RESEARCH ARTICLE OPEN

Fluid-Structure Interaction Study on an Aircraft wing model using ANSYS Coupling system

S. Srividhya, B.E[1], K.Nehru, M.Tech, (Ph.D)[2], M.Subramanian, M.Tech, Ph.D[3]
[1]*Department of Aeronautical Engineering, SNS College of Technology,Sathy Main Road, Coimbatore, India – 641035*
[2]*Assistant Professor, Department of Aeronautical Engineering, SNS College of Technology, Coimbatore, India*
[3]*Prof & Head of the Department Department of Aeronautical Engineering, SNS College of Technology, Coimbatore, India*

ABSTRACT
The wing of an aircraft is the most critical part of an aircraft structure as it greatly influences the aerodynamic performance of the aircraft. Maintaining aircraft stability involves a lot of factors including the aeroelastic factors in an aircraft wing. It is paramount that the structural integrity of the aircraft body and the wing is maintained to avoid structural failure. The interaction of aerodynamic forces and elastic forces in a structure leads to aeroelasticity problems. Hence, these problems can be addressed by solving the fluid-structure interaction on an aircraft wing. This paper deals with solving the Fluid-Structure interaction on an aircraft wing using ANSYS Workbench components. In this project, an aircraft wing is designed and modeled using CATIA whose material is defined as Aluminium Alloy. The 2-way FSI analysis is carried out using Coupling method analysis in ANSYS Workbench which combines the data from ANSYS Structural and ANSYS Fluent to determine the structural deformation as well as aerodynamic effects on the aircraft wing which aids in studying the static aeroelastic effects in the given aircraft wing.
Keywords *–Aircraft wing, ANSYS, CATIA, CFD, Coupling, Deformation, FEA, Fluid flow.*

Date of Submission: 29-08-2020 Date of Acceptance: 14-09-2020

I. INTRODUCTION

Fluid-structure Interaction is an interdisciplinary multifaceted physic phenomenon occurring in a system where fluid flow impacts the deformation of a solid structure, which also leads to the changes in the boundary condition of the fluid flow. An elastic solid body coming in contact with a flowing fluid is also subjected to pressure due to the flow which in turn causes deformation in the structure. Now, because of the deformed structure, the initial flow is also affected. Ultimately, the subsequently altered fluid flow exerts another type of pressure on the structure, and the process keeps repeating. This kind of interaction is termed as Fluid-Structure Interaction (FSI). During such interactions, the forces on the solid object can lead to different deformations. These deformations can be quite large or very small and it depends on the pressure and velocity of the flow as well as the material properties of the subject structure. FSI is of fundamental importance and a prominent research area in many aerospace-engineering applications on account of its manifestation in aeroelasticity, aerothermoelasticity, and heat transfer.

A. Computational FSI

The study of FSI is done by various methods and one of the most common methods in Engineering practices s Computational mechanics. The use of numerical simulations can minimize the amount of time spent and manpower required for experimental methods to evaluate a large number of design alternatives based on the problem. A better understanding and realization of the problem and concept can be obtained through Computational methods because of the large range of information gathered during the process. Especially in cases of FSI which involves both Fluid Mechanics and Structural mechanics, Computational methods involving both Computational Fluid- Dynamics and Computational Structural Dynamics have many advantages with the study by Numerical Simulations.

CFD and CSD are the two most prominent research areas in aerospace applications and now with the technological advancement in the past decade, it has reached a maturity level in aiding solving large industrial and academic problems that were not plausible in the past. With the technological growth in past decades, it is visible that high-performance computers and various coupling algorithms, the study of FSI by numerical simulations have been made accessible and reliable and there are numerous ways to solve FSI cases with various flexible structures and adaptive flow conditions.

There are numerous solvers for carrying out such Coupling methods and one such solver is ANSYS Workbench and this project is based on performing a coupling method in the said solver. The solution of such dependently coupled FSI problems is carried out by system coupling which applies implicit sequential coupling by employing an iterative solution between CFD and FEA solvers. During each time step of the FSI solution, there are many mesh updates and data transfers within the participant solvers along with the multiple iterations between them. Convergence for the time step is achieved only when participating solvers are converged individually.

B. Coupling Method

A system coupling component system in any solver connects a structural and fluid flow system for solving FSI. In ANSYS Workbench, the system coupling enables us to couple a transient or static structural and fluid flow in Fluent or CFX to form transient or steady-state FSI. Results can be viewed in individual components or an integrated post-processing result system is also available for CFD and structural results.

The fluid properties and Structural conditions on the wing are mutually exclusive and a coupling system is required to determine the effects of one over the other. The new boundary for coupling is obtained by calculating generalized force values and the deflection of the wing in static equilibrium. Fluid computations are based on the Navier-Stokes equation. The structural mesh and flow field mesh deformation is carried out by the Trans-Finite Interpolation method (TFI), and the data exchange of structural and fluid on the System coupling interface was realized by the Constant Volume Transform method (CVT).

C. FSI in ANSYS

The main objective of this project is to conduct the FSI numerical simulation on ANSYS Workbench and study how the solver is efficient. The primary reason for using this solver system is because the ANSYS Fluent provides many different mesh deformation schemes, which include layering, smoothing, and remeshing, that account for significant updates to the fluid domain during the FSI solution or coupling method. These FSI parameters also require the advanced structural capabilities that are available in ANSYS Mechanical components like Transient structural and Static structural accounting for nonlinear geometric effects, say large strain & large deformation, nonlinear material behaviors, and nonlinear contacts that require an accurate simulation to read their behavior.

The ANSYS Workbench system coupling solver which is a conjunction of both ANSYS Fluent and ANSYS Mechanical provides a platform of advanced software tools that aids us to robustly solve the industry's most complex fluid-structure interaction challenges. The ability to carry out such advanced FSI applications with an efficient and intuitive workflow allows the engineers to proficiently perform fluid-structure interaction simulations. Such practices enable the engineers to account for the Multiphysics problems during product development.

D. Applications of FSI

Fluid-structure interaction exists in its many forms in both natural systems as well as man-made objects. It is also primal to realize the degree of severity in such interaction between the solid and fluid characteristics as it varies between different problems. The most common FSI problems in Aerospace applications are aero-elasticity, hydroelasticity, flow-induced vibration, thermal deformation, etc.

E. Aeroelasticity

One of the main applications of aerospace FSI is the flow of air around an airplane wing structure causing the wing to deform. When the wing deforms, the structural deformation causes the change in air patterns around it. The most important FSI phenomenon on an airplane structure is Aeroelasticity. Aeroelasticity is a subject that studies, analyses, and harnesses the interactions among aerodynamic forces, elastic forces (structural deformation), and inertial forces (motion/ dynamics of aerodynamic lifting surfaces). Aeroelastic interactions determine the airplane loads and influence the performance of flight in four primary areas:

- wing and tail surface lift redistribution
- lift effectiveness
- control effectiveness
- aileron reversal

Also, Aircraft structural dynamic response causing structural instability due to buffeting and flutter. Aeroelastic phenomena may be divided according to the Collars triangle as shown in Fig. 1. The sides of the triangle illustrate the relationships among the particular pairs of forces representing the said areas of mechanics. On the other hand, the triangle's interior represents the interference of all three groups of forces leading to dynamic aeroelastic phenomena. When inertial forces are excluded, the interaction is characterized by unidirectional structural deformation, which as a result cause Static aeroelasticity. Meanwhile, dynamic aeroelastic phenomena do include inertial forces in their oscillatory characteristics of structural deformation.

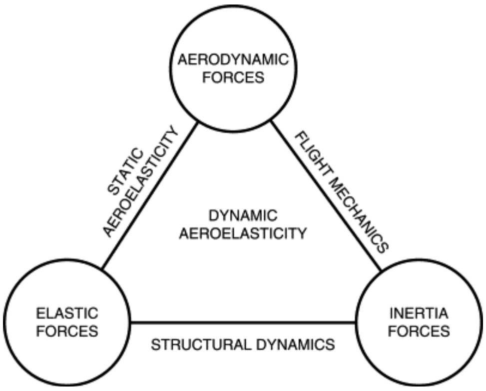


Fig. 1 Collar's triangle

The purpose of this project is to illustrate the effect of aerodynamic load and structural deflection interactions on the aircraft wing model by studying the interactions between the structural internal forces and the external aerodynamic forces on the wing. The typical section model development is followed by the development of a multi-degree of freedom model to illustrate the mathematical criteria for the static stability of an aeroelastic system.

In ANSYS Workbench, the coupling component is used to achieve the CFD/FEA coupling system to study the Fluid-Structure Interaction on the chosen aircraft wing.

II. LITERATURE REVIEW

Xiangying Chen,Ge-Cheng Zha, Ming-Ta Yang (2007) The authors have developed a numerical methodology with fully coupled fluid-structural interaction for determining the flutter characteristics in a 3-D transonic wing. It involves coupling Navier–Stokes equations and structural modal equations. They have employed the dual-time step implicit unfactored Gauss-Seidel iteration with the Roe scheme in the flow solver. A modal approach structure solver is used to simulate the wing's response. The flow and structure solvers are fully coupled via successive iterations within each physical time step. They have used ANSYS Workbench for this method of approach and have verified the accuracy of the solver. They obtained the first 5 modes of the chosen wing to study the modal response with the coupled system. They have done the study on the flutter boundary of AGARD wing 445.6 with free-stream Mach numbers ranging from 0.499 to 1.141 and their results are proved to be compared well with the experimental data.

Bocheng Zhang, Weilong Ding, Shengcheng Ji &Jiazhen Zhang (2016) In this paper, the authors have proposed a method based on Euler equations for predicting transonic flutter boundary in this paper. Euler equations are considered for the fluid field and boundary layer equations are considered inside the boundary layers, taking viscosity into account. The prediction of the transonic flutter boundary is performed based on the traditional method in the frequency domain using generalized aerodynamic coefficients matrices. Also, the simulation results are compared with the experimental results in this paper. The comparisons between the simulation and experimental results of the AGARD 445.6 wing show that the simulation results are following the experiment results for Mach numbers less than 1. They have encountered a transonic dip of the flutter boundary of the AGARD 445.6 wing which is located at a Mach number of around 0.954, which is also the Mach number when the shock wave appears on the wing surface. Also, they have proved that this frequency-domain is efficient than the time-domain method.

J S Chaitanya, Arun Prasad, B Pradeep, P L N Sri Harsha, S Shali, and S R Nagaraja (2017) The main objective of this paper is to use the application of CFD in determining the vibrational characteristics of an aircraft wing. The analysis has been performed in ANSYS Workbench to obtain the vibrational characteristics of the AGARD 445.6 wing. The wing was studied under transonic flow conditions. CFD analysis is performed on the wing to have a basic understanding of the pressure variations on the surface of the wing. The transient analysis gives the time domain solution for the wing which is utilized to extract the structural frequency. The plot of Mach number and its corresponding flutter index is used to observe the behavior of the wing when it is in the transition zone between subsonic and supersonic speeds. From the flutter index diagram, the dip is observed in the transonic regime and at a certain Mach number, the wing is most unstable.

NatarajKuntoji, Dr. Vinay,and V. Kuppast (2017) The design of the aircraft wing using NACA standards has been discussed in this work. The wing analysis is carried out by using computer numerical analysis tools including CAD/CAE and CFD. The necessary inputs for carrying out the structural analysis with emphasis on the vibration are obtained by CFD analysis. The deformation of the wing structures is investigated concerning the standard airflow velocity. The Computer-Aided Design Tools and NACA standards have been accomplished to design the wing structure. The vibration characteristics of the wing structures are studied by modal analysis to find the natural frequency of the wing structures. The CFD results revealed that the pressure on the upper surface of the wing for all the wing section planes is less, about -4.97e3N/mm2, as compared to the pressure on the lower surface, about 1.08e4 N/mm2, which satisfy the theory of lift generation. The pre-stressed modal analysis shows the correlation of the stress, deformation, and the corresponding mode of vibration.

Han Jinglong, Cui Peng (2011) This paper involves the study of aeroelastic behavior of the subject wing model using numerical simulation and comparing the results with experimental results. The wing used was the MAVRIC wing. They have adapted a high-fidelity method of approach to solve the structural and aerodynamic factors of the wing. They have used the CFD approach derived from Euler equations to solve the aerodynamic properties and CSD for structure. Since the aeroelastic behavior requires both the factors, they have developed a coupling solver to solve the aeroelastic phenomenon. Flutter and LCO behavior of the basic transport wing was predicted first, and the results were compared with the existing experiment. It was found that large-amplitude shock-wave motion provided the proper physical mechanism for the LCO. Then, flutter analyses of winglet transport wing and C-wing were conducted.

Jingyuan Yang, Yilang Liu, and Weiwei Zhang (2018) A fast static aeroelastic analysis method, coupling with the modal method and Kriging surrogate model, is proposed in this paper. The deflection of the wing is described by the modal method, and the Kriging surrogate model is utilized to model the generalized forces under different deformations, angles of attack, and Mach numbers to replace the CFD solver. They have analyzed the static aeroelasticity of the HIRENASD wing in the transonic flow field by coupling with the generalized force model by the static equilibrium equation. The results were compared with those of the experimental data. The developed model is aimed at replacing the CFD solver and is more time saving than the CFD/CSD method when it comes to a large quantity of the static aeroelastic analyses. It has a good perspective for engineering applications for the aircraft design period.

JainishTopiwala, Gaurav Mistry, Sandip Patel, and Pratik Umrigar (2016) The paper is a study that lightened up the basic theories behind fluid-structure interaction. That includes various methods to solve FSI Problem, types of Coupling, different ways to approach FSI& detailed application areas of it. The paper talks about the importance of FSI when designing any aerodynamics parts or other engineering components. The paper focuses on the interface condition that is required to consider for safe design. Also, the paper deals with using ANSYS Fluent, and Structure simulation for FSI is solved analytically by importing loads from fluent to structure. Their results suggest that either object will go into the plastic limit or elastic deformation.

Jong-Hwan Kim, Jae-Sung Bae, and Jai-Hyuk Hwang (2016) In this paper, they have developed a static/dynamic aeroelastic analysis through CFD-CSD coupled method on a UAV structure. A Computational Fluid Dynamic is used to compute the aerodynamics and the Finite Element Method is used for structural analysis. The CFD-CSD coupled analysis for static and dynamic analysis were performed for the high AR wing of the solar-powered UAV. FEM model and CFD model of the present high AR wing are established and they are verified. Static and dynamic aeroelastic analyses of the wing using the CFD-CSD coupled method are performed. Their results show that the present wing is aeroelastically stable for both static and dynamic cases.

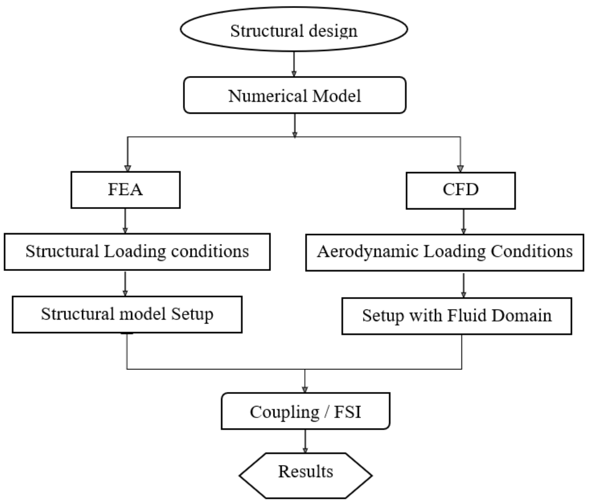
KakumaniSureka and R Satya Meher (2015) this paper deals with finding suitable material for a given wing ode which is the A300 wing since it is one of the most widely used aircraft wings. The main purpose of this project is to find out which material either AL alloy or Al alloy 7068 is best suited for making the wing of flight. The CAD model of the A300 wing with spares and ribs using is modeled using the software CATIA V5 R20 and the structural analysis is carried out using ANSYS WORKBENCH. From the obtained results they have concluded that the difference between the values of deformation, equivalent stress, max principle stress, stress intensity, and shear stress with Al alloy and Aluminum alloy7068 is minimal, and the results obtained are validated and verified. Since the difference in values was minimal, they have proposed that Aluminum Alloy 7068 should be used in the place of Aluminium alloy for the better structural integrity of the wing structure.

Ramindla Praveen, ElumagandlaSurendar, and K ShyamKumar(2018) The main objective of this paper is to achieve a reduction in the weight of an aircraft by using different materials including some composite materials, in which Aluminium as base material and mixed with some other materials at different proportions. A suitable wing profile NACA 4412 is selected and modeled in CATIAV5 R20. The generated wing profile is imported to ANSYS WORKBENCH. To examine the structural effectiveness of the designed wing, 3-D finite element analysis was performed using ANSYS software to compute the critical stresses, displacements, strains and to test the wings against

Von- Misses failure criterion. The conditions are given based on the inputs obtained from experimental examinations on the selected wing. The materials were assigned differently and the results were compared. The first 6 order natural frequency and the vibration modes are obtained. The results are used to study the structural performance of the wing with different materials.

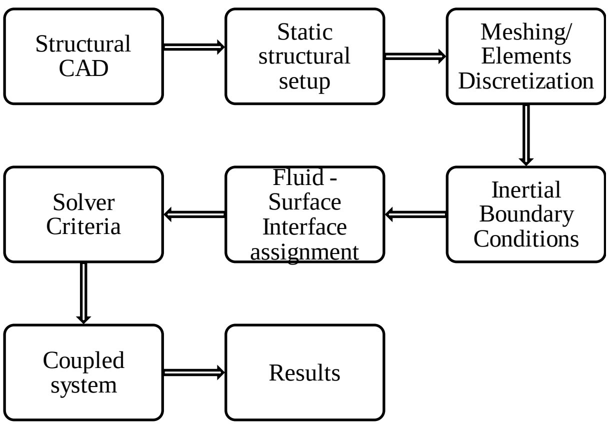
III. METHODOLOGY

A. Method of Approach Overview



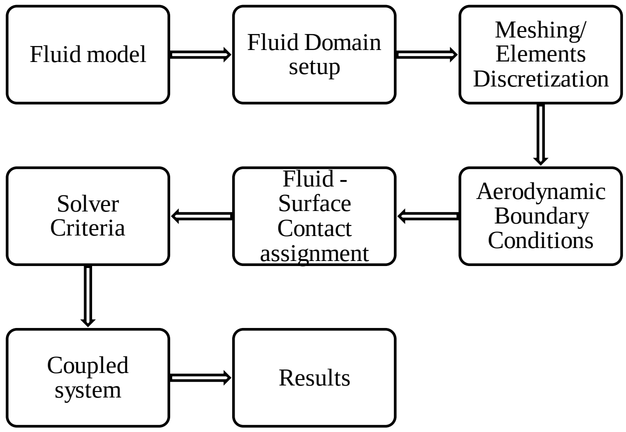
Flow chart 1 *Method of approach*

A. Finite Element Analysis



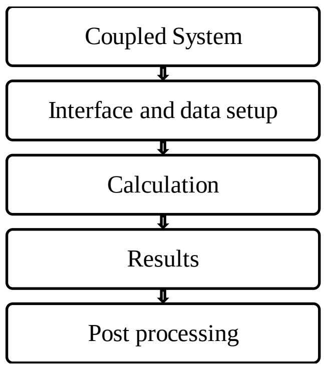
Flow chart 2 *Structural Analysis*

B. Computational Fluid Dynamics



Flow chart 3 *Fluid Dynamic analysis*

C. Fluid-Structure Interaction



Flow chart 4 *Coupling method for FSI*

IV. MODELLING OF WING

The performance and operation of any aircraft are fundamentally dependent upon the type of wing, its parameters, and characteristics. Hence, the wing of an aircraft is the most important part of an aircraft structure. The aircraft wing is modelled using CATIA V5 wherein the airfoil coordinates are imported through Microsoft Office Excel Macros. The wing model used in this paper is a tapered wing and the angle of attack maintained with the wing model is 10 degrees. The specifications of the wing model used are as follows.

A. Airfoil

An airfoil is the 2D representation of the aircraft wing or in other words, t is the cross-sectional view of the wing. It is a fluid determined body and is moved by aerodynamic forces. The front of the airfoil is named the leading edge and the rear of the trailing edge.

Airfoil used: NACA a65112

Maximum thickness ratio is 12% at 40% of the chord
Maximum camber is at 1.1% at 50% of the chord

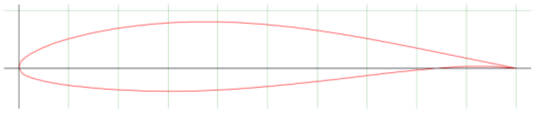


Fig. 2 Airfoil Plot

B. Wing Dimensions

The wing used for this project is inspired bythe AGARD wing. However, the dimensions are changed for research purposes. The dimensions of the wing used in this paper are as follows.

Root Length = 559 mm
Tip Length = 356 mm
Wing Span = 1542 mm

C. CAD Model

The airfoil data imported into the CATIA Software is displayed in the below figure. The points seen in the figure are the coordinates of the above-mentioned airfoil and they are connected to form the curve of an airfoil.

Fig. 3 Airfoil Plot imported in CATIA

The above figure displays the airfoil curve for the Root of the wing structure used in this project. Similarly, the airfoil coordinates for the tip are also generated by altering the data and dimensions of the coordinates and are generated using MS Excel macros. Once both the airfoil curves of Root and Tip are generated, the curves are joined using the Multi-Section method to generate the Wing Surface. The isometric view of the wing used is displayed in the following figure.

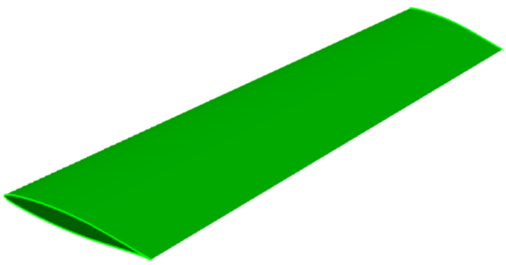


Fig. 4 CAD Model of the wing

D. Materials Used

The material properties of a structure or fluid have a great influence on the results of both structural and fluid simulations. The material of the wing structure is defined with **Aluminum Alloy**. Aluminum is the most common material used in the aerospace field due to its lightweight structural property and high strength. Aluminum Alloy constitutes mixtures of materials including Copper, Iron, Manganese, Silicon, Magnesium, Zinc, and Lithium. The Mechanical Properties of the chosen Aluminium allow exhibits lower weigh or density than any other high strength materials. Aluminium is much reliable for aerospace applications, especially for the high strength low weight property. The properties of the Aluminum material used in this paper are represented in Table 1.

TABLE 1
MATERIAL PROPERTIES

Density (g/cm3)	2.6898
Modulus of Elasticity (GPa)	68.3
Poisson's Ratio	0.34

Also, In ANSYS Fluent **air** is used for inlet surface condition at a speed of 115 m/sec. The properties of air are maintained at the atmospheric conditions with a density of 1.225 kg/m^3, the default in ANSYS Fluent.

V. FINITE ELEMENT ANALYSIS

Finite Element Analysis is a numerical simulation method in which a mathematical representation of a physical system is produced. A structural model is created by applying material properties, and applicable boundary conditions which collectively are referred to as PRE-PROCESSING of the analysis. It is followed by the solution of that mathematical representation, which is referred to as SOLVING. Finally, the study of the results of that solution and plotting virtual representations of the results are collectively referred to as POST- PROCESSING. On the other hand, Computational Static structural is one of the classical core disciplines in FEA that deals with the problems involving elastic properties of the materials and the corresponding deformations based on the given problem. ANSYS is an analysis systems software that is capable of performing structural, thermal, vibrational and fluid analysis. ANSYS Workbench Static Structural solver is used for the FEA analysis part of the FSI Study.

A. Ansys Static Structural

Ansys Static Structural is one of the FEA tools available in ANSYS Workbench. Ansys structural analysis software is used to solve complex structural engineering problems involving in design and development of a structure. With the FEA tools available in this component, the solutions can be customized and automated based on the requirement and severity of the structural mechanics' problems and parameterize them to analyze and visualize different and multiple design scenarios. We have chosen ANSYS Solver systems for this project and the structural portion of the analysis, Static Structural component in Workbench is used.

B. Meshing

The structural model of this project constitutes the 3D model of the wing structure. The primary part of the model setup in any numerical simulation sis meshing and the structural model is meshed using the Setup component in ANSYS Workbench. Meshing is an important part of the simulation as the elements discretized contribute integrally to the final results. A fine mesh is required for good results, the default mesh provided by ANSYS may not be uniform and proper in many cases. Especially for a structure with airfoil and wing parameters, the meshing is complicated.

A structured mesh is necessary for better results in such numerical simulations. ANSYS Workbench provides a default unstructured mesh for any geometry that is defined in the solver. So, a meshing method is introduced to form a structured mesh for this analysis. The meshing method used in our problem is the automatic method with the 2nd order quadratic elements. The whole body is uniformly defined with a 10mm element size and the mesh is solved.

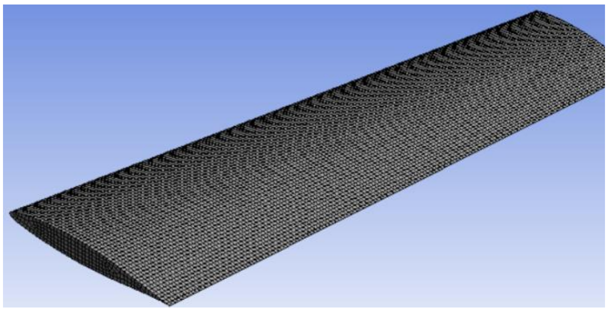


Fig. 5 Mesh of the model

The growth rate and smoothing of the above model are defined as high to attain a smooth mesh of the model. The statistics of this particular mesh are as follows,

Number of Nodes : 141894
Number of Elements : 30030

C. Boundary Conditions

In this project, the wing model is setup at Cantilever Beam's condition. The condition is based on the objective that the Root of the wing structure is attached to the body of the fuselage in an aircraft and hence it is fixed. The Root chord or the root surface of the wing is applied with fixed support. So, the structural model used in this problem exhibits the cantilever beam condition.

Standard Earth gravity is assigned in the y-direction according to the coordinate orientation if the model that resembles the actual position of a wing in cruise having gravity act downwards in the wing. An external load pressure of 500 Pa is distributed on the top surface of the wing as the C label indicates in the figure.

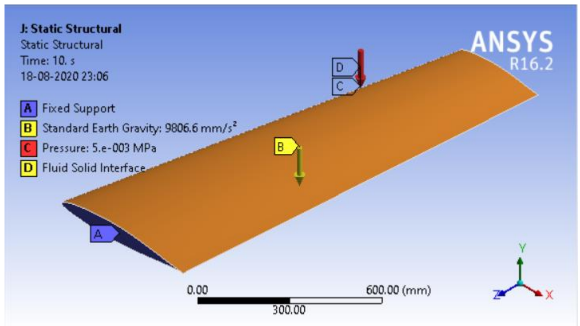


Fig. 6 Boundary conditions

The upper surface, lower surface, and the Tip surface are assigned as Fluid-Structure Interface. The Large deformations option in the analysis settings is turned on to obtain the results with more accuracy. In this particular type of solving, the minor deformations, or the changes in the result is also accounted for the result. The model is solved under Pre-Stress Environment in ANSYS. Hence the large deformation is selected. Also, the required results components are inserted into the solution for post-processing.

VI. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is the numerical solution method involving applied mathematics, physics, and computational software to solve the flow of a gas or a liquid and the effect of said fluid around the object through which they pass. Computational Fluid Dynamics is based on the Navier-Stokes equations. These equations control the CFD solver and describe how the velocity, pressure, temperature, and density of a moving fluid are connected in a system. For solving the fluid characteristics in this study, ANSYS Fluent solver, the Computational Fluid Dynamics solver system available in ANSYS is used. A fluid-domain is created for the wing model and the boundary conditions are applied based on the aerodynamic loads the structure is subjected to. Also, **Dynamic Meshing** is applied to the surfaces for the FSI purpose.

A. ANSYS Fluent

ANSYS provides us a comprehensive suite of a computational fluid dynamics solver system for modeling fluid flow and other equivalent and related physical phenomena. ANSYS solver system offers an unparalleled fluid flow analysis capability by providing all the tools that are necessary to design and optimize any kind of fluids equipment. It also helps in studying various scenarios by troubleshooting existing solving systems. The primary ANSYS solvers available in ANSYS in the area of the fluid and CFD are ANSYS Fluent and ANSYS CFX.

Both components provide a wide range of Fluid problem scenarios that enable many types of Engineering practices. With these solver systems, We can simulate various kinds of phenomena including aerodynamics, hydrodynamics, combustion, mixtures of liquids/solids/gas, reacting flows, particle dispersions, heat transfer, etc,. Both Steady-state and transient flow phenomena can be solved in ANSYS System. The graphic results of an ANSYS FLUENT CFD software simulation will illustrate the flow of a fluid, movement of the particle flow, heat transfer simulations, chemical reactions in a system, etc.

ANSYS Fluent has been used for this project for solving the aerodynamic forces of the wing. Fluent is much reliable than CFX in cases of system coupling method. The governing equations of any CFD Solver are as follows and the ANSYS Fluent also follows the same set of governing equations to solve any problem given.

B. Fluid Domain Model

A separate fluid domain around the model is required to define the role of the structural model inside the fluid atmosphere. In such cases, a separate fluid domain Boundary box is modeled. This can be done using CAD Modelling software or the design modeler in ANSYS also allows for such operations. This operation requires to draw a box around the fluid domain as required and subtracting the wing model structure inside the box so that the shape follows as that the region around the wing.

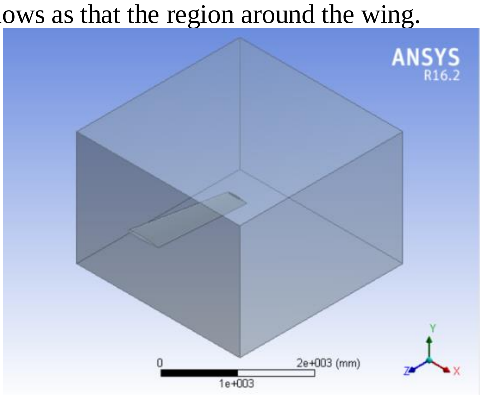


Fig. 7 Fluid- Domain Model

The Fig. 7 clearly illustrates the box that surrounds the wing model is the fluid domain of the given wing model. A required domain is designed and extruded to form a box, Also, a Boolean operation is performed to subtract the wing area from the fluid region such that the air or any other fluid flowing through the domain will not interfere with the interior portion of the wing. The air flows around the wing.

C. Meshing

The CFD model in this project comprises a fluid domain region surrounding the structure of the wing. Hence the meshing is a bit complicated than the structural model. We have to make sure that the entire fluid domain is distributed with uniform meshes while maintaining a smooth mesh around the contact region of the wing i.e., the FSI surfaces inside the boundary box. The 5 flat surfaces of this fluid domain are defined with second-order Triangular elements. The meshing method used for the wall face of the domain is the automatic method with the 2nd order quadratic elements as shown in the figures below, Fig. 8 and Fig. 9.

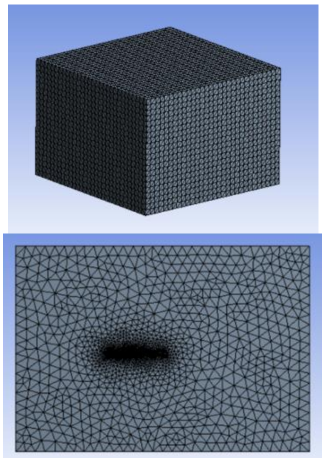


Fig. 8&9 Mesh of the Fluid Model

The growth rate and smoothing of the above model are defined too high to attain a smooth mesh of the model. The statistics of this particular mesh are as follows,

Number of Nodes	:123926
Number of Elements	:677511

D. Boundary Conditions

Boundary conditions consist of flow inlets and exit boundaries, wall, repeating, and pole boundaries, and internal face boundaries in the solution domain and it determines the flow characteristics. The applied boundary conditions in this set of the fluid model are tabulated.

TABLE 2
FLUENT BOUNDARY CONDITIONS

Surface	Type of boundary
Inlet surface	Inlet Velocity - Air
Outlet surface	Pressure outlet
Bottom face	Symmetry
Top face	Symmetry
Face behind	Symmetry
Front Wall / Wing attached face	Wall
Wing surfaces	Wall
The region inside the domain	Interior

The inlet surface of the fluid domain is defined with air inlet velocity with a velocity magnitude of 115 m/s. The initial gauge pressure is set to 0 at the inlet. The outlet surface in this model is defined as a Pressure outlet with default gauge pressure settings. As mentioned in the table, the front wall is assumed to be the region in which the root of the wing is attached to the fuselage body and hence assigned as a wall with No-Slip condition. The contact region is also defined as a wall since it is a structure and the fluid is flowing around the structure. The remaining 3 surfaces are applied as symmetry. The interior section is the region covered inside the volume of the fluid domain boundary box and so the region is applied as the interior.

VII. COUPLING SYSTEM

The FSI study requires a coupling system to transfer the data in Static structural (FEA data) and Fluent (CFD data) to solve the Fluid-Structure Interaction occurring in this set of problems. Ansys offers a System Coupling method through which we can manage to the exchange of data and coordinate independent solver executions to solve the complex interactions between physical models. In ANSYS software it is simulated in separate solvers with the transferred data and provides the results. The System Coupling in ANSYS manages the exchanged data transferred between the solvers as defined to model the application and coordinate both the solvers. This form of execution between them ensures smooth convergence of any Multiphysics simulation especially, high-fidelity Multiphysics simulations like FSI.

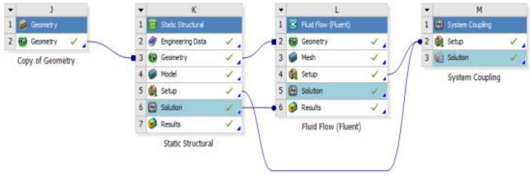


Fig. 9 ANSYS components Interaction

In ANSYS Static Structural, the upper surface and the lower surface of the aircraft wing is defined as **Fluid-Structure Interface**. This enables the setup in the Coupling system to read the data for Fluid-Structure Interaction.In ANSYS Fluent, the wing surfaces or the contact region as mentioned earlier is set to the Coupling method as the operation on these surfaces is entirely dependent on the coupling system and coupled boundary condition.

The data transfers allow the regions to be setup in the coupling system. The user must define the region with Structural – Fluid coupling and Fluid – Structural coupling. The properties are assigned by defining the source and target for the Fluid-surface interaction regions that were previously conditioned. The source and target are defined based on the FSI surface assigned in Static structural and Coupling contact defined in Dynamic Meshing in ANSYS Fluent. Dynamic Meshing was applied to all the surfaces in ANSYS Fluent based on the operation of each surface.

VIII. SOLUTION

The data in the Static Structural component and Fluent is transferred to the System coupling and the 2-way coupling method interchanges the data to both the solver components. The time step size and the number of time steps are calculated based on the problem and the solution model is setup. The numerical simulation is solved through the coupling solver. The results are observed separately from the Static Structural component and Fluent component respectively for better visualization.

A. Structural results

The ANSYS Workbench solves the given problem with the applied boundary conditions, solution definitions, and yield the results as required. The coupling method accounts for the aerodynamic load on the Surfaces of the aircraft wing. In this project, the structural results determined are Deformation along the y-axis in the wing, Equivalent Stress, Equivalent Strain, and shear stress along the surface of the wing. The Total deformation in this wing is illustrated in the figure below. The maximum is found to be at the tip of the wing where the wing is deformed up to 0.4 mm while the root of the wing doesn't deform and the value goes to very less in the range of 0 to 0.04 mm. This deformation is influenced by the structural load as well as the aerodynamic loads applied from ANSYS Fluent.

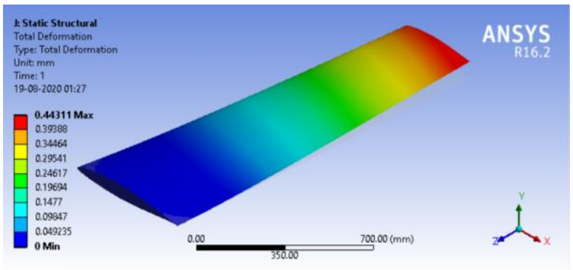


Fig. 10 Total Deformation

As we can see in the previous figure, the applied pressure causes the wing to deflect in the direction normal to the wing structure and this is accounted as the wing divergence under these circumstances and boundary conditions. The equivalent stress and strain are illustrated in the following figures.

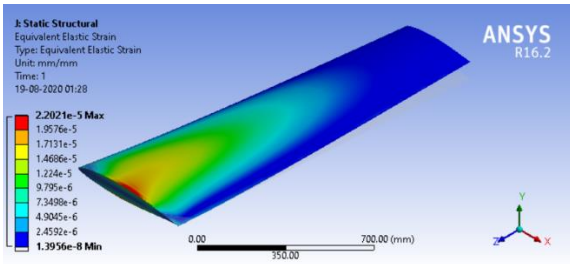


Fig. 11 Equivalent Elastic Strain

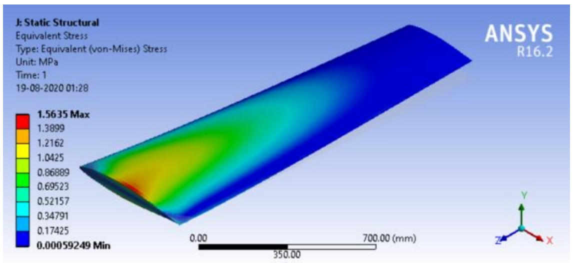


Fig. 12 Equivalent Stress

Shear stress is the force that is occurring tangentially over the surface area of the plane. Shear strain is the displacement of the plane over the distance of that surface from the opposite plane. The shear stress along the surface of the wing can be obtained by solving the shear stress on the XZ Plane in this particular model. Fig. 13 illustrates the Shear stress along the upper surface of the Wing. The shear stress is maximum along the leading edge of the wing surface near the Root of the wing.

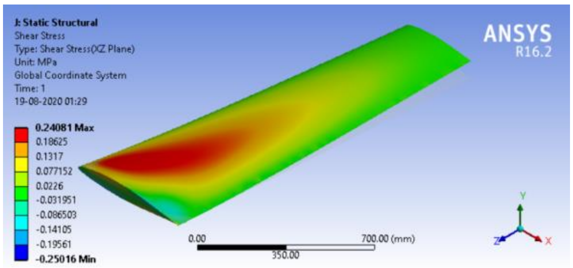


Fig. 13 Shear stress along the upper surface

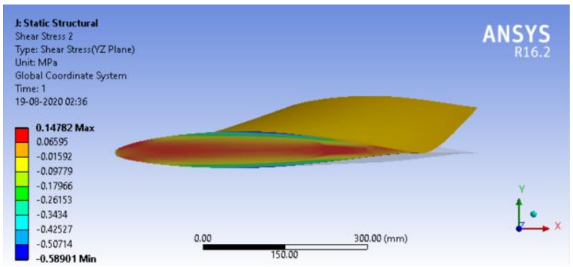


Fig. 14 Shear stress across the chord

The shear stress is intense near the root of the wing and it alleviates. Also, the shear stress solution along the YZ Plane gives the shear stress along the airfoil or cross-sectional area of the wing. The shear stress along the Root of the chord is represented in Fig. 14. Here, the shear stress is appeared to be maximum at the chord line and the camber line section for the given boundary conditions.

B. Fluent results

CFD Post is used for the Post-processing of the Fluent solver. Once the simulation is done, The CFD Post can be used to view the fluid flow, pressure distribution, and velocity vectors or other parameters required for any given problems. For this problem, we are studying the pressure distribution and velocity behaviours of the give fluid domain. Fig.15 below illustrates the pressure distribution over the developed wing.

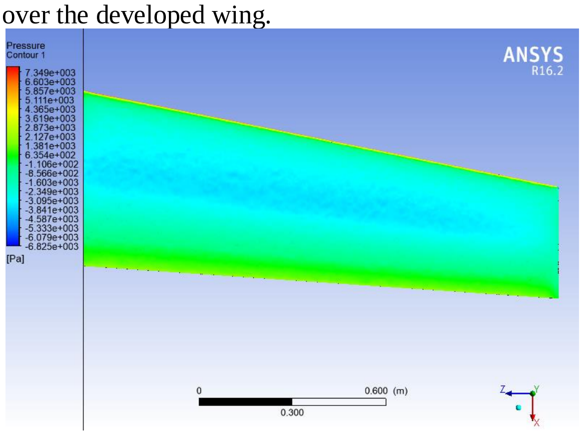


Fig. 15 Pressure distribution over the surface of the wing

The above image shows the pressure distribution along the surface of the wing. Here the upper surface is displayed. The pressure is maximum at the leading edge of the wing and it is uniform over the surface and again increases along the trailing edge. Fig.16 shows the pressure distribution contours concerning the wall that is attached to the wing.

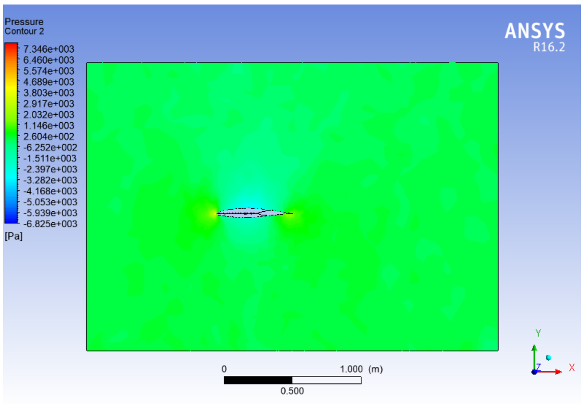


Fig. 16 Pressure distribution around the airfoil

Similar to that of pressure distribution observation, the velocity of the air inside the fluid domain is also observed in the post-processing of the Fluent. The air is set to flow from the inlet surface and it flows around the wing surface as it comes in contact with the wing area. The velocity vectors of this particular operation are illustrated in Fig. 17 and Fig.18.

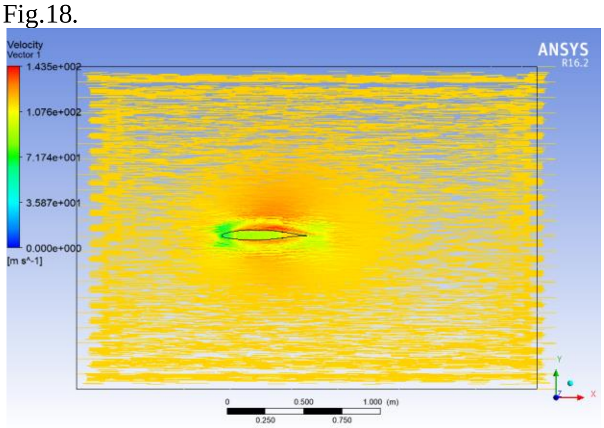


Fig. 17 Velocity contours around the airfoil

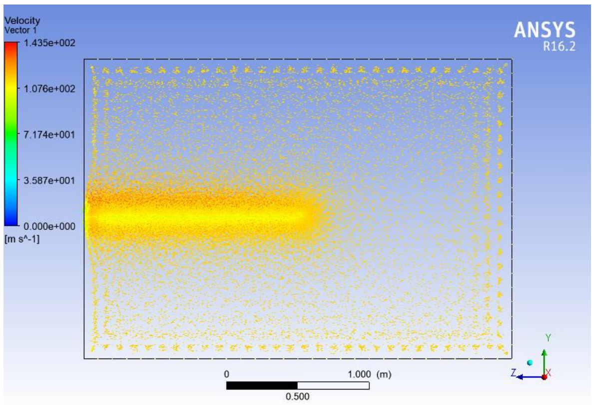


Fig. 18 Velocity contours around the domain

IX. RESULTS

The above solutions produced are the results of the model that is coupled under the System coupling method where the structural loads are transferred to ANSYS Fluent and the aerodynamic loads given in Fluent are transferred to Static Structural. But, the main objective of this paper is to study the deflection of the wing structure under these structural loads and aerodynamic loads combined. This is obtained by Total Mesh Displacement in CFD- Post.

Total Mesh displacement along the y-axis is determined and is represented in Fig. 19. The value of the deflection of the wing is **0.4 mm.** The deformation is along the tip of the aircraft wing under the given loads. The contours in the image illustrate the deflection of the wing.

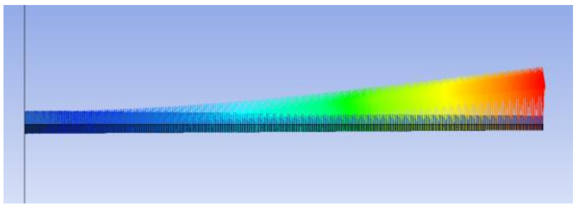


Fig. 19 Total Mesh displacement (Tip of the wing)

X. CONCLUSION

The main objective of this paper is to conduct the 2- way FSI Analysis using ANSYS components. The aircraft wing is modelled using CATIA V5 CAD software and is imported into ANSYS Workbench. The structural model and Fluid domain model are developed respectively based on the requirements. The boundary conditions for both the FEA Model and CFD model are defined in Static structural and ANSYS Fluent separately along with the assignment of contact surface conditions for coupling. The system coupling in ANSYS Workbench is used to perform the 2-way FSI Interaction for the components. The data is transferred as Fluid-Structure and Structure-Fluid and solved. The results are observed in post-processing.

REFERENCES

[1]. Xiangying Chen, Ge-Cheng Zha, and Ming-Ta Yang, *Numerical simulation of 3-D wing flutter with fully coupled fluid–structural interaction*,University of Miami, Pratt and Whitney, USA, (2006).

[2]. Bocheng Zhang, Weilong Ding, Shengcheng Ji &Jiazhen Zhang, *Transonic flutter analysis of an AGARD 445.6 wing in the frequency domain using the Euler method*, Beijing Aeronautical Science & Technology Research Institute of COMAC, Beijng, China, (2016).

[3]. J S Chaitanya, Arun Prasad, B Pradeep, P L N Sri Harsha, S Shali and S R Nagaraja, *Vibrational Characteristics of AGARD 445.6 Wing in Transonic Flow*, Amrita University, India, (2017).

[4]. NatarajKuntoji, Dr. Vinay and V. Kuppast, *Study of Aircraft Wing with Emphasis on Vibration Characteristics*, Basaveshwar Engineering College, Bagalkot, India (2017).

[5]. Cui Peng and Han Jinglong, *Prediction of flutter characteristics for a transport wing with wingtip devices*, Institute of Vibration Engineering Research, State Key Lab of Mechanics and Control for Mechanical Structures, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China, (2016).

[6]. Jingyuan Yang, Yilang Liu and Weiwei Zhang, *Static Aeroelastic Modeling and Rapid Analysis of Wings in Transonic Flow*, School of Aeronautics, Northwestern Polytechnical University, No. 127 Youyi West Road, Xi'an 710072, China (2018).

[7]. JainishTopiwala, Gaurav Mistry, Sandip Patel and Pratik Umrigar,*Fluid-Structure Interaction: Fundamentals and Application - A Review*, C. G. Patel Institute of Technology, UTU, India (2016).

[8]. Jong-Hwan Kim, Jae-Sung Bae, and Jai-Hyuk Hwang, Static and Dynamic Aeroelastic Analysis of a High Aspect Ratio Wing through CFD-CSD Coupled Method,Korea Aerospace University Goyang, Republic of Korea (2016).

[9]. KakumaniSureka and R Satya Meher, *Modeling and structural analysis on A300 flight wing by using ANSYS*, QIS College of Engg& Technology, Ongole, India (2015).

[10]. Ramindla Praveen, Elumagandla Surendar and K Shyam Kumar, Mulyono, *Design, Static Structural and Modal Analysis of Aircraft Wingb (NACA 4412) Using Anasys Workbench 14.5*, Warangal Institute of Technology & Science,Warangal, India (2018)