performance of the aerofoil wing was assessed using a range of materials, and its construction was analyzed statically to determine its structural integrity. Throughout the course of this investigation, a number of different types of glass, including EGR glass, R glass, and S glass, were used. When exposed to a pressure of 0.005 MPa, the data suggest that S Glass is the material that is most capable of creating the highest amount of deformation as compared to the other options the. While the other three materials employed in this experiment all generated the same maximum von-Mises stress of 0.49 MPa, this one only produced a deformation of 0.049 mm. From the analysis that was presented before, the following conclusion may be derived:

- Because it has a greater performance on deformation, which may also help to minimise the deformation that happens under high pressure, it is probable that S Glass is one of the best alternative materials among all of the others. This is because it has a higher performance on deformation.

- The results of this inquiry showed that the performance of all three of the materials that were selected for it was the same for this particular circumstance. However, the results may have been different for other types of working environment
- While S Glass can withstand a maximum von Mises stress of 0.049 MPa, the material can only deform to a depth of 0.049 mm at most. This suggests that the material performs fairly well in general across the board.

REFERENCE

[1] Mohamad (2022) “Buckling Analysis of a Thin-Walled Structure Using Finite Element Method and Design of Experiments” https://doi.org/ 10.3390/.

[2] Nasir et al., (2022) “Performance Evaluation of Unmanned Aerial Vehicle Wing Made From Sterculiasetigeradelile Fiber and Pterocarpus Erinaceus Wood Dust Epoxy Composite Using Finite Element Method Abaqus And Structural Testing” www.elsevier.com/locate/ijmecsci.

[3] Arijeet Nath et al., (2021) “Modelling and Finite Element Analysis of an Aircraft Wing using Composite Laminates” doi:10.1088/1757-899X/ 1183/1/012006.

[4] Diaa Emad et al., (2021) “Low-Computational-Cost Technique for Modeling Macro Fiber Composite Piezoelectric Actuators Using Finite Element Method” Www.Irjet.Net.

[5] Can Kandemġr (2020) “Weight Optimization of An Aircraft Wing Composite Rib Using Finite Element Method” https://app.emaze.com

[6] Sarah David et al., (2020) “Application of the Finite Element Method in the Analysis of Composite Materials: A Review” www.mdpi.com /journal/polymer.

[7] J. N. Reddy (2019) “Introduction to the Finite Element Method” National Research Council, Research Directions in Computational Mechanics, National Academies Press, Washington, DC, 1991

[8] Mongkol (2019) “Optimization of a Hybrid Carbon/Glass Composites Afterbody of the Amphibious Plane with Finite Element Analysis”http://www.engj.org/ DOI:10.4186/ej.2019.23.5.125.

[9] O.Nurihan et al.,(2019) “The Impact Behaviour of Carbon Fiber-Epoxy Composite Leading Edge using Finite Element Method” ISSN: 2277-3878, Volume-8 Issue-2.

[10] J. Naveen et al., (2019) “Finite Element Analysis of Natural Fiber-Reinforced Polymer Composites” https://doi.org/10.1016/B978-0-08-102289-4.00009-6.

[11] Salu Kumar &, Sandipan (2018) “Finite Element Analysis of Aircraft Wing Using Carbon Fiber Reinforced Polymer and Glass Fiber Reinforced Polymer.” doi:10.1088/1757-899X/402/1/0120 77.

[12] Yuchen Wen et al., (2018) “Feasibility Analysis of Composite Fuselage Shape Control Via Finite Element Analysis”

[13] https://www.researchgate.net/publication/322907 145.

[14] Galla & Dr. B Venkata (2018) “Modeling and Finite Element Analysis of Delaminated Composite Beams”

[15] Yogita U. & Poonam S (2017) “Comparative Analysis of Composite Materials by Using Finite Element Method” http://www.ijert.

[16] Opukuro S (2017) “The Effect of Ply Orientation on The Vibration Characteristics of Stiffen Composites Panel: A Finite Element Study” http://creativecommons.org/licenses/bync/4.0/.

[17] ADITYA MILIND (2017) entitled “Finite Element Analysis of Composite Aircraft Fuselage Frame "http://aerospaceengineeringblog.com/composite -manufacturing.

[18] Mohamed Mahran et al., (2016) “Aero-Elastic Analysis of Composite Plate Swept Wings Using The Finite Element Method” http://www. nisasoftware.com/.

[19] ASHAWESH. (2016) “Experimental and Finite Element Analysis for Composite Aircraft Structures.”https://doi.org/10.3390/ ma1415431 6

[20] Xuan Wang, et al., (2015) “Isogeometric Finite Element Method for Buckling Analysis of Generally Laminated Composite Beams with Different Boundary Conditions” https://www. researchgate.net/publication/283308891