International Journal of Scientific Engineering and Technology ISSN : 2277-1581
Volume No. 12, Special Issue No. 1 29 Dec. 2023

Modeling and Analysis of Two Stroke Piston of Alloy using FEA

[1]kumar Niraj, [2]Ramnarayan Sahu, [3]Yogesh Mishra
[1]Research Scholar MTech, (Machine Design and Robotics), NIIST, Bhopal
[2]Assocoate Professor Department of Mechanical Engineering, NIIST, Bhopal
3Professor & Head Department of Mechanical Engineering, NIIST, Bhopal
Email: k.nirajvoith@gmail.com

Abstract: The purpose of this final project is to design and calculate a piston for a large two-stroke diesel engine. They are used in railroad, marine or stationary services. They are very big and heavy; all the parts have to be very well studied and calculated. Modelling process has different parts but it has to be understood what we want to achieve, the optimal shape of the piston to obtain maximum performance. The piston itself has four different main parts that should be taken into account: the crown, piston rings, hubs and the skirt. The crown takes a very important part in the whole process. It is the part that supports most of the loads and heat. As a consequence, is the one that suffers the hardest deformation. Convection, temperature, heat flus and total deformation analysis were performed on piston using two Al alloy materials. The above analysis is carried out with the help of using SOLIDWORKS and ANSYS. In order to achieve to see different results it is interesting to simulate the piston crown with heat exchange. It was seen, later on, that the correct way to simulate the piston was introducing convection from the gases to the piston. The value of the mean of heat convection was fixed to 268,35 W/m^2. The value obtained was 268.35 w/m^2·^{o}C. Temperature distribution is very important to see which of the parts are sumited to a higher temperature. If some is larger than the maximum the material can stand. The first simulations were made with the temperature of the piston crown fixed. The value of the top of the piston crown was 650^{o}C. The piston cannot work in these conditions. The pipe and the convection of the piston itself could not absorb that heat.

Keywords: Piston, Solid work, ANSYS, IC Engine, Optimization, Analyses, 2 Stroke Diesel engine

1. Introduction:

Automobile components are in great demand these days because of increased use of automobiles. The increased demand of components is due to improved performance and reduced cost. The production of low weight component will be one of the major activities of the R&D Labs. R&D and testing engineers should develop critical components in shortest possible time to minimize launch time for new products. The necessitate understanding of new technologies and quick absorption in the development of new products.

A piston is a component of reciprocating engines, reciprocating pumps, gas compressors and pneumatic cylinders, among other similar mechanisms. It is the moving component that is contained by a cylinder and is made gastight by piston rings. In an engine, its purpose is to transfer force from expanding gas in the cylinder to the crankshaft via a piston rod and/or connecting rod. In a pump, the function is reversed and force is transferred from the crankshaft to the piston for the purpose of compressing or ejecting the fluid in the cylinder. In some engines, the piston also acts as a valve by covering and uncovering ports in the cylinder wall. The piston used in internal combustion engine is usually of trunk type. Such pistons are open at one end. Trunk piston consists of:

1. Head or crown to carry the cylinder pressure
2. Skirt to act as a bearing for connecting rod side thrust
3. A piston pin to connect it to the connecting rod
4. Piston ring to seal the cylinder

Piston is a cylindrical component fitted into the cylinder forming the moving boundary of the combustion system. It fits perfectly into cylinder providing gas tight space with the piston rings and the lubricant. It forms the first link in transmitting the gas forces to the output shaft. Piston is a simple machine element which forms the combustion chamber as well as it is the one which receive the combustion thrust which is to transferred to the crankshaft via connecting rod. Piston transfers the gas load from cylinder to the connecting rod which in turn transfers the load to the crankshaft in order to obtain mechanical energy. It will be able to transfer the load to connecting rod only when it is able to sustain the load that is supplied on it. Apart from the gas load transfer, it has the conduct to necessary amount of heat to the cooling medium in order to avoid material melting.

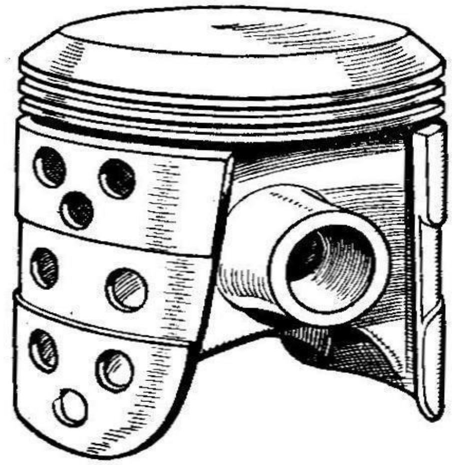

Fig. 1. Slipper Piston

At the same time, it has to retain enough amount of heat to sustain combustion and also to provide good thermal efficiency. Therefore, piston should be capable to sustain both thermal and structural distortions that are induced due to gas force and combustion energy.

A slipper piston is a piston for a petrol engine that has been reduced in size and weight as much as possible. In the extreme case, they are reduced to the piston crown, support for the piston rings, and just enough of the piston skirt remaining to leave two lands so as to stop the piston rocking in the bore. The sides of the piston skirt around the gudgeon pin are reduced away from the cylinder wall. The purpose is mostly to reduce the reciprocating mass, thus making it easier to balance the engine and so permit high speeds. A secondary benefit may be some reduction in friction with the cylinder wall, since the area of the skirt, which slides up and down in the cylinder is reduced by half. However, most friction is due to the piston rings, which are the parts which actually fit the tightest in the bore and the bearing surfaces of the wrist pin, the benefit is reduced. The slipper piston is as shown in Fig. 1..

2. Literature Review:

An exhaustive literature review has been done on simulation analysis of the piston related problem in this chapter. On the basis of literature review available in the field of simulation on piston, the contribution of various authors has been incorporated in this research work.

Azzawiet. al. (2023) Internal combustion engines are thermal engines in which a fuel-air mixture undergoes combustion within the combustion chamber. The high-pressure and hightemperature gases generated during this process expand, exerting force upon the pistons and converting chemical energy into mechanical energy. This retraction movement is transformed into a rotational motion that generates the necessary torque to rotate the wheels via the crankshaft. Common examples of these engines include four-stroke and two-stroke piston engines, gas turbines, jet engines, and most rocket engines. This study aims to develop a model of the piston-connecting rod crank shaft system and investigate the thermal and mechanical stresses resulting from its operation. Utilizing the finite element method in conjunction with ANSYS software, the simulation of thermal and mechanical stresses is carried out, while random vibrations are applied to represent potential imperfections and errors during operation. The SOLIDWORKS program is employed for the design and illustration of the system. Simulation results indicate that maximum vertical stress and shear stress are primarily distributed along the piston heads and connecting rods.[1]

Mithilesh Kumar Sahuet. al. (2022) The exhaust gas from an internal combustion engine carries away about 30% of the heat of combustion. The energy available in the exit stream of many energy conversion devices goes to waste, if not utilized properly. The major technical constraint that prevents the successful implementation of waste heat recovery is due to its intermittent and time mismatched demand and availability of energy. In this regard, the present work focused on investigating the thermal potential available at the diesel engine exhaust and its utilization for various applications. In this work, experimental investigations have been performed to identify the thermal potential available at diesel engine exhaust for different load conditions. Now to recover the available thermal potential, a double pipe heat exchanger of counter flow type has been modeled in ANSYS Design Modular. The same has been numerically simulated in ANSYS workbench environment to get the outlet temperatures of hot and cold fluids. The results have been evaluated for heat exchanger, operating at engine exhaust temperature of 478 K and cold water temperature of 298 K. Simulation results shows cold outlet temperature as 435.11 K.[2]

PagadalaSiddiraju and KoppulaVenkateswarareddy (2021) In the present work describes the stress distribution and thermal stresses of Five different materials for piston by using finite element method (FEM), testing of mechanical properties. The parameters used for the simulation are operating gas pressure, temperature and material properties of piston. The specifications used for this study of these pistons belong to four stroke single cylinder engine of Pulsar 220cc motorcycle. The results predict the maximum stress and critical region on the different materials piston using FEA. Design by using catia v5 software and analysis by using Ansys software in Ansys 16.0 Static and thermal analysis is performed. The suitable material is selected based on results of structural and thermal analysis on these Al-sic graphite, A7075, A6082, A4032, AL-ghy 1250 materials.[3]

Siddharth Boon (2020) The reliability of any system, during which the linear displacement of a piston is regenerate into the rotation of a power transmission shaft, strongly depends on the reliability of the crankshaft. The crankshaft is that the vital element and any damage occurring to the crankshaft might put the system out of order. Crankshaft is massive volume production component with a complex geometry within the ICE. This converts the reciprocating displacement of the piston into a rotation of the crank. The crank shaft takes the power from piston that is generated because of combustion method within the combustion chamber of the cylinder. Throughout the power transmission method, the load acts at a specific crank angle to the max and thus the connecting rod is analyzed for the stress developed, due to load conditions and the changes mentioned. The existing works design first the 3D model of the engine parts are built in the software ‘CATIA V5’ and are then transferred to ‘ANSYS’. The analysis of crank throw distortion and stress provides a conceptual support to enhance the design by weigh reduction. The proposed research works the 3D model of crankshaft system, obtained from CATIA V5 software is analyzed in ANSYS to assess the motion and loads acting on the crankshaft.[4]

S P Venkatasenet. al. (2020) The design geometry of exhaust manifold plays a vital role in smooth combustion and emission reduction of the spark ignition engine. In this work, by analysing and comparing the exhaust gas back pressures and its velocities of different types of manifold models chosen at different operating load conditions of the engine, the best exhaust manifold of model 5 has been found and recommended to use in the multi-cylinder engine to control the engine emission and protect the environment. The analysis is done with a virtual model of manifold. Modelling and analysis of exhaust manifold are carried out by CATIA v5 and ANSYS software.[5]

NiteshVaishnav and Satya Narayan Dubey (2019) In order to cool the engine, fins are provided on the cylinder to increase the rate of heat transfer. Present research work is based on investigations on fin thickness and improved material selection for an internal combustion’s engine assembly. For this purpose, simulation approach was adopted in which dimensions of fins from a standard engine were taken, model is prepared and analysis of model was conducted on a wellknown simulation software ANSYS 15.0. The main purpose of analysis thermal properties, three criteria namely, heat flux; thermal gradient and average temperature were selected. The varying parameters were fin thickness, and fin materials. Heat flux show varied results, and therefore on this criterion rankings of different materials with fin thicknesses were investigated. Results show the suitability of Al alloy 204 with fin thickness of 3 mm for the purpose.[6]

Vishal Sapkal and Kamal Ukey (2018) Cylinder head is a critical part of an I C engines cylinder head is used to seal the working ends of the cylinder and accommodates combustion chamber in its cavity, spark plug and valves. The heat generated in combustion chamber is highly dynamic and allows very little time (few micro seconds) to transfer the heat if not distributed will lead to squeezing of piston due to overheating. Hence an effective waste heat distribution through cylinder head plays a very important role in smooth function of I C engine. Heat Transfer through cylinder head consists of conduction through walls and convective heat transfer due to surrounding air flow. As the shape of cylinder head is complex and temperature within the combustion chamber is still unknown. Conventional methods of evaluating heat transfer are very complex this project aims at evaluating heat transfer through cylinder head using finite element analysis as well as the structural analysis. Geometrical models of Cylinder head with and without fins are developed in Auto CAD software. Thus, developed models are exported to ANSYS software, and finite element model for thermal analysis done in ANSYS. Effect of fins on heat transfer through cylinder is evaluated. The proficiency of any automobile engine is deals with the structural strength of its cylinder and cylinder head. Cylinder and cylinder head are most important parts of an engine because the piston moving inside the cylinder, so friction between cylinder wall and piston is very higher and due to this the mechanical load or fatigue load acting on the cylinder. So that structure of cylinder should be stronger. 3Dmodel of cylinder and cylinder head were created using Pro/Engineer software and ANSYS was used to analyze the thermal and structural analysis. So finally design considerations, material specifications, failure analysis, these all are reviewed successfully over here.[7]

Novi ShobiHendret. al. (2018) Frigate Class Vessel is one of the flagships of the Indonesian Navy. The average air temperature after repowering in the engine room is ranged from 60°C-65°C, while the maximum air temperature recommended based on the Lloyds Register is below 45°C. This condition affects the performance of equipment and machine operators in it. The in and out air circulation of the engine room is not sufficient for the air required. The and out Duct design is designed to keep the room temperature following standard requirements specified. This can be known by simulation using Ansys Computational Fluid Dynamics (CFD). A total of 24 outlet ducts of the ducting design was obtained by conducting the simulation using Ansys CFD. It took two blowers to supply engine room and two engine room suction blowers with an air capacity of 33.876 CFM or equivalent to 57,555.69 m3 /hr and power of 40 HP when the was sailing. However, the capacity and specification of the old blower installed on the operational use were respectively 16,627.32 CFM or equivalent to 28,250 m3 /hr with the power of 15 HP; thus, it could not be used to supply the air needs and to keep the temperature in the engine room in ideal conditions.[8]

2,1Objective:

The piston is one of the most critical components of an engine. Therefore, it must be designed to withstand from damage that is caused due to extreme heat and pressure of combustion process. The value of stress that causes the damages can be determined by using Finite Element Analysis (FEA). Thus, it can reduce the cost and time due to manufacturing the components and at the same time it can increase the quality of product.

The objective of this study is:

i) To calculate the equivalent (Von-Mises) stress.

ii) To calculate the total deformation by considering the gas load.

iii) To optimize the piston model for mass reduction.

iv) To calculate the total heat flux.

3, Methodology:

This method was developed at the time of 1960, especially to analyse the structural, dynamics problems. It is based on the weigh residual method. This method is a beneficial over the difference method as it can handle complex geometries and use arbitraries on irregular shapes [5]. The finite element method (FEM) (its practical application often known as Finite Element Analysis (FEA) is a numerical technique for finding approximate solutions to partial differential equations (PDE) and their systems, as well as (less often) integral equations. In simple terms, FEM has an in-built algorithm which divides very large problems (in terms of complexity) into small elements which can be solved in relation to each other. FEM solves the equations using the Galerkin method with polynomial approximation functions. The solution is obtained by eliminating the spatial derivatives from the partial differential equation. Many problems in engineering and science, including the piston model, involve complicated systems of differential equations which are too difficult to exactly solve due to their complex geometry. Finite element analysis, which has evolved over the past three decades, involves a method of breaking up a continuum into discrete, coupled components that approximate the overall solution. The geometric domain is broken into these nonoverlapping coupled components called elements. These elements are represented by a linear combination of polynomial functions with undetermined coefficients, which form the approximate numerical solution to the governing differential equations. The undetermined coefficients are represented by nodes, which are located on the element. The solutions are found at these nodes using the polynomial functions and prescribed boundary conditions. The linear combination of the assumed algebraic polynomials forms the continuous solution. Thus, the simplified local representations are patched together to form an approximate global solution. Because the local form of the solution needs to be kept simple, accuracy is increased by making the elements as small as possible.

This makes the approximation defined by a larger number of equations, which increases with every increase in the number of elements used. However, it reduces the differential equation into many algebraic equations, which leads to the possibility of

ISSN : 2277-1581
29 Dec. 2023

solving more complicated problems. Although finite element analysis (FEA) was originally done with personally written computer programs to carry out the analysis, there are many commercially available computer programs now which eliminate the need to write an individual code.

4, Result and Discussions:

Results are obtained for static structural analysis and thermal analysis by using ANSYS 14.5 software. In static structural analysis we found total deformation and equivalent (Von-Mises) stress for two Al alloy materials i.e., Al alloy 2618. In thermal analysis we found temperature distribution and total heat flux in Al alloy 2618. Results are shown in following fig. given below:

Convection between piston skirt and cylinder walls is not uniform. It changes along the axis of the piston. Understanding in the top is bigger due to the temperature gradient and in the bottom is lower. In this simulation is assumed the convection coefficient is constant all along the piston. And the value is $h_{c1}=5\cdot10^{-5}$ W/m^2 ·°C and temperature T_{c1}=200°C.

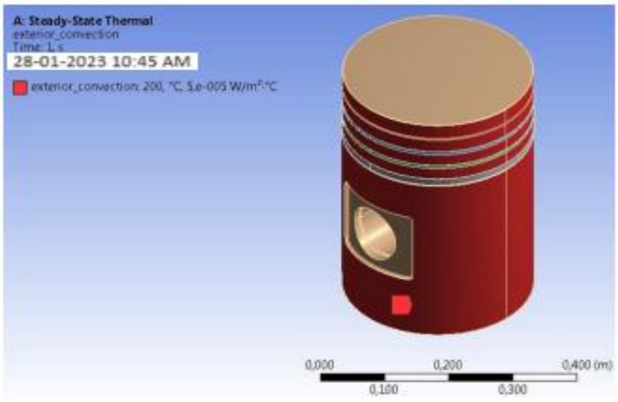

Figure 2 Convection coefficient distribution

Piston rings

The three piston rings used in this piston they should have different convections values and conduction values to be well defined.

In order to simplify the process, it is defined a temperature for each one. The temperatures are the following:

First sealing ring: T_{s1}= 150°C

Second sealing ring: T_{s2}= 140°C

Scraping ring: T_{sc}= 130°C

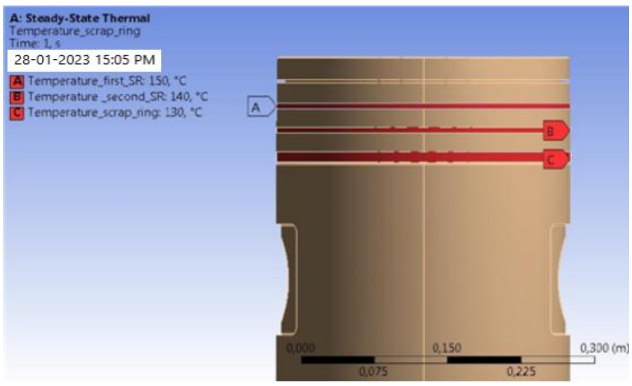

Figure 3 Temperature definition in the piston rings

Piston crown surface

The piston crown surface is one of the most important parts of the piston. Where the combustion takes place and where the maximum pressure is held by the combustion's gases.

As far the pressure is concerned no simplification is made, it is a steady state analysis so to take the worst case the maximum pressure is taken. The value is p = 3 MPa.When comes to the heat exchange between the gases and the piston crown surface, everything becomes a little bit more challenging. First approach made was to fix a temperature in the piston crown surface, afterwards it was seen that this led to incorrect results. In order to correct this mistake, the Woschni's formula has been used to get the proper convection coefficient. Woschni'scorrelation, can be summarized as:

$$h_c(w/m^2\cdot K) = 3{,}26B(m)^{-0{,}2}p(kPa)^{0{,}8}8T(K)^{-0{,}55}w(m/s)^{0{,}8} \quad (29)$$

Being w: average gas velocity, p: instantaneous cylinder pressure, T: instantaneous working-fluid temperature and B: Cylinder diameter. Given real working conditions it is possible to obtain a real curve of the convection in one cycle which corresponds to 720° of crankshaft rotation as shown in figure.

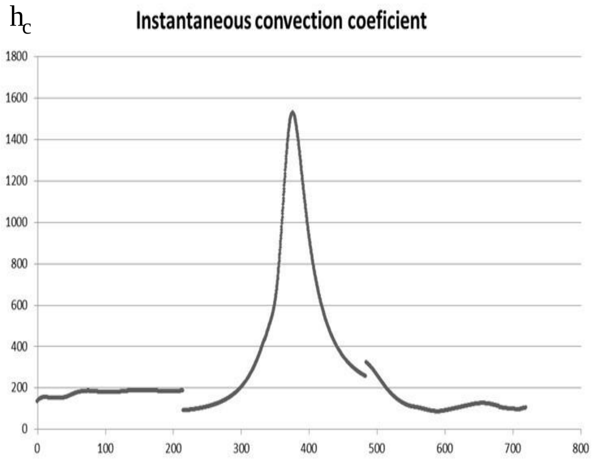

Figure 4 Value of convection coefficient in every position of the crankshaft

This graphic was calculated with the program of W. Mitianiec. "Engine4s", which calculates thermodynamic engine parameters with taking into account nonsteady gas flow in the inlet and outflow pipes. In the program the Woschni model of heat convection coefficient in order to calculate heat flow to the engine walls (cylinder, piston and cylinder head walls). What is interesting for this study is the mean of this value because the study is done in steady conditions. When calculating the obtained value is: 268.35 w/m^2·°C. and the temperature 750°C.

5 Conclusions:

- The piston analysed in this work is a really large piston, used in large two-stroke diesel engines. The pressures and heat applied are very large. Some especial devices are needed in order to maintain the piston in good working conditions.
- This system is not sealed so there is no much problem in designing it. Further ahead it should be studied the amount of flow of the oil that goes through the pipe. Depending of the loads applied in the piston.
- As it can be seen in the simulation the stresses in the piston are not too high. This leads to think that a greater

improvement could be done in the terms of materials usage. Some parts do not need as much material as it is used.

- Simulation showed that deformation at the top of the cylinder was 1mm. Consequently, barrel shape was adopted. The nominal radius at the top of the piston crown surface was reduced by 1 mm.
- This is very important because now there is no problem in the clearances of the piston.
- Piston ovalization was extrapoled from previous designs. Assuming that linear extrapolation was correct. This way piston deformation should compensate normal forces acting against it.

Reference:

- *Azzawi, Mohammed Mousa; Hadi, Auday Shaker; Abdullah and AtheerRaheem, Finite Element Analysis of Crankshaft Stress and Vibration in Internal Combustion Engines Using ANSYS, Mathematical Modelling of Engineering Problems, Jun2023, Vol. 10 Issue 3, p1011-1016. 6p.*
- *Mithilesh Kumar Sahu, Ajeet Kumar Singh and TusharChaudhary, Experimental investigation of thermal potential at diesel engine exhaust and numerical simulation of heat recovery in heat exchangers, Material Today Proceedings, 2022.*
- *PagadalaSiddiraju and KoppulaVenkateswarareddy, Design and analysis of the piston by using five different materials, International Journal of Advance Scientific Research And Engineering Trends, Volume 6 Issue 2 February 2021.*
- *Siddharth, Crankshaft analysis and modeling using ansys software, Institutional Repository of intellectual Contributions, July 2020.*
- *S.P. Venkatesan, S. Ganesan and R. Devraj, J. Hemanandh, Design and analysis of exhaust manifold of the spark ignition engine for emission reduction, International Journal of Ambient Energy, Volume 41, 2020.*
- *NiteshVaishanv and Satya Narayan Dubey, Thermal Analysis of I C Engine Fins using ANSYS Software, International Journal of Modern Engineering & Management Research, Volume 7 Issue 2, June 2019.*
- *Vishal Sapkal and Kamal Ukey, Design and analysis of cylinder head of an engine, International Research Journal of Engineering and Technology (IRJET), Volume: 05 Issue: 05 | May-2018.*
- *Novi ShobiHendri, Ahmadi, Okol S Suharyo and Arica DwiSusanto, The Air Flow Analysis in Engine Rooms at Frigate Class Ship with CFD Approach (Computational Fluids Dynamics), International Journal of Recent Engineering Science (IJRES), Volume 5 Issue 4, 11-18, Jul-Aug 2018.*
- *PathipatiVasu, Dr. M.Sri Rama Murthy, P.S. Amarnadh and S.V. +-+Gopal Krishna, Design and Analysis of IC Engine Piston with Different Materials, International Journal of Research, Volume 05 Issue 07, March 2018*
- *PraveshTediya, Shahnawaz Ansari, CFD Modelling and Analysis of Dual Fuel (Diesel + Methanol) Combustion Engine with various blend grade using ANSYS (Fluent), International Research Journal of Engineering and Technology (IRJET), Volume: 05 Issue: 10 | Oct 2018.*
- *Viswabharathy, N.Jeyakumar, P.kannan and A.Vairamuthu, Design and Analysis of Piston in Internal Combustion Engine Using ANSYS, International Journal of Emerging Technologies in Engineering Research (IJETER), Volume 5, Issue 3, March (2017).*