1. **Design and Analysis of Wing for Next Generation UAV’s**

**ABSTRACT**

A morphable / adaptive wing is the one that can change its geometry to accommodate multiple flight regimes. In this paper, we propose mechanisms to continuously morph a wing from a lower aspect ratio to a higher aspect ratio and to further extremities of a gull configuration. The morphable wing’s two-link structure is telescopic in nature. The telescopic actuation is performed by a linear actuator consisting of a rack and pinion arrangement. The cross sectional area remains almost constant but the aspect ratio does change due to the telescopic action. The 3D-CAD model developed in CREO and is imported into ANSYS 14 Work bench. The detailed structural analysis of the complete wing will be obtained by using ANSYS Work bench. Available aspect ratio wing would try to incorporate the high speed and maneuverability benefits of low aspect ratio wings, and increased range and fuel efficiency from the large aspect ratio wings. This type of wing system is to improve the range of achievable flying conditions for an unmanned aerial vehicle (UAV).

1. **Statistical Analysis OF Friction Stir Welding of Dissimilar Materials**

**ABSTRACT**

**Friction-stir welding (FSW)** is a solid-state joining process (the metal is not melted) and is used when the original metal characteristics must remain unchanged as much as possible. It mechanically intermixes the two pieces of metal at the place of the join, then softens them so the metal can be fused using mechanical pressure, much like joining clay, dough, or plasticine. It is primarily used on aluminum, and most often on large pieces that cannot be easily heat-treated after welding to recover temper characteristics.The aim of this paper is to investigate analytically friction stir welding of two materials aluminum alloys 6005-T1 and 6053-T6using different pin profiles, square, round, taper, thread and triangle. The analysis is done using different rotational speeds 700rpm, 1150rpm and 1350rpm. The plate sizes are 50mmX80mmX2mm. The tool material is H13 steel. The tool shoulder dia is 24mm, tool pin dia is 5mm, pin length is 1.7mm.

Modeling is done in solidwors and analysis is done in Ansys.

1. **Design and FEA Analysis of an Air Craft Nose Cone with Different Materials Using Ansys**

**ABSTRACT**

The aim of this paper is to design and analysis of an aircraft nose cone and analysis made with some different materials which are having slightly different properties by using simulation of ANSYS software. They are Aluminum alloy, Structural steel, stainless steel and titanium alloy. And concentrated firstly on deformation of nose cone under the air pressure of 18700 (Pa) and secondly the comparison of the materials mentioned above with their capability to withstand for the deformation, when the cone section of aircraft travelling at a height of 40,000 (Ft). This experiences the temperature around -45 0C to -54 0C through a compressible fluid medium. by using the ANSYS software, it is paved to analyze and created the replica of a nose cone section of aircraft as parabolic among many shapes because of less mean shear stress distribution, lowest tip temperature and Mach number having 0.8 to 1.0(which is very small and required for efficiency). Out of these materials the titanium material is having optimum deformation range of 49.674 (Min) and 238.14 (Max). Along with these materials titanium alloy having better properties which is now a day’s most popularly used in various fields is chosen for nose cone of an air craft for yielding good results.

1. **Design and FEA of Drill Jig for Head and Cover Part of the Actuator**

**ABSTRACT**

The manufacturing industry mainly small scale and medium scale provides wide range of products to fulfill the market needs. To face the many challenges of market these industries should increase their production rate with good quality and accuracy. Hence, the time required for the production should be decreased to as small as possible. As per the company’s present requirement they need such a technique for drilling operation which can be efficiently used to reduce the cost of production, improve the quality of the product, increase the production rate and reduce the operation time. Therefore, this study aims to design a drill jig. The main purpose of making this drill jig is to perform drilling operation without any need of shifting the job regularly.

The objective of this project work is to design a drill jig for head and cover part of the cylinder actuator. The creo software is used to model the drill jig, and analysis work is carried out on clamp plates to determine the stress, strain and deformation by using creo and ANSYS Workbench. Based on design model of the drill jig, the design drawings of each part are created by using AutoCad software. According to the dimension, all parts are manufactured and assembled to test its performance.

1. **Static Thermal Analysis of Exhaust Manifold using FEA**

**ABSTRACT**

An exhaust system carries waste gases and other combustion products away from an automobile engine. It allows the vehicle to operate with minimal noise, smoke and pollution transmitted to the environment. The exhaust discharges the gases at a very high temperature, thus it is important to study the distribution of temperature along the whole exhaust for its effective working.

In this study, the impact of temperature effect on the exhaust of an automotive is scrutinized. The objective of the analysis is to find the stresses induced in the manifold due to thermal growth. Firstly, the distribution of temperature giving rise to thermal stresses which are encountered due to varying load conditions in the manifold is analyzed using ANSYS with temperature field boundary conditions. Secondly, the same is analyzed analytically and after comparison it is found that the error which lies between the two results in less than 10%.

1. **Static Structural Analysis of Hybrid Composite Small Aircraft Connecting Rod**

***ABSTRACT***

In IC engine, reciprocating motion of the piston is converted into rotary motion by using connecting rod (CR). It acts as the intermediate link between the piston and crank. Most of automobile connecting rods are made of steel, now a day’s connecting rod are made of aluminum composite materials are used in race cars. The gas pressure inside the combustion chamber creates axial stress and inertial force due to reciprocation creates tensile and compressive stress on the connecting rod. In the present work, an investigation on structural behavior of connecting which is made of aluminum hybrid composite at different loading conditions. The Analysis done by using ANSYS WORKBENCH and model is created in Pro/E WF. Finallys comparison of analytical and FEA results are done.

1. **Finite Element anlysis of Bulb Turbine using ANSYS**

***ABSTRACT***

Hydro turbine is a rotary engine that extracts energy from a fluid flow by transferring the potential energy to electricity generation. Depending on head and water flow rate, variation of pressure and momentum cause the runner blades to rotate.

This research studied the effects of turbine runner blade which help in improving turbine efficiency. Objective of this project is to find the best material to increase the efficeiency and to maintain the profile of bulb turbine blades using ANSYS software. The bulb turbine consist three runner blade, 1685 runner speed, guide vane angle 350 and 18 no of guide vanes.

1. **Design and Structural Analysis of 3-Wheeled Hybrid Vehicle FRAME**

**ABSTRACT**

The use of energy acquired from fossil fuels to propel automobiles has always had its negative effects on the environment. Research on substitutes for fossil fuels has existed for a few decades now. This paper emphasizes on a 2 seated vehicle which is powered by human effort and by an electric motor, which can be employed both individually as well as simultaneously.

In this paper FRAME will be designed and model analyses with different Hybrid composite materials. The simulation and analysis of this vehicle has been carried out in **CREO** and **Ansys** softwares.

1. **Analysis of Knuckle Joint of 30C8 Steel for Automobile Application**

***ABSTRACT***

The aim of the present paper is to study calculate the stresses in Knuckle joint using analytical method. Further study in this direction can made by using various directions of the pin and the capacity to withstand load. It is also to be noted that instead of mild steel pin we can also use high strength high modulus steel pin that can further enhance the capacity to withstand higher loads. The shape of the knuckle joint can be changed for improved properties. One can carry out the analysis by changing the shape in the part in the knuckle joint in order to conserve materials and energy. The knuckle joint is proposed to develop in the present study is for an applied force of 25 KN. The diameter of the pin is proposed to be around 23 mm. The material of the knuckle joint is considered as mild steel grade 30C8, in order to do the stress analysis; mesh was developed for the knuckle joint. ANSYS software was run and the stress contour, displacement contour, strain energy contour were obtained. Based on the ANSYS analysis it shows that a pin of 23mm diameter can withstand a load of 25 kN if we use a factor of safety of 2. Further optimization of the diameter of pin, it depicts that a pin of 12 mm is enough to withstand a load of 25 kN. however if we use a pin of 25 mm the range of pulling load can be enhance to even 80 kN.

1. **Thermo structural analysis of high pressure cryogenic tank**

**ABSTRACT**

*In this paper, thermo-structural analysis of high pressure cryogenic tank (liquid hydrogen tank) is presented. The chill-down process is performed on a 9.6m3 Water Capacity (WC) and 22 MPa Maximum Allowable Working Pressure (MAWP) hydrogen tank with controlled rate of cooling. The finite element analysis (transient thermal and structural) of hydrogen tank is performed with ANSYS Software. This analysis incorporates temperature dependant material properties, temperature and pressure variations across the height of the tank during chill-down with liquid Nitrogen (LN2) followed by liquid Hydrogen (LH2) for analyzing behavior of the tank. Temperature distribution of the tank and magnitude are obtained and used for estimation of transient heat transfer, induced thermal stress, structural stress, distortion in the material due to chill-down and pressurization. In order to avoid thermal crack, chill-down at controlled rate of cooling restricting temperature difference across the tank height is maintained.*

1. **Validation of Hydraulic Design of a Metallic Volute Centrifugal Pump using CFD**

**ABSTRACT**

Centrifugal pump is a most common pump used in industries, agriculture and domestic applications. The design of a centrifugal pump impeller demands a detailed understanding of the internal flow during design and off design operating conditions. The present paper describes the simulation of the flow in a centrifugal pump impeller at four different flow rates viz. 3.33, 7.91, 12.52 and17.06 kg/s, with working fluids as Petrol/ Diesel and VG-32.The numerical solution of the deiscretized three-dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with CFD package ANSYS-CFX. For each flow rates, performance results, blade loading plots, mass averaged total pressure and static pressure, area averaged absolute velocity are presented.

1. **Optimization and Analysis of Lathe Machine Bed Structure Based On Structural Bionics**

**ABSTRACT**

A structural bionic design process is systematically presented for lightweight mechanical structures. By mimicking biological excellent structural principles, the structure of lathe bed or the stiffening ribs of a lathe bed were redesigned for better load-bearing efficiency. In this paper,

a machine bed (Manufacturer: Indira Machine Tools Ltd.) is selected for the complete analysis for static loads. Then investigation is carried out to reduce the weight of the machine bed, reduce the stress induced in the lathe bed and to reduce the displacement. In this work, the 3D CAD model for the existing bed model and the optimized bed model has been created by using commercial 3D modeling software CREO. The analyses were carried out using ANSYS. The results were discussed.

1. **Analysis of Loader Arm of Pneumatic High Speed Loader**

**ABSTRACT**

The pneumatic high – speed loader is employed to load and unload the auto sheet components in high speed metal forming press machine. Cycle time of 3 second for loading is required. Loader consists of tubular bridge frames; the tubular frames are mounted on the columns of the press machine. The loader consists of cross travel saddle, and vertical travel saddles, which are provided to have x-y axis, adjusted for the loading arm. The high – speed pneumatic arm is mounted on the vertical saddle. It has a rapid loading movement in and out of the press machine.

The loader arm also has up and down motion to pick and place the metal sheet in and out of the press machine; a travel time of 1500mm is completed in less than 1.2 sec. the pneumatic arm then places the metallic sheet on the press tool rapidly completing a loading cycle in complete 3 sec.

The 3D models of the loader arm are done in Pro/Engineer by changing the limb lengths. The limb lengths are decreased from 900mm to 700mm and 500mm. In the present thesis, Static analyses are done on the structure for maximum loads. The analysis is performed on the loader arm to optimize the arm from minimum stress and deflection. The model of the loader arm is changed by changing the limb lengths. The present used material is steel; it is replaced with Aluminum alloy 7075.

1. **Modeling and Analysis of Bracket Assembly in Aerospace Industry**

**ABSTRACT**

This project deals with the Model and static Analysis of Bracket assembly used in Aerospace Industry. This bracket assembly is used for placing components for various purposes like carrying fuel and air. The mounting bracket assembly consists of a circular plate with nine lugs for which three different tanks are mounted. The individual components i.e., circular plate, lugs etc are modeled and assembled through CREO Software. The loads are transferred by the tanks to the bracket are considered as pressure loads. To reveal the stresses induced due to pressure loads, Finite Element (FE) Analysis is performed with the help of ANSYS. Then the occurrence of max stress is found and factor of safety is calculated. This project provides a methodology for analysis of an assembly consisting of components made of composite materials and metal components.

1. **Modal Analysis of Engine Supporting Bracket using Finite Element Analysis**

***ABSTRACT***

*The Engine supporting bracket plays a vital role by reducing noise, vibration and harshness. The current work describes the finite element approach for modal and static structural analysis of engine supporting bracket. CAD model of engine supporting bracket was created in CREO software and same was analyzed for stress and vibration analysis using ANSYS workbench 15.0. Both initial design and modified design was compared for output responses in terms of equivalent Von-Mises stress, deformation and strain energy absorbed. In modal analysis, bracket was considered for vibration studies. The sole aim of modal analysis was to check whether the self excitation frequency of engine supporting bracket was less than natural frequency. Four alternative materials (Gray C.I., Aluminum alloy, Magnesium alloy and ERW-1) were analyzed. Stress analysis results suggest that deformation and Von-Mises stresses observed in EEW-1 materials was less (0.495 mm and 164.87 MPa). Further natural frequency of modified design was found to be 257.83 Hz which was well within the range below self-excitation frequency and less than the natural frequency (268.59 Hz) of initial design. It was found that aluminum bracket limit its use for the said application due to greater deformation and less stiffness. Magnesium bracket can be the option to ERW-1 steel for the Engine supporting bracket application but it cannot be deployed as it is highly susceptible to corrosion. From the results, it can be concluded that ERW-1 material best suit the requirement of the desired application and can be deployed with some safety standards.*