

Investigation and Optimization of Piston with Different Compositions in Existent Combustion Circumstance

Mr.V. Karthikeyan¹, Vengadesan. S², Kabilan. K. K³, Vijay. S⁴

¹Assistant Professor, ^{2,3,4}UG Scholar,

Department of Mechanical Engineering, Excel Engineering College, Namakkal, Tamil Nadu, India

ABSTRACT

Article Info

Volume 9, Issue 2

Page Number : 255-265

Publication Issue :

March-April-2022

Article History

Accepted : 01 April 2022

Published: 10 April 2022

This Project is to improve the compression ratio in particular in Diesel Engine and reduce heat rejection in cylinders. It is acquired by changing the material property by composition. Thermal barrier coatings have been applied to the internal combustion engine in the combustion chamber for simulating adiabatic changes. It is not only for reduction of heat rejection in-cylinder and thermal fatigue protection of underlying metallic surfaces, but it also for reduction of engine emissions and brake specific fuel consumption. Thermal Barrier Coatings (TBCs) in diesel engines lead to higher power density, fuel efficiency, and multifuel capacity due to higher combustion chamber temperature Using TBC can increase engine power. The application of TBC in the components is to enhance high temperature durability by reducing the heat transfer and lowering temperature of the underlying metal.

Keywords : FEA analysis, Piston, Static analysis, Thermal Barrier coatings, adiabatic change

I. INTRODUCTION

Functionally graded materials are of widespread interest because of their superior properties such as corrosion, erosion and oxidation resistance, high hardness, chemical and thermal stability at cryogenic and high temperatures. These properties make them useful for many applications, including Thermal Barrier Coating (TBC) on metallic substrates used at high temperatures in the fields of aircraft and aerospace, especially for thermal protection of components in gas turbines and diesel engines.

Thermal barrier coatings have been successfully applied to the internal combustion engine, in

particular the combustion chamber in order to simulate adiabatic changes. The objectives are not only for reduced in-cylinder heat rejection and thermal fatigue protection of underlying metallic surfaces, but also for possible reduction of engine emissions and brake specific fuel consumption. The application of TBC reduces the heat loss to the engine cooling-jacket through the surface exposed to the heat transfer such as the cylinder head, liner, piston crown and piston rings. The insulation of the combustion chamber with ceramic coating affects the combustion process and, hence, the performance and exhaust emissions characteristics of the engines improve. On the other hand, the desire of increasing the thermal efficiency or

reduce fuel consumption of engines lead to the adoption of higher compression ratios, in particular for diesel engines, and reduced in-cylinder heat rejection. Both of these factors cause increased mechanical and thermal stresses of materials used in combustion chamber.

The application of TBC to the surfaces of these components enhances high temperature durability by reducing the heat transfer and lowering temperature of the underlying metal. Typical TBCs failure is by spalling of the ceramic top coat from the bond coat. There are many factors that influence the overall performance of coatings and cause spalling of the coating.

However oxidation and thermal mismatch are identified as two major factors influencing the life of the coating system. The coatings are permeable to the atmospheric gases and liquids resulting in the oxidation of the bond coat and spalling of the coating. The functionally graded coatings were used to reduce the mismatch effect. Therefore the thermal expansion and interfacial stresses are an alternative approach to conventional thermal barrier coatings.

Energy conservation and efficiency have always been the quest of engineers concerned with internal combustion engines. The diesel engine generally offers better fuel economy than its counterpart petrol engine. Even the diesel engine rejects about two thirds of the heat energy of the fuel, one-third to the coolant, and one third to the exhaust, leaving only about one-third as useful power output. Theoretically if the heat rejected could be reduced, then the thermal efficiency would be improved, at least up to the limit set by the second law of thermodynamics. Low Heat Rejection engines aim to do this by reducing the heat lost to the coolant.

Thermal Barrier Coatings (TBCs) in diesel engines lead to advantages including higher power density, fuel efficiency, and multifuel capacity due to higher

combustion chamber temperature Using TBC can increase engine power by 8%, decrease the specific fuel consumption by 15-20% and increase the exhaust gas temperature 200K. Although several systems have been used as TBC for different purposes, yttria stabilized zirconia with 7-8 wt. % yttria has received the most attention. High thermo mechanical stability to the maximum service temperature and at last but not least the thermal expansion coefficient (TEC).

The diesel engine with its combustion chamber walls insulated by ceramics is referred to as Low Heat-Rejection (LHR) engine. The LHR engine has been conceived basically to improve fuel economy by eliminating the conventional cooling system and converting part of the increased exhaust.

A large number of studies on performance, structure and durability of the LHR engine have been carried out since Kamo and Bryzik presented a new concept of the LHR engine 041 combined with the turbocompound system. Although promising the results of the investigations have been somewhat mixed. Most have concluded that insulation reduces heat transfer, improves thermal efficiency, and increases energy availability in the exhaust. However contrary to the above expectations some experimental studies have indicated almost no improvement in thermal efficiency and claim that exhaust emissions deteriorated as compared to those of the conventional water-cooled engines.

Plasma spray is the most common method of depositing TBCs for diesel applications. It creates a splat structure with 10-20 % volume fraction of voids and cracks. High porosity of this structure makes it an ideal choice for TBC. Widespread application has been limited by insufficient lifetimes and the cost.

TBCs for diesel engines have generally been accepted to improve engine thermal efficiency and reduce emissions as well as specific fuel consumption because of their ability to provide thermal insulation to the

engine components. The generally known principle that increased operation temperatures in energy conversion systems lead to an increase in efficiency, fuel savings and reduced emissions as particles, carbon monoxides (CO), hydrocarbons (HC) and limited reductions of NO_x emissions have, over many decades, promoted R&D activities in the field of TBCs development. In this project, the main emphasis is placed on the study of thermal behavior of functionally graded coatings obtained by means of using a commercial code, ANSYS on aluminum and steel piston surfaces and the results are verified with numerical and experimental works.

II. MATERIAL COATING

Coating is a covering that is applied to an object. The aim of applying coatings is to improve surface properties of a bulk material usually referred to as a substrate. One can improve appearance, adhesion, wettability, corrosion resistance, wear resistance, scratch resistance, etc.. They may be applied as liquids, gases or solids. Coatings can be measured and tested for proper opacity and film thickness by using a Drawdown card.

NANO COATING:

Nano-coating is a recently developed technology used for coating any kind of material in hard coating and low friction coating both in which coating is done at nano scale that is of the order of 10⁻⁹. The two major types of nano-coating are

1. Physical vapour deposition (PVD)
2. Chemical vapour deposition (CVD)

THE RANGE OF ZIRCONIA THERMAL BARRIERS METCO201B-NS, 201NS, 210,210 NS-1-G,210NS-1,333:

This is recommended for combustion applications. Abrasive wear resistant up to approx. 900°C (1650°F). Coatings resist wetting and corrosive effects of molten metal. Applications: diesel engine pistons, valves, cylinder heads and coatings for casting molds and troughs.

METCO205NS:

Ideally suited for severe thermal barrier applications and offers greater high temperature corrosion resistance against sodium, sulfur and chlorine contaminants compared to 8% yttria zirconia coatings.

METCO143

Coatings exhibit excellent scuff resistance, high temperature hardness, excellent thermal shock properties and resistance to sulfidation, chlorination and sodium hot corrosion. METCO 204B-NS, 204C-NS, 204NS, 204NS-1, 204NS-G, 204NS-AP

A flame-stabilized yttria zirconia that stabilizes when melted during the spray coating process.

EUAMDRY6643:

AMDRY 6643 is a stabilized Yttria Zirconia. The Zirconia is not flame stabilized. Coatings can be used at temperatures up to approximately 1350 °C (2460 °F), depending on operating conditions.

METCO202NS:

A flame stabilized yttria zirconia, designed to produce coatings that are stable at high temperatures. Recommended for resistance to erosion at temperatures above 845°C (1550°F). Applications: thermal barrier coatings in rocket and jet engines.

DESIGN AND MODELING OF PISTON

Generally for modeling, packages like Pro-Engineer, IDEAS will be used. In this analysis the model is created in the ANSYS itself to eliminate the data losses that will occur if standard data exchange formats like IGES, STEPS are used. To create the model the details about the key point locations are taken as the inputs.

To create an area in a place, which is away from the global origin then the working plane has to offset to that plane by selecting a key point on that plane. To create an area in a plane, which is not parallel to the X-Y plane, there is an option to align working plane with required plane.

For attaching two adjacent entities properly glue operation is used. In that line, areas and volumes can be attached to the nearby entities. This glue operation is very useful one while creating volumes by selected

areas and creating areas by selected lines. The user can alter the tolerance limit of the glue operation. The model was fully created using different ANSYS commands like points, lines, areas, volumes, etc.

Steps involved in modelling of piston:

1. Points have been created first with the graphical plot.
2. Lines are created on the consecutive Key points.
3. Area is created by using lines.
4. Solid model is created as shown in fig, extrude about axis command is used for creating model

III. ANSYS EVALUATION

ANSYS is a complete FEA simulation software package developed by ANSYS Inc – USA. It is used by engineers worldwide in virtually all fields of engineering.

- Structural
- Thermal
- Fluid (CFD, Acoustics, and other fluid analyses)
- Low-and High-Frequency Electromagnetic.

PROCEDURE:

Every analysis involves three main steps:

- Pre-processor
- Solver
- post processor

STRUCTURAL ANALYSIS

Structural analysis is probably the most common application of the finite element method. The term structural (or structure) implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

TYPES OF STRUCTURAL ANALYSIS

The seven types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements.

Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements. Structural analyses are available in the ANSYS Multi physics, ANSYS Mechanical, ANSYS Structural, and ANSYS Professional programs only.

STATIC ANALYSIS--Used to determine displacements, stresses, etc. under static loading conditions. Both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

MODAL ANALYSIS--Used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

HARMONIC ANALYSIS--Used to determine the response of a structure to harmonically time- varying loads.

TRANSIENT DYNAMIC ANALYSIS--Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

SPECTRUM ANALYSIS--An extension of the modal analysis, used to calculate stresses and strains due to a response spectrum or a PSD input (random vibrations).

BUCKLING ANALYSIS--Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

EXPLICIT DYNAMIC ANALYSIS--This type of structural analysis is only available in the ANSYS LS-DYNA program. ANSYS LS-DYNA provides an interface to the LS-DYNA explicit finite element program. Explicit dynamic analysis is used to calculate fast solutions for large deformation dynamics and complex contact problems.

In addition to the above analysis types, several special-purpose features are available:

- Fracture mechanics
- Composites
- Fatigue
- p-Method

• Beam Analyses

ELEMENTS USED IN STRUCTURAL ANALYSES

Most ANSYS element types are structural elements, ranging from simple spars and beams to more complex layered shells and large strain solids. Most types of structural analyses can use any of these elements.

MATERIAL MODEL INTERFACE

If we are using the GUI, we must specify the material we will be simulating using an intuitive material model interface. This interface uses a hierarchical tree structure of material categories, which is intended to assist in us choosing the appropriate model for our analysis.

TYPES OF SOLUTION METHODS

Two solution methods are available for solving structural problems in the ANSYS family of products: the h-method and the p-method. The h-method can be used for any type of analysis, but the p-method can be used only for linear structural static analyses. Depending on the problem to be solved, the h-method usually requires a finer mesh than the p-method. The p-method provides an excellent way to solve a problem to a level of accuracy while using a coarse mesh. In general, the discussions in this manual focus on the procedures required for the h-method of solution.

STRUCTURAL STATIC ANALYSIS

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. The kinds of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

PERFORMING A STATIC ANALYSIS

The procedure for a static analysis consists of these tasks:

- Build the Model
- Set Solution Controls
- Set Additional Solution Options
- Apply the Loads
- Solve the Analysis
- Review the Results

LOAD TYPES

All of the following load types are applicable in a static analysis.

DISPLACEMENTS (UX, UY, UZ, ROTX, ROTY, ROTZ)

These are DOF constraints usually specified at model boundaries to define rigid support points. They can also indicate symmetry boundary conditions and points of known motion. The directions implied by the labels are in the nodal coordinate system.

FORCES (FX, FY, FZ) AND MOMENTS (MX, MY, MZ)

These are concentrated loads usually specified on the model exterior. The directions implied by the labels are in the nodal coordinate system.

PRESSURES (PRES)

These are surface loads, also usually applied on the model exterior. Positive values of pressure act towards the element face (resulting in a compressive effect).

TEMPERATURES (TEMP)

These are applied to study the effects of thermal expansion or contraction (that is, thermal stresses). The coefficient of thermal expansion must be defined if thermal strains are to be calculated. We can read in temperatures from a thermal analysis [LDREAD], or we can specify temperatures directly, using the BF family of commands.

FLUENCES (FLUE)

These are applied to study the effects of swelling (material enlargement due to neutron bombardment or other causes) or creep.

GRAVITY, SPINNING, ETC.

These are inertia loads that affect the entire structure. Density (or mass in some form) must be defined if inertia effects are to be included.

APPLY LOADS TO THE MODEL

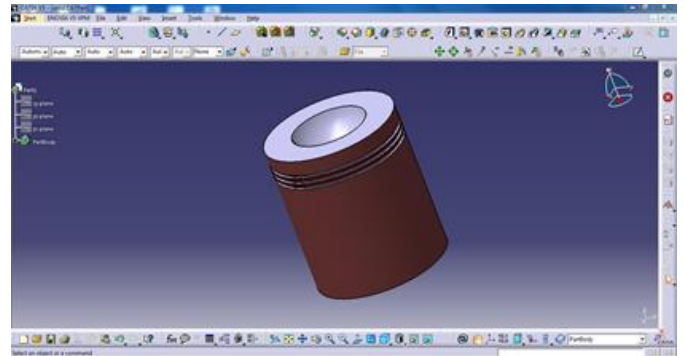
Except for inertia loads, which are independent of the model, we can define loads either on the solid model (key points, lines, and areas) or on the finite element model (nodes and elements). We can also apply boundary conditions via TABLE type array parameters. Applying Loads Using TABLE Type Array Parameters) or as function boundary conditions

APPLYING LOADS USING FUNCTION BOUNDARY CONDITIONS

“Loads Applicable in a Static Analysis” summarizes the loads applicable to a static analysis.

In an analysis, loads can be applied, removed, operated on, or listed.

CHOOSING THE PROPER ELEMENT TYPE



Proper element type

The following element types are available to model layered composite materials: SHELL99, SHELL91, SHELL181, SOLID46, and SOLID191. Which element we choose depends on the application, the type of results that need to be calculated, and so on. Check the individual element descriptions to determine if a specific element can be used in our ANSYS product. All layered elements allow failure criterion calculations.

SHELL99 - Linear Layered Structural Shell Element

SHELL99 is an 8-node, 3-D shell element with six degrees of freedom at each node. It is designed to model thin to moderately thick plate and shell structures with a side-to-thickness ratio of roughly 10 or greater. For structures with smaller ratios, we may consider using SOLID46. The SHELL99 element allows a total of 250 uniform-thickness layers. Alternately, the element allows 125 layers with thicknesses that may vary bilinearly over the area of the layer. If more than 250 layers are required, we can input our own material matrix. It also has an option to offset the nodes to the top or bottom surface.

SHELL91 - Nonlinear Layered Structural Shell Element

SHELL91 is similar to SHELL99 except that it allows only up to 100 layers and does not allow us to input a material property matrix. However, SHELL91 supports plasticity, large-strain behavior and a special sandwich

option, whereas SHELL99 does not. SHELL91 is also more robust for large deflection behavior.

SHELL181 - Finite Strain Shell

SHELL181 is a 4-node 3-D shell element with 6 degrees of freedom at each node. The element has full nonlinear capabilities including large strain and allows 255 layers. The layer information is input using the section commands rather than real constants. Failure criteria are available using the FC commands.

SOLID46 - 3-D Layered Structural Solid Element

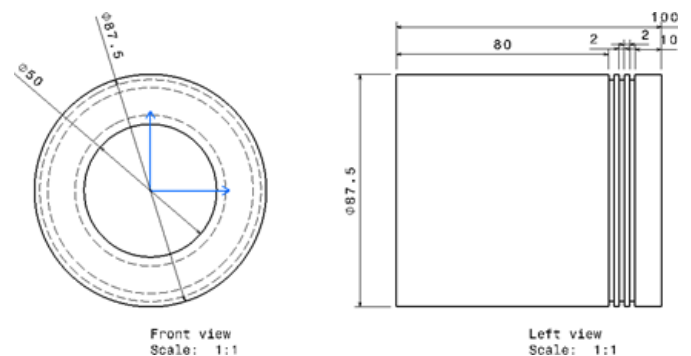
SOLID46 is a layered version of the 8-node, 3-D solid element, SOLID45, with three degrees of freedom per node (UX, UY, UZ). It is designed to model thick layered shells or layered solids and allows up to 250 uniform-thickness layers per element. Alternately, the element allows 125 layers with thicknesses that may vary bilinearly over the area of the layer. An advantage with this element type is that you can stack several elements to model more than 250 layers to allow through-the-thickness deformation slope discontinuities. The user-input constitutive matrix option is also available. SOLID46 adjusts the material properties in the transverse direction permitting constant stresses in the transverse direction. In comparison to the 8-node shells, SOLID46 is a lower order element and finer meshes may be required for shell applications to provide the same accuracy as SHELL91 or SHELL99.

SOLID191 - Layered Structural Solid Element

SOLID191 is a layered version of the 20-node 3-D solid element SOLID95, with three degrees of freedom per node (UX, UY, UZ). It is designed to model thick layered shells or layered solids and allows up to 100 layers per element. As with SOLID46, SOLID191 can be stacked to model through-the-thickness discontinuities. SOLID191 has an option to adjust the material properties in the transverse direction permitting constant stresses in the transverse direction. In spite of its name, the element does not support nonlinear materials or large deflections.

In addition to the layered elements mentioned above, other composite element capabilities exist in ANSYS, but will not be considered further in the chapter:

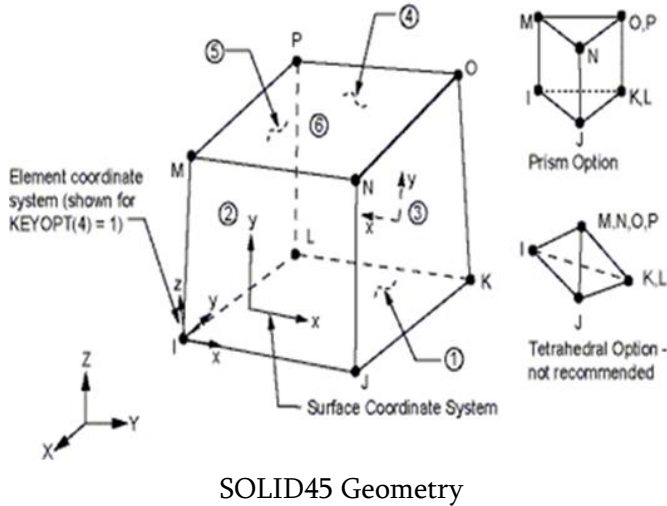
- SOLID95, the 20-node structural solid element, with KEYOPT(1) = 1 functions similarly to a single layered
- SOLID191 including the use of an orientation angle and failure criterion. It allows nonlinear materials and large deflections.
- SHELL63, the 4-node shell element, can be used for rough, approximate studies of sandwich shell models. A typical application would be a polymer between two metal plates, where the bending stiffness of the polymer would be small relative to the bending stiffness of the metal plates. The bending stiffness can be adjusted by the real constant RMI to represent the bending stiffness due to the metal plates, and distances from the middle surface to extreme fibers (real constants CTOP, CBOT) can be used to obtain output stress estimates on the outer surfaces of the sandwich shell. It is not used as frequently as SHELL91, SHELL99, or SHELL181, and will not be considered for anything other than sandwich structures in this section.
- SOLID65, the 3-D Reinforced Concrete Solid Element, models an isotropic medium with optional reinforcing in 3 different user-defined orientations.
- BEAM188 and BEAM189, the 3-D finite strain beam elements, can have their sections built up with multiple materials.



ELEMENT TYPE USED

SOLID45 Element Description

SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

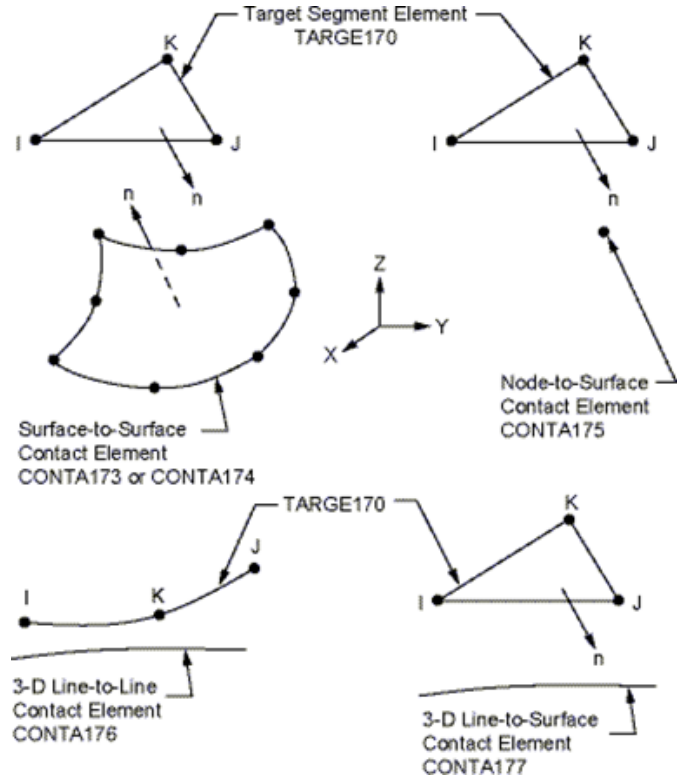
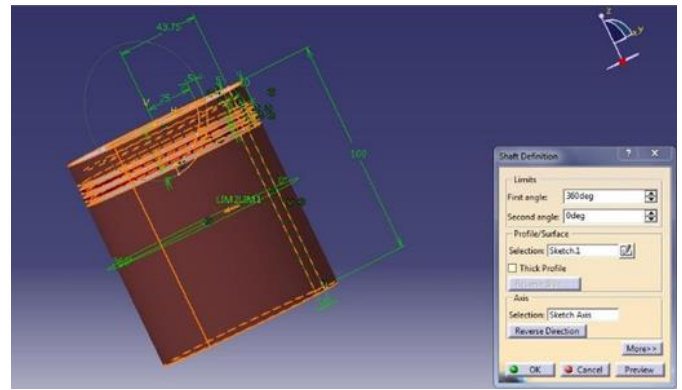


SOLID45 Geometry

TARGE170 Element Description

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements (CONTA173, CONTA174, CONTA175, and CONTA176 & CONTA177). The contact elements themselves overlay the solid, shell, or line elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. We can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. We can also impose forces and moments on target elements.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid, shell, or line elements describing the boundary of the deformable



TARGE170 Geometry

MESHING

The collection of nodes and elements form the finite element mesh. Each element is of simple shape for which the finite element program has information to write the governing equations in the form of stiffness matrix. The unknowns at each element are the stress and displacement at the node points, which are the points at which the elements are connected. The finite element program will assemble the stiffness matrix for the entire model. This stiffness matrix is solved for the unknown displacements, stress given the known forces and boundary

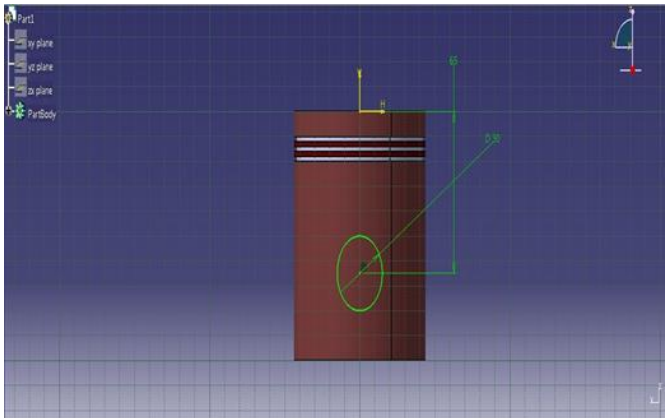
Conditions. From the displacements at the nodes, the stress in each element can be calculated.

STEPS INVOLVES FOR MESHING

The steps followed to mesh the modeled cross section of the piston are explained briefly as follows.

Element type

Before meshing model, element attributes should be specified based on the material, analysis and so on the required results. The following are the elements used in this analysis.



Element type for base material – SOLID 86

This element is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is now available in two forms: Structural Solid and Layered Solid. The element is well suited to modeling irregular meshes (such as those produced by various CAD/CAM systems).

This element is used for modeling and meshing of basic piston material.

Element type for coating material – SHELL 93

Shell 93 is particularly well suited to model curved shells. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z- axes. The

deformation shapes are quadratic in both in-plane directions. The element has plasticity, stress stiffening, large deflection, and large strain capabilities.

After defining element type material properties should be defined. Material properties used for aluminum is defined in table.

Table: Material properties of Aluminium

Material properties	Values
Young’s modulus	70 e9 N\mm ²
Poisson’s ratio	0.31 (no unit)
Coeff of thermal expansion	23 e-6 /K
Coeff of thermal conductivity	234 W/Mk

Material properties Values

- Young’s modulus 70 e9 N\mm²
- Poisson’s ratio 0.31 (no unit)
- Coeff of thermal expansion 23 e-6 /K
- Coeff of thermal conductivity 234 W/Mk

Table After defining material properties mesh attribute should be defined.

Material properties Values

- Young’s modulus 205 e9 N\mm²
- Poisson’s ratio 0.35 (no unit)
- Coeff of thermal expansion 10 e-6 /K
- Coeff of thermal conductivity 2 W/mK

STEPS INVOLVED FOR CREATING COATING

1. Define element type 1 for coating i.e. shells 93.
2. Define element 2 for solid piston i.e. solid 186.
3. Define material properties for coating as material model no 1 and define base material as material model no 2
4. Set element as 1 and material model 1 in element attributes.
5. Create area in upper side of piston.
6. Now mesh upper layer.

7. Change element attributes, changes material model as 2 an

8. Now mesh the solid.

Material properties	Values
Young's modulus	205 e9 N/mm ²
Poisson's ratio	0.35 (no unit)
Coeff of thermal expansion	10 e-6 /K
Coeff of thermal conductivity	2 W/mK

THERMAL ANALYSIS

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution is performed via ANSYS which calculates nodal temperatures, and then uses the nodal temperatures to obtain the other thermal quantities. Only the ANSYS Multi physics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN programs support thermal analyses. The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

ANSYS supports two types of thermal analysis for analyzing the component with the influence of temperature:

Steady-state thermal analysis

A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.

The ANSYS Multi physics, ANSYS Mechanical, ANSYS FLOTRAN, and ANSYS Professional products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often

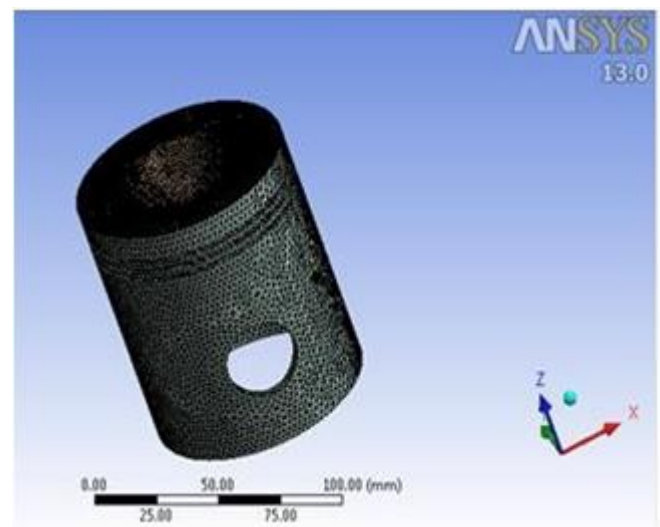
Perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

Steady-state thermal analysis is used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

ANSYS RESULTS & COMPARISON GRAPHS



IV. CONCLUSION

From the obtained literature survey the strengthening point of view the ceramic coated piston strength will be higher than the all other coated piston. So that the tribological property will be increased in terms of wear and thermal barrier also, this will improve the further overall engine efficiency due to the good combustion takes place. Due to more strengthening of the ceramic coated piston surface, the corrosion and wear properties also improved. This will further helpful for the complete combustion. This will improve the combustion properties and reduces the emissions.

V. FUTURE SCOPE

To better understand the correlations between friction and surface roughness a future study must include a study of different ceramic materials like alumina, zirconia and titanium materials. The structural and thermal analysis will be reported in the study are rather small in comparison to the normal wear found by coated piston. A future study should include a numerical investigation about with and without coated piston in a diesel engine. This investigation will going to compare the coated and uncoated piston material surface.

VI. REFERENCES

- [1]. Ekrem Buyukkaya, "Thermal Analysis of functionally graded coating AlSi alloy and steel pistons", Surface and coatings technology (2007)
- [2]. Reinhold H. Dauskardt, "THERMAL BARRIER COATINGS" (1999).
- [3]. M. Foy, M. Marchese, G. Jacucci, Engineering plasma spray films by knowledge based simulations proceedings of IFIP conference on, in: Arthur B. Baskin, et al., (Eds.), Cooperative Knowledge Processing for Engineering Design, Kluwer Academic Publishers, The Netherlands, 1999, p. 289.
- [4]. J.S. Lee, T. Matsubara, T. Sei, T. Tsuchiya, J. Mater, Preparation and properties of Y2O3-doped ZrO2 thin films by the sol-gel process, Surface Coating Technology (1997).
- [5]. J.N.Reddy, "An introduction to finite element analysis".
- [6]. Zhang T, Xu L, Xiong W, et al. Experimental research on the thermal contact resistance between Cu-Cu in vacuum and low temperature. Cryogenics, 1999.
- [7]. www.azom.com - "Functionally Graded Materials (FGM) and Their Production Methods".
- [8]. D.Robinson,R.Palaninathan.Thermal analysis of piston using 3D finite element method.2001Elsevier B.V
- [9]. A.R.Bhagat, Y.M.Jibhakate.Thermal Analysis of I.C Engine Piston Using Finite element method(IJMERE)Vol.2,Issue4,July-Aug 2012.

Cite this article as :

V. Karthikeyan, Vengadesan. S, Kabilan. K. K, Vijay. S, "Investigation and Optimization of Piston with Different Compositions in Existent Combustion Circumstance", International Journal of Scientific Research in Science, Engineering and Technology (IJSRSET), Online ISSN : 2394-4099, Print ISSN : 2395-1990, Volume 9 Issue 2, pp. 255-265, March-April 2022. Available at doi : <https://doi.org/10.32628/IJSRSET229237> Journal URL : <https://ijsrset.com/IJSRSET229237>