Air Dynamics of roof ventilator turbine for wind energy harvest using CFD

Abstract:

Harvesting energy using roof turbine ventilator and electroactive material has been investigated to verify its performance. Since electric power gained from a single piece of regular size is usually small, auxiliary device to vibrate multiple pieces of electroactive materials in order to harvest more power is required. In this paper, an attempt of using the developed nozzle wind collector associated with the popular roof turbine ventilator employed with gear mechanism to impact and vibrate a group of electroactive material to generate electricity is proposed. Number of blade and blade angle of the roof turbine ventilator are influential to the effectiveness of wind collection. Also, number of electroactive material employed on the turbine ventilator under the wind speed in environment eventually determines the efficiency of wind harvest. A simple model is derived to estimate the minimum driving force from the wind power that needs to overcome the inertia of the turbine ventilator mechanism and the electromechanical energy conversion of electroactive materials. Wind drag force is calculated by using CFD is assumed to provide such driving force. Various combinations of the blade angle, number of blade and electroactive material actuators are investigated in simulations. Optimum design concerning the environment wind resource and configuration of turbine ventilator is discussed. According to several case studies, a few of design trends is addressed for better efficiency of energy harvest. Since multiple electroactive materials are employed, circuitry design with parallel input sources is implemented to sum up the current and integrate the power.

Static structural analysis of roof ventilator turbine blades using ANSYS

The current study examines the efficiency of roof ventilator turbine blades made of three distinct materials: aluminium, zinc, and high-density polyethylene. (HDPE). This will help the user select the right roof ventilator turbine blade material and understand the turbine blade's load-carrying capacity under varied wind loads. Prior works in the field of this study have been thoroughly reviewed in this study. Roof ventilator turbine blades can be studied using ANSYS software to analyse material variations and wind loads affect their structural performance. The performance of aluminium, zinc, and high-density polyethylene (HDPE) roof ventilator turbine blades is examined and contrasted. Roof ventilator turbine blades with aluminium material show less deformation and the same elastic strain for the applied wind load compared to the other two materials. But, roof ventilator turbine blade with zinc material gives higher Equivalent Von-Misses stress for the applied wind load when compared with other two roof ventilator turbine materials.

Design of a 24V Electrical System Battery Pack for Electric Motorcycle

Abstract:

Abstract:

In many industrial electric motorcycle battery packs on the market with different voltages and different charging communication systems. The advantage of the results of the battery pack design in this study is that it has the same dimensions and weight as battery packs on the market but has twice the capacity of 40 Ah so that electric motorbikes can travel farther than those in circulation.

The advent of electric vehicles, which run on electricity rather than fossil fuels, was the big breakthrough. There have been many investigations and appreciated structures as well as outcomes for wirelessly powering electric automobiles, both passively and rapidly. This might result in a more rapid acceptance of electric vehicles globally, as it is an emerging technology in automotive manufacturing that has not yet been widely used.

This paper outlines the design, development and implementation of this system. To solve the problem of battery overheating during charging while plugged in, as well as to extend the vehicle's battery life, the suggested model is employed. Batteries temperature handling is the primary concern for EVs due to the increased heat losses while plugged-in charging, which in turn reduces the battery's lifespan.

There is a considerable decrease in thermal output while using composite Battery Pack case in air cooling while charging. The proposed system's primary objective is to provide heat transfer using inductive coupling as its basis.

PROPOSED: Composite Material 3D print or Natural Fiber Composites

DESIGN AND MATERIAL OPTIMIZATION OF BLOW MOULDING DIE FOR PET BOTTLE

ABSTRACT: - Blow moulding is the process of forming a molten tube (preform) of a thermos plastic material and placing the preform with in a mould cavity and filling the tube with compressed air to get the shape of cavity and cools the part before removing from the mould. Blow moulding is used to produce hollow objects like PET (POLY ETHYLENE TERE PHTHALATE) bottles or HDPE (HIGH DENSITY POLY ETHYLENE) bottles for storing of water, oil, soft drinks, for house hold use and chemicals in pharmaceuticals use etc.

The main aim of the project is to design and analysis of blow moulding die. The materials used for analysis of die are EN31, Mild Steel and Aluminium. In this will analyse the structural analysis for finding the best material of die by using ANSYS.

Designing of blow moulding die is done with the help of CREO software. The mould is prepared first by modelling the bottle and splitting the core and cavity then converting the core of the mould into STEP or IGES file and importing the file into “ANSYS” software for analysis and finding the best results of die.

DESIGN AND CRASH ANALYSIS OF CAR BODY STRUCTURE USING EXPLCIT DYNAMICS

ABSTRACT

The performance of the vehicle can be optimized by various means, one such method is by reducing the weight to increase its performance. With the recent development in the field of composites it is evident that composite materials can be the best alternative material to reduce the weight but we must also be careful to maintain its strength. The research and development over Composite materials have benefited greatly as they have higher strength and lower weight, more recyclable. In this thesis, a car body structure is designed using CREO software and a comparative Computer-aided engineering analysis for its crash worthiness is done using ANSYS workbench. The main advantage of CAE is the car body structure can be tested in various fields for optimum performance and durability of the component without actual manufacturing of the original component. The car body structure is implemented with three different materials such as AA7075 T651, Stainless steel, T300 carbon fibre composite and analysed of its crash worthiness The weight of the car body is being optimized by optimization technique for mass reduction without compromising the stiffness and durability of the structure. The CAE analysis and weight reductions are done in ANSYS workbench software. The results will be compared and discussed with the equivalent stress and total deformation value to find out the better materials that reduces weight and withstands stiffness and durability

Design, Analysis and Topology of crane structure via buckling analysis

Abstract—

Generally structural components are subjected to Complex loadings in their working life. These structure must be capable of taking the desired load. Buckling is one of the important parameter which needs to be considered separately as it may cause the complete structure to fail. Plate buckling causes much problem in Bridge girder of EOT Crane; designers are increasing plate thickness of girder to avoid plate buckling which further increases the weight & cost of crane. But instead of increasing the plate thickness of compressed plate, its stability can be increased four times more by adding a suitable stiffener section along longitudinal direction which bisect the plate. Detail theory & example of plate buckling without stiffener & with stiffener is discussed. Manual & finite element analysis report for plate buckling is presented to reduce overall weight & cost of bridge girder of EOT Crane. Results obtained from finite element analysis, Manual calculations and Experimental analysis are compared & presented in table.

Design, Modelling, Simulation, and evaluation of an Electric Scooter Energy Consumption Model

ABSTRACT

These motorcycles have been known as a source of air pollution. Therefore, shifting the motorcycle to electric-driven based technology is inevitable. However, it is quite challenging as the residents prefer higher performance (in terms of power and speed) of the electric model which is unavailable in the market. Unfortunately, the demand for higher performance models creates other problems, such as the requirement for a bigger battery. This is because there is a range of anxiety phenomena in-vehicle usage. However, the capability to accurately estimate the electric motorcycle's range does not exist. Therefore, this paper focuses on how to develop an electric scooter model to simulate its performances, especially its range estimation. The modeling approach was the use of an electric scooter with bumper model developed in CREO environment. Based on the dimensions and targeted performance, the developed model was simulated in ANSYS to determine its Explicit Dynamics. It is then validate using a fabrication on-road conditions. it can be concluded the developed model is valid and can be used as a basis for the next development of any electric motorcycle.

E-Scooter

The commercial Xiaomi M365 was chosen as a template for the e-scooter model.

Structural, Material Optimisation and Drop Analysis of Solid & Hollow Tension Helical Spring used in LIFT By FEA

Abstract

Helical springs play a pivotal role in various engineering applications, ensuring controlled motion and stability in mechanical systems. This study presents a comprehensive analysis of solid and hollow tension helical springs using advanced numerical simulations conducted in ANSYS Software.

The investigation encompasses geometric modelling, material characterization, finite element analysis (FEA) setup, parameter monitoring and result analysis.

Simulation and Analysis of Venturi Tubes for Use in Duct Wind Turbine Power Plants

Abstract:

This paper discusses the venturi tube used as a duct wind turbine power plant. The design of the power plant using duct wind turbine method is another alternative in exploiting the potential of wind energy as an environmentally friendly power plant. Before the design of the electric generator is carried out, a simulation and analysis of the geometry of the venturi tube is carried out. The simulations is to find the optimum design for wind speed and energy generated. In this paper, four designs are made, namely a straight design, a bent design on the input channel, a bent design on the input and output channels, and a bent design without sharp corners. To help analyze the designs, the Ansys Fluent is used. The simulation results show that the highest airflow velocity is located in the throat channel and the average velocity ratio between the input channel and the venturi channel is around 5x. Venturi Tubes with Bent Shape on the input and output channels without sharp angles can reduce turbulence in all parts of the channel.

Cooling Performance analysis of an EV lithium iron phosphate battery pack and ceramic coating

Abstract:

Lithium-ion battery packs comprise a significant share of an electric vehicle's cost, especially for low-cost variants such as those used for public transportation (e.g. jeepneys in the Philippines). These can easily occupy 40% of the vehicle's cost. In this regard, it is very important to ensure the longevity of the battery cells. Lithium-ion cells which are poorly-managed thermally risk having to be replaced sooner than their intended usable life. Thus, proper attention must be given to the design of the battery packs to allow effective and efficient cooling. This study performed a cooling simulation on prismatic lithium iron phosphate cells using ANSYS Workbench. The simulation looked into (1) the effect of the layout of the cells; (2) the thickness of cooling fins; and (3) the temperature and flow rate of the cooling fluid to the thermal profile of the battery pack. The simulations successfully showed that the recommended operating temperature can be obtained even with a compact layout and using thin cooling fins. Practical considerations such as space constraints on the vehicle and weight of the battery pack outweigh the technical benefits of improved heat transfer efficiency using a more spaced-out battery layout and thicker cooling fins.

Finite element analysis of Leaf Spring Systems with Hybrid Material Arrangement

Abstract:

Deformations that may occur on multi-level parabolic leaf spring systems due to external effects such as vibration and force should be analyzed before the system is designed. In this work, ANSYS-based deformation analyzes were carried out for 10-7 parabolic leaf spring systems designed by using stainless steel and titanium alloy materials in hybrid structure. From the deformation analyzes performed under the force values of 5000 N, 10000 N and 15000 N, it was observed that the highest total deformation values were obtained in the Group 6 structure as 1.0042 e-4 m, 2.0084 e-4 m and 3.0125 e-4 m, respectively. On the other hand, among the leaf spring system structures with different material arrangements, it was observed that the lowest total deformation values were obtained in the Group 5 structure as 5.1221 e-5 m, 1.0244 e-4 m ve 1.5366 e-4 m, respectively.

Design Optimization and cooling performance of Water-Cooling jacket Based on ANSYS Workbench

Abstract:

Water cooling jacket has been widely used in petroleum, chemical, food, energy and other industries. In the exhaust gas treatment device of the chemical experimental platform, bulge phenomenon was found in the inner cylinder. This paper completed the analysis and calculation by ANSYS Workbench and completed the upgrade and optimization of the water-cooling jacket structure. At present, the optimization mechanism has been upgraded and reformed, and passed the test and inspection, and provided a number of engineering experiment data, for the reference of relevant technical designers.

Static structural material comparison and Vibration Analysis of Manned Lunar Rover Frame Based on Ansys

Abstract:

As lunar exploration projects continue to progress, increasing expectations are being placed upon the efficiency and detection range of lunar surface exploration. For the next stage of lunar exploration projects, a breakthrough technology regarding the utilization of manned lunar rovers is requisite. In view of the driving environment of the lunar characterized by low gravity, uneven surface, and complexity, it is paramount to ensure the safety and stability of manned lunar vehicles in operation. Therefore, the static structure should be checked, and the environment for the frame of the manned lunar vehicle should be set to conduct vibration analysis for the lunar surface. Utilizing the finite element method in conjunction with existing data relating to manned lunar rovers, a finite element analysis model is established. Besides, the Ansys software is utilized to conduct static structural analysis, non-rigid body modal analysis, and vibration analysis of lunar sea driving when the manned lunar rover is fully loaded. The results indicate that the strength and stiffness of the manned lunar rover frame meet the requirements, and the maximum stress and maximum deformation caused by vibration are within the acceptable limits. This study provides a theoretical reference for further research endeavors involving the manned lunar rover.