DESIGN AND INVESTIGATION OF FOUR STROKE PETROL ENGINE EXHAUST VALVE

[1]V.Karthikeyan, [2]C.Arunkumar, [3]B.Keeerthivasan, [4]P.Prakash, [5]A.Gokulraj
[1] Assistant Professor, [2,3,4,5] Final Year Students
Department Of Mechanical Engineering
Excel Engineering College, Komarapalayam, Erode, Tamilnadu, India

ABSTRACT: Due to high amount of heat dissipation in four stoke petrol engine the efficiency is lesser and the valve get damaged. In order to reduce the heat dissipation that cause damage to the valves and heads of Four stroke petrol engine exhaust valve has been designed and analyzing with various alloy material like nickel, titanium cobalt, stainless steel and In conel alloys. The heat distribution over the values has been analyzed to reduce the valve damage.

KEYWORDS: Heat Transfer, Exhaust Valve, Four Stroke Petrol Engine, Inconel Alloy

OBJECTIVE

This paper presents a method for analysis of the thermal behavior of exhaust valve of four stork petrol engine. The purpose of analysis is study the characteristics of exhaust valve under given thermal load. The results obtained from analysis was verified. Now a days model analysis is become a common tool for studying the behavior of common mechanical structures. The approach of the software validation results is more convergent to the experimental results but the conditions in the ANSYS model should be as close to reality as possible to ensure acceptable accuracy. The main objective of this paper is to carry out Finite Element Analysis (FEA) in view to predict temperature distribution across the exhaust valve. Several specific features of the original design then is modified depending upon results, in the view to reduce the temperature of the exhaust valve. This paper also includes drawing 3D model of the valve, meshing and analysis by using ANSYS.

PROBLEMS IN THE PRESENT VALVES

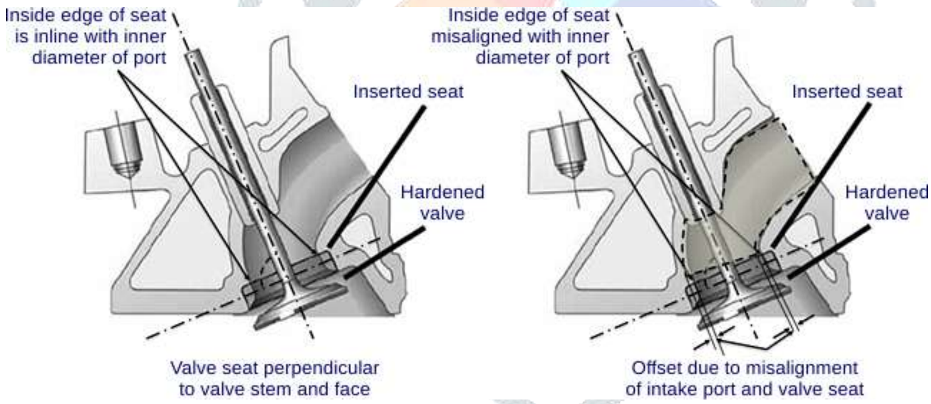


Fig no: 1
Arrangement of the Exhaust Valve

VALVES

Definition: A poppet valve (also called mushroom valve) is a valve typically used to control the timing and quantity of gas or vapor flow into an engine.

It consists of a hole, usually round or oval, and a tapered plug, usually a disk shape on the end of a shaft also called a valve stem. The portion of the hole where the plug meets with it is referred to as the 'seat' or 'valve seat'. The shaft guides the plug portion by sliding through a valve guide. In exhaust applications a pressure differential helps to seal the valve and in intake valves a pressure differential helps open it. The poppet valve was most likely invented in 1833 by E.A.G. Young of the Newcastle and Frenchtown Railroad. Young patented his idea but the Patent Office fire of 1836 destroyed all records of it.

Fig no: 2
Poppet valves

FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters.

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

FINITE ELEMENT ANALYSIS WORKING

FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements.

A wide range of objective functions (variables within the system) are available for minimization or maximization:

- Mass, volume, temperature
- Strain energy, stress strain
- Force, displacement, velocity, acceleration
- Synthetic (User defined)

There are multiple loading conditions which may be applied to a system. Next to, some examples are shown:

- Point, pressure, thermal, gravity, and centrifugal static loads
- Thermal loads from solution of heat transfer analysis
- Enforced displacements
- Heat flux and convection
- Point, pressure and gravity dynamic loads

Each FEA program may come with an element library, or one is constructed over time. Some sample elements are:

- Rod elements
- Beam elements
- Plate/Shell/Composite elements
- Shear panel
- Solid elements
- Spring elements
- Mass elements
- Rigid elements

- Viscous damping elements

Many FEA programs also are equipped with the capability to use multiple materials within the structure such as:

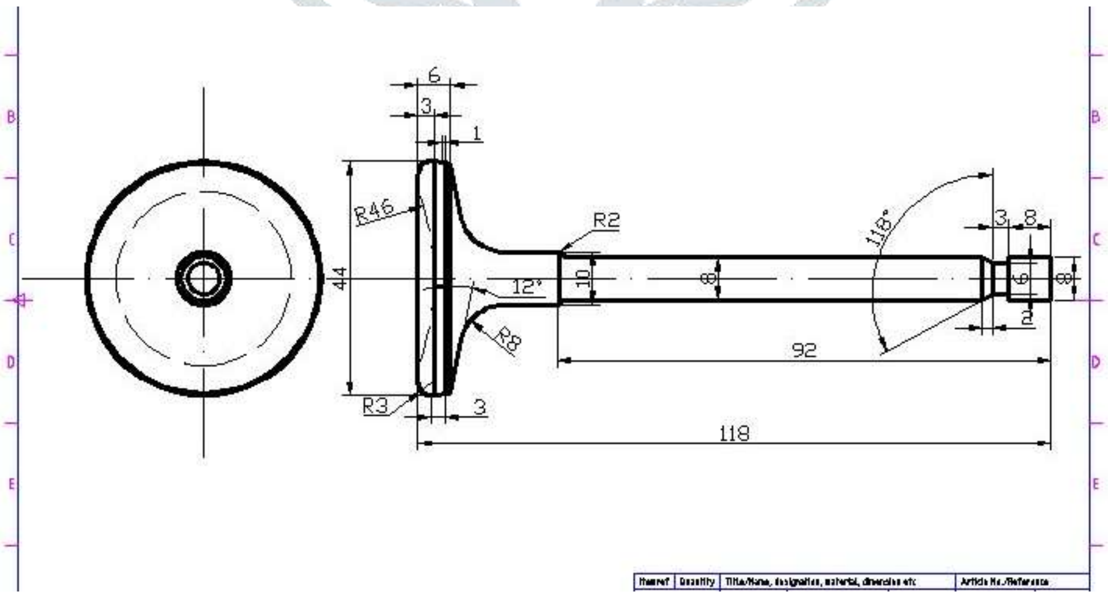
- Isotropic, identical throughout
- Orthotropic, identical at 90 degrees
- General anisotropic, different throughout

Heat Transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady-state or transient transfer. Steady-state transfer refers to constant thermo properties in the material that yield linear heat diffusion.

METHODOLOGY

Import the beam file from CATIA V6 as described in last lecture
1. Define the analysis as a thermal analysis
Main Menu - Preference - check “Thermal” - ok
2. Select the element type as Solid 87
Main Menu - Preprocessor - Element Type - Add/Edit/Delete - Click “Add” in the element dialog box - Choose “Solid” under “Thermal” in the left scroll box of the library of element types- Choose in the right scroll box - Click “ok” to close the library - Click “close” to close the element dialog box
3. Input the material properties
Main Menu - Preprocessor - Material Props - Material Models - In the “Define Material Model Behavior” dialog box, double click “Thermal” on the right box -double click “Conductivity” - double click “Isotropic” - Input in the box for KXX- Click “ok” to add the properties - double click “Specific Heat” - Input in the box for C - Click “ok” to add the property - double click “Density” - Input in the box for DENS – Click “ok” to add the property - Close the “Define Material Model Behavior” dialog box
Main Menu - Preprocessor - Material Props - Temperature Units - Change the unit to Celsius in the pop up window - ok
4. Mesh the structure
Main Menu - Preprocessor - Meshing - Mesh Tool -
In the “Mesh Tool” dialog box, check “Smart Size” and move the size down to fine “1” - Click on "Mesh" - In the dialog box of pick, click "Pick All" - After meshing, click on "Refine" in the "Mesh Tool" dialog box - Click "Pick All" - Change the level of refinement to "2" - Click ok - Click "Close" to close the Mesh Tool dialog box.
5. Apply the load
Main Menu - Solution - Define Loads - Apply - Thermal - Temperature - On Areas -Pick the left end area by mouse click (You can go to File Menu – PlotControls - Pan Zoom Rotate, to find the area, check "Dynamic Mode" to rotate the model by mouse) - Click "ok" - Choose "Temp" and put "25" as the value
Main Menu - Solution - Define Loads - Apply - Thermal - Heat Flux - On Areas - Pick the right end area by mouse click - Click "ok" - Input 1000 into the value
6. Solve the problem
Main Menu - Solution - Current
7. Review the results
Main Menu - General Postprocessor- Plot Results -Contour Plot - Nodal Solution- Choose the results review, for example, DOF Solution - Nodal Temperature - Click "ok

DIMENSIONING AND DESIGN



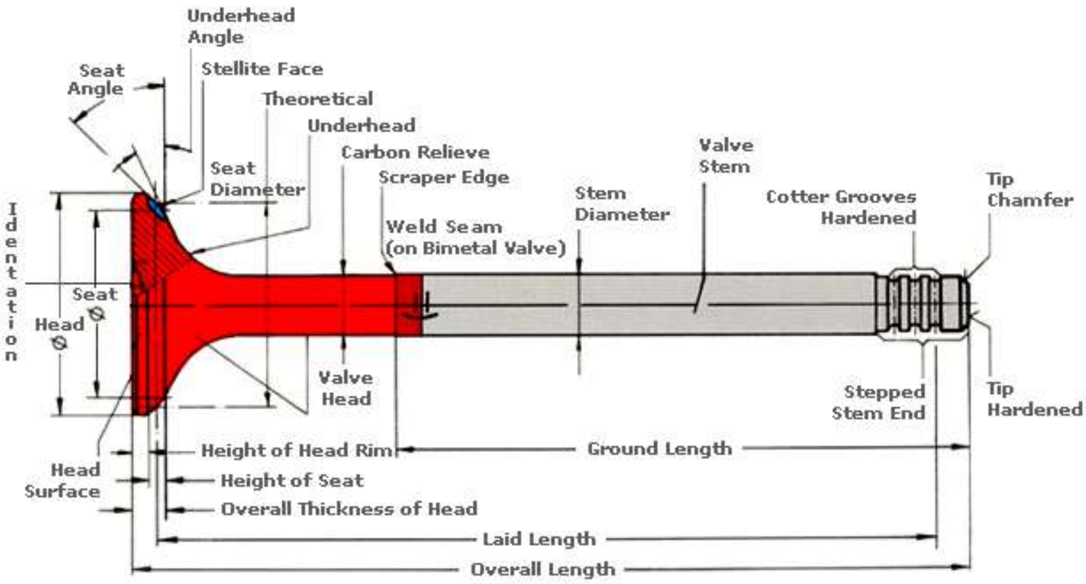
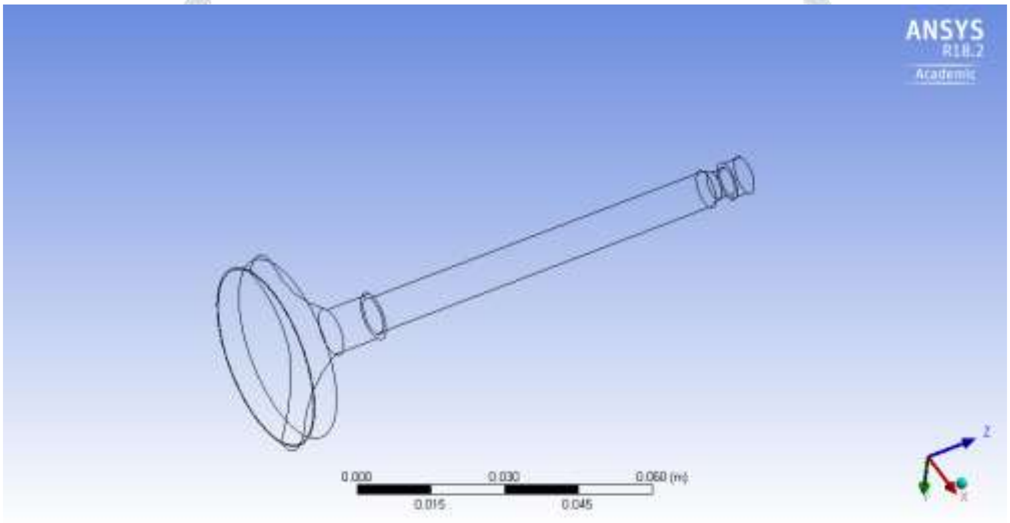
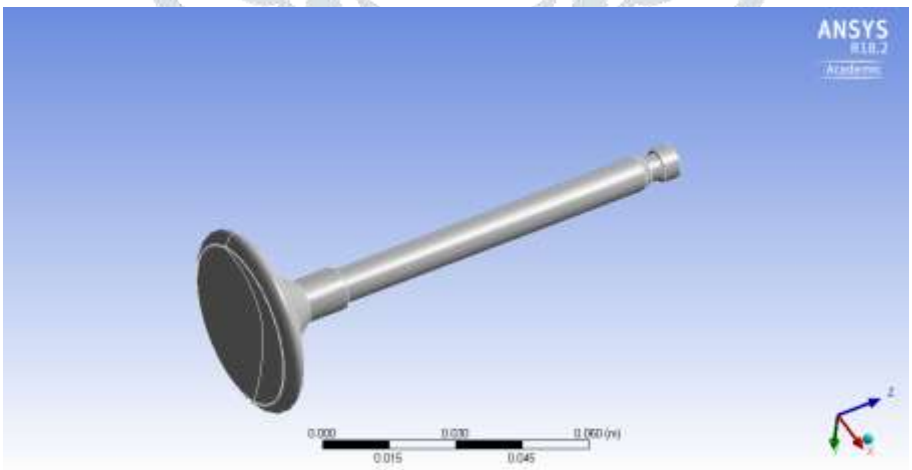


Fig no: 3
Dimensioning and design of Exhaust valve





PROCEDURE AND METHODOLOGY

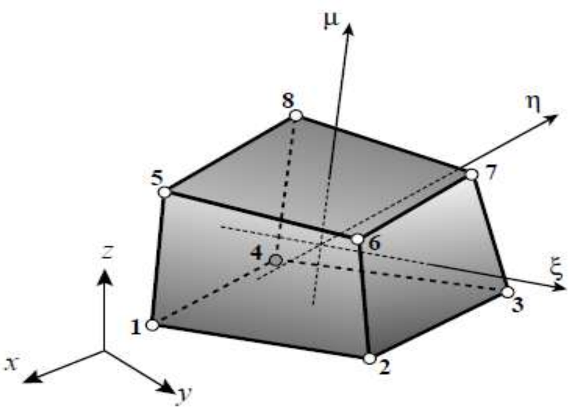
Step by step procedure for analyzing

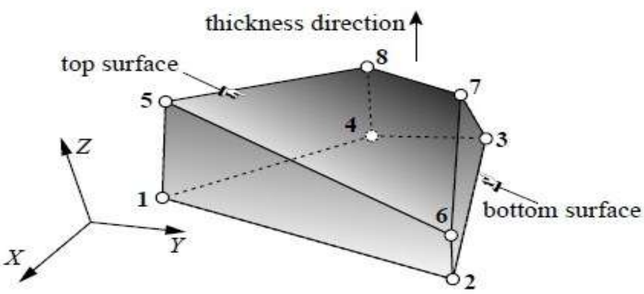
1. Preference → thermal
2. Preprocessor → element type → ADD → solid → brick node 70
3. Material property → material model → thermal → isotropic →
   - ✓ Thermal conductivity
   - ✓ Specific heat
   - ✓ Density
4. Meshing → mesh tool → free → pick all → ok
5. Load → define load →apply → thermal → temperature → on areas

6. Solution → solve → current LS
7. General postprocessor → plot result → contour plot →nodal solution
   - ✓ Nodal temperature
   - ✓ Thermal gradient

Fig no: 4
Meshing Diagram

11.2. MATERIAL PROPERTIES





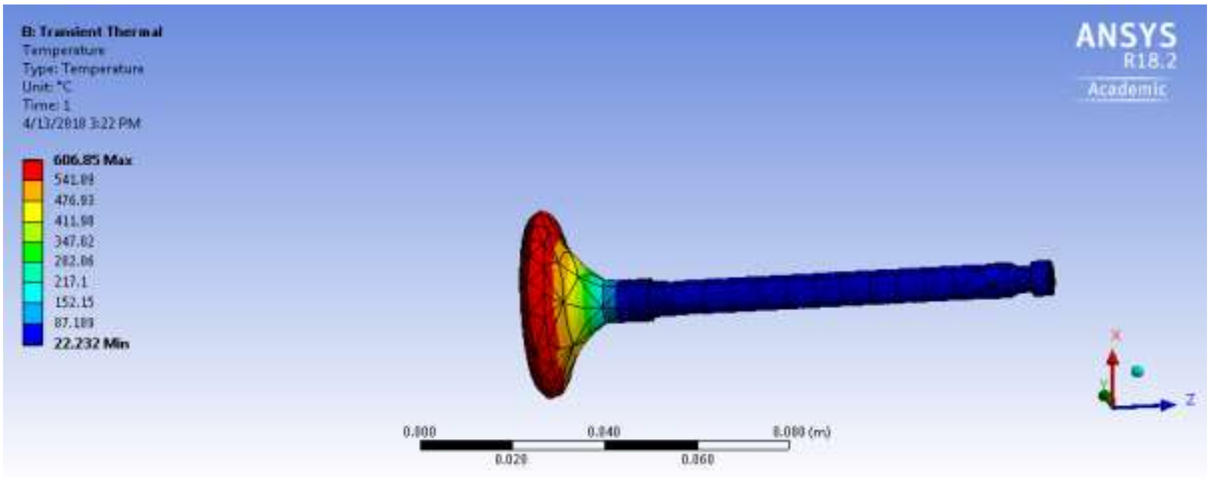
An eight-node "brick" finite element for three dimensional analysis has three iso parametric natural coordinates called ξ ,η and μ. These coordinates vary from −1 at one face to +1 at the opposite face, Construct the (trilinear) shape function for node 1 (follow the node numbering of the figure). The equations of the brick faces are:

$$\begin{array}{ll} 1485: \ \xi = -1 & 2376: \xi = +1 \\ 1265: \ \eta = -1 & 4378: \eta = +1 \\ 1234: \ \mu = -1 & 5678: \mu = +1 \end{array}$$

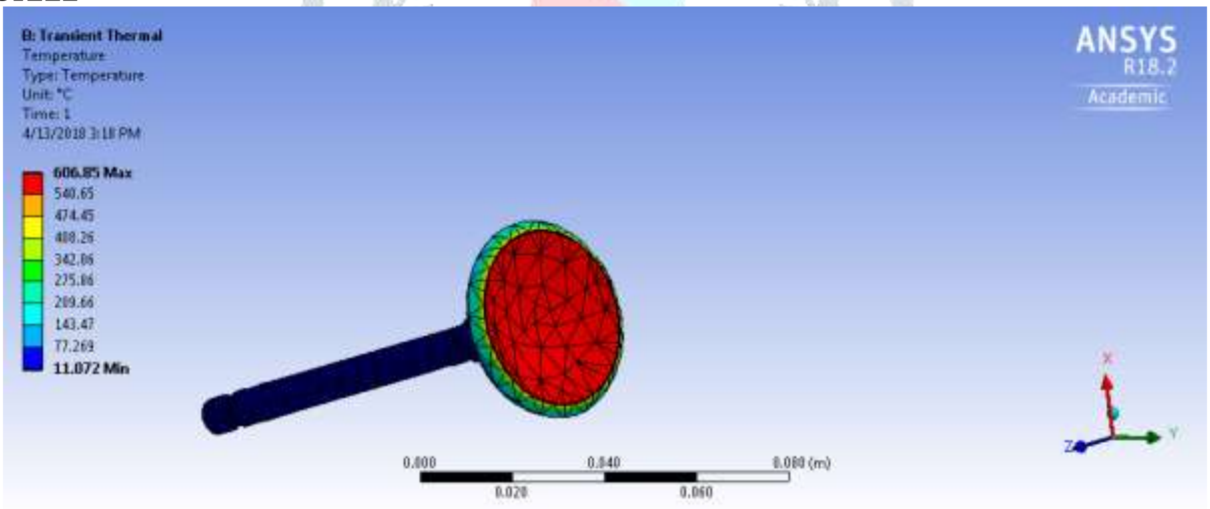
ANALYZED RESULTS

NODAL TEMPERATURE DISTRIBUTION& TEMPERATURE GRADIENT RESULTS

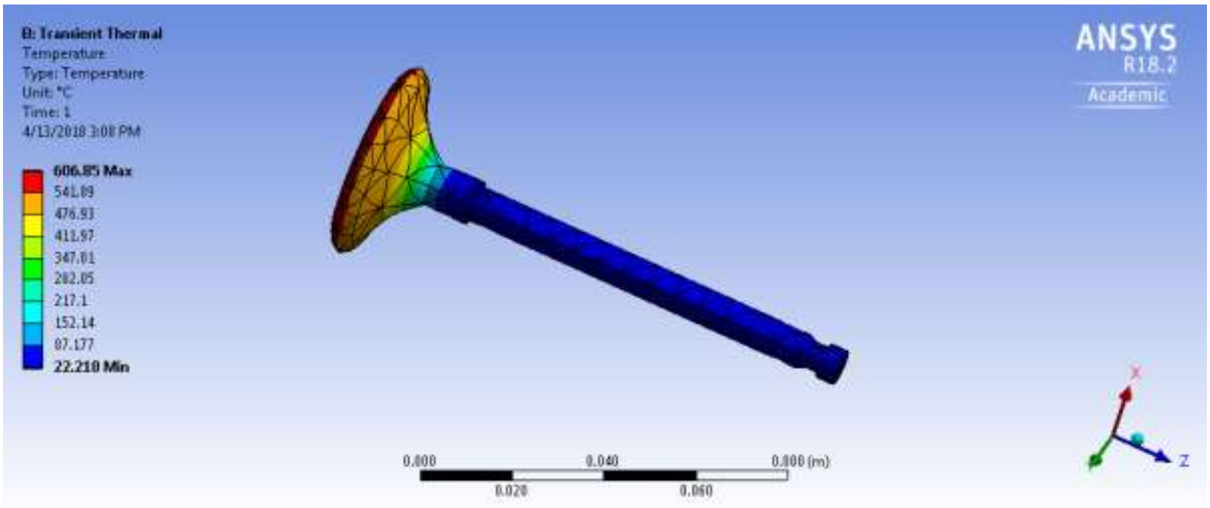
INCONEL



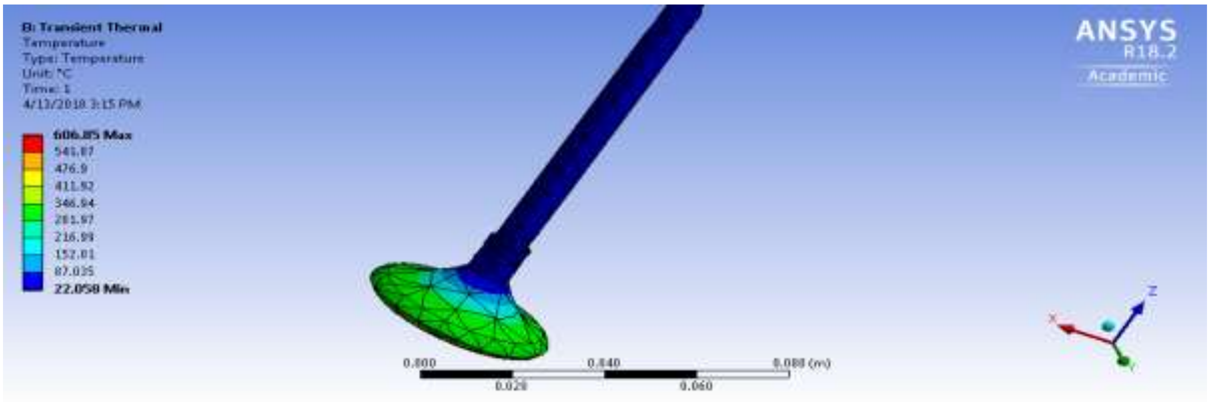
STAINLESS STEEL



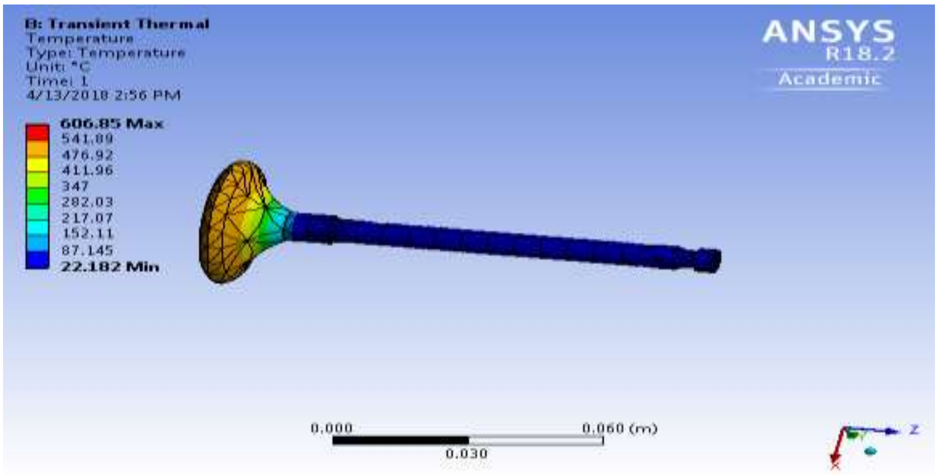
COBALT



TITANIUM



NICKEL



PLOT RESULTS

INCONEL

Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature

Time [s]	Minimum [°C]	Maximum [°C]
1.e-002	-141.93	606.85
2.e-002	-73.071	
2.8661e-002	-36.608	
3.7322e-002	-8.597	
5.7232e-002	1.6185	
9.0798e-002	4.6074	
0.15901	22.018	
0.25901	22.044	
0.35901	22.069	
0.45901	22.094	
0.55901	22.12	
0.65901	22.145	
0.75901	22.171	
0.85901	22.196	
0.95901	22.221	
1.	22.232	

STAINLESS STEEL

Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature

Time [s]	Minimum [°C]	Maximum [°C]
1.e-002	-272.77	606.85
2.e-002	-224.09	
3.4517e-002	-184.78	
6.0962e-002	-154.65	
0.10414	-111.52	
0.17147	-60.151	
0.27147	-17.075	
0.37147	2.0051	
0.47147	0.50793	
0.57147	0.69797	
0.67147	2.0721	
0.77147	4.2591	
0.87147	7.0138	
0.93573	8.9692	
1.	11.072	

COBALT

Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature

Time [s]	Minimum [°C]	Maximum [°C]
1.e-002	-145.66	
2.e-002	-79.037	
2.8787e-002	-41.184	
3.7574e-002	-13.597	
5.7699e-002	1.7446	
9.1054e-002	3.7921	
0.1576	19.324	
0.2576	22.04	606.85
0.3576	22.064	
0.4576	22.088	
0.5576	22.112	
0.6576	22.136	
0.7576	22.16	
0.8576	22.184	
0.9576	22.208	
1.	22.218	

TITANIUM

Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature

Time [s]	Minimum [°C]	Maximum [°C]
1.e-002	-217.94	
2.e-002	-167.45	
3.1735e-002	-135.79	
5.1735e-002	-89.514	
8.3187e-002	-43.121	
0.13073	1.4181	
0.20351	1.4272	
0.30351	5.2939	606.85
0.40351	12.526	
0.50351	21.43	
0.60351	22.026	
0.70351	22.034	
0.80351	22.042	
0.90351	22.05	
1.	22.058	

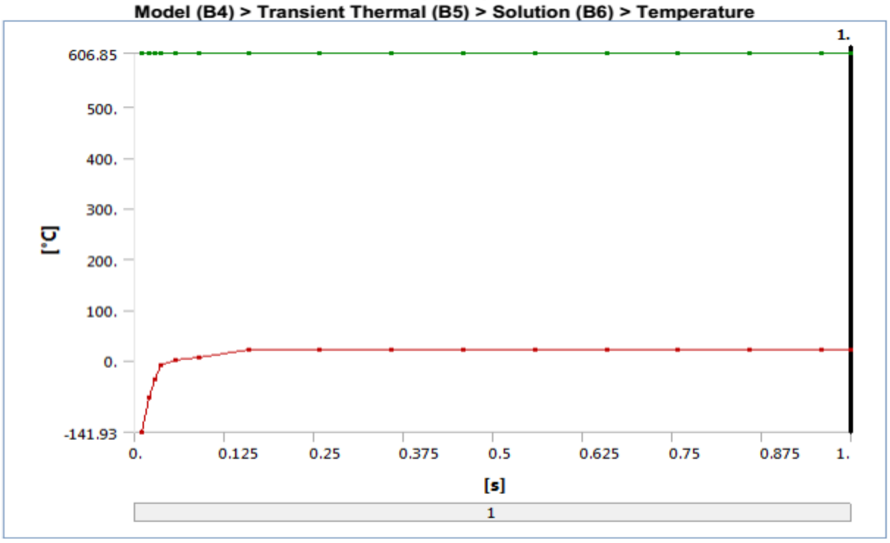
NICKEL

Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature

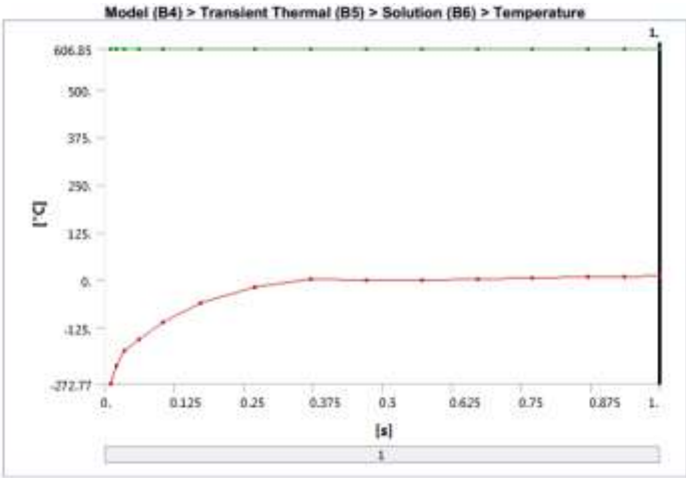
Time [s]	Minimum [°C]	Maximum [°C]
1.e-002	-155.97	
2.e-002	-95.858	
2.9174e-002	-54.038	
3.8348e-002	-27.885	
5.9256e-002	2.5043	
9.2495e-002	2.159	
0.15488	12.388	
0.25488	22.03	606.85
0.35488	22.05	
0.45488	22.071	
0.55488	22.091	
0.65488	22.112	
0.75488	22.132	
0.85488	22.153	
0.95488	22.173	
1.	22.182	

GRAPHS
INCONEL

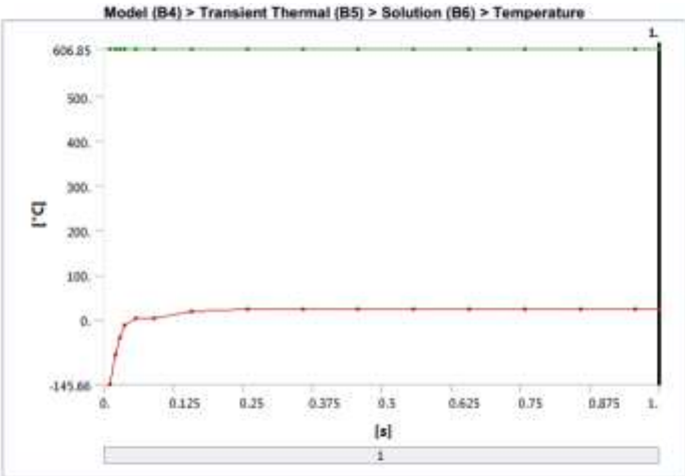
Model (B4) > Transient Thermal (B5) > Solution (B6) > Temperature



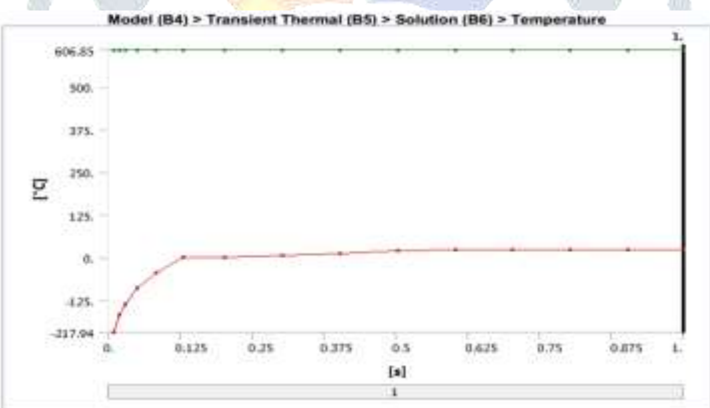
STAINLESS STEEL



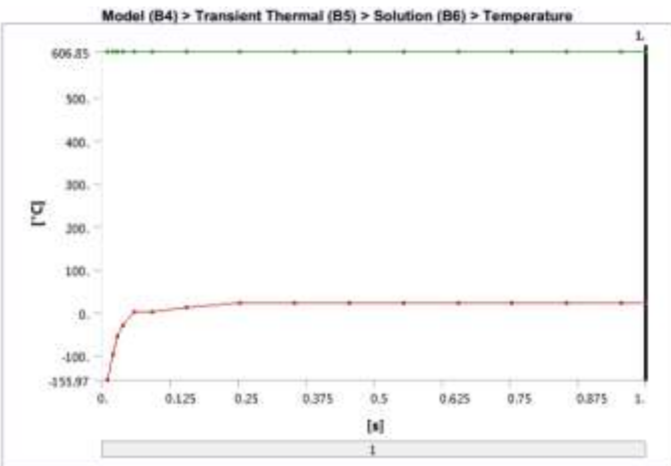
COBALT



TITANIUM

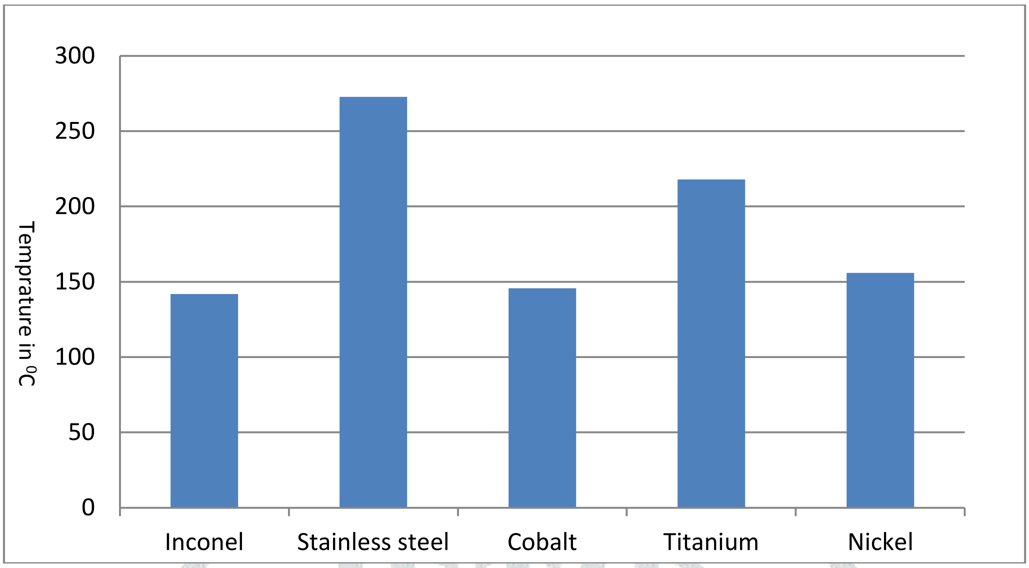


NICKEL



CONCLUSION

In this transient analysis of the exhaust valve with four different material concludes that the heat transfer in the stainless steel is very higher than the other materials so then previously used Inconel alloy is also little greater than the cobalt materials, hence the higher heat transfer of the exhaust valve leads to the reckless damage of the valve during the operation. From this the cobalt materials possess less amount of heat withstanding capacity other than the all materials. In cost wise the cobalt material is double the amount higher than Inconel alloy.



Result plot graph

FUTURE SCOPE

- In future the materials along with the composite material going to be analyzed
- To increase the heat transfer without damaging of the valve design should be changed
- Furthermore the material can be alloyed with orthotropic composition.

REFERENCE

[1] "Amended analysis of fins", V.Karthikeyan, "International Journal of scientific research and development".
[2] Balat, M., and Balat, H, "Progress in biodiesel processing", Applied Energy 87,(2017), 1815-1835.
[3] Heywood J. Internal combustion engines fundamentals. McGraw Hill book company; 2014.
[4] Assanis, D.N., Z.S. Filipi, S.B. Fiveland, et al., A predictive ignition delay correlation under steady-state and transient operation of a direct injection diesel engine. J. Eng. Gas Turbines Power, 125,(2003): p. 450.
[5] Lyn WT, Valdamanis. Effects of physical factors on ignition delay, SAE 680102, 1968.
[6] Watson N., Pilley A. D., and Marzouk M., A combustion correlation for diesel engine simulation, SAE paper 800029, 1980.
[7] Sahoo, P. K., Das, L. M., Babu, M. K., Arora, P, Singh, V. P, Kumar, N. R, and Varyani. T. S, "Comparative evaluation of performance and emission characteristics of jatropha , karanja and polanga based biodiesel as fuel in a tractor engine", Fuel 88,(2009), 1698-1707.