**DESIGN AND ANALYSIS PROJECT TITLES 2023-24**

1. **Modeling And Finite Element Analysis of Gas Turbine Blade**

**Abstract**

* In this project, the failure analysis of gas turbine blades and to overcome those failures the development of materials have been discussed. The properties of a gas turbine blades should have exhibit have been reported in the current review. In addition to the above, the importance of martial selection in gas turbine blade has been listed. After the brief investigation of the failure analysis and materials developed it is observed that still there are some properties to be developed to obtain an optimum gas turbine blade.

A gas turbine blade for the elongations is a major consideration in their properties because they are subjected to high tangential, axial, centrifugal forces during their working conditions. Several methods have been suggested for the better enhancement of the mechanical properties of blades to withstand these extreme conditions. This project summarizes the modeling and analysis of Gas turbine blade, on which CREO 2.0 is used for deign of solid model of the turbine blade with the help of the spline and extrude options ANSYS 14.0 software is used for model meshing of the blade and thereby applying the boundary condition.

* This project specifies how the program makes effective use of the ANSYS and applies boundary conditions to examine steady state thermal & structural performance of the blade for Nickel Alloy A286 & Inconel 725 materials with Titanium Alloys. Finally found the best suited material among the TWO from the report generated after analysis. From this the results are stated and reported.

Existing Material for Blade: *Nickel Alloy A286 &* Inconel 725

Proposed Material For Blade : **titanium alloys**

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.

Existing Material for Blade: Aluminium Alloy

Proposed Material For Blade : Hybrid Composites Aluminium

1. **Modeling and FE Analysis of Single Point Cutting HSS Tool inserts coating for low thermal conductivity**

**ABSTRACT**

*In this project, Heat Flow Rate at tool-tip interface is determined, generated in high-speed machining operations. In addition, analyses are done of a High Speed Steel and of a coated Carbide Tip Tool machining process results produced as part of this study. An investigation of heat generation in cutting tool is performed by varying cutting parameters at the suitable cutting tool geometry.*

*The experimental results reveal that the main factors responsible for increasing cutting temperature are cutting speed (v), feed rate (f), and depth of cut (d), respectively. It is also determined that change in cutting speed and depth of cut has the maximum effect on increasing cutting temperature. Various researches have been undertaken in measuring the temperatures generated during cutting operations. Investigators made attempt to measure these cutting temperatures with various techniques during machining.*

*In this project, while in turning operation temperature to be 700°c maximum at the cutting edge is used for measuring temperature at tool-tip interface. Single point cutting tool coated inserts has been solid modeled by using CAD Modeler CREO and FEA carried out to find the Properties of low thermal conductivity compare to other coatings by using ANSYS Workbench 14.*

Coating Thickness: 2.5 micro meter

Existing Material for (insert) : Carbide Tip

Proposed Material for insert coating : CrAlSiN/Cr2O3

1. **Heat loses simulation of a piston crown used in a Cooled bare diesel engine for agriculture**

**ABSTRACT**

* *The goal of this paper is to determine both temperature and thermal stress distributions in a plasma sprayed magnesia-stabilized zirconia coating AND CrAlSiN/Cr2O3 on an aluminum piston crown to improve the performance of a diesel engine. Effects of the coating thickness on temperature and thermal stress distributions are analysed, including comparisons with results from an uncoated piston by means of the finite element method (ANSYS). Temperature and thermal stress analyses are performed for various coating thicknesses from 0.4 & 1.2 mm including the bond coat layer.*
* *Temperature at the coated surface is significantly higher than that of the uncoated piston. It is observed that the coating surface temperature increases with coating thickness by decreasing rate. Increase in the maximum temperature according to the uncoated piston for 1.2 mm thick coating. The higher combustion chamber temperature provided by means of coating results in the better thermal efficiency of the engine. It also provides for a reduction in the substrate surface temperature. Main factor of this project is to maintain the piston height while in coating (ie. Piston crown surface to be faced depends upon to coating thickness).*

Existing coating Material : magnesia-stabilized zirconia

Proposed coating Material : CrAlSiN/Cr2O3

Coating thicknesses 0.4 & 1.2 mm

1. MODELING and ANALYSIS OF COMPOSITE LEAF SPRING USING FEA FOR LIGHT VEHICLE MINI TRUCK

**ABSTRACT**

*The Automobile Industry has shown increase interest for replacement of steel leaf spring with that of composite leaf spring, since the composite material has high strength to weight ratio, good corrosion resistance and tailor-able properties. The paper describes static analysis of steel leaf spring and laminated composite Multi leaf spring. The objective is to compare the load carrying capacity, stiffness and weight savings of composite leaf spring with that of steel leaf spring. The dimensions of an existing conventional steel leaf spring of a Light design calculations. Static Analysis of 3-D model of conventional leaf spring is performed using ANSYS. Same dimensions are used in composite multi leaf spring using CARBON FIBRE or sandwich composites STEEL NYLON STEEEL unidirectional laminates. The load carrying capacity, and weight of composite leaf spring are compared with that of steel leaf spring. The design constraints are stresses and deflection.*

( The most commonly used **grades** of **spring steel** are 55 Si 7, 60 Si Cr 7, 50 Cr V4.

Alloy Steel 5160, also sold as AISI 5160 )

Existing Material for Leaf Spring = **Alloy Steel 5160**

carbon/Epoxy and Graphite/Epoxy

Proposed Material for Leaf Spring =

CARBON fiber Or

Sandwich composite AISI 5160 STEEL NYLON 6 AISI 5160 STEEL

1. **CAD MODELLING AND OPTIMIZATION OF INDUSTRIAL ROBOT LOWER ARM**

**ABSTRACT**

* *In a competitive market it has become necessary to deliver products in a fast and efficient way. The mechanical design process needs to be modified in a way to be able to deliver reliable and fast products. One way of modifying the process is by reducing the amount of manual work by automating the repetitive tasks using scripts built in the computer tools.*
* *To validate its properties, the arm’s mass properties are compared with the real non-parametric model of the Industrial Robot IRB6640.*
* *Created robot arm models and analysed with real time boundary conditions with 2 different materials ( AL- 356, ARAMID epoxy). And calculated results of deformation and stress, and shear stress and strain values for both models.*
* *The modeling is done in the CAD software CREO and is evaluated structurally in the FEM software ANSYS.*
* *From all these results here we are going to conclude which material has less weight and which material has fewer amounts of stress values. From all these results we can get an idea with which material we should use for different conditions like less weight or less stress producing robot*

Existing Material Used = **AL356**

Proposed Material = ARAMID epoxy

1. **Finite element analysis and optimization of wind turbine blade**

**ABSTRACT**

* *The wind turbine blade is a very important part of the rotor. Extraction of energy from wind depends on the design of the blade.*
* *At present, the majority of wind turbine blades are constructed with glass fiber reinforced plastic (GFRP) and carbon fiber. The use of composite materials eventually have solved some of the problems associated with efficient operation of horizontal axis wind turbines (HAWTs) such as gravitational forces due to weight but there are other unresolved problems such as long term material property degradation, local shape deformation of the profile of the wind turbine blades etc.*
* *This project aims to address the adverse structural response of the blade profile with the variation of operational parameters such as wind velocity and material properties on blade’s performances. For this reason, the shape memory alloy (SMA), which is Nitinol (NiTi) has been embedded in the blade to alleviate the load. A parametric blade model utilising the ANSYS finite element program has been developed to efficiently predict the deflection of the blade.*

Existing Material Used = GFRP, carbon fiber

Proposed Material = Nitinol (NiTi) or **PLA**

1. **DESIGN AND MODEL AND VIBRATION MODE ANALYSIS OF END EFFECTOR**

**ABSTRACT**

* *To design and Analysis of end Effectors for handling and holding irregular objects, by using under actuation mechanism. The project is motivated by the requirement for grasping of objects of arbitrary shape and size.*
* *The key issues consider here are the gripper should be able to grasp the object of any shape, size and weight with a maximum limit stability of the object held during manipulations should not dependent of frictional forces between gripper and object.*
* *In this concept the finger is under actuated and is able to hold the irregular objects. The finger is of the snapping action. it is fixture that can be mounted on the machining centers.*
* *It is the concept that the three fingers are mounted on the table with the help of springs underactution is possible.*
* *To Analysis the End effecter hand using ANSYS14 workbench*

Existing Design = Design Implemented

Proposed system = Design Safety Factor

1. **DESIGN AND MODEL ANALYSIS OF INDEXING CONVEYOR**

**ABSTRACT**

*Indexing conveyors are the most effective way to move or elevate the material and transfer it from point to point with indexing. The design, layout configurations, service duty rating, size of the casing and flight assembly are all components that make a significant difference on the quality, durability and longevity of any given conveyor. This report is an attempt to put in words the experience of our exposure to Design and Analysis of indexing Conveyor System. The aim and objective was to optimize existing indexing Conveyor System. The design involves working on heavy structure, high scale vibration, and overweight bottom chassis. Procedure of selecting material played an important role in achieving the objectives. All the procedures and operations are done keeping in mind the dimensional restrictions along with economic views. We tried to give the direction according to developing industrial sector in India and provide best alternatives. Finite Element Analysis is also done for better understanding to optimize the load or weight. Model objective is achieved by tool SOLIDWORKS.*

Existing Design : Asm4340 Steel / Aluminum Alloy (2024-T6)

Proposed Design : Titanium Alloy (Ti-6Al-4V)

1. **DESIGN AND STATIC STRUCTURAL AND MODEL ANALYSIS FREE STANDING I BEAM JIB CRANE**

**Abstract**

*A jib crane is a type of crane having cantilevered beam with hoist and trolley and it is either attached to a building column or cantilever vertically from an independent floor mounted column. This paper will mainly concentrates floor mounted jib cranes here the trolley hoist moves along the length of the boom and the boom spin allowing the lifted load to be skillfully about in a relatively circular area. While designing a jib crane several factors have to be considered in these most important factors are own weight of the crane, the weight of the goods.*

*The aim of this project investigates by Static structural analysis the stress and deformation of jib crane with different materials and the work is carried out by designing reinforcement to overcome those stresses in the component. With the analytical design dimensions models are prepared in modeling software CREO 2.0 and the analysis is performed on the models by finite element solver ANSYS 14.0 with suitable conditions and results are compared.*

Existing Material Used = ASTM A36 CARBON steel

Proposed Material = ASTM A516 CARBON steel

1. **CHARACTERISATION OF Aluminum 7075-T6 COMPOSITE FOR PISTON ANALYSIS BY CAE TOOLS**

**ABSTRACT**

* *This paper presents the piston static structural analysis for unreinforced alloy and reinforced composite with different weight percentage of reinforcements by considering the combined loading of both gas pressure force and inertia force.*
* *The force is applied on crown of piston and model is constrained for all DOF on piston pin mounting hole.*
* *In present paper the mechanical behavior or performance of a* ***Aluminium 7075-T6*** *composite is compared with the conventional material.*

Existing Material Used = Aluminum 6061

Proposed Material = **Aluminum 7075-T6**

1. **DESIGN AND ANALYSIS OF CYLINDRICAL PRESSURE VESSEL BY USING DIFFERENT FRP COMPOSITE MATERIALS**

**ABSTRACT**

*The pressure vessel is in wide range, A pressure vessel is a holder intended to hold gasses or fluids at a pressure considerably not quite the same as the ambient pressure. The pressure differential is perilous, and deadly accidents have occurred in the authentic background of pressure vessel progression and operation. Pressure vessels can theoretically be any shape, yet shapes made of regions of circles, barrels, and cones are normally used.*

*This project will present the development, application and results of this new composite material which shows better performance results than the existing.in this dissertation work ANSYS software is used to carry out the structural analysis of steel and FRP composite pressure vessels. On the basis of analysis it is found that steel pressure vessel has less strength than FRP pressure vessel and it is also concluded that the pressure inside the vessel can be reduced up to 75% by replacing steel with FRP material. For the same geometrical parameters of the steel pressure vessel the stress state of FRP composite pressure vessel is calculated under different interval pressures experimentally. Finally pressure and structural efficiency of composite pressure vessel is compared with steel pressure vessel. In this project pressure vessel is modeled used CREO design software and static analysis is carried out by applying inconel, s2 glass, saffil and stainless steel in ANSYS 14 software.*

Existing Material Used = stainless STEEL or Nickel Alloy

Proposed Material = **FRP composites – PP, PVC, NYLON**

1. **STATIC STRUCTURAL COMPARATIVE ANALYSIS OF MILLING CUTTER USING FEA**

**ABSTRACT**

*Milling is a process of producing flat and complex shapes with the use of multi-tooth cutting tool, which is called a milling cutter and the cutting edges are called teeth. The axis of rotation of the cutting tool is perpendicular to the direction of feed, either parallel or perpendicular to the machined surface. The machine tool that traditionally performs this operation is a milling machine. Milling is an interrupted cutting operation: the teeth of the milling cutter enter and exit the work during each revolution. This interrupted cutting action subjects the teeth to a cycle of impact force and thermal shock on every rotation. The tool material and cutter geometry must be designed to withstand these conditions. Cutting fluids are essential for most milling operations .In this Paper the design aspects of milling cutter is analyzed. The objective considered is the design and modeling of milling cutter and to analyze various stress components acting on it. By taking two different material i.e. HSS and Cemented Carbide with coating to check stress and deformation. The design and analysis is carried out using the software like CREO and ANSYS.*

Existing Material : **HSS**

Proposed Material : **Cemented Carbide**

1. DESIGN AND ANALYSIS OF AUTOMATED 2-AXIS WELDING MACHINE USING ANSYS

ABSTRACT

*CNC automated welding allows creating and maintaining a quality welding workflow, ensuring flexibility, high efficiency and high output rates. The MIG welding machine has a relay switch to control ON/OFF operation of welding torch. The essential feature of the process is the small diameter electrode wire, which is fed continuously into the arc from a coil. The base metal used for welding is carbon steel. Through this report our effort will be to completely eliminate human contact during welding which will no doubt be safer but will also be more efficient. The machine is used to weld work pieces in flat position since higher deposition rates are achieved in this position and is a convenient approach. The project will be able to weld any contour in 2D profile that is generated over a graphic user interface in addition to butt, edge, and lap joints.*

*In this report, the emphasis is given on design calculation and structural analysis. The analysis of L section base frame and bridge of material AISI 1020 is done considering the weight of 18kgs. Setup makes use of three stepper motors of 10 kg-cm capacity for efficient movements in three axes. MIG welding is used in the process in order to attain proper trade-off between quality of welding and the price. The report gives step wise development and analysis of the entire designing process for automatic welding equipment.*

Existing Material : AISI 1020

Proposed Material : **AISI 1045**

1. DESIGN AND STATIC STRUCTURAL ANALYSIS OF EXCAVATOR BUCKET THE TOOTH OF THE BUCKET

Abstract

*There is rapid growth in the earth moving machine industries as the construction work is rapidly growing is assured through the high performance of construction machines. This paper focuses on the evaluation method of digging forces required to dig the terrene for light duty construction work. This method gives the force calculation and further it is used for the carrying out the fatigue analysis to calculate fatigue life of bucket and its failure. Further the work regarding the optimization of bucket to give maximum fatigue life for the digging at the desired force conditions. An analytical approach provided for static force analysis of excavator bucket.*

Existing Material : SM50A - JIS [Japan. Structural steel](http://www.splav-kharkov.com/en/e_z_mat_class.php?type_id=3&zcountry=22)

Proposed Material : **SCNCrM2B -** LOW CARBON STEEL

1. DESIGN AND FINITE ELEMENT ANALYSIS OF REAR WHEEL FOR CAR

**ABSTRACT**

*The paper describes process of design and structual analysis for rear wheel hub. Wheel hub in automotive system are attached wheel to motor shaft (axle) and provide the support to the tie rods, connect the trailing arm from chassis to the rear wheel, fastening of brake caliper respectively. while we designing the wheel hub that time have to work on mainly overall shape, material specification, size, surface finish and appearance, easy to fastening & handling. Because rear wheel hub is undergoing radial load, axial load, tangential load, fatigue load during running condition in the various automotive system. The Finite element analysis (FEA) is used after designing process for checking factor of safety and what would be possible changes that can provide adequate design of it. And by using FEA we can select the exact material from the others by which we can made light weight wheel hub with adequate properties so, it will survive against different load condition with higher factor of safety.*

Existing Material : AISI 1026, Al alloy

Proposed Material : CARBON STEEL **AISI 1045 or Al 2014**

1. Design and Material Optimization of Ladder Chassis instead of MS

**ABSTRACT**

*Chassis is the foremost component of an automobile that acts as the frame to support the vehicle body. Hence the frame ought to be very rigid and robust enough to resist shocks vibrations and stresses acting on a moving vehicle. Steel in its numerous forms is commonly used material for producing chassis and overtime aluminium has acquired its use. However, in this study traditional materials are replaced with ultra light weight carbon fibre materials. High strength and low weight of carbon fibres makes it ideal for manufacturing automotive chassis. This paper depicts the modal and static structural analysis of TATA 407 fire truck chassis frame for steel as well as carbon fibres. From the analyzed results, stress, strain and total deformation values were compared for both the materials. Since it is easy to analyze structural systems by finite element method, the chassis is modified using PRO-E and the Finite Element Analysis is performed on ANSYS workbench*

Existing Material : CARBON STEEL

Proposed Material : PLA

1. **FINITE ELEMENT MODAL AND VIBRATION MODE ANALYSIS OF AUTOMOTIVE CRANK SHAFT**

**ABSTRACT**

*An attempt is made in this paper to study the Static analysis on a crankshaft from a 4 cylinder diesel Engine. The modelling of the crankshaft is created using CREO Software. Finite element analysis (FEA) is performed to obtain the variation of stress at critical locations of the crank shaft using the ANSYS software and applying the boundary conditions. Then the results are drawn Von-misses stress induced in the crankshaft is compared with three different materials and optimization of crank shaft by vibration mode analysis.*

*The stress analysis and modal analysis of a 4-cylinder crankshaft are discussed using finite element method in this paper. Three-dimension models diesel engine crankshaft and crank throw were created using CREO software .The finite element analysis (FEM) software ANSYS was used .*

|  |  |  |
| --- | --- | --- |
| Existing Material | **45CrMg** | **Magnesium alloy** |

Proposed Material : **En-8D**

1. **Design and Analysis of Automotive Bumper Using Composite Materials**

*Automobile bumper subsystem is a frontal and rear structure of the vehicle that has the purpose of energy absorption during low velocity impact. The bumper beam is the main structure for absorbing the energy of collisions. Automotive bumper beam is one of the key system in car. Bumper beam design to prevent or reduce physical damage to front or rear end of the Motor vehicle in collision conditions they protect the hood, trunk, grill, fuel, exhaust and cooling system as well as safety related equipment such as the parking light, head lamp and tail light etc. A good design of the car bumper must provide safety for passengers and should have low weight.*

Existing Material : CARBON STEEL

Proposed Materials : CARBON FIBRE **& E- Glass Epoxy Fibre**

1. **FINITE ELEMENT ANALYSIS AND OPTIMIZATION OF CONNECTING ROD USING ALFASIC COMPOSITES**

**ABSTARCT**

*Connecting rod is the intermediate link between the piston and the crank. And is responsible to transmit the push and pull from the piston pin to crank pin, thus converting the reciprocating motion of the piston to rotary motion of the crank. Generally connecting rods are manufactured using carbon steel and in recent days aluminum alloys are finding its application in connecting rod. In this work connecting rod is replaced by aluminum based composite material reinforced with silicon carbide and fly ash. And it also describes the modeling and analysis of connecting rod.FEA analysis was carried out by considering two materials. The parameters like von miss stress, von misses strain and displacement was obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of 6 % of weight.*

* *FEA analysis was carried out by considering two materials. The parameters like von miss stress, von misses strain and displacements were obtained from ANSYS software. Compared to the former material the new material found to have less weight and better stiffness. It resulted in reduction of weight and reduction in displacement*

1. **Design and fabrication of Power generation using Shock Absorber with Buckling Analysis of Shock Absorber Using Ansys**

**ABSTRACT**

In this project, we Design and fabricate a power generator using shock absorber. These can be done by adding a rack and pinion type of gear with a generator attached to the combination of the gears mentioned above. These are some of the sample calculations with an average weight of 2 person. All the basic components related to the topic are elaborated in the write-up. The further research on the same is in progress.

In this project a shock absorber is designed and a 3D model is created using CREO. Buckling analysis and Structural analysis is done on the shock absorber by varying material for spring, Structural Steel and Titanium alloy. The analysis is done by considering loads, bike with 2 persons. Structural analysis is done to validate the strength and displacements for different materials. Comparison is done for two materials to verify best material for spring in Shock absorber. Modeling is done in CREO and analysis is done in ANSYS

Existing Material : ASTM A227

|  |
| --- |
| Proposed Material : ASTM A302 |
|

**FINITE ELEMENT ANALYSIS AND OPTIMIZATION OF ENGINE MOUNT ANALYSIS USING ANSYS**

**Abstract.**

In this paper, the capabilities of three different material analytical models to measured Structural properties of elastomeric engine mounts of a passenger car are investigated. ANSYS is used in identifying the mechanical characteristics of each material by static structural analysis. Here linear and orthotropic material properties are considered for Evaluation. The frequency response functions of updated models are compared In Ansys by modal analysis.

FEA analysis was carried out by considering three materials. The parameters like von miss stress, von misses strain and displacements were obtained by static structural analysis from ANSYS software. And frequency obtained by vibration mode analysis using modal analysis in ANSYS Compared to the former material the new material found to have better stiffness and arresting vibration. It resulted in reduction of vibration and increasing stability.

Materials - POLYURETHANE, NEOPRENE, NATURAL RUBBER

**Thermal Analysis and material Optimization of Processor Cooling Fins**

**Abstract**

In this study, thermal analysis of electronic processor fins is proposed and an effort is made to decrease the maximum temperature in processor by employing circular fins which aid in rapid heat removal to the surroundings for ensuring the optimal working of the processor. Removal of heat generated in the processor gets augmented by the application of fins to it. Comparative study is presented by selecting particular material for electronic processor, rectangular in shape, enclosed by a casing with fins are attached. Modelling and Analysis is carried out using the Finite Element Method (FEM) based software, ANSYS. Heat flows out from the processor to the surrounding through the casing and then to fins attached to it. The results report the temperature distribution and heat flux contour for variation in materials. Conclusion is drawn from the results pertaining using the appropriate material to be used to optimize the maximum temperature in processor. Results show that ANSYS can be used effectively and efficiently to solve the challenge of heat transfer.

**Materials: AL6063, AL7075 – T6, GREY CAST IRON, STAINLESS STEEL**

**MODELING and ANALYSIS OF COMPOSITE LEAF SPRING USING FEA**

**ABSTRACT**

*The Automobile Industry has shown increase interest for replacement of steel leaf spring with that of composite leaf spring, since the composite material has high strength to weight ratio, good corrosion resistance and tailor-able properties. The paper describes static analysis of steel leaf spring and laminated composite Multi leaf spring. The objective is to compare the load carrying capacity, stiffness and weight savings of composite leaf spring with that of steel leaf spring. The dimensions of an existing conventional steel leaf spring of a Light design calculations. Static Analysis of 3-D model of conventional leaf spring is performed using ANSYS. Same dimensions are used in composite multi leaf spring using CARBON FIBRE or sandwich composites STEEL NYLON STEEEL unidirectional laminates. The load carrying capacity, and weight of composite leaf spring are compared with that of steel leaf spring. The design constraints are stresses and deflection.*

( The most commonly used **grades** of **spring steel** are 55 Si 7, 60 Si Cr 7, 50 Cr V4.

Alloy Steel 5160, also sold as AISI 5160 )

Existing Material for Leaf Spring = **Alloy Steel 5160**

Proposed Material for Leaf Spring =

Sandwich composite = STEEL, ACRYLIC, POLYCARBONATE

**MODELING, ANALYSIS AND MATERIAL OPTIMIZATION OF BUS BODY STRUCTURE BY STATIC ANALYSIS**

# Abstract:

Transports are the preeminent method of road transportation. The look of the vehicle body depends mainly on driving the execution requirement underneath varying sorts of stacking and managing conditions separated from those of the street conditions. The model investigation moreover, static analysis of an enunciated urban transport chassis, was completed with the FEM. Static analysis was carried out for the bus body made up of STEEL but CHASSIS made up of **Magnesium Alloy** AZ91**, Aluminum 6061-T6 and** SPFH540to determine the equivalent stress and deformation on considering the self weight of the materials. From the results obtained it can be concluded that which material was found to be better material compared to other materials. In this paper, three kinds of materials (**Magnesium Alloy** AZ91**, Aluminum 6061-T6 and** SPFH540) replacement along with section design optimization strategy is proposed to develop a lightweight chassis structure to satisfy the safety while reducing weight. Using Ansys static structure, the design optimization problem is solved; comparing the results of each step, structure of the base flame is optimized for lightweight.

The design model of the bus body is generated in CREO and later foreign to ANSYS for analysis. We are conducting above analysis for the existing design Best of the result we will consider for the chassis design.

**Design and Computational Analysis of Leading-Edge Jet Impingement Cooling With Different Conical Converging Hole For Blade Cooling Using CFD**

**ABSTRACT**

Leading edge of the gas turbine blades is subjected to highest temperature. Jet impingement cooling is an efficient technique for leading edge blade cooling. In this present study, a single baseline cylindrical hole and the proposed converging conical hole is numerically investigated for convective heat transfer using CFD. The converging angle of the conical hole is 10°, 20°, 30° and the hole diameter is 2.15 cm and the study is carried out at Reynolds number ranging from 11000 to 50000 for a jet impinging length of R/2. The target surface is maintained at constant heat flux of 10000 W/m2. Steady-state simulations are performed using Reynolds Averaged Navier Stokes equation along with kω-SST turbulence model for closure. Based on the Nusselt number and temperature distribution, converging conical hole is observed to give a better heat transfer thereby cooling the blades effectively.

Around 186 % increase in Nu and 13% decrease in surface temperature of the concave surface is observed at jet impingement point for Re = 23000.

**DESIGN AND FABRICATION OF WORK HOLDING DEVICE USING COMPRESSED AIR FOR DRILLING MACHINE**

**SYNOPSIS**

Work holding and releasing is the most essential act to carry out machining. These are commonly used in the machine shop to operate the machine vices by compressed air. Normally machine vices are operated by rotating the screw threaded shaft by lever. In this system compressed air is used to operate the machine vice. The movable jaw was moved by compressed air and the work was hold.To hold the job in proper position. To release the job quickly. To hold the job rigidly. To prevent vibration of the job while the machining is carried out.

The objective of this project work is to design a drill jig for drilling machine to radial drill in a cylindrical part. 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.

**MODELING AND FABRICATION OF COMPOSITE MATERIAL DRIVE SHAFT IN TWO-WHEELER**

**ABSTRACT:**

Drive shaft is a mechanical instrument which is used to transfer the rotary motion to the back wheels. This rotary motion is used to run the rear wheels. the details of propeller shaft to substitute its material with composite material and suitability of material is analyzed by evaluating and comparing stress distribution & deformation within the shaft to replace the steel drive shaft with a piece of composite materials with the help of material properties. This was achieved by reducing the weight of the Propeller shaft with the use of different composite materials. The Propeller shaft for determining the dimensions, which were then used for creating a model in CREO. And tested in ANSYS for optimization of design or material check and providing a best material.

This project is developed for the rotate with back wheel of two-wheeler using propeller shaft. Usually in two wheelers chain and sprocket method is used to drive the back wheel. But this project composite material of propeller shaft is used to connect back wheels. These avoid the usage of chain and sprocket method.