ICTIMES 2019

5th INTERNATIONAL CONFERENCE ON

Trends in Information, Management, Engineering and Sciences

Proceedings of International Conference on Future Technologies in Mechanical Engineering (ICFTME)

- ISBN 978-93-88808-60-6

Convenor
Dr. A. Karthikeyan
HOD-Dept. of MECH

Editor
Dr. P. Velmurgan
Professor - MECH

Editor in Chief
Dr. P. John Paul
Principal

Organized by

Estd :2005

MALLA REDDY COLLEGE OF ENGINEERING

Approved by AICTE - New Delhi, Affiliated to JNTUH, Accredited by NBA & Accredited by NACC.
ISO 9001:2015 Certified Institution, Recognition of College under Section 2(f) & 12 (B) of the UGC Act,1956.

website: www.mrce.in

ROLE OF CAD/CAM IN DESIGNING AND MANUFACTURING OF NEW PRODUCTS

Mohammed affan khan
mtech mechanical cad/cam
(19Q91D0406)
mallareddy college of engineering & technology
khanaffan364@gmail.com

Mohammed asif
mtech mechanical cad/cam
(19Q91D0407)
mallareddy college of engineering & technology
mohd.asif6741@gmail.com

Abstract-
Smart CAD/CAM technologies for superior product modeling within the intelligence of designing complete product variants become additional pertinent in future. several design techniques to assist knowledge base design actions in numerous engineering domains additionally to subsequent processes need to be developed. A necessary job to achieve this aim is to permanently investigate this state of the art, rising trends, new approaches, additionally to industrial issues and needs concerning the complete CAD/CAM area. With the aim of direct future analysis and development activities as close as possible to the unendingly rising an needs of a worldwide market we have a tendency to disbursed a wide-ranging national study in cooperation with one amongst the Germans leading CAD/CAM magazines. during this method, it became potential to achieve a representative quantity of users, to get their experience based mostly assessments on today's most significant aspects of CAD/CAM technology. The results of this examination are summarized during this paper to grant system developers, engineers, and researchers an overview of this condition as well on function a direction for call manufactures with in the design.

***Keywords: CAD/CAM; CIM; CAPP; Product Development ; design ; manufacture*.**

1.INTRODUCTION

In a globally competitive environment, time compression strategies in product development are of critically importance. Certain products have long development cycle times. Examples are aircraft and automobiles. In few products like computers, technological obsolescence keeps a constraint on the time required for product development.

Whenever a new microprocessor is released in the market, the manufacturer companies of the computers link with each other to market computers based on the new processor.

Frequent making of newer microprocessors have consistently narrowed down the product life cycle of computers. The pursuit of good in performance has resulted in new technologies having developed be further refined. Here focus is on manufacturing planning, data management, supply chain management.

In entertainment electronics. The life cycle of computers and entertainment electronic products is thus reduced, thereby necessitating new products being delivered to the market at reduced intervals. The time compression in development has additionally necessitated

- Avoidance of design errors, make over of parts and tooling,
- Better information management,
- Improved provide chain management,
- Attaining higher and better levels of performance,
- Providing quality levels superior to what's offered by competitors,
- Above all provision the merchandise at the bottom potential price.

2.STAGES IN PRODUCT DEVELOPMENT

The need to be right 1st time anytime has modified the approach to style. The initial section of style consists of abstract style, style analysis and performance simulation.

The section is extremely repetitious as shown in (fig. 1) The techniques like coincident engineering, failure mode and impact analysis etc., area unit accustomed guarantee a reliable and quality style at this stage. this is often followed by careful style, tool style, prototype manufacture and analysis and documentation.

In (Fig. 2) consequent section of development concerned second section of engineering wherever the planning might activities in product development through seamless data transfer.

In (Fig. 3) CAD/CAM technologies help to simulate and the manufacturing methodologies in the following ways.

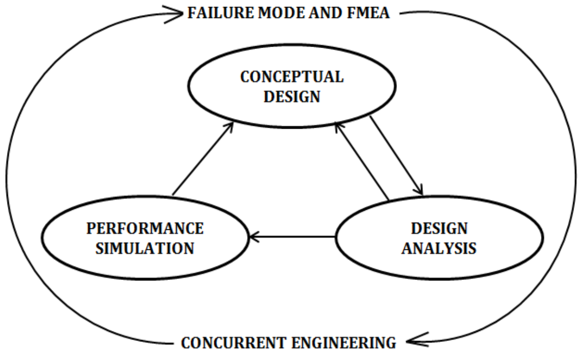


Fig. 1: Product Development Scenario

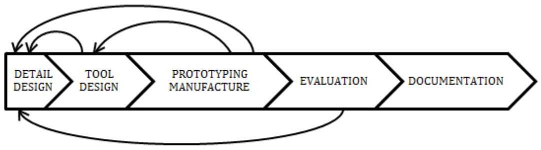


Fig. 2: Prototyping Stage of Product Development

3. PRODUCT DEVELOPMENT AND MANUFACTURE

CAD/CAM as an enabling technology for product development and manufacture Developments in computers and software relating to CAD/CAM have made CAD/CAM an indispensable enabling technology for time compression in product development. This is made possible by an integrated approach to carry out different activities in product development through seamless data transfer. (Fig. 3) CAD/CAM technologies help to simulate and the manufacturing methodologies in the following ways.

3.1 Assemble Analysis

With the assistance of today's CAD/CAM technology, style team will add a prime down and bottom up manner to form an entire electronic product model. Once an assembly is completed, solids based mostly kinematic analysis will be used to simulate advanced motions of mechanisms additionally to hold out tolerance analysis.

3.2 CAD/CAM in Aid to Manufacture through higher Tool style and Optimize manufacturing Processes

Manufacturing simulation uses a set of powerful CAD/Cam tools that ask for to create virtual manufacturing environment. several uncertainties which can lead to time delay work on an production of defective components will be eliminated through simulation or manufacturing, whether or not it's CNC machining, plastic injection moulding, casting, forging or welding.

3.3 Rapid Prototyping technology is being additional wide employed to verify and improve designs, fast tooling additionally as initial prototypes.

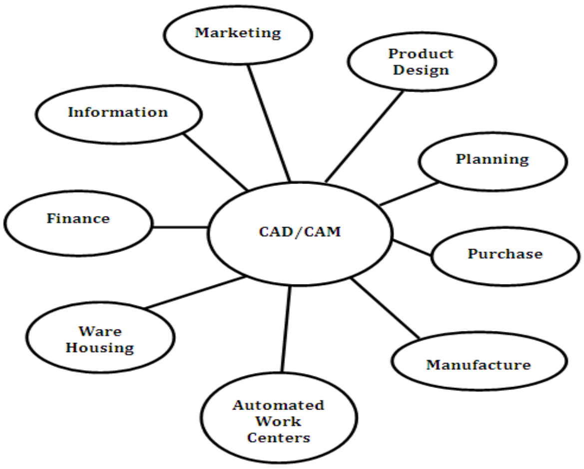


Fig. 3: CAD/CAM Database

3.4 Agile Manufacturing

Agile Manufacturing is oriented in the direction of high mix/ low volume, flexibility and adding velocity in the production process. It is applied to nature somewhere customizable order, suggest a required improvement. Consequently, that manufacturing has been one of most important strategies of new enterprises. In the atmosphere of the market ongoing to vary the quality, speed, quick responds, at very low cost by improving its agility of the manufacturing firm. A work of many highly developed technologies in Agile Manufacturing atmosphere has been researched through a few examinations. Many of them comprise computer-aided design, CIM, computer-aided manufacturing, IT, computer-aided process planning (CAPP). A few number of papers include the researchers investigative the integration of such highly developed technologies in Agile Manufacturing environment.

3.5 Agile Manufacturing Conception and Enabling Technologies

Even though there are many definitions of Agile Manufacturing brought out as a result of the researchers, the most familiar definition is, Agile Manufacturing is the ability of a manufacturing association to manufacture a range of products contained by a short period of time also in a cost effectiveness approach. Agile Manufacturing is an idea to standardize general manufacturing data, CAD/ CAM structure, research data, and join together them into a network. a standardized research data base and a general manufacturing data base are very critical for agility and can considerably decrease planning period and the product design period.

3.6 Characteristics of Agile Manufacturing

There are many characteristics of agile manufacturing such as show in following:

- Rapid new product development,
- Short lead times, cycle times,
- Use of superior CAD/CAM,
- Modular design and technology,
- highly flexible machines and equipment,
- Short and fast order processing,
- Fast supplier deliveries,
- Very Short time to market,
- Short guide times and short cycle times,
- Highly flexible and responsive processes,
- Modular assembly,
- Use of Solids model.

3.7 mould industry

In recent machinery manufacturing industry, mould industry has developed into the start industry for national economic system. many innovative product development and production depends deeply on mould manufacturing experience, particularly within the lightweight industry, automotive industry, and region and physics industries. the potential of mould manufacturing and stage of subsequent technique has end up to be a major pointer of a nation's level of mechanical producing technique. It straight affects many sectors of the nation's economy. mould CAM/ CAD is developed from the origin brought regarding by the autonomous development of mould CAM and mould CAD. it's a unique jump within the wide-ranging application of mould producing and technology. The quick development of CAD/CAM technology and therefore the more development of software system and hardware level provided well-built technical support for mould business and brought a mount up the standard of production level, endeavor product style and producing. it's become the simplest possibility for a contemporary enterprise networking, integration and knowledge.

3.8 mold CAD/CAM

Desgin flow By suggests that of the speedy development {of producing|of producing} technology and technology there square measure growing issues on the way to shorten machining production period and mould design time and to reinforce manufacturing quality. mould technology is additionally migrating often from manual style, counting on manual data and normal machine process skill to mould package, assisted engineering and assisted producing technology. The United States has pioneered implementing technology on mould business, realizing mould CAD/CAE/CAM incorporated system and achieving functions of enhancing mould manufacture quality, boosting production time and design effectiveness.

4. CONCLUSIONS

This paper concluded the results of a study with reference to advanced CAD/CAM technologies in reference to product development and manufacture. This paper conferred this methodologies are being employed and therefore the future oriented methodologies are going to be preferred. CAD/CAM users additionally as designers are asked to rate many sensible CAD/CAM technologies in relevance development and manufacture. moreover, issues in reverence to the consciousness of product variant style are mentioned. The Constant development of product style and manufacturing progressively bring down impacts upon sensible CAD/CAM technologies, proposing larger necessities for the analysis on and growth of CAD/CAM.

REFERENCES

[1] Paul S. Goodman, Terri L. Griffith, "A process approach to the implementation of new technology", Journal of Engineering and Technology Management, Volume 8, Issues 3-4, December 1991, Pages 261-285.

[2] Suk-Hwan Suh, Sung-Kee Noh, Yong-Jong Choi, "A PC-based retrofitting toward CAD/CAM/CNC integration", Computers & Industrial Engineering, Volume 28, Issue 1, January 1995, Pages 133-146.

[3] Bjørn Moseng, Bjarte Haaøy, "Nes Integration of CAD/CAM as seen from the production planners' point of view", Computers in Industry, Volume 5, Issue 4, December 1984, Pages 341-350.

[4] Guido Gürtler, "CAD Standard parts file - A DIN project in the standards committee on tabular layouts for article characteristics", Computer Standards & Interfaces, Volume 6, Issue 4, 1987, Pages 473-481.

[5] Jakob Vlietstra, "Integration aspects in CAD and CAM", Computers in Industry, Volume 5, Issue 4, December 1984, Pages 295-296.

[6] D.A. Linkens, "CAD for control systems-a review of PC software", Computer-Aided Design,

Volume 20, Issue 9, November 1988, Pages 564-565.

[7] Yoshihiro Ochiai, Tsuyoshi Sekiya, “Generation of free-form surface in CAD for dies”, Advances in Engineering Software, Volume 22, Issue 2, 1995, Pages 113-118.

[8] Lex Lennings, “CAD/CAM integration in practice: Two cases of computer aided toolmaking”, Computers in Industry, Volume 18, Issue 2, 1992, Pages 127-134.

[9] Ziga Turk, “Object-oriented modelling and integrated CAD”, Automation in Construction, Volume 1, Issue 4, March 1993, Pages 323-337.

[10] Emad S. Abouel Nasr, Ali K. Kamrani, “A new methodology for extracting manufacturing features from CAD system”, Computers & Industrial Engineering, Volume 51, Issue 3, November 2006,
Pages 389-415.

Design and CFD Analysis of Solar Flat Plate Collector by Using CREO

MOHD TABRAIZUDDIN

Tabraizuddin7@gmail.com

MALLA REDDY COLLEGE OF ENGINEERING

Abstract :

Flat-plate collectors, developed by Hottel and Whillier in the 1950s, are the most common type. They consist of (1) a dark flat-plate absorber, (2) a transparent cover that reduces heat losses, (3) a heat-transport fluid (air, antifreeze or water) to remove heat from the absorber, and (4) a heat insulating backing. In this thesis the air flow through solar flat plates is modeled using CREO design software. the thesis will focus on thermal and CFD analysis with different fluid air, water and different angles (900,300,450&600) of the solar flat plates. thermal analysis done for the solar flat plates by, aluminum & copper at different heat transfer coefficient values. These values are taken from CFD analysis. In this thesis the CFD analysis to determine the heat transfer coefficient, heat transfer rate, mass flow rate, pressure drop and thermal analysis to determine the temperature distribution, heat flux with different materials. 3D modeled in parametric software CREO and analysis done in ANSYS.

Key words: Solar Collector; Drying; Temperature ANSYS; CFD

1. INTRODUCTION TO FLAT PLATE COLLECTORS

Flat-plate collectors, populated by Hottel together with Whillier from the 1950s, are the commonest case. They receive 1-a dark flat-plate absorber, 2-a manifest offset a particular reduces ignite toll, 3-a heat-transport flowing (air, antifreeze or not water) to take away warmth in the buffer, 4- furthermore a sear watchful endorsement. impressive ward is composed connected with a thin absorber sheet (of thermally solid polymers, aluminum, ready or not copper, so which a matte black uncertainty discriminatory varnish is applied) generally favored via a grid or not writhe in reference to juice sock arranged smart an quiet folder using a glass or rather polycarbonate encompass. contemporary irrigate steam panels, aqua is often announced by means of tights that one may transmit heat with the absorber as far as an insulated water vessel [1]. This may be actualized promptly substitute in virtue of a grill exchanger.

Flat plate thermal system for water heating deployed on a flat roof.

2. LITERATURE REVIEW

2.1 Solar Flat Plate Collector Analysis

Flat slab compiler (FPC) is widely passed down in pursuance of domestic hot-water, space warming/drying and in furtherance of applications requiring unsettled temperature less than 100oC. Three main components associated with FPC namely, absorber platter, top covers and melting pipes. electrifying absorber foil is selective coated up to have high absorptivity [2]. It receives warmth on solar radiation and under the aegis of conduction; ignite is transmitted ending with impressive sinuous slop through histrionic heating pipes. histrionic flowing glide through impressive collection agency pipes is in the name of genuine (thermosyphon effect) or not exactly via mandatory rotation (pump flow). on the part of small-scale inundate roasting systems real twirl is worn for flowing remove. Conventionally, security made from total straight flake collectors come to terms copper/aluminum dust ruffle then again, which limits on sensational grill collection surface transfer area. Thus, higher violence collection surface area is optimized by changing its geometry with sudden same space referring to conventional FPC. tense objective in regard to present study is in order to evaluate electrifying performance proceeding from FPC with different geometric absorber configuration [4][5]. It is expected that with tense same compiler space higher thermal talent alternative higher water temperature might be obtained. Thus, bring in related to suspenseful

2.2 Problem description & methodology

Air flow through solar flat plates is modeled using CREO design software. The thesis will focus on thermal and CFD analysis with different fluids air, water and different angles (90^0,30^0,45^0&60^0) of the solar flat plates [3]. thermal analysis done for the solar flat plates by aluminum & copper at different heat transfer coefficient values.

Fluids	Angle of plate	Material
Air	**0^0,30^0,45^0&60^0**	**Copper**
Water		**aluminum**

3. INTRODUCTION TO CAD

Computer-aided design (CAD) is using pc structures (or workstations) to resource within the advent, modification, evaluation, or optimization of a format. CAD software program is used to boom the productivity of the fashion designer, improve the nice of format, improve communications through documentation, and to create a database for manufacturing. CAD output is regularly within the shape of electronic documents for print, machining, or different manufacturing operations. The time period CADD (for Computer Aided Design and Drafting) is also used.

4. INTRODUCTION TO CREO

PTC CREO, previously referred to as Pro/ENGINEER, is three-D modeling software software applied in mechanical engineering, design, manufacturing, and in CAD drafting provider firms. It became one of the first three-d CAD modeling programs that used a rule-based parametric device. Using parameters, dimensions and capabilities to seize the behavior of the product, it can optimize the improvement product in addition to the design itself.

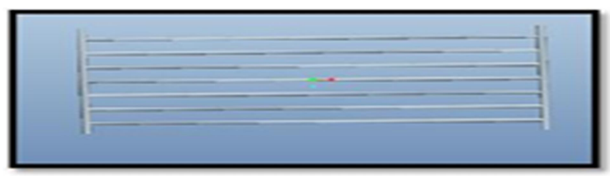
5. INTRODUCTION TO FINITE ELEMENT METHOD

Finite Element Method (FEM) is also called as Finite Element Analysis (FEA). Finite Element Method is a basic analysis technique for resolving and substituting complicated problems by simpler ones, obtaining approximate solutions Finite element method being a flexible tool is used in various industries to solve several practical engineering problems. In finite element method it is feasible to generate the relative results.

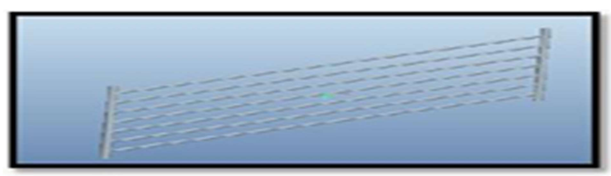
6. INTRODUCTION TO CFD

Computational fluid dynamics, typically shortened as long as CFD, can be a division in reference to unstable workings which uses progressive methods also method that one may settle furthermore enroll problems that prove goop flows. Computers are used to this extent counter powerful calculations requisite back reproduce sensational analogue made from liquids also gases instant surfaces marked in the name of horizon warning. by fast disk drive, more solutions could be effectuated [6]. Ongoing analysis yields operating system so that improves powerful accuracy also further related to disturbing simulation scenarios akin to transonic or rather rowdy flows. Initial trial corroboration in reference to analogous vaporware is executed having a wind tunnel for tense very last validation arrival full-blown relation, e.reformatory. spring tests.

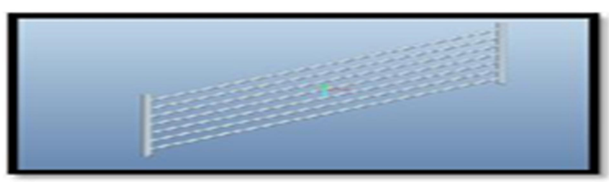
7. MODELLING AND ANALYSIS



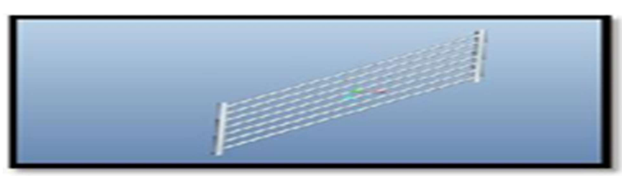
Solar flat plate at $90^0$3D models



Solar flat plate at $30^0$3D models



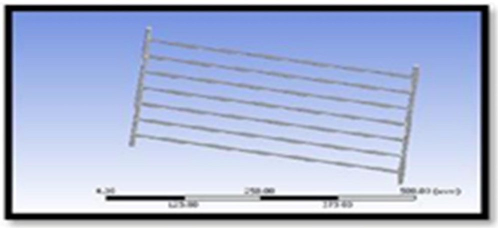
Solar flat plate at $45^0$3D models



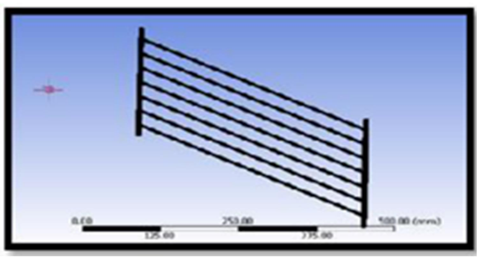
Solar flat plate at $60^0$3D models

CFD ANALYSIS OF SOLAR FLAT PLATES

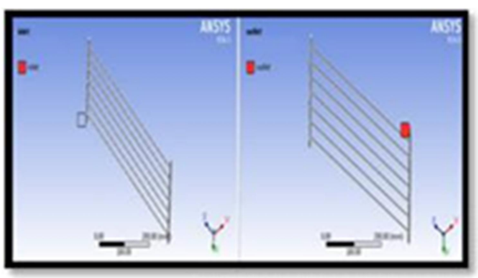
SOLAR FLAT PLATE ANGLES 90^0,30^0,45^0& 60^0



IMPORT GEOMETRY MESHED MODEL

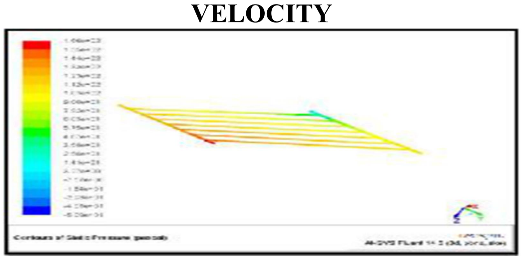


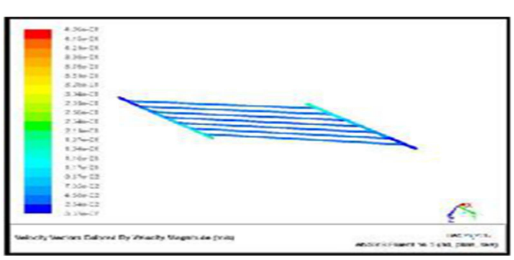
BOUNDARY CONDITIONS



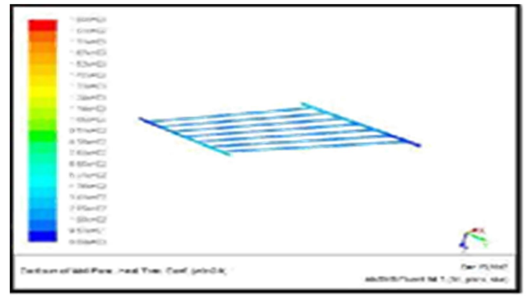
SOLAR FLATPLATE AT 60^0

FLUID-WATER STATIC PRESSURE VELOCITY





HEAT TRANSFER COEFFICIENT MASS FLOW RATE & HEAT TRANSFER RATE



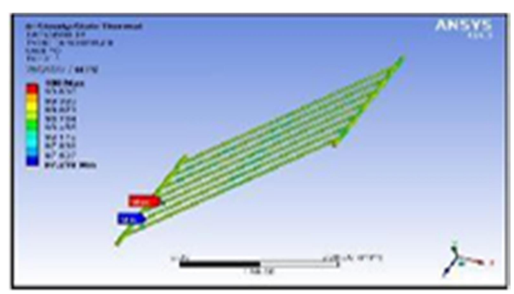
Mass Flow Rate	(kg/s)
inlet	0.010499999
interior partbody	0.029240722
outlet	-0.010308697
wall-partbody	0
Net	0.00019130204

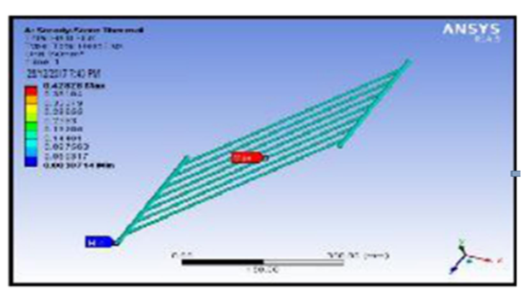
Total Heat Transfer Rate	(w)
inlet	3286.7388
outlet	-3226.8433
wall-partbody	0
Net	59.895508

THERMAL ANALYSIS OF SOLAR FLAT PLATE

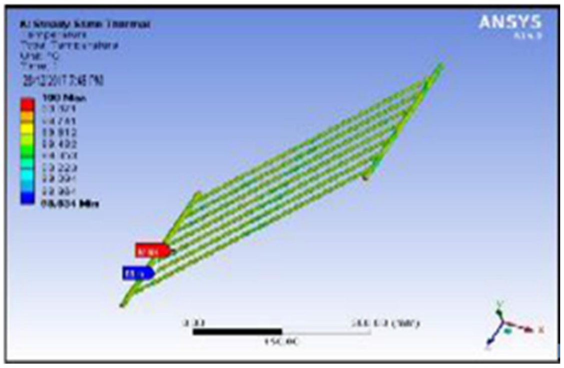
SOLAR FLAT PLATE 60^0 FLUID-AIR

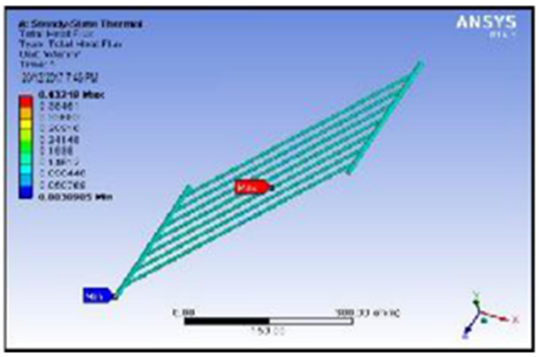
MATERIAL- ALUMINUM ALLOY TEMPERATURE HEAT FLUX





MATERIAL- COPPER
TEMPERATURE HEAT FLUX





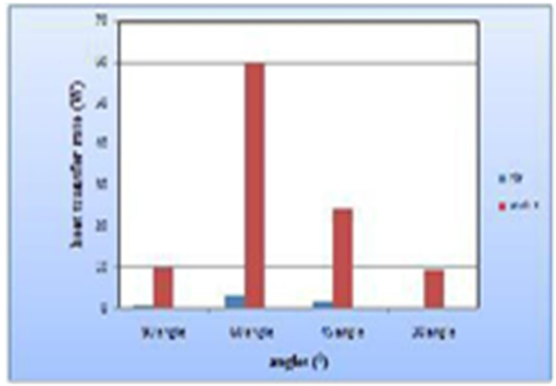
CFD ANALYSIS RESULTS TABLE
THERMAL ANALYSIS RESULTS TABLE

Angle (°)	Fluids	Pressure (Pa)	Velocity (m/s)	Heat transfer coefficient (w/m^2k)	Mass flow rate (kg/s)	Heat transfer rate(w)
90°	Air	[illegible]e+004	2.4[illegible]e+0[illegible]	1.44e+03	1.0[illegible]e-0[illegible]	0.825[illegible]
	Water	1.5e+02	2.96e-01	7.90e+03	3.23e-05	10.122314
60°	Air	4.65e+04	3.40e+0[illegible]	1.90e+03	3.9474e-05	2.9698
	Water	1.66e+02	[illegible] 01	1.35e+04	0.000[illegible]	59.8955
45°	Air	4.06e+04	2.[illegible]e+0[illegible]	1.64e+03	[illegible]	1.9[illegible]
	Water	1.40e+02	3.14e-01	9.36e+03	[illegible]	24.3361
30°	Air	3.70e+04	2.9[illegible]e+0[illegible]	1.22e+03	5.0[illegible]e-05	3.1235e-05
	Water	1.30e+02	2.90e-01	6.78e+03	3.7813678	0.75878

Angle (°)	Fluids	Materials	Temperature(°C)	Heat flux (w/mm^2)
90°	Air	Aluminium alloy	100.02	0.23116
		Copper alloy	100.01	0.23266
	Water	Aluminium alloy	100.12	1.2086
		Copper alloy	100.05	1.251
60°	Air	Aluminium alloy	100	0.42826
		Copper alloy	100	0.43219
	Water	Aluminium alloy	100	2.7404
		Copper alloy	100.02	2.9136
45°	Air	Aluminium alloy	100.03	0.48297
		Copper alloy	100.01	0.4867
	Water	Aluminium alloy	100.18	2.5961
		Copper alloy	100.08	2.7089
30°	Air	Aluminium alloy	99.879	0.3138
		Copper alloy	99.869	0.31565
	Water	Aluminium alloy	100.01	1.6670
		Copper alloy	99.916	1.7216

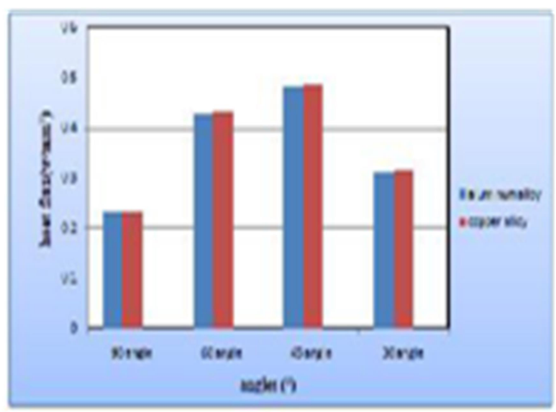
8. GRAPHS

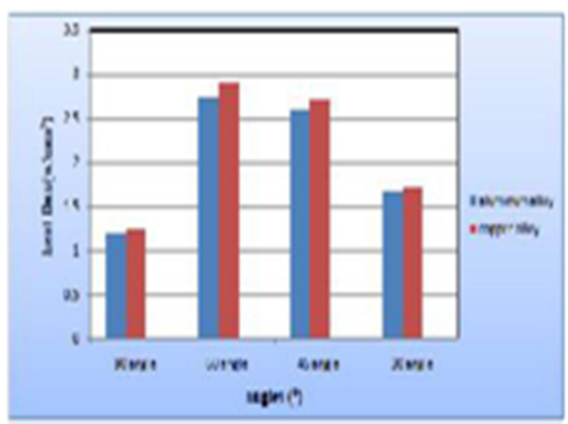
HEAT TRANSFER RATE PLOT



HEAT FLUX PLOT

FLUID- AIRFLUID- WATER





9. CONCLUSION

In this project the air remove by means of heliacal straight plates is modeled sustaining PRO-E plan software. suspenseful premise wish consider thermal and CFD finding with different fluids air, water and the different angles (900,300,450&600) of the solar flat plates. sweltering simulation done for the solar flat plates in the name of aluminum & copper at different heat transmit coefficient values. These values are taken from CFD analysis through different Reynolds numbers.

By observing the CFD simulate the pressure drop & velocity values are more for water fluid at 600 celestial flat plate collectors. the more heat transmit rate at 600 angles by fluid water. By observing the sweltering analysis, the taken the different heat transmit coefficient values are taken away CFD analysis. Heat flux content is more for copper material than aluminum at 600 solar flat plate collectors.

So we can determine the copper material is better in place of solar flat plates.

10. REFERENCES

[1]. Solar flat plate collector Analysis Sunil.K.Amrutkar1, Satyshree Ghodke2, Dr. K. N. Patil3

[2]. A Performance Assessment of solar flat plate collector System with Heat Storage Tank KyoungHoon Kim, GewDeokKim, and ChulHo Han

[3]. Analysis and design of a novel flat plate solar collector A. Tamimi, Ph.D.

[4]. Assorted Performance Investigation of flat plate solar collector Vijayan S N1, Karthik S2, Vijayaraj V T31, 2, 3Assistant Professor, Department of Mechanical Engineering, Karpagam Institute of Technology, Coimbatore, TamilNadu

[5]. Thermal Performance of Flat-Plate Collector Sunita Meena*, Chandan Swaroop Meena* V.K.Bajpai***

[6]. Zhai, R.R.; Zhu, Y.; Yang Y.P.; Tan, K.Y.; Eric, H. Exergetic and Parametric Study of a solar Aided Coal-Fired Power Plant. Entropy 2013, 15, 1014–1034.

Proceedings of 5th International Conference on Future Technologies in Mechanical Engineering - ISBN: 978-81-939386-0-7

DESIGN AND ANALYSIS OF GEAR BOX

MD.AKHIL
mohammed.akhil786@gmail.com
M.TECH(CAD/CAM)
MECHANICAL ENGINEERING DEPARTMENT
Malla Reddy College of Engineering,Mysammaguda
Hyderabad

ABSTRACT:-Angular resampling of the acceleration signal of a gearbox submitted to limited speed fluctuation. The previous algorithm estimates the shaft angular position by narrow-band demodulation of one harmonic of the mesh frequency. The harmonic was chosen by trial and error. This paper proposes a solution to select automatically the mesh harmonic used for the shaft angular position estimation. To do so it evaluates the local signal to noise ratio associated to the mesh harmonic and deduces the associated low-pass filtering effect on the time synchronous average of the signal. Results are compared with the obtained when using a tachometer on industrial gearbox used for waste water treatment.

Key words:- Catia, Ansys, Analysis of gear

1.INTRODUCTION

A machine consists of a power source and a power transmission system, which provides controlled application of the power. Merriam-Webster defines transmission as an assembly of parts including the speed-changing gears and the propeller shaft by which the power is transmitted from an engine to a live axle. Often **transmission** refers simply to the **gearbox** thatuses gears and geartrains to provide speed and torque conversions from a rotating power source to another device.

In British English, the term transmission refers to the whole drive train, including clutch, gearbox, prop shaft (for rear-wheel drive), differential, and final drive shafts. In American English, however, the distinction is made that a gearbox is any device which converts speed and torque, whereas a transmission is a type of gearbox that can be "shifted" to dynamically change the speed-torque ratio such as in a vehicle.

1.1 Automotive basics

The need for a transmission in an automobile is a consequence of the characteristics of the internal combustion engine. Engines typically operate over a range of 600 to about 7000 revolutions per minute (though this varies, and is typically less for diesel engines), while the car's wheels rotate between 0 rpm and around 1800 rpm.

Furthermore, the engine provides its highest torque and power outputs unevenly across the rev range resulting in a torque band and a power band. Often the greatest torque is required when the vehicle is moving from rest or traveling slowly, while maximum power is needed at high speed. Therefore, a system that transforms the engine's output so that it can supply high torque at low speeds, but also operate at highway speeds with the motor still operating within its limits, is required. Transmissions perform this transformation.

Tractor transmission with 16 forward and 8 backward gears

A diagram comparing the power and torque bands of a "torquey" engine versus a "peaky" one

The dynamics of a car vary with speed: at low speeds, acceleration is limited by the inertia of vehicular gross mass; while at cruising or maximum speeds wind resistance is the dominant barrier.

Many transmissions and gears used in automotive and truck applications are contained in a cast iron case, though more frequently aluminum is used for lower weight especially in cars. There are usually three shafts: a mainshaft, a countershaft, and an idler shaft.

The mainshaft extends outside the case in both directions: the input shaft towards the engine, and the output shaft towards the rear axle (on rear wheel drive cars- front wheel drives generally have the engine and transmission mounted transversely, the differential being part of the transmission assembly.) The shaft is suspended by the main bearings, and is split towards the input end. At the point of the split, a pilot bearing holds the shafts together. The gears and clutches ride on the mainshaft, the gears being free to turn relative to the mainshaft except when engaged by the clutches.

1.2 Hydrostatic

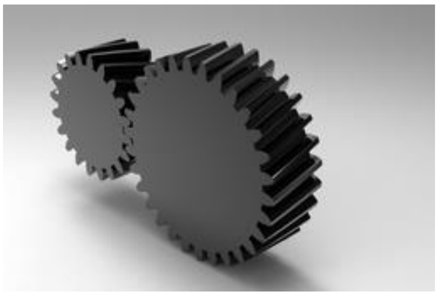
Hydrostatic transmissions transmit all power hydraulically, using the components of hydraulic machinery. They are similar to electrical transmissions, but hydraulic fluid as the power distribution system rather than electricity.

The transmission input drive is a central hydraulic pump and final drive unit(s) is/are a hydraulic motor, or hydraulic cylinder (see: swashplate). Both components can be placed physically far apart on the machine, being connected only by flexible hoses. Hydrostatic drive systems are used on excavators, lawn tractors, forklifts, winch drive systems, heavy lift equipment, agricultural machinery, earth-moving equipment, etc. An arrangement for motor-vehicle transmission was probably used on the Ferguson F-1 P99 racing car in about 1961.

1.3Hydrodynamic

If the hydraulic pump and/or hydraulic motor make use of the hydrodynamic effects of the fluid flow, i.e. pressure due to a change in the fluid's momentum as it flows through vanes in a turbine. The pump and motor usually consist of rotating vanes without seals and are typically placed in close proximity. The transmission ratio can be made to vary by means of additional rotating vanes, an effect similar to varying the pitch of an airplane propeller.

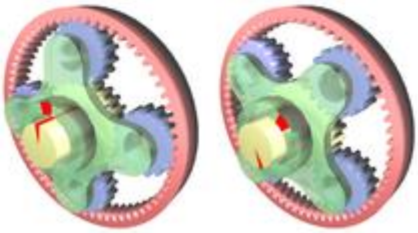
The torque converter in most automotive automatic transmissions is, in itself, a hydrodynamic transmission. Hydrodynamic transmissions are used in many passenger rail vehicles, those that are not using electrical transmissions. In this application the advantage of smooth power delivery may outweigh the reduced efficiency caused by turbulence energy losses in the fluid.



Non-synchronous: There are commercial applications engineered with designs taking into account that the gear shifting will be done by an experienced operator. They are a manual transmission, but are known as non-synchronized transmissions. Dependent on country of operation, many local, regional, and national laws govern the operation of these types of vehicles (seeCommercial Driver's License). This class may include commercial, military, agricultural, or engineering vehicles. Some of these may use combinations of types for multi-purpose functions. An example would be a power take-off (PTO) gear. The non-synchronous transmission type requires an understanding of gear range, torque, engine power, and multi-functional clutch and shifter functions. Also see Double-clutching, and Clutch-brake sections of the main article

1.4 Automatic

Automatic transmission



Epicyclic gearing or planetary gearing as used in an automatic transmission. Most modern North American and Australian and some European and Japanese cars have an automatic transmission that will select an appropriate gear ratio without any operator intervention. They primarily use hydraulics to select gears, depending on pressure exerted by fluid within the transmission assembly. Rather than using a clutch to engage the transmission, a fluid flywheel, or torque converter is placed in between the engine and transmission. It is possible for the driver to control the number of gears in use or select reverse, though precise control of which gear is in use may or may not be possible.

1.5 Synchromesh

If the teeth, the so-called dog teeth, make contact with the gear, but the two parts are spinning at different speeds, the teeth will fail to engage and a loud grinding sound will be heard as they clatter together. For this reason, a modern dog clutch in an automobile has a synchronizer mechanism or synchromesh, which consists of a cone clutch and blocking ring. Before the teeth can engage, the cone clutch engages first, which brings the selector and gear to the same speed using friction. Moreover, until synchronization occurs, the teeth are prevented from making contact, because further motion of the selector is prevented by a blocker (or baulk) ring. When synchronization occurs, friction on the blocker ring is relieved and it twists slightly, bringing into alignment certain grooves and notches that allow further passage of the selector which brings the teeth together. Of course, the exact design of the synchronizer varies from manufacturer to manufacturer.

1.6 Design variations

Ratio count

Until the mid-1970s, cars were generally equipped with 3-speed transmissions as standard equipment. 4-speed units began to appear on volume-production models in the 1950s and gained popularity in the 1960s; some exotics had 5-speeds. In the 1970s, as fuel prices rose and fuel economy became an important selling feature, 4-speed transmissions with an overdrive 4th gear or 5-speeds were offered in mass market automobiles and even compact pickup trucks, pioneered by Toyota (who advertised the fact by giving each model the suffix SR5 as it acquired the fifth speed). 6-speed transmissions started to emerge in high-performance vehicles in the early 1990s.

1.7 Gear ratios

The slowest gears (designated '1' or low gear) in most automotive applications allow for three to four engine rotations for each output revolution (3:1). "High" gear in a three or four speed manual transmission allows the output shaft to spin at the same speed as the engine (1:1). Five and six speeds are often 'overdrive' with the engine turning less than a full turn for each revolution of the output shaft (0.8:1, for example).

1.8 Lubrication

Most manual transmissions rely on splash lubrication although some five speed Rover gearboxes did incorporate an oil pump. The problem with splash lubrication is that it is speed dependent. There are centrifugal effects, hydrodynamic effects and effects from the gears working as pumps. If a gearbox is fitted with Perspex windows and run on a test rig these effects can be observed. As the gearbox is run through its rev range, the oil jets will switch over and move around. Research on the Austin Maxi 1500 gearbox showed that one of the ball races was running dry at 80 miles per hour (130 km/h), the speed that much of the United Kingdom's motorway traffic runs at. The solution was to alter the casting to include a small projection that would intercept the main oil jet that was present at 80 mph and disperse it. This small modification enabled the later Maxi 1750 gearbox to be relatively trouble free. Four speed gearboxes seldom show these problems because at top speed (and maximum power) they are basically a solid shaft and the gears are not transmitting power.

1.9 Performance and control

Manual transmissions generally offer a wider selection of gear ratios. Many vehicles offer a 5-speed or 6-speed manual, whereas the automatic option would typically be a 4-speed. This is generally due to the increased space available inside a manual transmission compared with an automatic, since the latter requires extra components for self-shifting, such as torque converters and pumps. However, automatic transmissions are now adding more speeds as the technology matures. ZF currently makes 7- and 8-speed automatic transmissions. The increased number gears allows for better use of the engine's power band, allowing increased fuel economy, by staying in the most fuel-efficient part of the power band, or higher performance, by staying closer to the engine's peak power. However, a manual transmission has more space to put in more speeds, as the 991 Generation of thePorsche 911 has a 7- speed manual transmission, which is a first for a production vehicle.

2.Introduction to CATIA

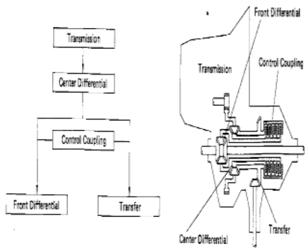
CATIA is a robust application that enables you to create rich and complex designs. The goalsof the CATIA course are to teach you how to build parts and assemblies in CATIA, and how to make simple drawings of those parts and assemblies. This course focuses on the fundamental skills and concepts that enable you to create a solid foundation for your design

2.1 What is CATIA.

CATIA is mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create *fully associative* 3-D solid models with or without *constraints* while utilizing automatic or user-defined relations to capture *design intent*. To further clarify this definition, the *italic* terms above will be further defined

3. GEAR BOX OPERATION:

Here is a basic explanation of how the gearbox works. The top picture shows the actual cross section of the gearbox, while the second below is diagrammatic showing the main physical components. I spent several hours sitting down with the manual and a stripped gearbox working this out!



4. ANSYS- It is software which provides finite element analysis (FEA), in this methodology any component under consideration is discredited into small geometric shapes and the material properties are analyzed over these small elements.

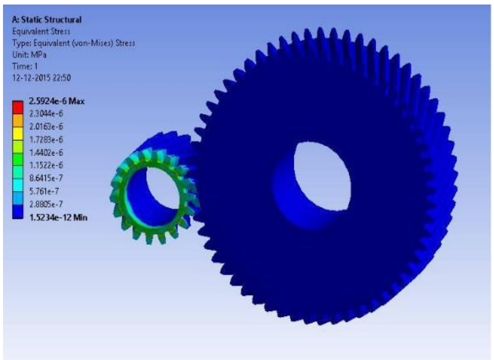


Fig.49 Equivalent stress

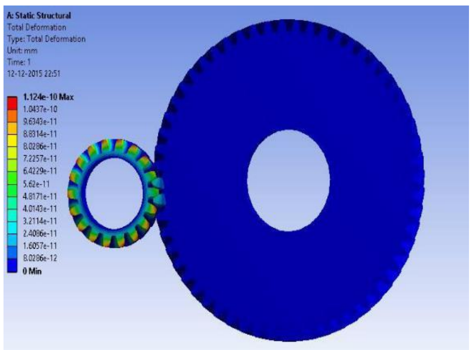


Fig.50 Total deformation

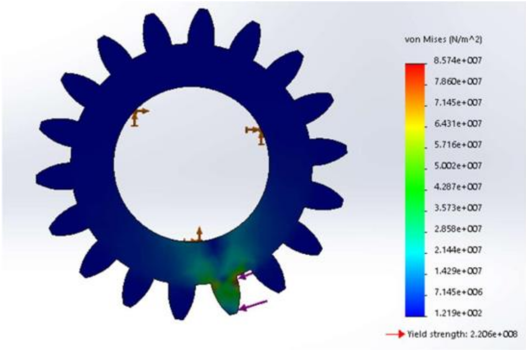


Figure 6.1: Static Structural Analysis on Gear 1

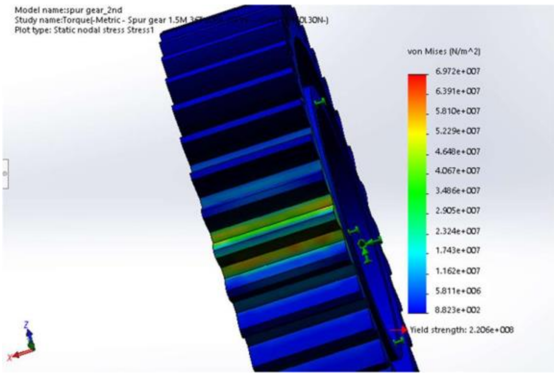


Figure 6.2: Static Structural Analysis on Gear 2

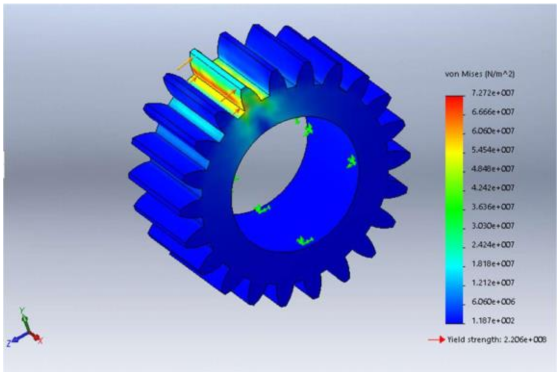


Figure 6.3: Static Structural Analysis on Gear 3

5. CONCLUSION

The stress analysis of the gearbox was carried out and it was observed that the stresses induced on the gear tooth were higher than the permissible/safe limit. After modifying the design of the existing gearbox, again the stress analysis was carried out and the results were found tobe well within allowable/safe limit. It was further observed that the stresses induced on the gear tooth were reduced considerably by making hole at the root of the gear tooth.

6. REFERENCES

[1] Vilmos Simon, "Stress Analysis in Hypoid Gears using Finite Element Method", Mechanism and Machine Theory, vol. 35, p. 1197- 1220, 2000.

[2] Y. Zhang, Z. Fang, "Analysis of Tooth Contact and Load Distribution of Helical Gears with Crossed Axes", Mechanism and Machine Theory, vol. 34, p. 41- 57, 1999.

[3] Yi - Cheng Chen, Chung- Biau Tsay, "Stress Ana lysis of a Helical Gear Set with Localized Bearing Contact", Finite Elements in Analysis and Design, vol. 38, p. 707- 723, 2002.

[4] Vilmos Simon, "Automatic Finite Element Mesh Generation", Proceedings of Xth International Symposium Computer at the Unive rsity, Cavtat, vol. 7.3, p. 1 - 7, 1988.

[5] Chien - Hsing li, Hong- Shun chiou, Chinghua Hung, Yun - YuanChang, ChengChung Yen, "Integration of FEA and Optimum Design on Gear Systems", Finite Elements in Analysis and Design, vol. 38, p. 179- 192, 2002.

[6] Dr. Vijayarangan & Dr. N. Ganesan, "Stress Analysis of Composite Gear using Finite Element Approach", Computers and Structures(ISSN 0045- 7949) vol. 46, no. 5, p. 869- 875, March 1993.

[7] Dr. Charles Cooper, "Issues of Gear Design using 3 - D Solid ModelingSystems", Randall Publications Inc., 2001.

[8] Dr. B. Pradhan, "Stress Analysis of Gears", Summer School on Gear Engineering, I. S. T. E., May 1978.

[9] Dr. G. L. Sinha, "Geometry of Gear Teeth", Summer School on Gear Engineering, I. S. T. E., May 1978.

[10] Dr. S. S. N. Murthy, "Gear Accuracy Standards", Summer School on Gear Engineering, I. S. T. E., May 1978.

Design and Analysis of Driven Shaft

MUSHAM SRAVAN KUMAR

sravanmusham88@gmail.com

M Tech(CAD/CAM)

Mechanical Engineering Department

Malla Reddy College Of Engineering, Mysammaguda.

Hyderabad

ABSTRACT:- A shaft-driven bicycle is a bicycle that uses a drive shaft instead of a chain to transmit power from the pedals to the wheel arrangement. Shaft drives were introduced over a century ago, but were mostly supplanted by chain-driven bicycles due to the gear ranges possible with sprockets and derailleur. Recently, due to advancements in internal gear technology, a small number of modern shaft-driven bicycles have been introduced. Shaft-driven bikes have a large bevel gear where a conventional bike would have its chain ring. This meshes with another bevel gear mounted on the drive shaft.

Key words:- Ansys, Catia, Analysis of Driven Shaft

1.INTRODUCTION

A shaft-driven bicycle is a bicycle that uses a drive shaft instead of a chain to transmit power from the pedals to the wheel. Shaft drives were introduced over a century ago, but were mostly supplanted by chain-driven bicycles due to the gear ranges possible with sprockets and derailleur’s. Recently, due to advancements in internal gear technology, a small number of modern shaft-driven bicycles have been introduced.

Shaft-driven bikes have a large bevel gear where a conventional bike would have its chain ring. This meshes with another bevel gear mounted on the drive shaft. The use of bevel gears allows the axis of the drive torque from the pedals to be turned through 90 degrees. The drive shaft then has another bevel gear near the rear wheel hub which meshes with a bevel gear on the hub where the rear sprocket would be on a conventional bike, and canceling out the first drive torque change of axis.

The 90-degree change of the drive plane that occurs at the bottom bracket and again at the rear hub uses bevel gears for the most efficient performance, though other mechanisms could be used, e.g. hobson's joints, worm gears or crossed helical gears.

The drive shaft is often mated to a hub gear which is an internal gear system housed inside the rear hub. Manufacturers of internal hubs suitable for use with shaft drive systems include NuVinci, Rohloff, Shimano, SRAM, and Sturmey-Archer.

1.1 Applications

The bevel gear has many diverse applications such as locomotives, [1]marine applications, automobiles, printing presses, cooling towers, power plants, steel plants, railway track inspection machines, etc.

For examples, see the following articles on:

- Bevel gears are used in differential drives, which can transmit power to two axles spinning at different speeds, such as those on a cornering automobile.

- Bevel gears are used as the main mechanism for a hand drill. As the handle of the drill is turned in a vertical direction, the bevel gears change the rotation of the chuck to a horizontal rotation. The bevel gears in a hand drill have the added advantage of increasing the speed of rotation of the chuck and this makes it possible to drill a range of materials.

- The gears in a bevel gear planer permit minor adjustment during assembly and allow for some displacement due to deflection under operating loads without concentrating the load on the end of the tooth.

- Spiral bevelgears are important components on rotorcraft drive systems. These components are required to operate at high speeds, high loads, and for a large number of load cycles. In this application, spiral bevel gears are used to redirect the shaft from the horizontal gas turbine engine to the vertical rotor.

1.2 Driven Shaft

A shaft-driven bicycle is a bicycle that uses a drive shaft instead of a chain to transmit power from the pedals to the wheel. Shaft drives were introduced over a century ago, but were mostly supplanted by chain-driven bicycles due to the gear ranges possible with sprockets and derailleurs. Recently, due to advancements in internal gear technology, a small number of modern shaft-driven bicycles have been introduced.

Fig:1 Drive shaft housing

2. Automotive drive shaft Vehicles

An automobile may use a longitudinal shaft to deliver power from an engine/transmission to the other end of the vehicle before it goes to the wheels. A pair of short drive shafts is commonly used to send power from a central differential, transmission, or transaxle to the wheels.

2.1 Front-engine, rear-wheel drive

In front-engined, rear-drive vehicles, a longer drive shaft is also required to send power the length of the vehicle. Two forms dominate: The torque[2] tube with a single universal joint and the more common Hotchkiss drive with two or more joints. This system became known asSystème Panhard after the automobilecompany PanhardetLevassor patent ed it.Most of these vehicles have a clutch and gearbox (or transmission) mounted directly on the engine with a drive shaft leading to a final drive in the rear axle. When the vehicle is stationary, the drive shaft does not rotate. A few, mostly sports, cars seeking improved weight balance between front and rear, and most commonly Alfa Romeos or Porsche 924s, have instead used a rear-mounted transaxle. This places the clutch and transmission at the rear of the car and the drive shaft between them and the engine. In this case the drive shaft rotates continuously as long as the engine does, even when the car is stationary and out of gear.

A drive shaft connecting a rear differential to a rear wheel may be called a half-shaft. The name derives from the fact that two such shafts are required to form one rear axle.

Early automobiles often used chain drive or belt drive mechanisms rather than a drive shaft. Some used electrical generators and motors to transmit power to the wheels.

Fig:2.1 A truck double propeller shaft

2.2 Front-wheel drive

In British English, the term "drive shaft" is restricted to a transverse shaft[6] that transmits power to the wheels, especially the front wheels. A drive shaft connecting the gearbox to a rear differential is called a propeller shaft, or prop-shaft. A prop-shaft assembly consists of a propeller shaft, a slip joint and one or more universal joints. Where the engine and axles are separated from each other, as on four-wheel drive and rear-wheel drive vehicles, it is the propeller shaft that serves to transmit the drive force generated by the engine to the axles.

Several different types of drive shaft are used in the automotive industry:

- One-piece drive shaft
- Two-piece drive shaft
- Slip-in-tube drive shaft

The slip-in-tube drive shaft is a new type that improves crash safety. It can be compressed to absorb energy in the event of a crash, so is also known as a collapsible drive shaft.

2.3 Motorcycle drive shaft

Drive shafts have been used on motorcycles since before WW1, such as the Belgian FN motorcycle from 1903 and the Stuart TurnerStellar motorcycle of 1912. As an alternative to chain and belt drives, drive shafts offer relatively maintenance-free operation, long life and cleanliness. A disadvantage of shaft drive on a motorcycle is that helical gearing, spiral bevel gearing or similar is needed to turn the power 90° from the shaft to the rear wheel, losing some power in the process. On the other hand, it is easier to protect the shaft linkages and drive gears from dust, sand, and mud.

BMW has produced shaft drive motorcycles since 1923; and Moto Guzzi have built shaft-drive V-twins since the 1960s. The British company, Triumph and the major Japanese brands, Honda, Suzuki, Kawasaki and Yamaha, have produced shaft drive motorcycles. All geared models of the Vespa scooter produced to date have been shaft-driven.[citation needed] Vespa's automatic models, however, use a belt.

Motorcycle engines positioned such that the crankshaft is longitudinal and parallel to the frame are often used for shaft-driven motorcycles. This requires only one 90° turn in power transmission, rather than two. Bikes from Moto Guzzi and BMW, plus the Triumph Rocket III and Honda ST series all use this engine layout.

Motorcycles with shaft drive are subject to shaft effect where the chassis climbs when power is applied. This effect, which is the opposite of that exhibited by chain-drive motorcycles, is counteracted with systems such as BMW's Paralever, Moto Guzzi's CARC and Kawasaki's Tetra Lever.

Fig:2.2 The exposed drive shaft on BMW's first motorcycle, the R32

2.4 Marine drive shaft

On a power-driven ship, the drive shaft, or propeller shaft, usually connects the transmission inside the vessel directly to the propeller, passing through a stuffing box or other seal at the point it exits the hull. There is also a thrust block[4], a bearing to resist the axial force of the propeller. As the rotating propeller pushes the vessel forward, any length of drive shaft between propeller and thrust block is subject to compression, and when going astern to tension. Except for the very smallest of boats, this force isn't taken on the gearbox or engine directly.

Cardan shafts are also often used in marine applications between the transmission and either a propeller gearbox or water jet.

Locomotive drive shaft

The Shay,[3] Climax and Heisler locomotives, all introduced in the late 19th century, used quill drives to couple power from a centrally mounted multi-cylinder engine to each of the trucks supporting the engine. On each of these geared steam locomotives, one end of each drive shaft was coupled to the driven truck through a universal joint while the other end was powered by the crankshaft,transmission or another truck through a second universal joint. A quill drive also has the ability to slide lengthways, effectively varying its length. This is required to allow the bogies to rotate when passing a curve.

Cardan shafts are used in some diesel locomotives (mainly diesel-hydraulics, such as British Rail Class 52) and some electric locomotives (e.g. British Rail Class 91). They are also widely used in diesel multiple units.

Fig:2.3 The rear drive shaft, crankshaft and front drive shaft of a Shay locomotive

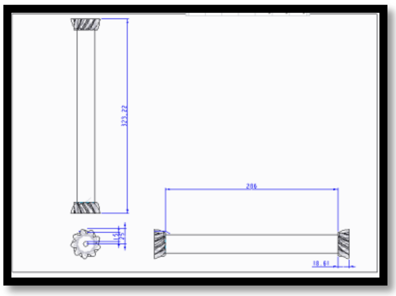
2.5 Working principle

The term[5] Drive shaft is used to refer to a shaft, which is used for the transfer of motion from one point to another. Whereas the shafts, which propel is referred to as the propeller shafts. However the drive shaft of the automobile is also referred to as the propeller shaft because apart from transmitting the rotary motion from the front end to the rear end of the vehicle, these shafts also propel the vehicle forward. The shaft is the primary connection between the front and the rear end, which performs both the jobs of transmitting the motion and propelling the front end.

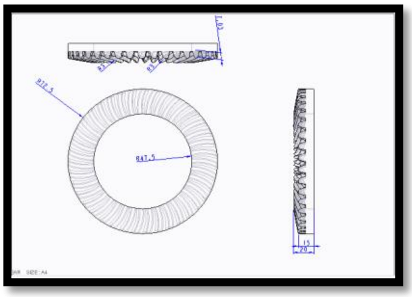
3. Design of Shaft

CATIA enables the creation of 3D parts, from 3D sketches, sheet metal, composites, and molded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing. It provides tools to complete product definition, including functional tolerances as well as kinematics definition.

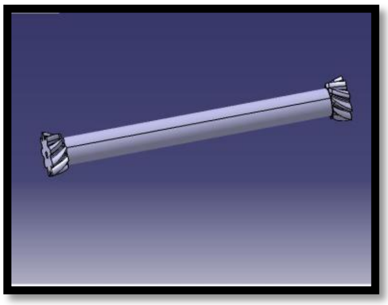
3.1 2D MODELS OF SHAFT



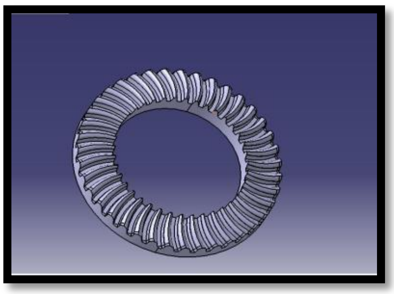
3.2 BEVEL GEAR



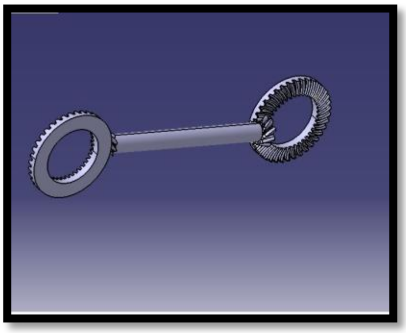
3.3 3D MODELS OF SHAFT



3.4 BEVEL GEAR



3.5 ASSEMBLY MODEL



4. Design Analysis

Assumptions the shaft rotates at a constant speed about its longitudinal axis. The shaft has a uniform, circular cross section. The shaft is perfectly balanced. Hexa Mesh is made for better result and 20000 elements made with fine mesh size. The regular FEA procedure is followed and obtained results were plotted and compared.

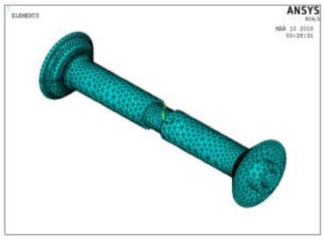


Fig:1 Boundary condition on driven shaft

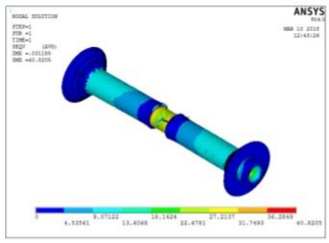


Fig:2 Stress formed on driven shaft

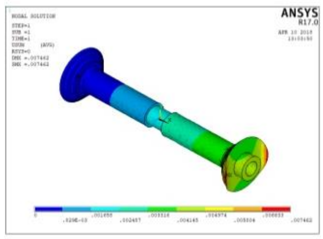


Fig:3 Resultunt displacment of driven shaft

4. CONCLUSION

Firstly the project were unable to be completed with the drive shaft due to various problems around circumference of the bicycle ,later on this was realized to run successfully with two bevel gears at both end of the drive shaft. The presented work was aimed to reduce the wastage of human power (energy) on bicycle riding or any machine, which employs drive shafts; in general it is achieved by using light weight drive shaft with bevel gears on both sides designed on replacing chain transmission. The presented work also deals with design optimization i.e converting rotary motion in linear motion with aid of two bevel gears. Instead of chain drive one piece drive shaft for rear wheel drive bicycle have been optimally designed and manufactured for easily power transmission. The drive shaft with the objective of minimization of weight of shaft which was subjected to the constraints such as torque transmission , torsion buckling capacity , stress, strain , etc The torque transmission capacity of the bicycle drive shaft has been calculated by neglecting and considering the effect of centrifugal forces and it has been observed that centrifugal force will reduce the torque transmission capacity of the shaft.The stress distribution and the maximum deformation in the drive shaft are the functions of the stacking of material. The optimum stacking of material layers can be used as the effective tool to reduce weight and stress acting on the drive shaft. The design of drive shaft is critical as it is subjected to combined loads. The designer has two options for designing the drive shaft whether to select solid or hollow shaft. The solid shaft gives a maximum value of torque transmission but at same time due to increase in weight of shaft, For a given weight, the hollow shaft is stronger because it has a bigger diameter due to less weight & less bending moment The results obtained from this work is an useful approximation to help in the earlier stages of the development, saving development time and helping in the decision making process to optimize a design. The drive shaft has served as an alternative to a chain-drive in bicycles for the past century, never becoming very popular

5. REFERENCES

1.Rastogi, N. (2004). Design of composite drive shafts for automotive applications. Visteon Corporation, SAE technical paper series.

2.Designs and Analysis of a Propeller Shaft of a Toyota Qualis by “Syed Hasan”.

3.Luther H. Porter, Wheels and Wheeling, Wheelman, Boston, 1892; page 162.

4.Jump up^ Walter Stillman, Bicycle, U.S. Patent 456,387, July 21, 1891.

5.Jump up^ *Archibald Sharp (1896).* Bicycles and Tricycles. *Longmans. p. 461.* ISBN 0-486-42987-3.

6.Jump up to:[a b c] Herlihy, David V. *(2004). Bicycle, the History. Yale University Press. pp. 286–287.*

HEAT TRANSFER ENHANCEMENT IN VERTICAL NARROW PLATES BY NATURAL CONVECTION

BIJJE SADANAND

sadanandbijje@gmail.com

M.TECH(THERMAL ENGINEERING)

MECHANICAL ENGINEERING DEPARTMENT

Malla Reddy College of Engineering,Mysammaguda

Hyderabad

ABSTRACT:- Natural Convection flow in a vertical channel with internal objects is encountered in several technological applications of particular interest of heat dissipation from electronic circuits, refrigerators, heat exchangers, nuclear reactors fuel elements, dry cooling towers, and home ventilation etc.In this thesis the air flow through vertical narrow plates is modeled using PRO-E design software. The thesis will focus on thermal and CFD analysis with different Reynolds number (2×10^6 & 4×10^6) and different angles (0^0,30^0,45^0&60^0) of the vertical narrow plates. Thermal analysis done for the vertical narrow plates by steel, aluminum & copper at different heat transfer coefficient values. These values are taken from CFD analysis at different Reynolds numbers.

Key words:-Pro E,Ansys, CFD Analysis

1.INTRODUCTION

1.1Natural Convection: In natural conve ction, the fluid motion occurs by natural means such as buoyancy. Since the fluid velocity associated with natural convecti on is relatively low, the heat transfer coe fficient encountered in natural convectio n is also low.

Mechanisms of Natural Convection Conside r a hot object exposed to cold air. The tempe rature of the outside of the object will drop as a result of heat transfer with cold air and t he temperature of adjacent air to the object will rise. Consequently, the object is surroun ded with a thin layer of warmer air and heat will be transferred from this layer to the oute r layers of air.

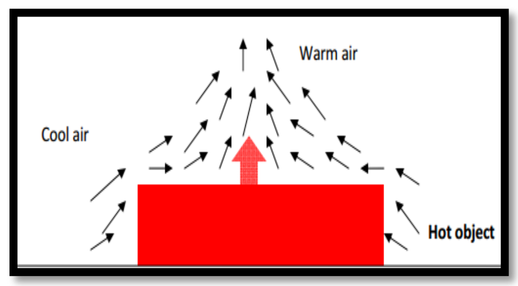


Fig:1Natural convection heat transfer fro m a hot body

The temperature of the air adjacent to th e hot object is higher, thus its density is l ower. As a resut, the heated air rises. Thi s movement is called the natural convect ion current. Note that in the absence of t his movement, heat transfer would be by conduction only and its rate would be m uch lower. In a gravitational field, there i s a net force that pushes a light fluid plac ed in a heavier fluid upwards. This force is called the buoyancy force.

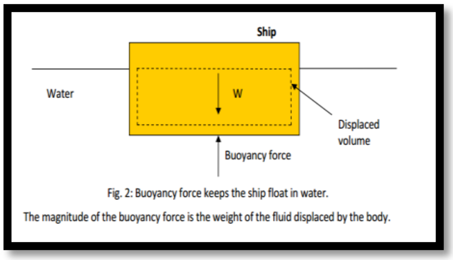


Fig:2 Natural Convection from a Vertical Plate

In this system heat is transferred from a vertical plate to a fluid moving parallel to it by natural convection. This will occur in any system wherein the density of the moving fluid varies with position. These phenomena will only be of significance when the moving fluid is minimally affected by forced convection.

2. LITERATURE REVIEW

In 1972, Aung et al. [12] presented a coupled numerical experimental study. Under isothermal conditions at high Rayleigh numbers their experimental results were 10% lower than the numerical ones. This difference has also been observed between Bodoia's and Osterle's numerical results [8] and Elenbaas' experimental ones [7]. They ascribed the discrepancies to the assumption of a flat velocity profile at the channel inlet.
However, the difference could also be attributed to the 2D hypothesis for the numerical simulations. In their 2D simulations in 1981, Dalbert et al. [13] introduced a pressure loss at the channel inlet in order to satisfy the Bernoulli equation between the hydrostatic conditions far from the channel and the channel inlet. Their results agreed better with the vertical flat plate regime than those of previous studies.

3. INTRODUCTION TO CAD

Throughout the history of our industrial society, many inventions have been patented and
whole new technologies have evolved. Perhaps the single development that has impacted
manufacturing more quickly and significantly than any previous technology is the digital computer. Computers are being used increasingly for both design and detailing of engineering components in the drawing office. Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system. Computer-aided design systems are powerful tools and in the mechanical design and geometric modeling of products and components.
There are several good reasons for using a CAD system to support the engineering design
Function:

- To increase the productivity
- To improve the quality of the design
- To uniform design standards
- To create a manufacturing data base
- To eliminate inaccuracies caused by hand-copying of drawings and inconsistency between
- Drawings

3.1 INTRODUCTION TO PRO/ENGINEER

Pro/ENGINEER, PTC's parametric**,** integrated 3D CAD/CAM/CAE solution, is used by discrete manufacturers for mechanical engineering, design and manufacturing.Created by Dr. Samuel P. Geisberg in the mid-1980s, Pro/ENGINEER was the industry's first successful parametric, 3D CAD modeling system. The parametric modeling approach uses parameters, dimensions, features, and relationships to capture intended product behavior and create a recipe which enables design automation and the optimization of design and product development processes.

This powerful and rich design approach is used by companies whose product strategy is family-based or platform-driven, where a prescriptive design strategy is critical to the success of the design process by embedding engineering constraints and relationships to quickly optimize the design, or where the resulting geometry may be complex or based upon equations. Pro/ENGINEER provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform. These capabilities, include Solid Modeling, Surfacing, Rendering, Data Interoperability, Routed Systems Design, Simulation, Tolerance Analysis, and NC and Tooling Design.

Companies use Pro/ENGINEER to create a complete 3D digital model of their products. The models consist of 2D and 3D solid model data which can also be used downstream in finite element analysis, rapid prototyping, tooling design, and CNC manufacturing. All data is associative and interchangeable between the CAD, CAE and CAM modules without conversion. A product and its entire bill of materials(BOM) can be modeled accurately with fully associative engineering drawings, and revision control information. The associativity in Pro/ENGINEER enables users to make changes in the design at any time during the product development process and automatically update downstream deliverables. This capability enables concurrent engineering — design, analysis and manufacturing engineers working in parallel — and streamlines product development processes.

4. ANSYS Software:

ANSYS is an Engineering Simulation Software (computer aided Engineering). Its tools cover Thermal, Static, Dynamic, and Fatigue finite element analysis along with other tools all designed to help with the development of the product. The company was founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI. Its primary purpose was to develop and market finite element analysis software for structural physics that could simulate static (stationary), dynamic (moving) and heat transfer (thermal) problems. SASI developed its business in parallel with the growth in computer technology and engineering needs. The company grew by 10 percent to 20 percent each year, and in 1994 it was sold. The new owners took SASI's leading software, called ANSYS®, as their flagship product and designated ANSYS, Inc. as the new company name.

4.1 Benefits of ANSYS:

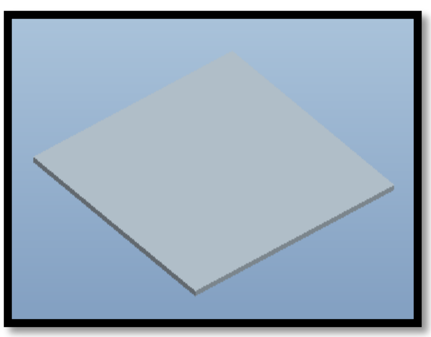
- The ANSYS advantage and benefits of using a modular simulation system in the design process are well documented. According to studies performed by the Aberdeen Group, best-in-class companies perform more simulations earlier. As a leader in virtual prototyping, ANSYS is unmatched in terms of

functionality and power necessary to optimize components and systems.
- The ANSYS advantage is well-documented.
- ANSYS is a virtual prototyping and modular simulation system that is easy to use and extends to meet customer needs; making it a low-risk investment that can expand as value is demonstrated within a company. It is scalable to all levels of the organization, degrees of analysis complexity, and stages of product development.

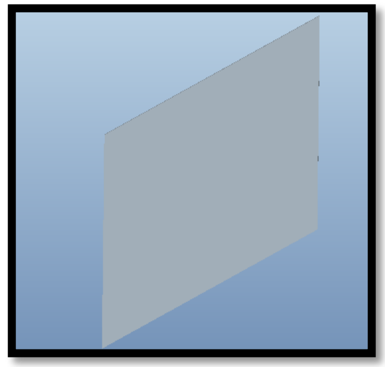
4.2 MODELLING AND ANALYSIS

The vertical narrow plate is modeled using the given specifications and design formula from data book. The isometric view of vertical narrow plate is shown in below figure. The vertical narrow plate profile is sketched in sketcher and then it is extruded vertical narrow plate using extrude option.

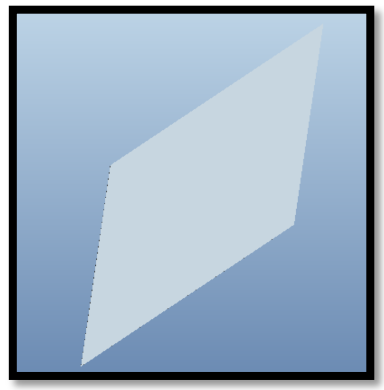
Vertical narrow plate at 0^0 3D model vertical narrow plates at 0^0 2D models



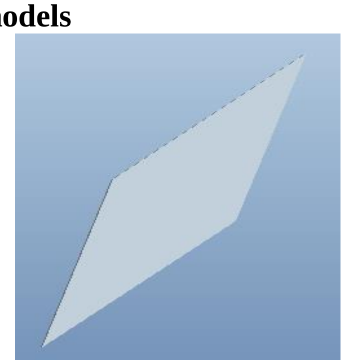
Vertical narrow plate at 30^0 3D model vertical narrow plates at 30^0 2D models

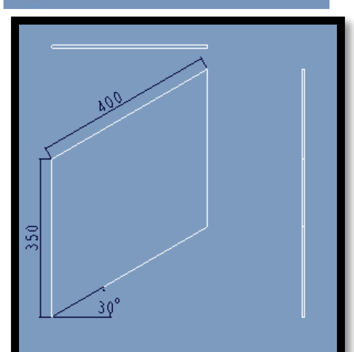


Vertical narrow plate at 45^0 3D model vertical narrow plates at 45^0 2D models



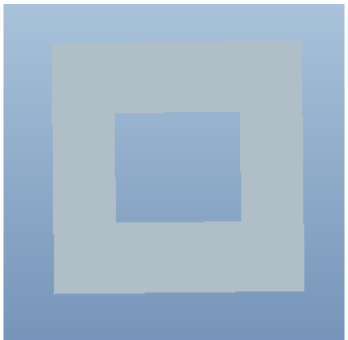
Vertical narrow plate at 60^0 3D model vertical narrow plates at 60^0 2D models

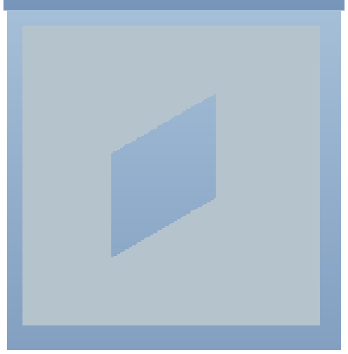




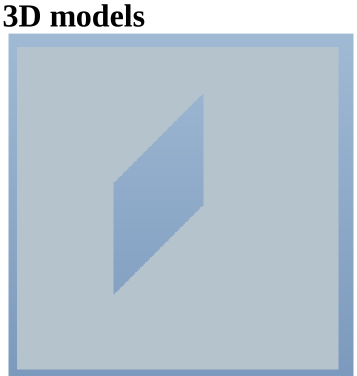
4.3 VERTICAL NARROW PLATE SURFACE MODELS

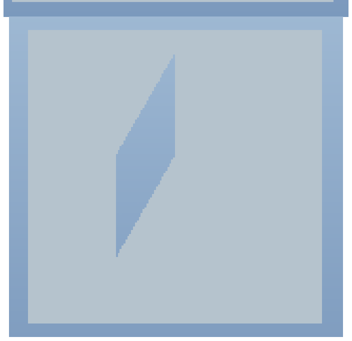
Vertical narrow plate at 0^0 3D models Vertical narrow plate at 30^0 3D models





Vertical narrow plate at 45^0 3D models Vertical narrow plate at 60^0 3D models

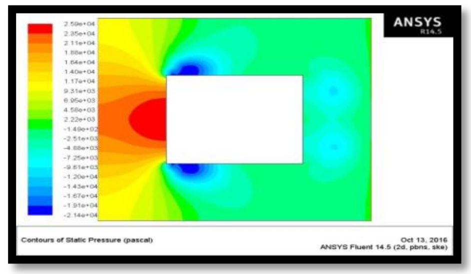




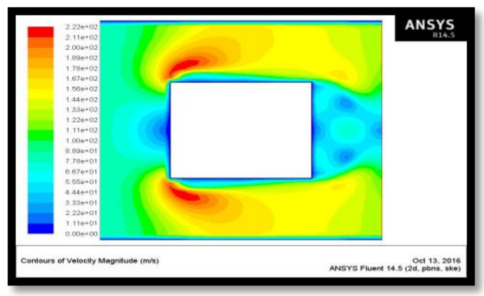
4.4 VERTICAL NARROW PLATE AT 0^0

REYNOLDS NUMBER - 2×10^6

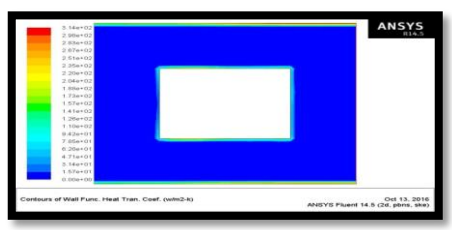
STATIC PRESSURE



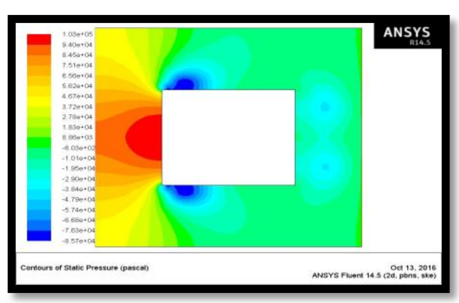
VELOCITY



HEAT TRANSFER COEFFICIENT

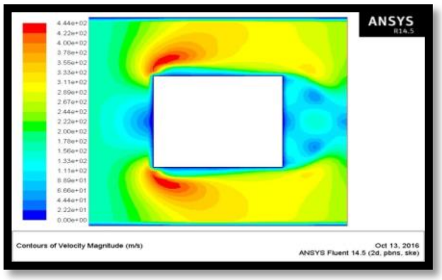


STATIC PRESSURE



.

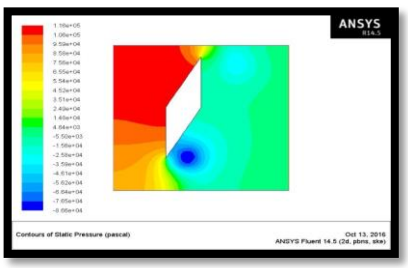
VELOCITY



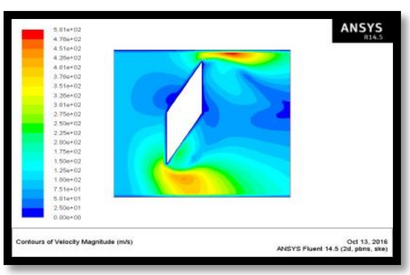
4.5 VERTICAL NARROW PLATE AT 60^0

REYNOLDS NUMBER - 2×10^6

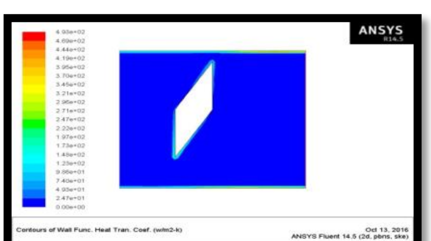
STATIC PRESSURE



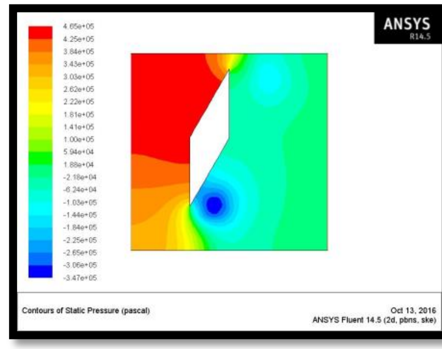
VELOCITY



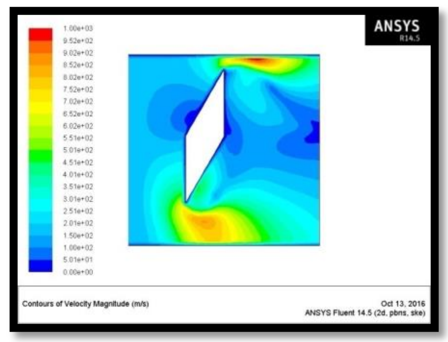
HEAT TRANSFER COEFFICIENT



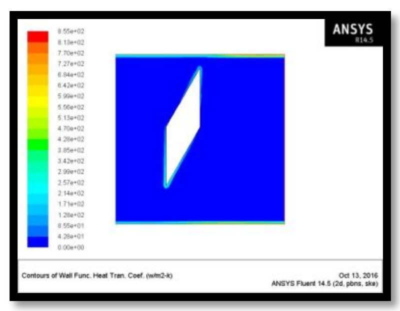
STATIC PRESSURE



VELOCITY



HEAT TRANSFER COEFFICIENT



5. CONCLUSION:

In this thesis the air flow through vertical narrow plates is modeled using PRO-E design software. The thesis will focus on thermal and CFD analysis with different Reynolds number (2×10^6 & 4×10^6) and different angles (0^0,30^0,45^0&60^0) of the vertical narrow plates. Thermal analysis done for the vertical narrow plates by steel, aluminum & copper at different heat transfer coefficient values. These values are taken from CFD analysis at different Reynolds numbers.

By observing the CFD analysis the pressure drop & velocity increases by increasing the inlet Reynolds numbers and increasing the plate angles. The heat transfer rate increasing the inlet Reynolds numbers, more heat transfer rate at 0^0 angles.

By observing the thermal analysis, the taken different heat transfer coefficient values are from CFD analysis. Heat flux value is more for copper material than steel& aluminum.

So we can conclude the copper material is better for vertical narrow plates.

6. REFERENCES

1.Arpaci, V. S., 1995, "Buoyant Turbulent Flow Driven by Internal EnergyGeneration," Int. J. Heat Mass Transfer, Vol. 38, pp.

2.761-2770.Asfia, F. J., and Dhir, V. K., 1994, "An 2.Experimental Study ofNarural Convectionin a Volumetrically Heated Spherical Pool With Rigid Wall," ASME Paper 94-WA/H7--26.

3.Cheung, F. B., i980a, "HeatSource-Driven Thermal Convection at Arbirrary Prandtl Numbers," J. Fluid Mech., YoL 97, pp. 743-758.

4. Cheung, F. B., 1980b, "The Boundry Layer Behavior in Transient Turbulent Thermal Convection Flow," ASME JoURNAL oF HEAr TRANSFERV, ol. 102, pp.373-375.

5.Cheung, F. B., 1978, "Turbulent Thermal Convection in a Horizontal FluidLayer With Time Dependent Volumetric Energy Sources." AIAA/ASME Thermo physics and Heat Transfer Conf.,78-HT-6, Palo Alto.

6. Cheung, F.8., 1977, "Natural Convection in a Volumerrically Heared FluidLayer at High Rayleigh Numbers," Int. J. Heat Mass Transfer, Vol. 20, pp.499-506.

7.Cheung, F. B., Shiah, S. W., Cho, D. H., and Tan, M. J., 1992, "Modeling ofHeat Transfer in a Horizontal HearGenerating Layer by an Effective DiffusivityApproach," ASME/HTD, Vol. 192, pp.55-62.

8.Dinh, T. N., and Nourgalier, R. R., 1997, "On Turbulence Modeling in LargeVolumetrically Heated Liquid Pools," Nncl. Engng. Design, in press.Fan, T. H., 1996, "Heat Transport Phenomena ofTurbulent Natural Convectionin a Melt Layer With Solidification," M.S. thesis, The Pennsylvania State University, University Park, PA.

9.Fielder, H. E., and Wille, R., 1970, "Turbulante Freie Konvektion in EinerHorizontalea Flussigkeitss chicht mit Volumen-Warmequelle," Paper NC 4.5,Proc. Fourth Int. Heat Transfer Conf., Vol. IV, pp. l-12.

DESIGN EVALUTION AND OPTIMIZATION OF NOZZLE USED IN DIESEL ENGINE

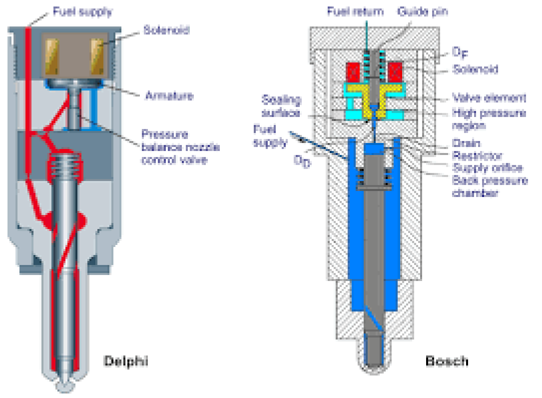
AREBOINA SANDEEP
areboinasandeep@gmail.com
M Tech(CAD/CAM)
Mechanical Engineering Department
Malla Reddy College Of Engineering , Mysammaguda. Hyderabad

Abstract —
The nozzle is used to convert the chemical thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and temperature. Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that exhaust from them. Nozzles come in a variety of shapes and sizes depending on the mission of the rocket, this is very important for the understanding of the performance characteristics of rocket. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. In this thesis the convergent divergent nozzle changing the different nozzle diameters and different fluids at different velocities. We modeled convergent divergent nozzle changing with different nozzle diameters and Analyzed the convergent divergent nozzle with different mass flow rates to determine the pressure drop, heat transfer coefficient, and velocity and heat transfer rate for the fluid by CFD technique.

Keywords — ANSYS, Catia, optimization of nozzle.

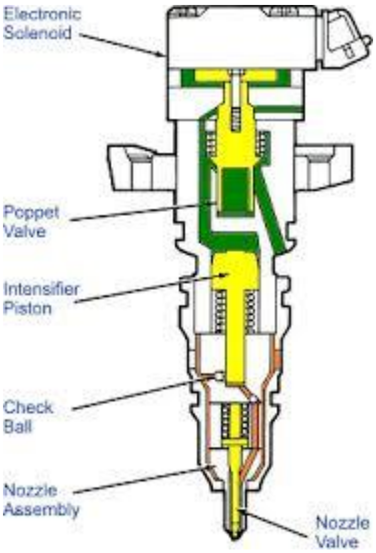
1.INTRODUCTION

The primary challenges towards developing new diesel engines for passenger cars lie in the strict future emission legislation in combination with the customer's demands for steadily 0improving performance. For example, the emission limitations of Tier 2 Bin 5 requires an advanced after treatment system and a robust combustion process that minimizes emissions in the process of them being formed. Advancements in the technology of Diesel Injection (DI) systems have played in important role in the improvements that have been made up to this point . Combining the reduction in nozzle orifice diameters through enhanced flow characteristics with increased injection pressures provides an opportunity to develop engines featuring high power density and reduced emissions. The primary drawback to these modern spray hole geometries is that they often suffer a reduction of power output during long term operation. Other studies have identified these critical formations of deposits as the main reason for this behavior.



Basic mechanisms can be used to explain the formation and removal of deposits in internal combustion engines These mechanisms act independently of the location of formed deposits (e.g. injection nozzles, heat changer) and of the combustion process (e.g. IDI, DI; diesel or gasoline).

The model described in the study illustrates the interaction of a wall with the enclosing flow regime. The transport of particles to the wall is based on the process of thermophores is This process results in the force of gas particles in the direction of the temperature depression. It is amplified with an increasing temperature differential between wall (cold)and fluid (hot). This process results is an increasing concentration of deposit-building particles near the wall.



High 2 turbulence near the wall may reduce the force of the aerosol again to a mean value, compensating for an increased temperature difference. The deposits are composed of attached particles (solid and liquid) and gas (Figure 1).

Condensation and adsorption of gaseous compounds at the cold wall promotes the formation process. At this point, the growth of the deposits is now mainly influenced by the sticking, impaction and incorporation of particles The adsorption of gaseous components and the chemical reactions (as pyrolysis, dehydration and polymerization, etc.), lead to the compaction of the deposits]. The removal of deposits has analogous physical mechanisms.

The chemical mechanism is oxidation destroying the organic compounds in the coating Evaporation and desorption reduce the gaseous fraction dissolved in the deposits. Abrasion is caused by strong aerodynamic forces and breaking-off, due to high temperature changes, resulting in inhomogeneous extensions of the wall and deposit layer.

The corresponding shearing stresses initiate the breaking-off process The soluble fraction of the deposits is washed off by solvents (e.g. water as solvent for salt compounds)

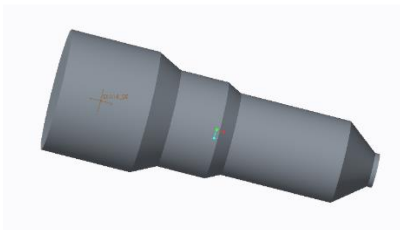
2.LITERATURE REVIEW

Design and Optimization of Fuel Injection System in Diesel Engine Using Biodiesel – A Review H. M. Pate

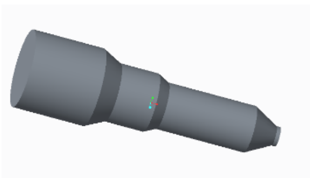
Fuel injection is systems for supplying high pressurize fuel to maximum mixing of fuel with air in an internal combustion engine. Direct Injection (DI) Systems as used in DI engines, in which the fuel is injected directly into a combustion chamber formed in the clylinder itself. The fuel injector directly injects fuel into the direct fuel injection system. The injector is a very complicated part, and massive research has been done to improve it.In my work indicating the development of fuel injector system to reduce chocking problem which is generally happen in bio diesel engine. The injection nozzles and their respective nozzle holders are vitally important components situated between the in-line injection pump and the diesel engine, its functions are as metering the injection of fuel, management of the fuel, defining the rate-of-discharge curve, Sealing-off against the combustion, chamber. Mechanical type injectors used in direct injection system. When biodiesel is used in the diesel engine choking problem is created in fuel injector. Therefore, we optimize the design of fuel injector component, and tried to prevent the chocking problem. The diesel fuel injector system directly injects fuel into the system without chocking.

M. Volmajer et al [4] had numerical and experimental results of the nozzle fuel flow analysis for a four-hole injection nozzle Bosch DLLA 148 S 311376 are presented. The fuel flow coefficients obtained from the experimental results at steady flow conditions in the nozzle are compared with the results of the CFD analysis. The fuel flow coefficients obtained from the experimental results at steady flow conditions in the nozzle are compared with the results of the CFD analysis. From the presented results the following conclusions could be made. Flow coefficient testing device constructed at the ERL yields sufficiently precision, with reasonable uncertainties of the measurement. To refine the precision of the measurement, by defining the exact value of the pressure difference, the pressure downstream of the nozzle should be measured, or the nozzle position should be changed so, that the fluid would be injected directly into the measuring Plexiglas. For the same purpose, Plexiglas cylinder with high ovalness should be replaced with the glasslPlexiglas cylinder with proper circle cross-section. the presented testing device also enables the measurement of the flow coefficient separately for each nozzle hole, which brings better comparison with the results of CFD analysis when the simplified models, introducing only one hole, are applied. Zhijun Li et al [5]had investigates the effects of manufacturing variations in fuel injectors on the engine performance with emphasis on emissions. The variations are taken into consideration within a Reliability-Based Design Optimization (RBDO) framework. A reduced version of Multi-Zone Diesel engine Simulation (MZDS), MZDS-lite, is used to enable the optimization study. The numerical noise of MZDS-lite prohibits the use of gradient-based optimization methods. Therefore, surrogate models are developed to filter out the noise and to reduce computational cost. Three multi-objective optimization problems are formulated, solved and compared: deterministic optimization using MZDS-lite, deterministic optimization using surrogate models and RBDO using surrogate models. The obtained results confirm that manufacturing variation effects must be taken into account in the early product development stages. The effects of manufacturing variations in fuel injectors on the engine performance with emphasis on emissions. The results obtained using deterministic and probabilistic optimization formulations demonstrated the need for RBDO to improve not only performance but also reliability. LI Minghai et al [7] had indicated forced lubrication is adopted for the new injector nozzle matching parts, which can reduce failure rate and increase service life. If the patented product is used widely, economic efficiency and social efficiency will be obtained. Benny Paul et al [8] had indicated effect of helical, spiral, and helical-spiral combination manifold configuration on air motion and turbulence inside the cylinder. Swirl inside the engine is important for diesel engine. Hence, for better performance they recommended a helical-spiral inlet manifold configuration.

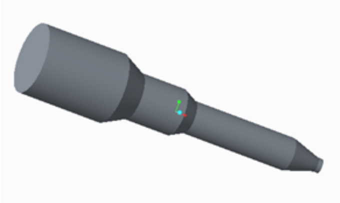
2.1 3D MODEL OF DIESEL NOZZLE WITH 50DIA



2.2 NOZZLE WITH 40DIA



2.3 NOZZLE WITH 30DIA



3. INTRODUCTION TO FEA

Finite element analysis is a method of solving, usually approximately, certain problems in engineering and science. It is used mainly for problems for which no exact solution, expressible in some mathematical form, is available. As such, it is a numerical rather than an analytical method. Methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met with in engineering. For example, engineering strength of materials or the mathematical theory of elasticity can be used to calculate analytically the stresses and strains in a bent beam, but neither will be very successful in finding out what is happening in part of a car suspension system during cornering.

One of the first applications of FEA was, indeed, to find the stresses and strains in engineering components under load. FEA, when applied to any realistic model of an engineering component, requires an enormous amount of computation and the development of the method has depended on the availability of suitable digital computers for it to run on. The method is now applied to problems involving a wide range of phenomena, including vibrations, heat conduction, fluid mechanics and electrostatics, and a wide range of material properties, such as linear-elastic (Hookean) behavior and behavior involving deviation from Hooke's law (for example, plasticity or rubber-elasticity).

Many comprehensive general-purpose computer packages are now available that can deal with a wide range of phenomena, together with more specialized packages for particular applications, for example, for the study of dynamic phenomena or large-scale plastic flow. Depending on the type and complexity of the analysis, such packages may run on a microcomputer or, at the other extreme, on a supercomputer. FEA is essentially a piece-wise process. It can be applied to one-dimensional problems, but more usually there is an area or volume within which the solution is required. This is split up into a number of smaller areas or volumes, which are called finite elements. Figure 1 shows a two-dimensional model of a spanner that has been so divided: the process is called discretisation, and the assembly of elements is called a mesh.

4.INTRODUCTION TO ANSYS

4.1 Structural Analysis

ANSYS Autodyn is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions.

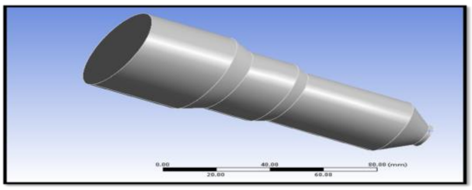
4.2 Fluid Dynamics

ANSYS Fluent, CFD, CFX, FENSAP-ICE and related software are Computational Fluid Dynamics software tools used by engineers for design and analysis. These tools can simulate fluid flows in a virtual environment — for example, the fluid dynamics of ship hulls; gas turbine engines (including the compressors, combustion chamber, turbines and afterburners); aircraft aerodynamics; pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, etc.

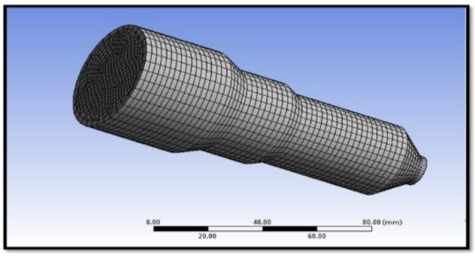
4.3 CFD ANALYSIS OF DIESEL ENGINE NOZZLE

FLUID- DIESEL

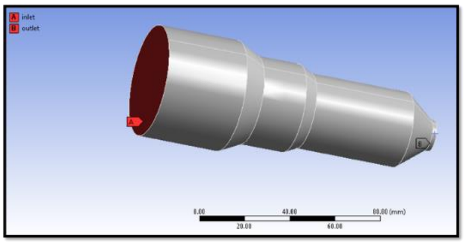
Velocity inlet = 200m/s, 300m/s & 400m/s



MESHED MODEL



SPECIFYING THE BOUNDARIES FOR INLET & OUTLET

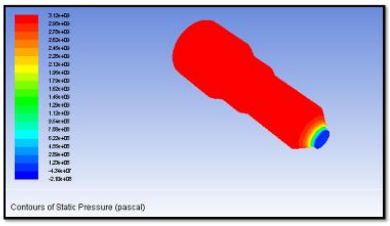


4.4 FLUID- DIESEL

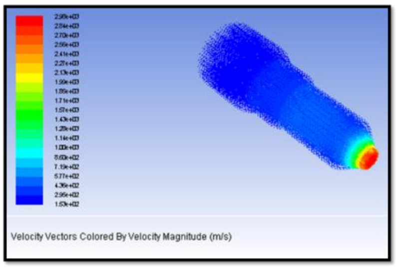
DIESEL ENGINE NOZZLE DIA. 50MM

VELOCITY INLET = 200m/s

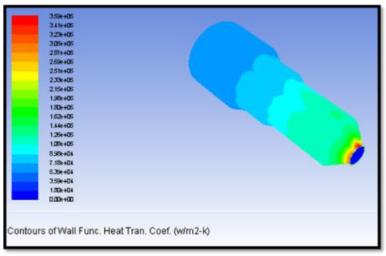
PRESSURE



VELOCITY

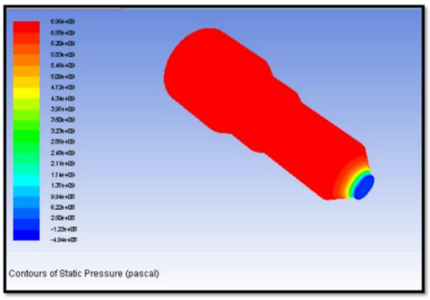


HEAT TRANSFER COEFFICIENT

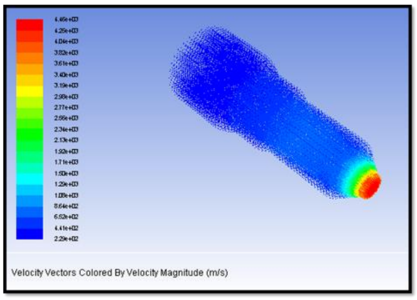


4.5 VELOCITY INLET = 300m/s

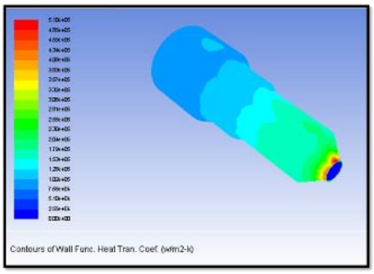
PRESSURE



VELOCITY



HEAT TRANSFER COEFFICIENT

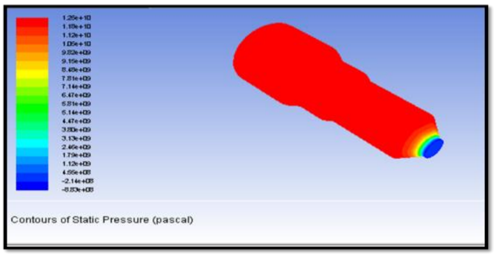


MASS FLOW RATE & HEAT TRANSFER RATE

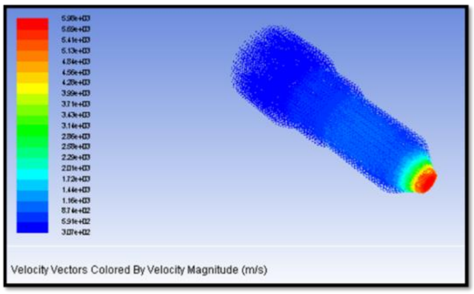
```
                Mass Flow Rate                  (kg/s)
-------------------------------- --------------------
                         inlet              427.56219
         interior-____msbr              26587.756
                        outlet             -427.85144
              wall-____msbr                      0
                ---------------- --------------------
                           Net           -0.28924561

otal Heat Transfer Rate                          (w)
-------------------------------- --------------------
                         inlet                4333986
                        outlet             -4336913.5
              wall-____msbr                      0
                ---------------- --------------------
                           Net                -2927.5
```

4.6 VELOCITY INLET = 400m/s

PRESSURE



VELOCITY



5. CONCLUSION

Nozzles come in a variety of shapes and sizes depending on the mission of the rocket, this is very important for the understanding of the performance characteristics of rocket. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber.

In this thesis the convergent divergent nozzle changing the different nozzle diameters and different fluids at different velocities. We modeled convergent divergent nozzle changing with different nozzle diameters.

By observing the cfd analysis of diesel engine nozzle the pressure, velocity, heat transfer rate and mass flow rate values are increases by increasing the inlet velocities and decreasing the nozzle dia.

By observing the thermal analysis, heat flux is more for aluminum alloy compared with brass material.

So it can be concluded the diesel engine nozzle efficiency were more when the nozzle dia. decreases.

6. REFERENCES

[1] J.B. Heywood, “Internal Combustion Engine Fundamentals”,McGraw-Hill Book Co, pp 493-494, 1988.

[2] D. Ing. H. Tschöke, “Diesel distributor fuel-injection pumps”, Robert Bosch GmbH, pp 12-53, 1999.

[3] B. Challen R. Baranescu, “Diesel Engine Reference Book” Reed Educational and Professional Publishing Ltd., Second Edition, pp.260-301, 1999.

[4] M. Volmajer, B. Kegl, “Experimental and numerical analysis of fuel flow in the diesel engine injection nozzle" , Journal of Kones. Combustion Engines, Vol. 8, No. 1-2, 2001.

[5] Z. Li, M. Kokkolaras, D. Jung, Panos Y. Papalambros and D. N. Assanis, “An Optimization Study of Manufacturing Variation Effects on Diesel Injector Design with Emphasis on Emissions”, SAE International, 2004.

[6] A.J.Von Wielligh, “Influence of fuel quality on diesel injector failures”, Fifth International Colloquium Fuels, Germany, 2005. Property Unit Jatropha oil Jatropha Oil methyl ester Diesel Density at 15°C kg/m3 918 880 850 Viscosity at 40°C mm2 /s 35.4 4.84 2.60 Flash point °C 186 162 70 International Journal of Innovations in Engineering and Technology (IJIET) Vol. 2 Issue 1 February 2013 76 ISSN: 2319 – 1058

[7] LI Minghai, CUI Hongjiang, W.Juan, G. Ying, “Improvement of fuel injection system of locomotive diesel engine”, Journal of Environmental Sciences, pp. S139-S141, 2009.

[8] B. Paul, V. Ganesan, “Flow field development in a direct injection diesel engine with different manifolds” International Journal of Engineering, Science and Technology Vol. 2, No. 1, pp. 80-91, 2010.

[9] K. K.Khatri, D. Sharma, S. L. Soni, S. Kumar, D Tanwar, “Investigation of Optimum Fuel Injection Timing of Direct Injection CI Engine Operated on Preheated Karanj-Diesel Blend” Jordan Journal of Mechanical and Industrial Engineering Volume 4, pp. 629-640, 2010.

[10] Bizhan Befrui, Giovanni Corbinelli, Mario D'Onofrio and Daniel Varble, “GDI Multi-Hole Injector Internal Flow and Spray Analysis”, SAE International, 2011.

DESIGN AND CFD ANALYSIS OF FLOW THROUGH VENTURI OF A CARBURETOR

PELIMALI RAVICHANDRA

Pelimeli09@gmail.com

M.TECH(THERMAL ENGINEERING)

MECHANICAL ENGINEERING DEPARTMENT

Malla Reddy College of Engineering,Mysammaguda

Hyderabad

ABSTRACT:The process of forming a combustible fuel-air mixture by mixing the right amount of fuel with air before admission to the cylinder of the engine is called carburetion and the device doing this job is called carburetor.Modern passenger vehicles with gasoline engines are provided with different compensating devices for fuel air mixture supply. One of the important factors that affect the fuel consumption is that design of carburetor. The venture of the carburetor is important that provides a necessary pressure drop in the carburetor device. Since different SI engine alternative fuels such as LPG, CNG are used in the present day vehicles to reduce the pollution and fuel consumption. Still for a better economy and uniform fuel air supply there is a need to design the carburetor with an effective analytical tool or software.In this project venture of carburetor is modeled in 3D modeling software Creo/Engineer. CFD analysis is done on the venture by varying the fuel discharge nozzle angle on the flow.

Key word: Ansys, Pro E,CFD Analysis

1.INTRODUCTION

Engine is a device that transforms one form of energy into another form. Heat energy is a device that transforms the chemical energy contained in a fuel to another form of energy and utilizes that energy for some useful work

1.1 Carburetor

The simple carburetor consists of the following basic parts
Float chamber

- Venturi
- Fuel discharge nozzle
- Metering orifice
- Choke
- Throttle valve

1.3 PRINCIPLE OF CARBURETION

Both air and gasoline are drawn into the cylinder due to suction pressure created by the downward movement of the piston. In the carburetor, the air passing into the combustion chamber picks up the fuel discharged by a fine orifice in a tube called the carburetor jet. The rate of discharge of the fuel depends on the pressure difference between the float chamber and the throat of the venturi of the carburetor and the area of the outlet of the tube. In order that the fuel is strongly atomized the suction effect must be strong and the nozzle outlet must be comparatively small. To produce a strong suction, a restriction is generally provided in the pipe in the carburetor carrying air to the engine. This restriction is called throat. In this throat due to increase in the velocity of the air the pressure is decreased and suction is created.

The venturi tube has a narrower path at the center so that the path through air is going to travel is reduced. As same amount of air must travel must travel through the path of the tube so the velocity of the air at the venturi is increased and suction is created.

Usually the fuel discharge jet is located at the point where the suction is maximum. So this is positioned just below the throat of the venturi. The spray of the fuel from the fuel discharge jet and the air are mixed at this point of the throat and a combustible mixture is formed. Maximum amount of fuel gets atomized and some part gets vaporized. Due to increase in the velocity of the air at the throat the vaporization of the fuel becomes easier.

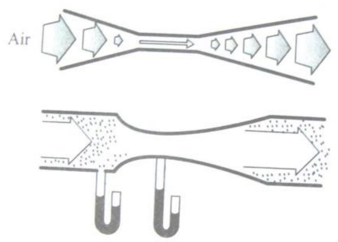


Fig:1 Operation of venture tube

2.PROJECT DESCRIPTION

In this project venture of carburetor is modeled in 3D modeling software Pro/Engineer. CFD analysis is done on the venture by varying the fuel discharge nozzle angle on the flow. The analysis was done for ϴ = 30, 35, 40 and 45 where ϴ is the angle between the axis of the fuel discharge nozzle and the vertical axis of the body of the carburetor.

And also CFD analysis is done to calculate the throat pressure for different angles of the throttle plate 45, 60, 75, 90.

Fig The simple carburetor

3.INTRODUCTION TO CREO

PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself.

The name was changed in 2010 from Pro/ENGINEER Wildfire to CREO. It was announced by the company who developed it, Parametric Technology Company (PTC), during the launch of its suite of design products that includes applications such as assembly modeling, 2D orthographic views for technical drawing, finite element analysis and more.

PTC CREO says it can offer a more efficient design experience than other modeling software because of its unique features including the integration of parametric and direct modeling in one platform. The complete suite of applications spans the spectrum of product development, giving designers options to use in each step of the process. The software also has a more user friendly interface that provides a better experience for designers. It also has collaborative capacities that make it easy to share designs and make changes.

There are countless benefits to using PTC CREO. We'll take a look at them in this two-part series.

First up, the biggest advantage is increased productivity because of its efficient and flexible design capabilities. It was designed to be easier to use and have features that allow for design processes to move more quickly, making a designer's productivity level increase.

Part of the reason productivity can be increased is because the package offers tools for all phases of development, from the beginning stages to the hands-on creation and manufacturing. Late stage changes are common in the design process, but PTC CREO can handle it. Changes can be made that are reflected in other parts of the process.

The collaborative capability of the software also makes it easier and faster to use. One of the reasons it can process information more quickly is because of the interface between MCAD and ECAD designs. Designs can be altered and highlighted between the electrical and mechanical designers working on the project.

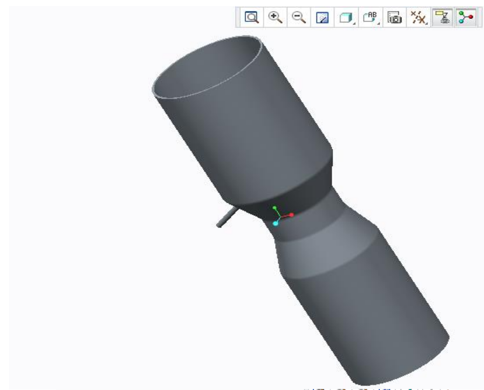
The time saved by using PTC CREO isn't the only advantage. It has many ways of saving costs. For instance, the cost of creating a new product can be lowered because the development process is shortened due to the automation of the generation of associative manufacturing and service deliverables.

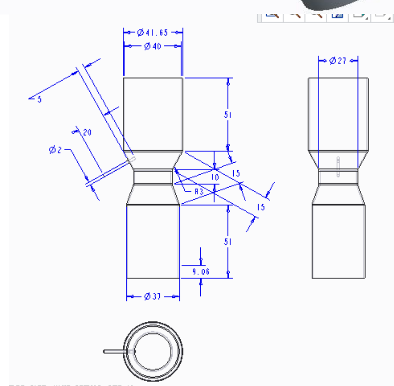
PTC also offers comprehensive training on how to use the software. This can save businesses by eliminating the need to hire new employees. Their training program is available online and in-person, but materials are available to access anytime.

A unique feature is that the software is available in 10 languages. PTC knows they have people from all over the world using their software, so they offer it in multiple languages so nearly anyone who wants to use it is able to do so.

3.1 MODELS OF VENTURE

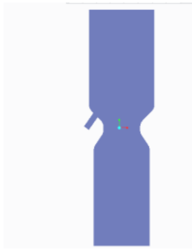
3D MODEL



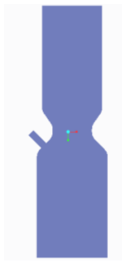


3.2 FUEL DISCHARGE NOZZLE ANGLE – 30^0

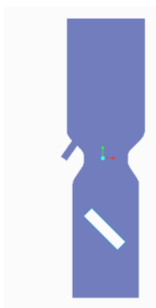
3.3 FUEL DISCHARGE NOZZLE ANGLE – 35^0



3.4 FUEL DISCHARGE NOZZLE ANGLE – 45^0



3.5 THROTTLE PLATE ANGLE – 45^0



.

4.INTRODUCTION TO ANSYS

4.1 Structural Analysis

ANSYS Autodyn is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions.

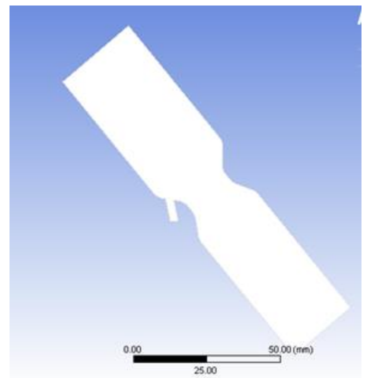
4.2 ANSYS Mechanical

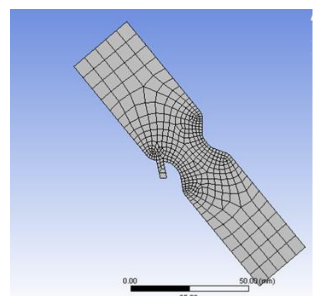
ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behavior, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis.

4.3 CFD ANALYSIS OF VENTURE OF CARBURETOR:

FUEL DISCHARGE NOZZLE ANGLE – 30^0:

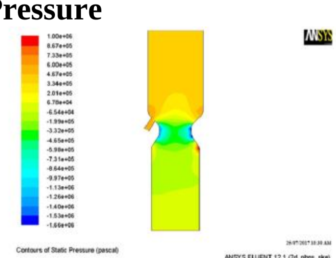
Save Creo- Model as .iges format



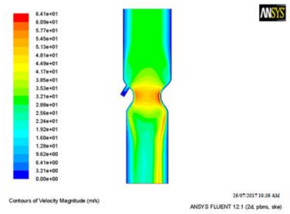


4.4 SPECIFYING BOUNDARIES FOR INLET AND OUTLET

Static Pressure

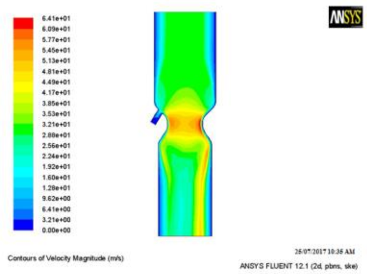


Velocity

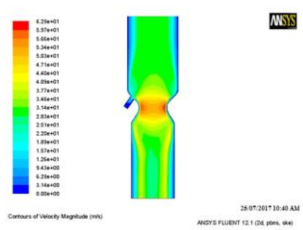


4.5 FUEL DISCHARGE NOZZLE ANGLE – 35^0

Static Pressure

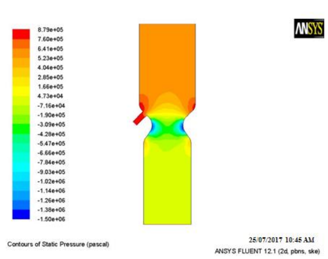


Velocity

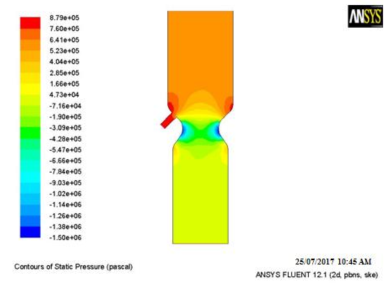


4.6 FUEL DISCHARGE NOZZLE ANGLE – 45^0

Static Pressure

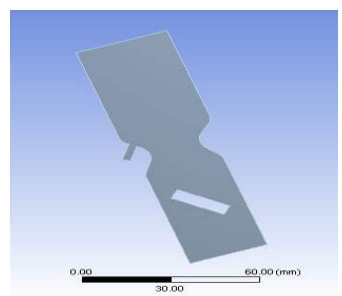


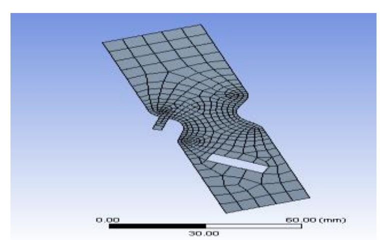
Velocity



4.7 THROTTLE PLATE ANGLE – 45^0

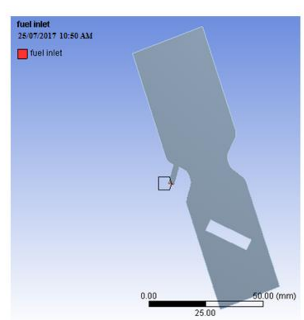
Save Creo Model as .iges format



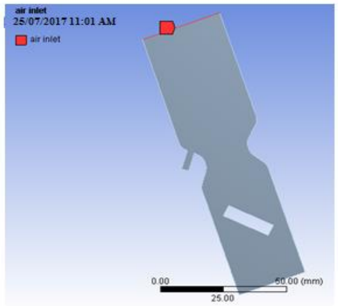


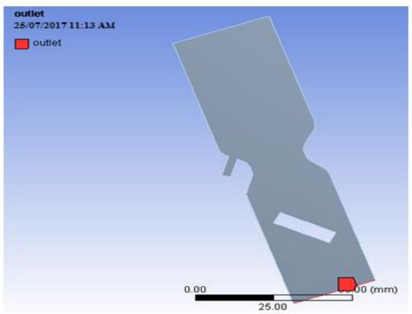
4.8 SPECIFYING BOUNDARIES FOR INLET AND OUTLET

Fuel Inlet

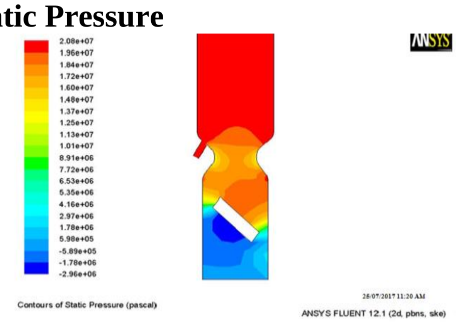


Air Inlet



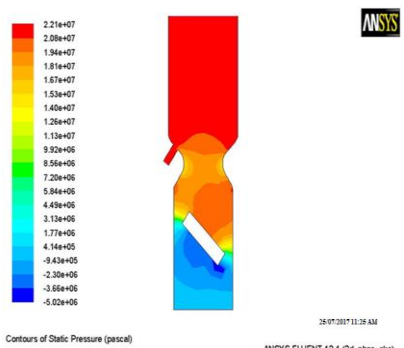


Static Pressure



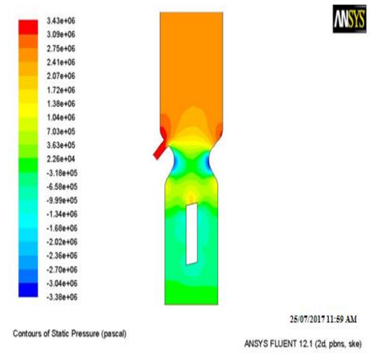
4.9 THROTTLE PLATE ANGLE – 60^0

Static Pressure



THROTTLE PLATE ANGLE – 90^0

Static Pressure



RESULTS TABLE

FUEL DISCHARGE ANGLE	**30**	**35**	**40**	**45**
STATIC PRESSURE (Pa)	$1e^6$	6.75 e^5	7.4 e^5	8.79 e^5
VELOCITY (m/s)	6.41 e^1	6.21 e^1	6.33 e^1	6.37 e^1

THROTTLE PLATE ANGLE	**45**	**60**	**75**	**90**
STATIC PRESSURE (Pa)	2.08^7	2.21 e^7	4.22 e^6	3.43 e^6

5. CONCLUSION

From the above analysis the conclusions obtained are

1. When the flow inside the carburetor was analyzed for different angles of throttle plate opening, it was found that the pressure at the throat of the venturi decreased with the increase in opening of the throttle plate. Because when the throttle plate opening increases then the flow of air through the carburetor increases but the fuel flow remains constant. So the mixture becomes leaner. But as obtained from the analysis above the pressure at the throat the throat also decreases with increase in opening of the throttle plate so the flow of fuel from the float chamber into the throat increases and hence the quality of the mixture tends to remain constant.

2. When analyzed for fuel discharge nozzle angle of 30^0, it was observed that the pressure distribution inside the body of the carburetor is quite uniform which leads to a better atomization and vaporization of the fuel inside the carburetor body. But in other cases like where the fuel discharge nozzle angle was 35^0, 40^0 or 45^0, the pressure distribution is quite non-uniform inside the body of the carburetor. So it is concluded that for gasoline operated engine the optimum fuel discharge nozzle angle is 30^0.

6. REFERENCES

[1] Ganeshan V. INTERNAL COMBUSTION ENGINES. New Delhi, TMH publication,2009

[2] Diego A. Arias. Numerical and experimental study of air and fuel flow in small engine carburetors. University of Wisconsin-Madison, 2005

[3] Heywood John B. Internal combustion engines fundamentals. Mc Graw Hill, Inc, 1988.

[4] Diego A. Arias, Timothy A. Shedd, Steady and non-steady flow in a simple carburetor, Inst Mech Eng (Lond) Proc , Volume 188, Issue 53, 1974, Pages 537-548.

[5] Fluent. FLUENT 5 User Guide, 1999.

[6] Gambit. GAMBIT 5 User Guide, 1999.

[7] Sayma Abdulnaser, Computational Fluid Dynamics, ISBN- 978-87-7681-938-4, Abdulnaser Sayma and Ventus Publishers, 2009

[8] D. L. Harrington. Analysis and digital simulation of carburetor metering. PhD thesis, University of Michigan, 1968.

[9] Martin Cook, Masud Behnia. Pressure drop calculation and modelling of inclined intermittent gas liquid flow, Chemical Engineering Science 55 (2000) page 4699-4708.

[10] Bhramara P., Rao V. D., Sharma K. V. , and Reddy T. K. K. CFD analysis of two phase flow in a horizontal pipe – prediction of pressure drop, International Journal of Mechanical,industrial and Aerospace Engineering 3:2 2009.

DESIGN AND ANALYSIS OF TURBINE WING

ALUGALA SRAVAN
sravanalugala6@gmail.com
M.TECH(CAD/CAM)
MECHANICAL ENGINEERING DEPARTMENT
Malla Reddy College of Engineering,Mysammaguda
Hyderabad

ABSTRACT:- A Turbine blade is the individual component which makes up the turbine section of a gas turbine. The blades are responsible for extracting energy from the high temperature, high pressure gas produced by the combustor. The turbine blades are often the limiting component of gas turbines. To survive in this difficult environment, turbine blades often use exotic materials like super alloys and many different methods of cooling, such as internal air channels, boundary layer cooling, and thermal barrier coatings.

Key words:-Analysis,Modeling,Design

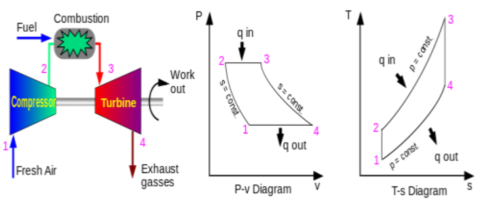
1.INTRODUCTION TO GAS TURBINE

A gas turbine, also called a combustion turbine, is a type of internal combustion engine. It has an upstream rotating compressor coupled to a downstream turbine, and a combustion chamber or area, called a combustor, in between.

The basic operation of the gas turbine is similar to that of the steam power plant except that air is used instead of water. Fresh atmospheric air flows through a compressor that brings it to higher pressure. Energy is then added by spraying fuel into the air and igniting it so the combustion generates a high-temperature flow. This high-temperature high-pressure gas enters a turbine, where it expands down to the exhaust pressure, producing a shaft work output in the process. The turbine shaft work is used to drive the compressor and other devices such as an electric generator that may be coupled to the shaft. The energy that is not used for shaft work comes out in the exhaust gases, so these have either a high temperature or a high velocity. The purpose of the gas turbine determines the design so that the most desirable energy form is maximized. Gas turbines are used to power aircraft, trains, ships, electrical generators, and tanks.

1.1 Theory of operation

In an ideal gas turbine, gases undergo four thermodynamic processes: an isentropic compression, isobaric (constant pressure) combustion, an isentropic expansion and heat rejection. Together, these make up the Brayton cycle.



In a real gas turbine, mechanical energy is changed irreversibly (due to internal friction and turbulence) into pressure and thermal energy when the gas is compressed (in either a centrifugal or axial compressor). Heat is added in the combustion chamber and the specific volume of the gas increases, accompanied by a slight loss in pressure. During expansion through the stator and rotor passages in the turbine, irreversible energy transformation once again occurs.

Fresh air is taken in, in place of the heat rejection.

If the engine has a power turbine added to drive an industrial generator or a helicopter rotor, the exit pressure will be as close to the entry pressure as possible with only enough energy left to overcome the pressure losses in the exhaust ducting and expel the exhaust. For a turboprop engine there will be a particular balance between propeller power and jet thrust which gives the most economical operation. In a jet engine only enough pressure and energy is extracted from the flow to drive the compressor and other components. The remaining high-pressure gases are accelerated to provide a jet to propel an aircraft.

The smaller the engine, the higher the rotation rate of the shaft(s) must be to attain the required blade tip speed. Blade-tip speed determines the maximum pressure ratios that can be obtained by the turbine and the compressor. This, in turn, limits the maximum power and efficiency that can be obtained by the engine. In order for tip speed to remain constant, if the diameter of a rotor is reduced by half, the rotational speed must double. For example, large jet engines operate around 10,000 rpm, while micro turbines spin as fast as 500,000 rpm.

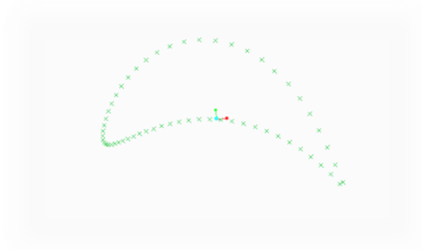
Mechanically, gas turbines can be considerably less complex than internal combustion piston engines. Simple turbines might have one main moving part, the compressor/shaft/turbine rotor assembly (see image above), with other moving parts in the fuel system. However, the precision manufacture required for components and the temperature resistant alloys necessary for high efficiency often make the construction of a simple gas turbine more complicated than a piston engine.

More advanced gas turbines (such as those found in modern jet engines) may have 2 or 3 shafts (spools), hundreds of compressor and turbine blades, movable stator blades, and extensive external tubing for fuel, oil and air systems.

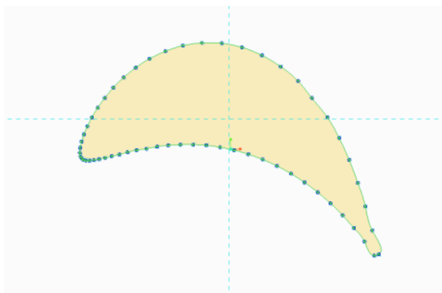
Thrust bearings and journal bearings are a critical part of design. They are hydrodynamic oil bearings or oil-cooled rolling-element bearings. Foil bearings are used in some small machines such as micro turbines and also have strong potential for use in small gas turbines/auxiliary power units.

2. 3D MODELS OF BLADES

2.1 Points

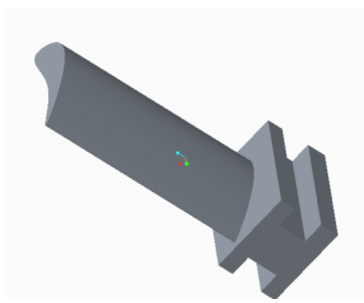


2.2 SKETCHER



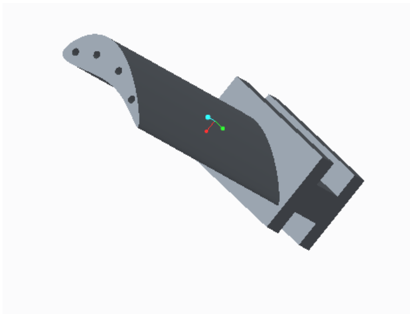
2.3 ORIGINAL MODEL

WITHOUT HOLES

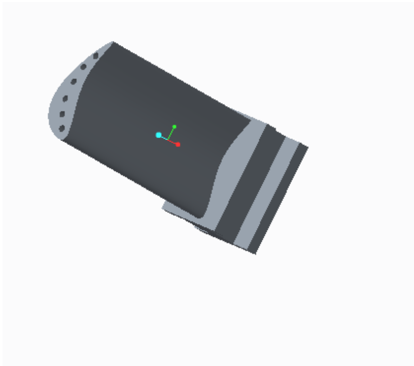


2.4 MODIFIED MODEL

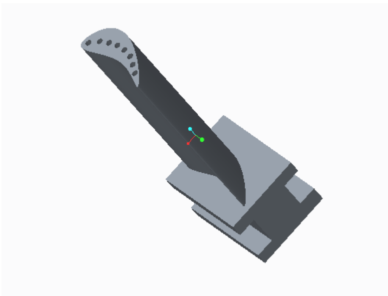
WITH 4 HOLES



2.5 WITH 6 HOLES



2.6 WITH 8 HOLES



3. INTRODUCTION TO FEA

Finite element analysis is a method of solving, usually approximately, certain problems in engineering and science. It is used mainly for problems for which no exact solution, expressible in some mathematical form, is available. As such, it is a numerical rather than an analytical method. Methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met with in engineering. For example, engineering strength of materials or the mathematical theory of elasticity can be used to calculate analytically the stresses and strains in a bent beam, but neither will be very successful in finding out what is happening in part of a car suspension system during cornering.

One of the first applications of FEA was, indeed, to find the stresses and strains in engineering components under load. FEA, when applied to any realistic model of an engineering component, requires an enormous amount of computation and the development of the method has depended on the availability of suitable digital computers for it to run on. The method is now applied to problems involving a wide range of phenomena, including vibrations, heat conduction, fluid mechanics and electrostatics, and a wide range of material properties, such as linear-elastic (Hookean) behavior and behavior involving deviation from Hooke's law (for example, plasticity or rubber-elasticity).

Many comprehensive general-purpose computer packages are now available that can deal with a wide range of phenomena, together with more specialized packages for particular applications, for example, for the study of dynamic phenomena or large-scale plastic flow. Depending on the type and complexity of the analysis, such packages may run on a microcomputer or, at the other extreme, on a supercomputer. FEA is essentially a piece-wise process. It can be applied to one-dimensional problems, but more usually there is an area or volume within which the solution is required. This is split up into a number of smaller areas or volumes, which are called finite elements. Figure 1 shows a two-dimensional model of a spanner that has been so divided: the process is called discretisation, and the assembly of elements is called a mesh.

4. INTRODUCTION TO ANSYS

Structural Analysis

ANSYS Autodyn is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions.

4.1 ANSYS Mechanical

ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behavior, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis.

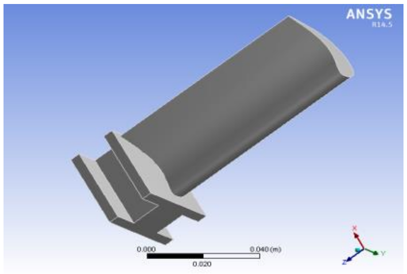
4.2 Fluid Dynamics

ANSYS Fluent, CFD, CFX, FENSAP-ICE and related software are Computational Fluid Dynamics software tools used by engineers for design and analysis. These tools can simulate fluid flows in a virtual environment — for example, the fluid dynamics of ship hulls; gas turbine engines (including the compressors, combustion chamber, turbines and afterburners); aircraft aerodynamics; pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, etc.

5. STRUCTURAL ANALYSIS OF GAS TURBINE BLADE

WITHOUT HOLES

TITANIUM ALLOY



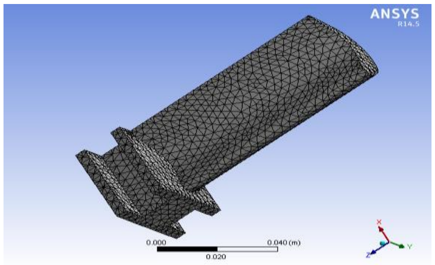
5.1 Material properties of Titanium Alloy

Density : 0.0000134 kg/mm^3

Young's modulus : 125000Mpa

Poisson’s ratio : 0.342

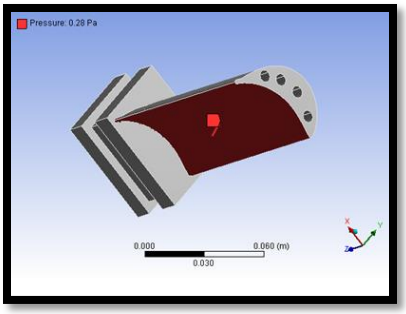
Meshed model



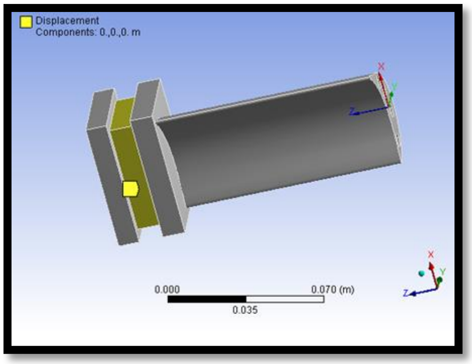
5.2 WITH 4 HOLES

TITANIUM ALLOY

Pressure

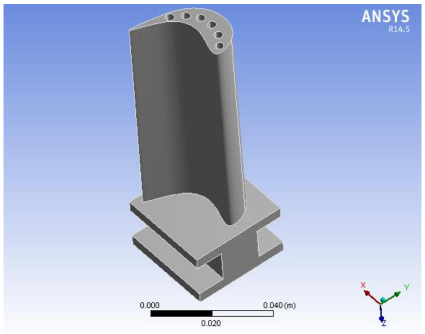


Displacement

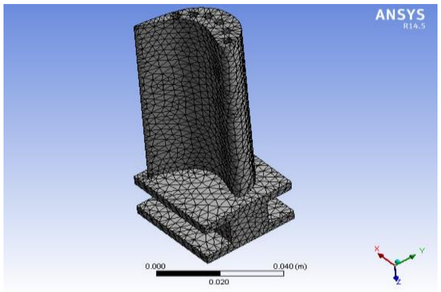


5.3 TITANIUM ALLOY

Save CREO Model as .iges format



Meshed model

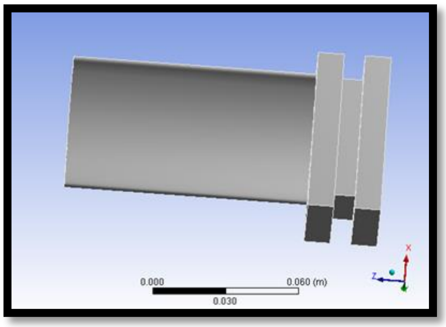


5.4 THERMAL ANALYSIS ON GAS TURBINE BLADE

TITANIUM ALLOY

WITHOUT HOLES

Import model



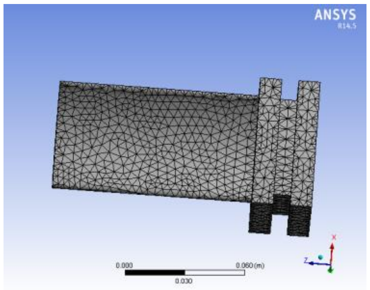
Material Properties of Titanium Alloy

Density : 0.00000484 kg/mm^3

Thermal conductivity : 10.9 w/mk

Specific heat: 670 j/g^^{0}c

Meshed model



6. CONCLUSION

In our project we have designed a turbine blade used in gas turbines and modeled in 3D modeling software Pro/Engineer. Two other models with 4 holes and 6 holes are also modeled.

We have done structural and thermal analysis on all the models of turbine blades using Titanium alloy and Nickel alloy. By observing the analysis results, the analyzed stress values are less than their permissible stress values. So using both the materials is safe. The stress and deformation values are more for Nickel alloy.

By observing the thermal results, thermal flux is more for Nickel alloy than titanium alloy. So using Nickel alloy is better than Titanium alloy. But the main disadvantage is its weight.

By comparing the results for all the models, thermal flux is increasing by increasing number of holes, so heat transfer rate is increased.

So we can conclude that by using Nickel alloy with 6 holes is better.

7. REFERENCES

1. Design and Analysis of Gas Turbine Blade by Theju V, Uday P S , PLV Gopinath Reddy, C.J.Manjunath
2. Heat Transfer Analysis of Gas Turbine Blade Through Cooling Holes by K Hari Brahmaiah , M.Lava Kumar
3. The design and analysis of gas turbine blade by Pedaprolu Venkata Vinod
4. Effect Of Temperature Distribution In 10c4/60c50 Gas Turbine Blade Model Using Finite Element Analysis by V.Veeraragavan
5. Film Cooling on a Gas Turbine Rotor Blade by K. Takeishi, S. Aoki, T. Sato and K. Tsukagoshi
6. Film Cooling of the Gas Turbine Endwall by Discrete-Hole Injection by M. Y. Jabbari, K. C. Marston, E. R. G. Eckert and R. J. Goldstein
7. An advanced impingement/film cooling scheme for gas turbines – numerical study by A. Immarigeon, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada), I. Hassan, (Department of Mechanical and Industrial Engineering, Concordia University, Montréal, Canada)
8. An experimental investigation on the trailing edge cooling of turbine blades by Zifeng Yang, Hui Hu
9. CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Film Cooling Effectiveness by Shridhar Paregouda, Prof. Dr. T. Nageswara Rao

ANALYSIS OF HEAT TRANSFER RATE BY VARYING COOLING FLUID FOR ENGINE CYLINDER FINS

MOHD IMRAN
Mohdimran9605@gmail.com
M TECH(CAD/CAM)
Malla Reddy College Of Engineering,Mysammaguda
Hyderabad

Abstract: The Engine cylinder is one of the major automobile components, which is subjected to high temperature variations and thermal stresses. In order to cool the cylinder, fins are provided on the cylinder to increase the rate of heat transfer. By doing thermal analysis on the engine cylinder fins, it is helpful to know the heat dissipation inside the cylinder.The principle implemented in this project is to increase the heat dissipation rate by using the invisible working fluid, nothing but air. We know that, by increasing the surface area we can increase the heat dissipation rate, so designing such a large complex engine is very difficult. The main purpose of using these cooling fins is to cool the engine cylinder by air.

Keywords:- Ansys,Catia,Engine Cylinder Fins

1. INTRODUCTION

The internal combustion engine is an engine in which the combustion of a fuel (normally a fossil fuel) occurs with an oxidizer (usually air) in a combustion chamber. In an internal combustion engine, the expansion of the high-temperature and -pressure gases produced by combustion applies direct force to some component of the engine, such as pistons, turbine blades, or a nozzle. This force moves the component over a distance, generating useful mechanical energy.

1.1 NECESSITY OF COOLING SYSTEM IN IC ENGINES

All the heat produced by the combustion of fuel in the engine cylinders is not converted into useful power at the crankshaft. A typical distribution for the fuel energy is given below:

It is seen that the quantity of heat given to the cylinder walls is considerable and if this heat is not removed from the cylinders it would result in the resignation of the charge. In addition, the lubricant would also burn away, thereby causing the seizing of the piston. Excess heating will also damage the cylinder material.

Keeping the above factors in view, it is observed that suitable means must be provided to dissipate the excess heat from the cylinder walls, so as to maintain the temperature below certain limits.

However, cooling beyond optimum limits is not desirable, because it decreases the overall efficiency due to the following reasons:

Thermal efficiency is decreased due to more loss of heat to the cylinder walls.

- The vaporization of fuel is less; this results in fall of combustion efficiency.
- Low temperatures increase the viscosity of lubrication and hence more piston friction is encountered, thus decreasing the mechanical efficiency.

Though more cooling improves the volumetric efficiency, yet the factors mentioned above result in the decrease of overall efficiency.

Thus it may be observed that only sufficient cooling is desirable and any deviation from the optimum limits will result in the deterioration of the engine performance.

2. METHODS OF COOLING

Various methods used for cooling of automobile engines are:

1.Air Cooling 2.Water cooling

2.1 AIR-COOLING

Cars and trucks using direct air cooling (without an intermediate liquid) were built over a long period beginning with the advent of mass produced passenger cars and ending with a small and generally unrecognized technical change. Before World War II, water cooled cars and trucks routinely overheated while climbing mountain roads, creating geysers of boiling cooling water. This was considered normal, and at the time, most noted mountain roads had auto repair shops to minister to overheating engines.

ACS (Auto Club Suisse) maintains historical monuments to that era on the Susten Pass where two radiator refill stations remain (See a picture here). These have instructions on a cast metal plaque and a spherical bottom watering can hanging next to a water spigot. The spherical bottom was intended to keep it from being set down and, therefore, be useless around the house, in spite of which it was stolen, as the picture shows.

During that period, European firms such as Magirus-Deutz built air-cooled diesel trucks, Porsche built air-cooled farm tractors, and Volkswagen became famous with air-cooled passenger cars. In the USA, Franklinbuilt air-cooled engines. The Czechoslovakia based company Tatra is known for their big size air cooled V8 car engines, Tatra engineer Julius Mackerle published a book on it. Air cooled engines are better adapted to extremely cold and hot environmental weather temperatures, you can see air cooled engines starting and running in freezing conditions that stuck water cooled engines and continue working when water cooled ones start producing steam jets.

2.2 LIQUID COOLING

Today, most engines are liquid-cooled.

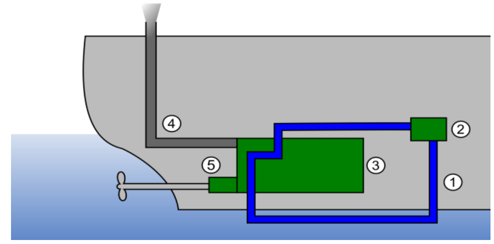


Fig:1 A fully closed IC engine cooling system

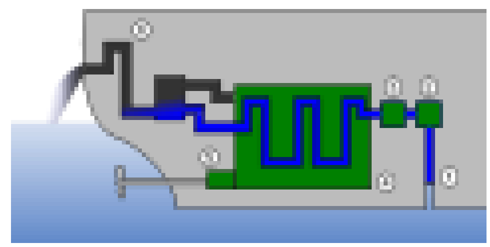


Fig:2 Open IC engine cooling system

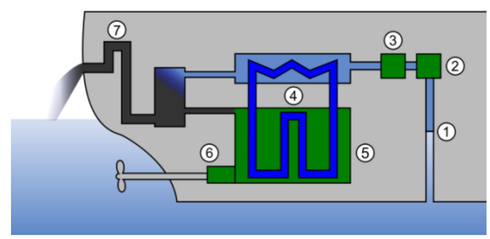


Fig:3 Semi closed IC Engine cooling system

Liquid cooling is also employed in maritime vehicles (vessels, ...). For vessels, the seawater itself is mostly used for cooling. In some cases, chemical coolants are also employed (in closed systems) or they are mixed with seawater cooling.

3. INTRODUCTION OF ANSYS :

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behavior of the product, be it electromagnetic, thermal, mechanical etc.

3.1 GENERIC STEPS TO SOLVING ANY PROBLEM IN ANSYS :

Like solving any problem analytically, you need to define (1) your solution domain, (2) the physical model, (3) boundary conditions and the physical properties. You then solve the problem and present the results. In numerical methods, the main difference is an extra step called mesh generation. This is the step that divides the complex model into small elements that become solvable in an otherwise too complex situation. Below describes the processes in terminology slightly more attune to the software.

Build Geometry

Construct a two or three dimensional representation of the object to be modeled and tested using the work plane coordinate system within ANSYS.

Define Material Properties

Now that the part exists, define a library of the necessary materials that compose the object (or project) being modeled. This includes thermal and mechanical properties.

Generate Mesh

At this point ANSYS understands the makeup of the part. Now define how the modeled system should be broken down into finite pieces.

Apply Loads

Once the system is fully designed, the last task is to burden the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution

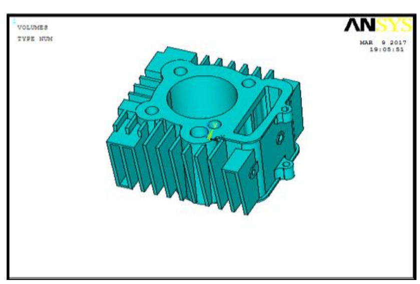
This is actually a step, because ANSYS needs to understand within what state (steady state, transient… etc.) the problem must be solved.

Present the Results

After the solution has been obtained, there are many ways to present ANSYS' results, choose from many options such as tables, graphs, and contour plots.

Thermal analysis of fin body:

Set Units - /units,si,mm,kg,sec,k
File- change Directory-select working folder
File-Change job name-Enter job name
Preferences-Thermal\preprocessor-Element type-add/edit/delete-Select Add-Solid 20 node 90



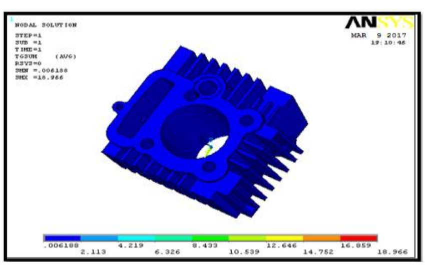
3.2MESHED MODEL OF ALUMINIUM ALLOY 204 3MM THICKNESS

Finite element analysis or FEA representing a real project as a “mesh” a series of small, regularly shaped tetrahedron connected elements, as shown in the above fig.And then setting up and solving huge arrays of simultaneous equations. The finer the mesh, the more accurate the results but more computing power is required.

Nodal temperature of Aluminium Alloy 204 3mm thickness

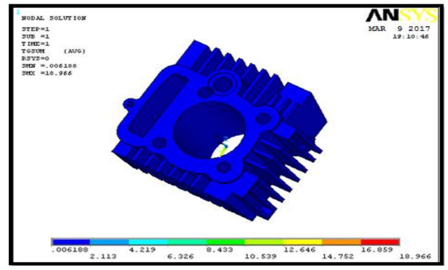
According to the contour plot, the temperature distribution maximum temperature at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the maximum temperature at bore and its distributed to outer surface of the fins.

General post processer- contour plot- Thermal Gradient-Thermal Gradient Vector Sum



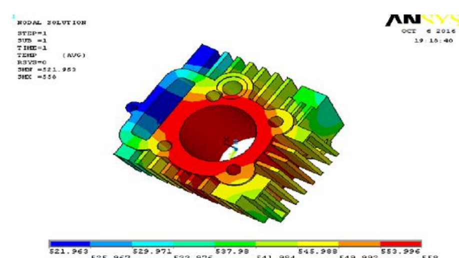
Thermal Gradient of Aluminium Alloy 204 3mm thickness

According to the contour plot, the thermal gradient maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum gradient at fins. According to the above contour plot, the maximum gradient is 18.966 k/m and minimum gradient is 0.006188 k/m.**General post processer- contour plot- Thermal Flux –Thermal Flux Vector Sum**



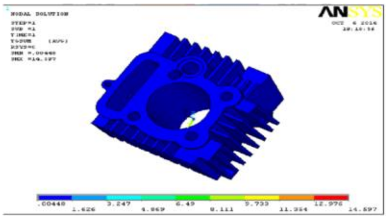
Nodal Temperature of Aluminium Alloy 6061 3mm thickness

According to the contour plot, the temperature distribution maximum temperature at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the maximum temperature at bore and its distributed to outer surface of the fins.



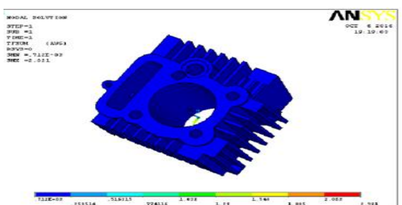
3.3 NODAL TEMPERATURE OF MAGNESIUM ALLOY 3MM THICKNESS

According to the contour plot, the temperature distribution maximum temperature at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the maximum temperature at bore and its distributed to outer surface of the fins.



3.4THERMAL GRADIENT SUM OF MAGNESIUM ALLOY 3MM THICKNESS

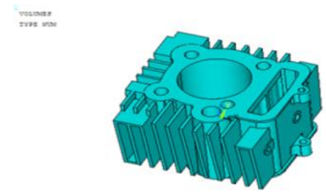
According to the contour plot, the thermal gradient maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum gradient at fins. According to the above contour plot, the maximum gradient is 14.597 k/m and minimum gradient is 0.00448 k/m.



3.5 THERMAL FLUX SUM OF MAGNESIUM ALLOY 3MM THICKNESS

According to the contour plot, the thermal flux maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum thermal flux at fins. According to the above contour plot, the maximum thermal flux is 2.321 k/m and minimum thermal flux is 0.712E-03 k/m.

ALUMINUM ALLOY 204 – 2.5mm THICKNESS



MODEL ANALYSIS OF ALUMINUM ALLOY 204 – 2.5mm THICKNESS

MODEL IMPORTED FROM PRO/ENGINEER

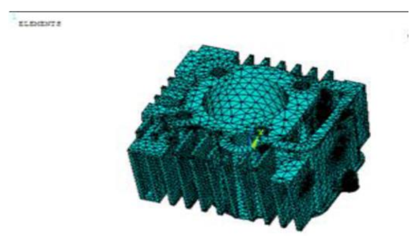
MATERIAL PROPERTIES

Thermal Conductivity – 120 w/mk

Specific Heat – 0.963 J/g ºC

Density – 2.8 g/cc

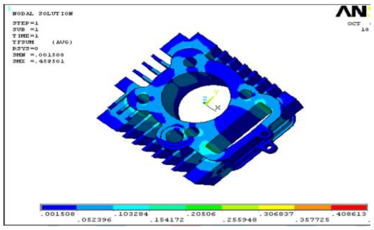
MESHED MODEL



3.6 NODAL TEMPERATURE OF ALUMINIUM ALLOY 204 2.5MM THICKNESS

According to the contour plot, the temperature distribution maximum temperature at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the maximum temperature at bore and its distributed to outer surface of the fins.

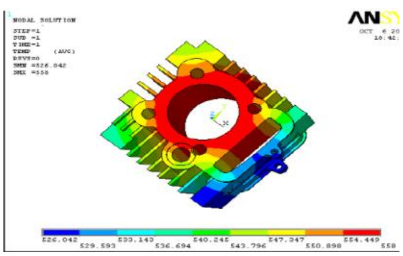
THERMAL FLUX SUM



3.7 THERMAL FLUX SUM SUM OF ALUMINIUM ALLOY 204 2.5MM THICKNESS

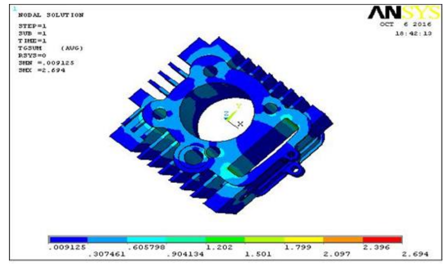
According to the contour plot, the thermal flux maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum thermal flux at fins.

According to the above contour plot, the maximum thermal flux is 0.459501 k/m and minimum thermal flux is 0.001508 k/m.



3.8 NODAL TEMPERATURE OF ALUMINIUM ALLOY 6061 2.5 MM THICKNESS

According to the contour plot, the temperature distribution maximum temperature at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the maximum temperature at bore and its distributed to outer surface of the fins.



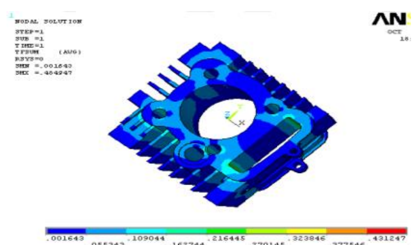
THERMAL GRADIENT SUM OF ALUMINIUM ALLOY 6061 2.5 MM THICKNESS

According to the contour plot, the thermal gradient maximum at bore because the operating temperature passing inside of the bore.

So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum gradient at fins.

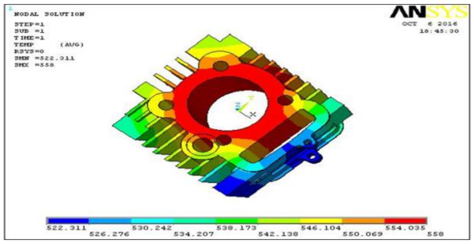
According to the above contour plot, the maximum gradient is 2.694 k/m and minimum gradient is 0.009125 k/m.

THERMAL FLUX SUM



3.9 THERMAL FLUX SUM OF ALUMINIUM ALLOY 6061 2.5 MM THICKNESS

According to the contour plot, the thermal flux maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum thermal flux at fins. According to the above contour plot, the maximum thermal flux is0.484947 k/m and minimum thermal flux is 0.001643 k/m.



THERMAL FLUX SUM OF MAGNESIUM ALLOY 2.5 MM THICKNESS

According to the contour plot, the thermal flux maximum at bore because the operating temperature passing inside of the bore. So we applied the temperature inside of the bore and applied the convection to fins. Then the minimum thermal flux at fins. According to the above contour plot, the maximum thermal flux is 0.477984 k/m and minimum thermal flux is 0.001604 k/m.

4. CONCLUSION

In this thesis, a cylinder fin body for a 150cc motorcycle is modeled using parametric software Pro/Engineer. The original model is changed by changing the thickness of the fins. The thickness of the original model is 3mm, it has been reduced to 2.5mm. By reducing the thickness of the fins, the overall weight is reduced.

Present used material for fin body is Aluminum Alloy 204. In this thesis, two other materials are considered which have more thermal conductivities than Aluminum Alloy 204. The materials are Aluminum alloy 6061 and Magnesium Alloy. Thermal analysis is done for all the three materials. The material for the original model is changed by taking the consideration of their densities and thermal conductivity.

By observing the thermal analysis results, thermal flux is more for Aluminum alloy 6061 than other two materials and also by reducing the thickness of the fin, the heat transfer rate is increased.

Thermal flux is also calculated theoretically. By observing the results, heat transfer rate is more when the thickness of the fin is 2.5mm.

So we can conclude that using Aluminum alloy 6061 and taking thickness of 2.5mm is better.

5.REFERENCES

[1] Thermal Engineering by I. Shvets, M. Kondak

[2] Thermal Engineering by Rudramoorthy

[3] Thermal Engineering by R.K. Rajput

[4] Thermal Engineering by Sarkar

[5] Online Materials

[6] Gibson, A.H., The Air Cooling of Petrol Engines, Proceedings of the Institute of Automobile Engineers, Vol.XIV (1920), pp.243 – 275.

[7] Biermann, A.E. and Pinkel, B., Heat Transfer from Finned Metal Cylinders in an Air Stream, NACA Report No.488 (1935).

[8] Thornhill, D. and May, A., An Experimental Investigation into the Cooling of Finned Metal Cylinders, in a

[9] Free Air Stream, SAE Paper 1999-01-3307, (1999). (4) Thornhill, D., Graham, A., Cunnigham, G., Troxier, P. and Meyer, R.,

[10] Experimental Investigation into the Free Air-Cooling of Air-Cooled Cylinders, SAE Paper 2003-32-0034, (2003). (5) Pai, B.U., Samaga, B.S. and Mahadevan, K., Some

[11] Experimental Studies of Heat Transfer from Finned Cylinders of Air-Cooled I.C. Engines, 4th National Heat Mass Transfer Conference, (1977), pp.137–144.

REUSING, RECYCLING AND UP-CYCLING OF BIOMASS

CH.SHIVA SAI NARESH
Shivasai36.chittiprolu@gmail.com

CH.BHASKAR
Bhaskar1438480@gmail.com

M.TECH.

Abstract:- **There is a growing interest in the utilisation of biomass for a range of applications. Coupled with this is the appeal of improving the circular economy and as such, there is a focus on reusing, recycling and upcycling of many materials, including biomass. This has been driven by society in terms of demand for more sustainable energy and products, but also by a paradigm shift in attitudes of the population to reduce their personal carbon footprint. Herein we have selected a number of types of biomass (woody, herbaceous, etc.) and surveyed the ways in which they are utilised. We have done this in combination with assessing some kinetic modelling approaches which been reported for the evaluation of different processes for the recycling, reuse and upcycling of biomass.**

Keywords:- General classifications of biomass, Processing of biomass resources, Practical applications, Reusing and recycling, Herbaceous biomass, Pyrolysis and bio-oil production, Other types of biomass sources upcycling

1.Introduction

Worldwide, there are issues around "waste" that are prevalent. Perhaps the most pressing issue is the attitudes and behavioural issues in terms of designating something as "waste". From a chemical, and in turn energy, perspective there is normally an intrinsic (carbon) value in such materials; so that it would be better to deem this as "resources" rather than waste. This way of thinking can be considered part of the circular economy, a model by which as much value as possible is retained via reuse, recycling, repurposing and up-cycling with aspirations towards zero waste. As such, there is interest in evaluating the potential value in the circular economy that was highlighted recently by Zacho [1].

1.1 General classifications of biomass

In general, biomass can be classified as primary, secondary and tertiary, where:

Primary biomass:- It is produced directly via photosynthesis and is taken directly from the land. For instance, herbaceous and woody biomass along with the seeds of oil crops and the residues after the harvesting of forest trees and agricultural crops (corn stover, limbs, bark and wheat straw) are all considered primary biomass source [1].

Secondary biomass:- It results from the processing of the primary bio- mass such as sawdust (physical processing), black liquor (chemical processing) or manure production by animals (biological processing) [1].

Tertiary biomass:- It is post-consumer byproducts such as animal fat, used vegetable oils, construction and demolition debris and packing by products [1].

1.2 Classification of biofuel generations

Biomass, when used as a biofuel, can also be classified to the first, second, third and fourth generation where:

(i) 1st generation biofuels come from edible biomass such as the ethanol production from fermenting sugars. There are major drawbacks of this generation in terms of the food versus fuel debate, low land use efficiency along with the geographical limitations

(ii) 2nd generation biofuels are derived from lignocellulosic biomass, either non-edible of food crop (rice husks) or non-edible whole plant biomass resources (grasses). The major challenge related to this generation is that processing can be expensive and a technological breakthrough is still required to make them more feasible.

(iii) 3rd generation biofuels are produced using non-arable land such as algae and typically need a high capital cost to operate

(iv) 4th generation biofuels can be made using non-arable land and do not require the destruction of biomass to be converted into fuel such as the photobiological solar fuels [1].

1.3 Processing of biomass resources

Typically biomass resources can be processed in a number of ways such as:

(i) Combustion, which is the burning of biomass in the air to produce hot gases, ash and consequently converting the stored chemical energy into heat which can be then converted to kinetic energy through heating water to produce vapour which is used for gas engines.

(ii) Gasification, which is the partial oxidation of biomass (for ex- ample rice husk) at high temperatures to convert the biomass to a combustible gas mixture which can be burnt directly or used as a fuel for gas engines, gas turbines or can be used for the production of chemicals in the case of the medium energy gases.

(iii) Pyrolysis, which is the thermal degradation of the biomass in an oxygen-free environment which results a char (solid product), condensable vapour (bio-oil) along with the gaseous products

(iv) Liquefaction, which is the conversion of biomass into stable liquid hydrocarbons at low temperatures and high hydrogen pressures [1].

2. Introductory piece to reusing and recycling approaches

2.1 Practical applications

Firstly, using biomass for combustion in the energy sector depends upon the quality, chemical and physical parameters of the feedstock or feedstocks in question. Mancini et al. developed a combined technique of near-infrared spectroscopy with multivariate analysis to predict the gross calorific value (GCV) and the ash content in the bio- mass feedstock, with a standard error of 0.23 $kJ\cdot g^{-1}$ and 0.4%, respectively. This approach could be used on-line or off-line for providing an indication of the quality of the feedstock and its suitability for the industrial application. Unlike coal, biomass is characterised by its larger volatile content while also being a porous material i.e. biomass allows the ingress of bacteria and O_2, thus, increasing the risk of self-heating and ignition in storage and transportation. Therefore, monitoring the temperature, volatile organic compounds (VOC), carbon monoxide (CO) and controlling the moisture is crucial to avoid the ignition [2].

2.2 Introductory piece on kinetic modelling

Understanding the thermal kinetic behavior of the thermal de- composition of biomass is crucial in order to identify the physicochemical combustion characteristics that hinder some of the energy generation applications. Thus, determining the activation energy (E_a) and the pre-exponential factor using either model-fitting or model-free (isoconversional) approaches, which are the two common methods for studying the Differential scanning calorimetry/Thermogravimetric analysis (DSC/TGA) solid-state kinetic data, is important. The model- fitting method is based on the best fit of the different models with the experimental thermogravimetric data using a single TGA/DSC curve, which might actually fit with more than one model. Conversely, the model-free isoconversional method is a simple method that eliminates the error related to the model fitting. In the isoconversional method, different heating rate curves are required to calculate the kinetic parameters as a function of the extent of conversion (α) i.e. the E_a is calculated for a serious of different conversion points. It is well known that the thermal analysis mechanism is complicated as it tends to take place in multiple steps with different reaction rates and, as such, an isoconversional method is the most suitable method and, consequently, the most commonly used in this case. As biomass is pre- dominantly composed of three different components (cellulose, hemi- cellulose and lignin), each behaves differently and separately during the biomass thermal decomposition. Thus, this complex behaviour of bio- mass decomposition should be considered in kinetic modeling [2].

3. Reusing and recycling

3.1. Woody biomass

There are significant opportunities to recycle and reuse wood. Many items, such as furniture, can be sold second-hand and used again; and there has been human behavioural/attitude changes with regards to secondhand consumption/use which had been typical during the19th Century, but became stigmatized in the 20th Century [3].

More recently there has been a marked increase in secondhand use, which may have been financially driven by the impact on consumers from the 2008 economic crisis. This was likely not the only driver in such a paradigm shift. Consumers are typically much more informed now than previous generations and are aware of environmental issues including land use concerns such as deforestation for agriculture, the eat versus heat debate and landfill capacity. As such, consumers are very conscious of their carbon footprint and, consequently, this is also being factored into their decisions. For example, it has been reported that the reuse/recycle pathways can help reduce GHG emissions, thereby reducing environmental impact

Generally, with the reuse and recycling of woody biomass there is little, if any, chemical conversion and so there are not many examples of kinetic studies or modelling for such processes. Given that ability of biomass to be converted to solid, liquid and gas fuels, biomass is a suitable energy resource for heat, power and transport.

While the utilisation of liquid and gaseous biofuels typically requires some chemical transformation/valorisation of the biomass, this is not necessarily the same for solid fuels [3].

In order to use unprocessed biomass as a solid fuel, it is normally blended with other fuels, such as coal, in order to compensate for the low calorific value and high moisture content inherent in the biomass source. Woody biomass, can, in theory, be combusted without much processing and as such, the application of woody biomass as a fuel can still be considered a low-grade use [3].

3.2 Herbaceous biomass

Herbaceous biomass usually contains components such as seeds, cones, leaves and stems that require further size reduction as a post-harvest handling procedure to increase the bulk density of the biomass, thus facilitating its storage and transportation alongside increasing the surface area, hence, increasing the chemical process reaction rate. As discussed previously, size reduction methods such as the grinding process require energy. Naimiet al, studied the required energy input for five herbaceous biomass (miscanthus, wheat straw, switch grass, corn stover and canolastraw) and found out that it was in the range of 22–35 $kWh\cdot t^{-1}$ at a lab scale while using an industrial grinder showed a required energy input of 59.4 $kWh\cdot t^{-1}$ [3].

Herbaceous biomass such as miscanthus is typically characterized based on its silica and inorganic contents. These components cause various problems during the thermal decomposition via valorization and melting of the low potassium silicates compounds. In the case of woody biomass this problem is less noticeable due to the low alkali and Si content in the biomass. Thus, herbaceous biomass is usually mixed with other woody fuel or coal to harness the energy inherent in these types of biomass. Furthermore, the high ash content in the herbaceous biomass is problematic as it catalyses the thermochemical reactions and affects the yield of the pyrolysis products and their subsequent composition as well. Forbes et al. studied the physicochemical characteristics of eight different biomass fuels such aspine, spruce, brash, Wr (riddled willow), Ws (open air dried chipped willow), miscanthus and commercially available wood pellets. They reported that the standard wood pellet fuel showed the best combustion performance while miscanthus showed the largest clinker formation due to the highest ash content [3].

Biomass-coal co-firing has been shown to diminish the dangerous greenhouse gas emissions and particulates derived from the coal-fired power plants. However, due to the low bulk, low energy density and wet nature of biomass along with the seasonal availability, it makes it a more challenging process. Co-HTC(miscanthus-coal) showed increased higher heating values (HHV)compared with miscanthus (27.3 and 16.81 MJ·kg−1, respectively).While it decreased regarding the pure coal, due to the low mass density with HHV of 28.10 MJ·kg^{-1} [4].

3.3 Other types of biomass

Species of seaweed are conventionally characterised by their physical colour. The contents of organic based constituents such as carbohydrates, fats and proteins vary considerably in different types and species of seaweeds. For instance, brown seaweed is very rich in carbohydrates with its protein content is relatively low, whereas approximately 33 wt% of red seaweed is protein. This feature of brown seaweeds may be useful in carbonisation based processes since carbohydrates can produce volatile gaseous compounds, as well as some char in the pyrolysis/carbonisation processes. Phaeophyta or "brown seaweed" as it is more commonly known, is the most common type of seaweed. Mannitol, fucoidan, laminarin and alginic acid are the main building blocks of this particular type of seaweed.

Conventionally, seaweed is classed as unsuitable for thermochemical conversions such as combustion and gasification unless pre-treatments are carried out or the application is carried out in conjunction with a co-existing feedstock. It is the inorganic matter that is contained within the seaweed that gives rise to some problems. Typically, this will lead to a higher variation in ash-forming elements, ash content and high levels of salts (e.g. sodium chloride) compared to other fuel sources available such as coal or diesel [5].

The combination of high Na and Cl concentrations promote the risk of alkali chloride-related operational problems. There are several studies reporting on the complex ash composition that arises from seaweed-based feedstocks. For example, compared to land-based biomass, the composition of seaweeds are heterogeneous with high concentrations of ash-forming elements, higher ratios of Na/K and Mg/Ca and higher Cl contents. These higher concentrations can be explained by the region and environment in which the biomass grows (i.e. sea water). Characterisation for this type of biomass and its ash content is extremely important because numerous problems (fouling, deposit formation or slagging) can arise when using as a fuel application. These problems will likely cause a financial and time burden in the form of potential shutdowns of apparatus or there being substantial periodical maintenance required. In most cases, seaweeds require a pre-treatment step prior to utilisation, and washing/leaching with water or weak acids are the most common practice to remove mineral matter and halogens

4. Up-cycling approach

4.1. Herbaceous biomass

(i) Pyrolysis and bio-oil production

Recently, the production of bio-oil from biomass has gained great attention as an alternative to fossil fuels. Usually bio-oil, which is typically dark-brown, is composed of different chemicals such as esters ,acids, alcohols, ketones, aldehydes, phenols and lignin-derived oligomers. Bio-oil is thus characterised by its high acidity (pH 2–3), high moisture and viscosity [212]. Bio-oil typically has low heating value compared with fossil fuel with HHV of approximately 20 and40 MJ·kg−1, respectively. In general, the pyrolysis of hemicellulose and cellulose occurred faster than that of lignin with weight loss in the range of 220–315 °C, 315–400°C and 160–900 °C, respectively [5] .

The pyrolysis of cellulose is endothermic while pyrolysis of hemicellulose and lignin is exothermic . Previous studies reported four stages during the pyrolysis; water desorption followed by three decomposition stages for cellulose, hemicellulose and lignin. The product distribution depends upon the feedstock chemical and biochemical composition, the biomass taxonomy and the secondary interaction between liquid and char [5].

In general, the highest yield of liquid (mostly an hydro sugars) is produced by the decomposition of cellulose[6]
along with the lowest char yield. While hemicellulose decomposes to the highest yield of gas and liquid (mainly water, ketones and phenols), lignin produces the phenols in the liquid form along with the highest yield in char.

Woody biomass decomposes slower than that of the herbaceous biomass with less evolution of volatile matters due to the larger contents of cellulose and hemicellulose within the herbaceous biomass. The produced biochar is characterised with higher calorific value than the raw biomass with values in the range of 25–26 MJ·kg−1 which is due to the increase in the fixed carbon content. The bio-oil composition and yield are unfavourably affected by the amount of the ash content in the biomass feedstock, therefore washing it with water, surfactant or either acid improves both the yield and the composition of the bio-oil. For instance, washing the biomass with water showed an effective way to reduce the inorganic materials such as (K, P and Cl) [5] and it is an ideal pre-treatment choice for woody biomass (low ash content). Again, washing with a surfactant such as Triton X-100 can effectively reduce the ash content in comparison to the untreated biomass. Washing the raw biomass with HCl reduced the metal content and consequently increased the volatile materials during the pyrolysis which in turn increased the bio-oil yield and also reduced the hemicellulose present in the polymer structure of the biomass [5].

Banks et al. reported that using stronger acid could lead to hydrolyse both cellulose and hemicellulose completely that eventually decreased the bio-oil yield [5].

4.2 Woody biomass

Woody biomass can be upcycled/valorised via torrefaction, gasification, hydrothermal carbonisation, biological treatments, pyrolysis, anaerobic digestion, fermentation and transesterification. Many of these processes have been described herein previously for recycling/
reusing and/or applied to other types of biomass.

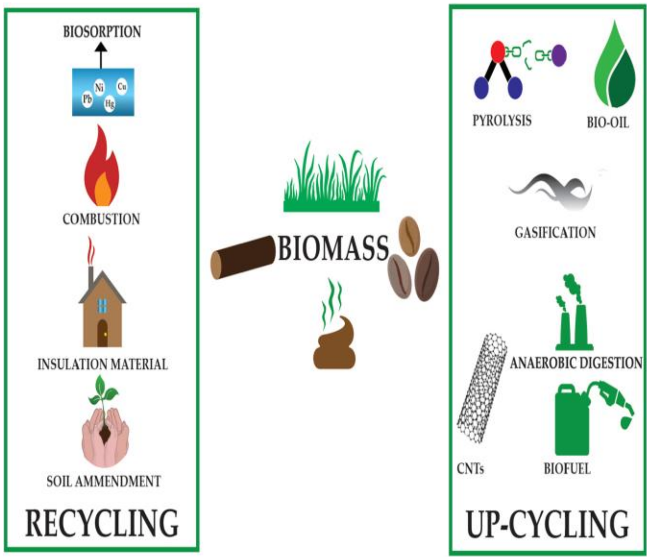
Primarily in this section, the focus will be on the bio-refinery and the extraction of platform and value-added chemicals from woody biomass. Woody biomass can be fed into anaerobic digesters for the conversion of solid/liquid heterogeneous biomass mixtures to biogas, though would typically require a higher residence time compared to some other types of biomass. Pyrolysis and hydrothermal liquefaction result in bio-oil which can then be upgraded to the value added chemicals or fuels. Pyrolysis is conducted between 450 and 500 °C for short residence times at atmospheric conditions and requires drying whereas the hydrothermal liquefaction is at lower temperatures (300–400 °C) at higher pressure (up to 25 MPa) and for longer times (up to 1 h) with the advantage of no drying step being required.

4.3 Other types of biomass sources upcycling

Ross et al. have noted that the char yield from the slow pyrolysis/carbonisation of macro-algae should not be neglected and should be utilised. The abundance of alkali metals in the biomass directly affects the char yield as it alters the pyrolysis mechanism as reported by Ross et al. themselves elsewhere [6].

In order to further predict the yield of pyrolysis products from slow pyrolysis and compare with that of fixed-bed reactor experiments, a mechanistic-based model was designed. This was an adaption of a model originally developed by Van de Velden et al.]. The model
was found to give good accuracy at predicting the pyrolysis products over the temperature range of 400–600 °C producing a mean squared error of 0.49, 16.34 and 13.37 for gas, bio-oil and char, respectively [6].

The overall bio-oil yield from the slow pyrolysis process of cassava peel was found to be 38.7–51.2 wt%, with the optimum yield of 51.2 wt%, corresponding to a temperature of 525 °C. Above this temperature, the yield was found to decrease and this was attributed to secondary thermal cracking. The resultant bio-oil properties satisfied all the ASTM D7544 standard requirements which suggest that bio-oil coming from the pyrolysis of cassava peel could be used as a resultant bio-fuel.



5. Prospective overview and conclusion

A number of different types of biomass have been considered, while various processes have been discussed for their reuse, recycling an upcycling as seen in Fig. 1. Additionally, the prospects of boosting the circular economy of biomass have been considered. There are significant literature contributions, of which the references of the current work are not an exhausted list, and it is clear that there is a strong desire for maximum utilisation of biomass. By also considering the kinetic modelling of some processes it is hoped to demonstrate that such work can and does play a vital role in the understanding of biomass processing technology. It has also demonstrated that there is still potential to improve on both the actual technologies as well as the kinetic models. Finally, while it is clear that biomass is still required for some lower grade uses, it is preferable to extract the maximum value from the biomass sources if we are truly to achieve a circular economy. It is clear, however, that there is still some progress which is required in this field due to the energy-intensive nature of some of the processes.

6. References

[1] K.O. Zacho, M. Mosgaard, H. Riisgaard, Capturing uncaptured values — a Danish case study on municipal preparation for reuse and recycling of waste, Resource. Conserv. Recycle. 136 (2018) 297–305.

[2] P. Schroeder, P. Dewick, S. Kusi-Sarpong, J.S. Hofstetter, Circular economy and power relations in global value chains: Tensions and trade-offs for lower income countries, Resource. Conservation. Recycle. 136 (2018) 77–78.

[3] Queen's University Belfast gets £1m to research plastic waste, https://www.bbc. co.uk/news/uk-northern-ireland-46648020 , Accessed date: 30 January 2019.

[4] J.M. Encinar, J.F. González, Pyrolysis of synthetic polymers and plastic wastes. Kinetic study, Fuel Process. Technol. 89 (2008) 678–686.

[5] A.I. Osman, J.K. Abu-Dahrieh, F. Laffir, T. Curtin, J.M. Thompson, D.W. Rooney, A bimetallic catalyst on a dual component support for low temperature total methane oxidation, Appl. Catal. B Environ. 187 (2016) 408–418.

[6] A.I. Osman, J. Meudal, F. Laffir, J. Thompson, D. Rooney, Enhanced catalytic activity of Ni on η-Al 2 O 3 and ZSM-5 on addition of ceria zirconia for the partial oxidation of methane, Appl. Catal. B Environ. 212 (2017) 68–79.

DESIGN AND FINITE ELEMENT ANALYSIS OF A PISTON OF INTERNAL COMBUTION ENGINE

Saiteja Singoju, Rama Krishna Pandiboina
Department of Mechanical Engineering,
Malla Reddy College of Engineering,
Jawaharlal Nehru Technological University, Hyderabad.
saitejasingoju@gmail.com
ramakrishnapandiboina@gmail.com

***Abstract*— This journal describes the stress distribution of the piston in four stroke engines by using FEM. Our main objectives is to Study and analyze the thermal stress and maximum or minimum principal stresses, Vanishes stresses distribution on engine piston during combustion process. The journal describes the optimization techniques with using finite element analysis method (FEM) to predict the higher stress and critical region on the component. The stress concentration on the piston head, piston skirt and piston sleeve are reduced by the optimization, using computer aided design (AUTO CAD), Pro- ENGINEER/ CREO software the structural model of a piston will be developed. Furthermore, the FEM analysis is done using Computer Aided Simulation software.**

Keywords— FEA, Pro-E, ANSYS, Piston crown, Piston skirt, stress concentration, Thermal analysis.

I. INTRODUCTION

A piston is a cylindrical engine component that Reciprocates in the cylinder bore by forces produced during the combustion process. The combustion chamber is made gas-tight by the piston rings. In an engine, its transfer force from expanding gas in the cylinder to the crankshaft via connecting rod. As a main part in an engine, piston endures the cyclic gas pressure and the inertial forces at work, and this real working condition may cause the fatigue damage of piston, such as piston skirt wear, piston head or crown cracks and so on. The investigations denote that the greatest stress appears on the upper end of the piston and stress concentration is one of the mainly reason for fatigue failure. On the other hand piston over heating-seizure can only occur when something burns or scrapes away the oil film that exists between the piston and the cylinder wall. Understanding this, it's not hard to visually why oils with exceptionally high film strengths are very desirable. Good quality oils will offer provide a film that stands up to the most intense heat and the pressure loads of a modern high output engine. Thermal analysis is a branch of materials science where the properties of materials are studied as they change with temperature. Finite element method (FEM) are commonly used for thermal Analysis. Due to the complicated working environment for the piston; on one hand, the finite element method (FEM) for the piston became more difficult, on the other hand, though there have many methods which are put forward to apply optimal design, the optimal parameters is not easy to determine. In this, the piston is used in low idle and rated speed gas engine. In order to enhance the engine performance.

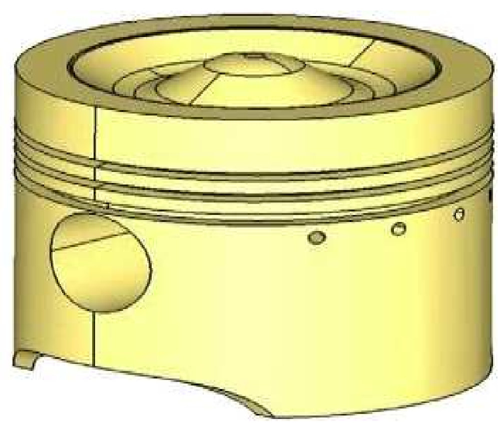


Fig.1 Piston of IC engine

It Is necessary for the piston to be optimized. The mathematical model of optimization is established firstly, and then the FEA is carried out by using ANSYS software. Based on the analysis of optimal result, the stress concentrates on the Upper end of piston has to be evaluated, which provides a better reference for redesign of the piston.

A. *Properties of a Piston Material :*

Aluminum silicon alloys are widely used in the production of pistons because of their

- High Strength,
- Low Density,
- High Thermal Conductivity,
- Good Cast Ability,
- Workability,
- Good Machinability ,
- High Temperature Resistance.

B. *Aspects of a IC Engine Piston:*

The Piston of an IC engine should comprises of the following characteristics:

- High Strength to resist gas pressure.
- Should have minimum weight.
- Should able to reciprocate with minimum noise.
- Should seal the gas from top and oil from the bottom.
- Should disperse the heat generated during combustion.
- Should have good resistance to distortion under high loads and high temperature.

In this study the piston material we choose is AlSi12CuMgNi cast alloy with eutectic microstructure. As all the engine components around the combustion chamber experience significantly high temperatures and temperature gradients, the temperature dependent property materials has to be used.
According to the thermal analysis results maximum piston temperature reaches approximately 374°C, therefore the cyclic behavior of material is considered at 20, 150, 250 and 350 °C.

C. *Engine Specifications:*

The engine we are considering in this article is a four-cylinder four -stroke air cooled type Bajaj Kawasaki diesel engine.
The engine specifications are mentioned below:

PARAMETERS	VALUES
Engine Type	Four stroke, diesel engine
Induction	TCIC
Number of cylinders	4 cylinder
Bore	74 mm
Stroke	70 mm
Length of connecting rod	97.6 mm
Displacement volume	99.27 cm^3
Compression ratio	16
Maximum power	21.6 KW at 7000 rpm
Maximum Torque	86 Nm at 3500 rpm
Number of revolutions/cycle	2

Table 1. Engine specifications

II. CAE APPROACH

Computer-Aided-Engineering (CAE) is the computer software to aid in engineering in analyzing tasks. In this standard approach conception ideas are converted into sketches or engineering drawing. With the assistance of this drawings, the prototypes i.e. product which appearance as that of final product are created. It is launched in the market after testing of prototype which gives acceptable results. The thing is, product is launched after doing several practical testing and many trial and error procedures which consumes more time and cost too [1]. Figure 2 depicts the flow process adopted for typical design approach.

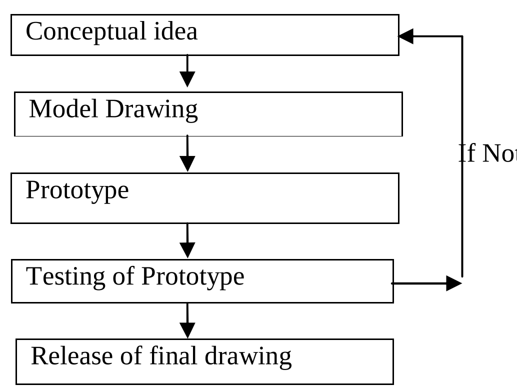


Fig.2 Flowchart of a Conventional Approach to Finite element analysis

FEA method is the mathematical idealization of authentic system. It is a computer predicated method that breaks geometry into element and link a series of equation to every, which are then solved simultaneously to evaluate the outward behavior of the complete system. It is utilizable for perplexed geometry, loading, and material properties where exact analytical solution are difficult to obtain. Most often utilized for structural, thermal, fluid analysis and simulation.

III. METHODOLOGY

During the working cycle operation, the piston is exposed to the high gas pressure and high temperature because of combustion. Simultaneously the piston is fortified by a minute terminus of the connecting rod with the Gudgeon pin. Therefore the methodology for analyzing the piston is considered as the gas pressure given 180 bar is applied uniformly over the crown (top surface/face of the piston) and all degrees of liberation for nodes at upper moiety of piston pin bossed in that piston pin is going to fine- tune. By Considering the fit between piston pin and piston is clearance fit. Only the upper moiety of piston pin boss is considered to be fine-tuning during the analysis.

A. Material properties of piston:

- Material of Piston: - Cast aluminum alloy 201.0
- Young’s Modulus [E] – 71 GPa
- Poisson’s ratio [µ] – 0.33
- Tensile strength – 485 MPa
- Yield strength – 435 MPa
- Shear strength –290 MPa
- Elongation – 7 %

B. The geometrical representation of an IC Engine piston used for FEA.:

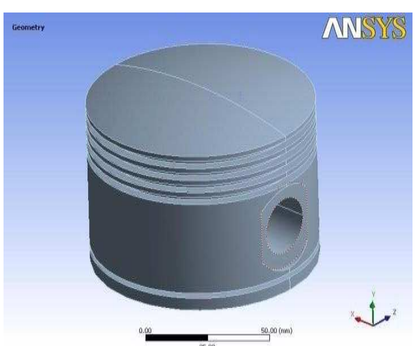


Fig. 4 Basic Geometry of a IC engine piston

A geometrical model of piston is prepared by modeling software’s like PRO-e/ CREO, CATIA V5 and be modeled and analysed in the analysis software ANSYS.

C. Finite element model:

The element selected for meshing of the piston model is SOLID187 tetrahedron type of element which is a higher order tetrahedral element. The mesh count for the selected model is 71,910 of nodes and 41,587 number of elements. The figure below is the meshed model of the piston.

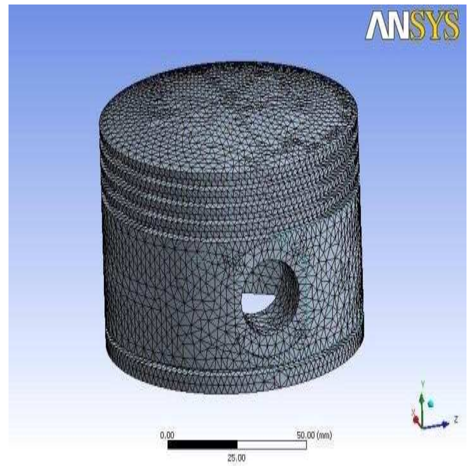


Fig.5 Meshed model of piston

D. Loading & boundary conditions:

The loading and boundary conditions considered for the analysis are showed in the below figure. The uniform pressure of 18 MPa is applied on crown of the piston (top red color) and the model is constrained on upper moiety of piston pin aperture as shown by violet color.

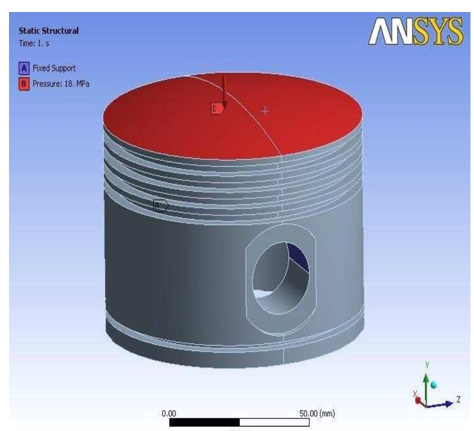


Fig.6 Loading and boundary conditions on piston

IV. RESULTS AND DISCUSSION

A. *Total deflection:*

The maximum deflection in the piston geometry due to the application of gas pressure observed at the central portion of the piston crown is is 0.29669 mm.

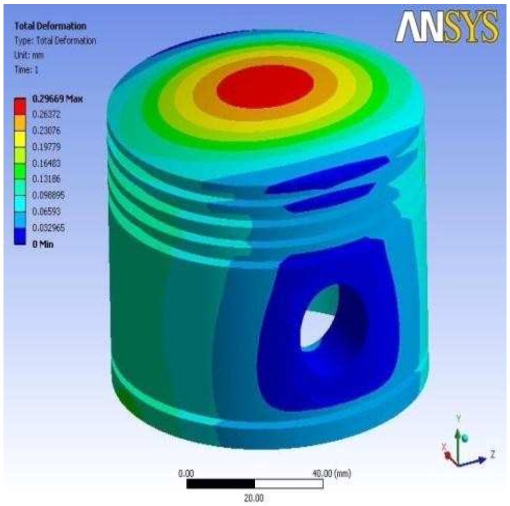


Fig.7 Total deflections on the piston head

B. *Maximum principal stress:*

The distribution of localized and observed at inner side of piston pin boss in the figure below. The overall maximum stresses in the piston body at the inner side of piston crown and piston boss is resulted as 231.25N/mm2.

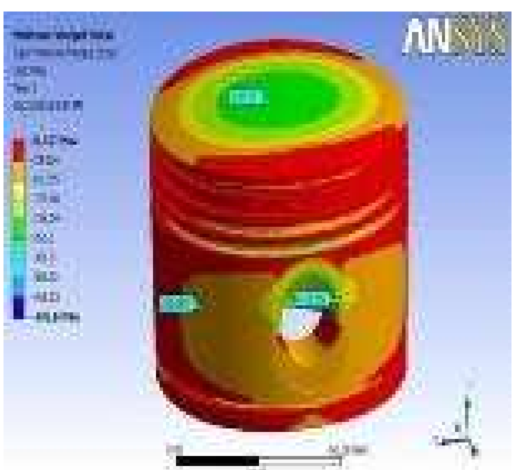


Fig.8 Maximum principal stress on piston

C. *Minimum principal stress:*

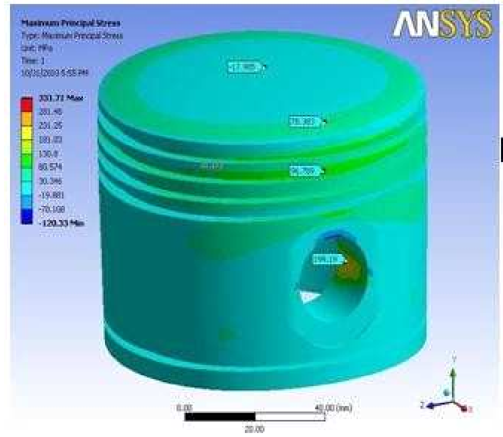


Fig.9 minimum principal stress on piston

The above figure shows the distribution of the minimum principle stresses induced within the piston body. The most maximum values of equivalent stresses are goes up to - 376.74 N/mm^2, which are highly localized and observed at inner side of piston crown & skirt junction. The overall maximum stresses in the piston body is - 250.5 N/mm2 at the top of piston crown.

D. *Von mises stresses:*

The above Figures 10.1 and 10.2 show the distribution of Von mises stresses induced within the piston body. The utmost maximum values of equivalent stresses are goes up to 200.97N/mm2.

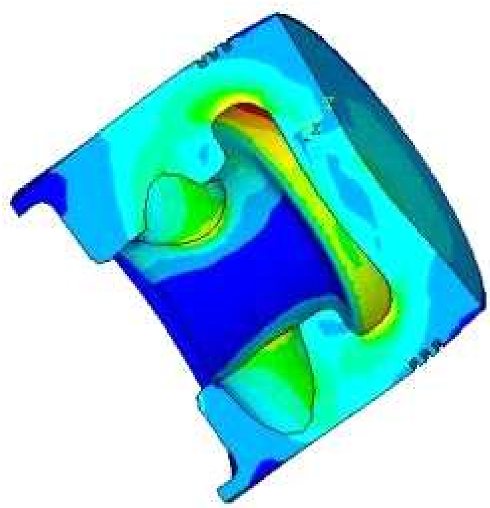


Fig. 10.1 Von mises stress

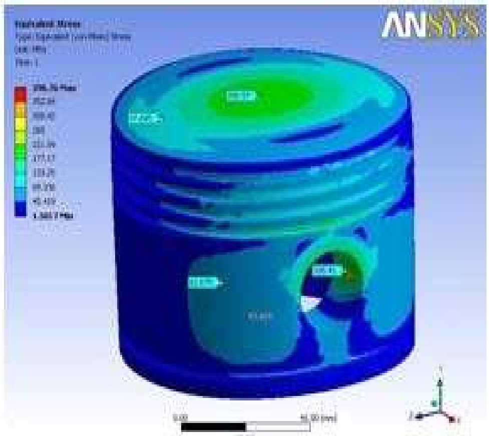


Fig. 10.2 Von mises stress

E. *Stress distribution on the piston body:*

The critical area is observed on the piston head and piston pin hole region. Figure 11 shows stress concentration at various point in piston.

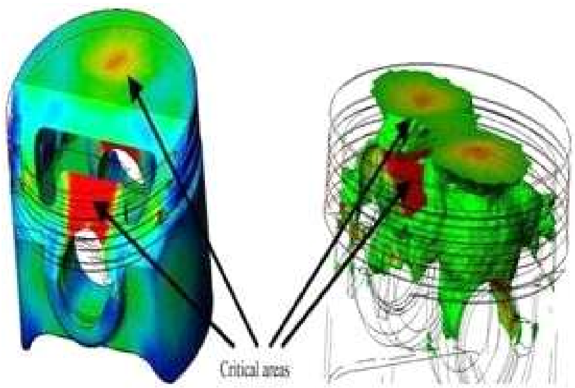


Fig.11 Typical stress distribution on an engine piston

V. CONCLUSION

The Piston skirt may appear deformation during the cyclic operation, which usually results in the crack on the upper end of piston head. Due to the deformation, the greatest stress concentration is caused on the crown, it may leads to the failure of the piston when the stiffness of the piston is not enough, and the crack generally appeared at the point A which may gradually expand and even cause splitting aacross the piston vertically. The stress distribution on the piston mainly depends on the deformation of piston. Therefore, in order to reduce the stress concentration, the piston crown should have enough stiffness.

Also from analysis various results are obtained like The maximum deflection occurred about 0.29669mm due to the application of 180bar gas pressure on crown of piston, 231.25N/mm^2 of maximum principal stress is ascertained, -250.5 N/mm2 of minimum principal stress is ascertained. Also von mises stress of 200.97N/mm2 is observed.

REFERENCES

"Computer Aided Engineering in Compression Molding" (Antoine Rios, Bruce Davis and Paul Grumman, the Madison Group: Polymer Processing Research Corporation 505 S. Ronsard. Madison, WI 53719)

C.H. Li. Piston thermal deformation and friction Considerations, SAE Paper 820086, 1982.

Properties and Selection:High- performance Alloys, ASM Handbook, vol. 1, ASM International, 1990.

Handbook of IC Engines, SAE International.
Martin, Gary G. Failure of stationary pump engine piston, J Failure Anal Prevention 2004;4(1):37–9.

"Fatigue on engine pistons–A compendium of case Studies", F.S.Silva *, 31 March 2005.

FE Analysis of a Vibrating Rigid Circular Piston in Water" by K. Jagadeesh and M. S. Vijaya2SASTECH Vol. VI, NO.1, in the year April 2007.

F.S. Silva, Fatigue on engine pistons – A compendium of case studies, 13 (2006) 480–492.

"Finite Element And MBD Analysis Of Piston To Predicts The Engine Noise" S.N Kurbet1,Vinay V Kuppast1 and Vijay kumar N Chalwa Int. J. Mech. Eng. & Rob. Res. 2013.

DESIGN AND ANALYSIS OF A FOUR WHEELER CRANK SHAFT BY DIFFERENT ALUMINUM ALLOYS

DHEERAVATH SHEKHAR
M.Tech(Thermal Engineering)
Mechanical Engineering Department,
Malla Reddy College Of Engineering,Hyderabad.

***Abstract*:** A crankshaft is used to convert reciprocating motion of the piston into rotary motion or vice versa. The crankshaft consists of the shaft parts which revolve in the main bearings, the crankpins to the big ends of the connecting rod are connected, the crank arms or webs which connects the crankpins and the shaft parts. The crankshafts are subjected to shock and fatigue loads. Thus material of the crankshaft should be tough and fatigue resistant. The common materials used for crankshaft are Carbon Steel or Nickel-Chrome-Moly alloy steel or Nickel-Chrome or special cast iron. The aim of the project is to design and manufacturing a crankshaft for a four cylinder IC engine by using theoretical calculations in design area for Aluminum alloys6061 and commercial grade. A 2D drawing is drafted for crankshaft from the calculations and a 3D model is created in the 3D modeling software CREO

Introduction to crankshaft:

The crankshaft, sometimes casually abbreviated to crank, is the part of an engine which translates reciprocating linear piston motion into rotation. To convert the reciprocating motion into rotation, the crankshaft has "crank throws" or "crankpins", additional bearing surfaces whose axis is offset from that of the crank, to which the "big ends" of the connecting rods from each cylinder attach. It typically connects to a flywheel, to reduce the pulsation characteristic of the four-stroke cycle, and sometimes a torsional or vibrational damper at the opposite end, to reduce the torsion vibrations often caused along the length of the crankshaft by the cylinders farthest from the output end acting on the torsional elasticity of the metal.

1) CRANKSHAFT MANUFACTURING PROCESSES:

Many high performance crankshafts are formed by the forging process, in which a billet of suitable size is heated to the appropriate forging temperature, typically in the range of 1950 - 2250°F, and then successively pounded or pressed into the desired shape by squeezing the billet between pairs of dies under very high pressure. These die sets have the concave negative form of the desired external shape. Complex shapes and / or extreme deformations often require more than one set of dies to accomplish the shaping.

Originally, two-plane V8 cranks were forged in a single plane, then the number two and four main journals were reheated and twisted 90° to move crankpins number two and three into a perpendicular plane. Later developments in forging technology allowed the forging of a 2-plane "non-twist" crank directly (**Figure 1**).

Figure:1
Two-Plane V8 Crankshaft Raw Forging

Crankshafts at the upper end of the motorsport spectrum are manufactured from billet. Billet crankshafts are fully machined from a round bar ("billet") of the selected material (**Figure 2**). This method of manufacture provides extreme flexibility of design and allows rapid alterations to a design in search of optimal performance characteristics. In addition to the fully-machined surfaces, the billet process makes it much easier to locate the counterweights and journal webs exactly where the designer wants them to be. This process involves demanding machining operations, especially with regard to counterweight shaping and undercutting, rifle-drilling main and rod journals, and drilling lubrication passages. The availability of multi-axis, high-speed, high precision CNC machining equipment has made the carved-from-billet method quite cost-effective, and, together with exacting 3D-CAD and FEA design methodologies, has enabled the manufacture of extremely precise crankshafts which often require very little in the way of subsequent massaging for balance purposes.

Figure:2
Billet Crankshaft Machining (Courtesy of Bryant Racing)

There is an old argument that a forged crank is superior to a billet crank because of the allegedly uninterrupted grain flow that can be obtained in the forging process. That might be true of some components, but with respect to crankshafts, the argument fails because of the large dislocations in the material that are necessary to move the crankpin and counterweight material from the center of the forging blank to the outer extremes of the part. The resulting grain structure in the typical V8 crank forging exhibits similar fractured grain properties to that of a machined billet. More than one crankshaft manufacturer has told me that there is no way that a forging from the commonly used steel alloy SAE-4340 (AMS-6414) would survive in one of today's Cup engines.

Some years ago, there was an effort at Cosworth to build a Formula One crankshaft by welding together various sections, which comprised the journals, webs and counterweights. The purported intent was to be better able to create exactly the shape and section of the various components, thereby reducing MMOI while achieving the same or better stiffness. While no one was willing to divulge details about the effort, it is rumored to have been run once or twice then abandoned due to the high cost and complexity compared to the measurable benefits.

In certain cases, there are benefits to the use of a built-up crankshaft. Because of the ‘master-rod’ mechanism necessary for the implementation of the radial piston engines that powered most aircraft until well into the second half of the 20th century, a bolted-together crankshaft configuration was used almost exclusively. **Figure 5** illustrates a typical two-row composite radial crankshaft and master-rod layout. The loose counterweights will be addressed later in this article.

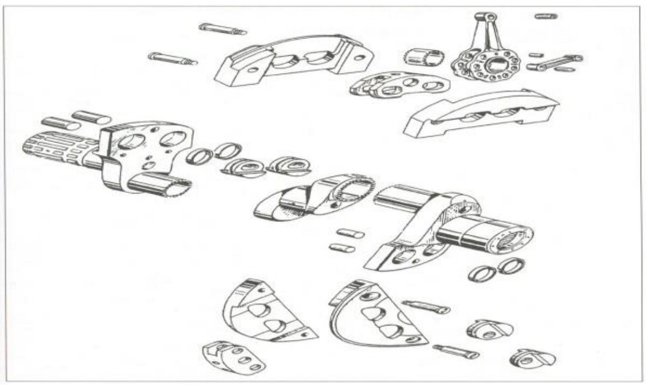


Figure:3

Built-up Radial Engine Crank

2) CRANKSHAFT MATERIALS:

The steel alloys typically used in high strength crankshafts have been selected for what each designer perceives as the most desirable combination of properties. **Figure 4** shows the nominal chemistries of the crankshaft alloys discussed here.

Medium-carbon steel alloys are composed of predominantly the element iron, and contain a small percentage of carbon (0.25% to 0.45%, described as ‘25 to 45 points’ of carbon), along with combinations of several alloying elements, the mix of which has been carefully designed in order to produce specific qualities in the target alloy, including hardenability, nitridability, surface and core hardness, ultimate tensile strength, yield strength, endurance limit (fatigue strength), ductility, impact resistance, corrosion resistance, and temper-embrittlement resistance. The alloying elements typically used in these carbon steels are manganese, chromium, molybdenum, nickel, silicon, cobalt, vanadium, and sometimes aluminium and titanium. Each of those elements adds specific properties in a given material. The carbon content is the main determinant of the ultimate strength and hardness to which such an alloy can be heat treated.

Chemistry of Crankshaft Alloys
Nominal Percentages of Alloying Elements

Material	AMS	C	Mn	Cr	Ni	Mo	Si	V
4340	6414	0.40	0.75	0.82	1.85	0.25		
EN-30B		0.30	0.55	1.20	4.15	0.30	0.22	
4330-M	6427	0.30	0.85	0.90	1.80	0.45	0.30	0.07
32-CrMoV-13	6481	0.34	0.55	3.00	<0.30	0.90	0.25	0.28
300-M	6419	0.43	0.75	0.82	1.85	0.40	1.70	0.07
Key:	C = Carbon		Mn = Manganese			Cr = Chromium		
	Ni = Nickel		Mo = Molybdenum			Si = Silicon		
	V = Vanadium			AMS = Aircraft Material Spec Number				

Figure: 4

In addition to alloying elements, high strength steels are carefully refined so as to remove as many of the undesirable impurities as possible (sulfur, phosphorous, calcium, etc.) and to more tightly constrain the tolerances, which define the allowable variations in the percentage of alloying elements. The highest quality steels are usually specified and ordered by reference to their AMS number (Aircraft Material Specification). These specs tightly constrain the chemistry, and the required purity can often only be achieved by melting in a vacuum, then re-melting in a vacuum to further refine the metal. Typical vacuum-processing methods are VIM and VAR.

There are other ultra-high-strength steels that are not carbon steels. These steels, known as "maraging" steels, are refined so as to remove as much of the carbon as possible, and develop their extreme strength and fatigue properties as a by-product of the crystalline structures resulting from the large amounts of nickel (15% and up) and cobalt (6% and up) they contain. These steels can achieve extreme levels of strength and maintain excellent levels of impact resistance. As far as I could determine, maraging alloys are not currently (2008) used for racing crankshafts but they have been used in certain extreme application conrods.

The material which is currently viewed as the ultra-extreme crankshaft alloy is a steel available from the French manufacturer Aubert & Duval, known as 32-CrMoV-13 or 32CDV13. It is a deep-nitriding alloy containing 300 points of chrome, developed in the mid-nineties specifically for aerospace bearing applications. It is available in three grades. GKH is the commercial purity and chemistry tolerance. GKH-W is the grade having higher purity (VAR) and tighter chemistry tolerance. GKH-YW is the extremely pure grade (VIM - VAR) and is said to cost twice as much per pound as the -W grade.

According to data supplied by Aubert & Duval, fatigue-tests of the -W and -YW grades, using samples of each grade heat treated to similar values of ultimate tensile strength, show consistently that the -YW grade achieves a dramatic improvement (over 22%) in fatigue strength compared to the -W grade, and the endurance limit is claimed to be just a bit short of the yield stress, which is truly amazing. I have been told that, because of the extreme stress levels on Formula One crankshafts, most of them use the -YW grade, while the lower stress levels of a Cup crank allow the successful use of the -W grade.

One well-known manufacturer (Chambon) has developed a process which allows the production of a deep case nitride layer in this alloy (almost 1.0 mm deep, as compared to the more typical 0.10 to 0.15 mm deep layer). They say this deeper case provides a far less sharp hardness gradient from the >60 HRc surface to the 40-45 HRc core, which improves the fatigue and impact properties of the steel. It says that its deep-case process requires several days in the nitriding ovens, but the depth allows finish-grinding after nitriding, using a very sophisticated process to remove the distortions which occurred during the nitriding soak.

No discussion of high-end crankshaft materials would be complete without mention of the ultra-high-strength alloy known as 300-M (AMS 6419). This alloy is a modification to the basic 4340 chemistry, in which a few more points of carbon are added (higher achievable hardness and strength), along with 170 points of silicon and 7 points of vanadium. The vanadium acts as a grain refiner, and the silicon enables the material to be tempered to very high strength (285 ksi) and fatigue properties, while retaining extremely good impact resistance and toughness.

This material (300-M) is expensive and sometimes hard to get, since it is preferred for heavy aircraft landing gear components. It has been used by a few manufacturers for extreme duty crankshafts and conrods as well as high-shock aircraft components. However, several of the manufacturers I spoke with told me that they consider their favorite materials to be much better than 300-M for crankshaft applications.

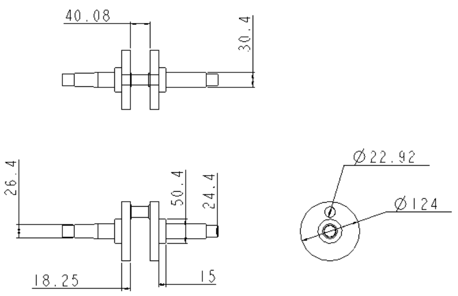
3 DESIGN OF CRANKSHAFT:

SPECIFICATIONS:-

Lml freedom Bore diameter or cylinder bore = D=69.6mm Stroke = 82mm Explosion pressure gas pressure =P =10.936 N/mm2 Maximum torque = 200 N-m @1750-3000 rpm Design of crank shaft when the crank is at the dead centre We know that piston gas load Now the various parts of the crank shaft are design as discussed below: 1. Design of crank pin: let We have Takin 41585.951= Let us now the included bending stress in the crank pin We know that bending moment at the crank pin M = Section modules of the crank pin Z = Bending stress induced The induced bending stress is within in the permissible limits of 560 Mpa, therefore design of crank pin is safe. 2. Design of bearings Let Let us take thickness of the crank web t = 0.6 length of bearing WKT bending moment at centre of bearing M = =41585.951(0.75) =6133927.773 Bending moment (M) WKT 6133927.773= 3.

Design of crank web Let w = width of the crank web in mm WKT bending moment on the crank web M = =41585.951(0.75)=2703086.815 Section modulus Z = Bending stress Total stress on the crank web = Total stress should not exceed permissible limit of 560 Mpa 560 = W = 17.390=20mm Design of shaft under the flywheel: Let First of all let us find the horizontal and vertical reactions at the bearings 1 and 2 Allowing for certain clearance the distance b = W+ 300+ 442mm and a = 0.75 WKT the horizontal reactions at bearings 1 and 2 due to the piston gas load () are Assuming The vertical reactions at bearings a1 and 2 ue to the weight of flywheel are W = weight of flywheel = 1.18kg=11.57N There is no belt tension the horizontal reactions due to the belt tension are neglected WKT horizontal bending moment at the flywheel location due to piston gas load = 3066964 There is no belt pull. There will be no horizontal bending moment due to the belt pull, i.e Total horizontal bending moment WKT vertical bending moment due to the flywheel weight Resultant bending moment = = 3066964.266 We know that bending moment 3066964.266.

2D DRAWING



INTRODUCTION TO CAD

Computer-aided design (**CAD**), also known as **computer-aided design and drafting** (**CADD**), is the use of computer technology for the process of design and design-documentation. Computer Aided Drafting describes the process of drafting with a computer. CADD software, or environments, provide the user with input-tools for the purpose of streamlining design processes; drafting, documentation, and manufacturing processes. CADD output is often in the form of electronic files for print or machining operations. The development of CADD-based software is in direct correlation with the processes it seeks to economize; industry-based software (construction, manufacturing, etc.) typically uses vector-based (linear) environments whereas graphic-based software utilizes raster-based (pixelated) environments.

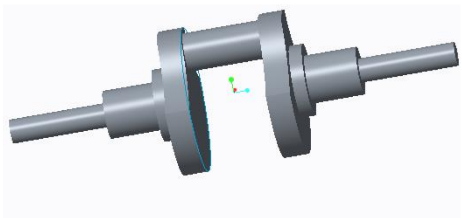
CADD environments often involve more than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) objects.

INTRODUCTION TO CREO

PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself. The name was changed in 2010 from Pro/ENGINEER Wildfire to CREO. It was announced by the company who developed it, Parametric Technology Company (PTC), during the launch of its suite of design products that includes applications such as assembly modeling, 2D orthographic views for technical drawing, finite element analysis and more.

The time saved by using PTC CREO isn't the only advantage. It has many ways of saving costs. For instance, the cost of creating a new product can be lowered because the development process is shortened due to the automation of the generation of associative manufacturing and service deliverables. PTC also offers comprehensive training on how to use the software. This can save businesses by eliminating the need to hire new employees.



ADVANTAGES OF CREO PARAMETRIC SOFTWARE

1. Optimized for model-based enterprises
2. Increased engineer productivity
3. Better enabled concept design
4. Increased engineering capabilities
5. Increased manufacturing capabilities
6. Better simulation
7. Design capabilities for additive manufacturing

CREO parametric modules:

- Sketcher
- Part modeling
- Assembly
- Drafting

INTRODUCTION TO FEA

Finite element analysis is a method of solving, usually approximately, certain problems in engineering and science. It is used mainly for problems for which no exact solution, expressible in some mathematical form, is available. As such, it is a numerical rather than an analytical method. Methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met with in engineering. For example, engineering strength of materials or the mathematical theory of elasticity can be used to calculate analytically the stresses and strains in a bent beam, but neither will be very successful in finding out what is happening in part of a car suspension system during cornering.

INTRODUCTION TO ANSYS

ANSYS Autodyn is computer simulation tool for simulating the response of materials to short duration severe loadings from impact, high pressure or explosions.

1. ANSYS Mechanical

ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behavior, and supports material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo-electric analysis.

2. Fluid Dynamics

ANSYS Fluent, CFD, CFX, FENSAP-ICE and related software are Computational Fluid Dynamics software tools used by engineers for design and analysis. These tools can simulate fluid flows in a virtual environment — for example, the fluid dynamics of ship hulls; gas turbine engines (including the compressors, combustion chamber, turbines and afterburners); aircraft aerodynamics; pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners, etc.

Definitions of Results obtained

Displacement - A vector quantity which refers to the distance which an object has moved in a given direction. It is measured as the length of a straight line between the initial and final positions of a body.

Von Mises Stress - The Von Mises criteria is a formula for combining these 3 stresses into an equivalent stress, which is then compared to the tensile stress of the material.

RESULTS

As per the analysis images

	Displacement (mm)	Von Mises Stress (N/mm^2)	Von-mises strain
Aluminum alloy 7475	0.004071	13.3622	0.191e-03
Aluminum alloy 6061	0.004154	13.6322	0.199e-03

Thermal analysis results table

	Nodal temperature(k)	Thermal gradient	Thermal flux
Aluminum alloy 7475	558	52.7603	7.20082
Aluminum alloy 6061	558	49.4276	8.5987

Manufacturing component

Casting Process

Consider the casting process at 645 B.C, the first traces of the Sand Molding was found. Now consider the state-of-the-art Electromagnetic casting process. Truly, the Casting process has traversed a long path and impacted human civilization for nearly five millennia. With technological advances, metal casting is playing a greater role in our everyday lives and is more essential than it has ever been.

Selecting the Right Metal Casting Process

- For any Metal Casting Process, selection of right alloy, size, shape, thickness, tolerance, texture, and weight, is very vital.
- Special requirements such as, magnetism, corrosion, stress distribution also influence the choice of the Metal Casting Process.
- Views of the Tooling Designer; Foundry / Machine House needs, customer's exact product requirements, and secondary operations like painting, must be taken care of before selecting the appropriate Metal Casting Process.
- Tool cost.
- Economics of machining versus process costs.
- Adequate protection / packaging, shipping constraints, regulations of the final components, weights and shelf life of protective coatings also play their part in the Metal Casting process.

Comparative Advantages, Disadvantages and Applications for Various Casting Methods:

Sand Casting		
Advantages	**Disadvantages**	**Recommended Application**
Least Expensive in small quantities (less than 100) Ferrous and non - ferrous metals may be cast Possible to cast very large parts. • Least expensive tooling	Dimensional accuracy inferior to other processes, requires larger tolerances Castings usually exceed calculated weight Surface finish of ferrous castings usually exceeds 125 RMS	Use when strength/weight ratio permits Tolerances, surface finish and low machining cost does not warrant a more expensive process
Permanent and Semi-permanent Mold Casting		
Less expensive than Investment or Die Castings Dimensional Tolerances closer than Sand Castings Castings are dense and pressure tight	Only non-ferrous metals may be cast by this process Less competitive with Sand Cast process when three or more sand cores are required Higher tooling cost than Sand Cast	Use when process recommended for parts subjected to hydrostatic pressure Ideal for parts having low profile, no cores and quantities in excess of 300
Plaster Cast		
Smooth "As Cast" finish (25 RMS) Closer dimensional tolerance than Sand Cast • Intricate shapes and fine details including thinner "As Cast" walls are possible • Large parts cost less to cast than by Investment process	More costly than Sand or Permanent Mold-Casting Limited number of sources Requires minimum of 1 deg. draft	Use when parts require smooth "As Cast" surface finish and closer tolerances than possible with Sand or Permanent Mold Processes
Investment Cast		
Close dimensional tolerance Complex shape, fine detail, intricate core sections and thin walls are possible Ferrous and non-ferrous metals may be cast As-Cast" finish (64 - 125 RMS)	Costs are higher than Sand, Permanent Mold or Plaster process Castings	Use when Complexity precludes use of Sand or Permanent Mold Castings The process cost is justified through savings in machining or brazing Weight savings justifies increased cost
Die Casting		
Good dimensional tolerances are possible	Economical only in very large quantities due to	Use when quantity of parts justifies the high tooling

CONCLUSION

In our project we have designed a crankshaft for a multi cylinder engine using theoretical calculations and modeled the crankshaft in parametric software creo. Pressure produced in the engine is also calculated.

Structural and modal analysis is done on the crankshaft to validate our design. Analysis is done for two materials aluminum alloy 6061 and Aluminum alloy7475.

By observing the stress values for both the materials, the analyzed stress values are less than their respective yield stress values. So our design is safe.

By comparing the stress results for both materials, it is less for Aluminum alloy7475 than aluminum alloy 6061.

So for our designed crankshaft, using Aluminum alloy 7475 is best.

REFERENCES

1. Design of machine member's vol.1&2 by R.S Kurmi, S.Chand publications.
2. C.M.Balamurugan.,R.Krishnaraj.,Dr.M.Sakthive.l, .Kanthave.l, DeepanMarudachalam M.G., R.Palani., 2011"Computer Aided Modeling and Optimization of Crankshaft, ''International Journal of Scientific & Engineering Research Volume 2, Issue 8 ISSN 2229-5518
3. Yu Ding and Xiaobo Li.,2011, " Crankshaft Strength Analysis of a Diesel Engine Using Finite Element Method," Asia-Pacific Power and Energy Engineering Conference
4. JianMeng., Yongqi Liu., Ruixiang Liu.,2011,"Finite Element Analysis of 4-Cylinder Diesel Crankshaft, " I.J. Image, Graphics and Signal Processing, 5, 22-29
5. Yu Gongzh.i, Yu Hongliang.,DuanShulin., 2011, "Crankshaft Dynamic Strength Analysis for Marine Diesel Engine," Third International Conference on Measuring Technology and MechatronicsAutomation

Heat Transfer Augmentation in Three Sides Dimple Roughened Solar Duct

Vikash Kumar[1], Meghavath Parsha[2]

[1,2] Assistant Professor, Department of Mechanical Engineering, Malla Reddy College of Engineering, Maisammaguda, Dhulapally, Kompally, Secunderabad-100, Telangana, India

Email: - (vikashism2012@gmail.com, parshamegavath@gmail.com)

Abstract

The present paper deals with the experimental results of heat transfer, friction factor and thermal efficiency of a novel type of three sides concave dimple roughened solar air heater under fully developed turbulent flow conditions. Three sides concave dimple roughened solar air heaters have higher values of heat transfer than those of one side concave dimple roughened solar air heaters in the range of 25-86% for varying relative roughness pitch and 21-81% for varying relative roughness height for the same values of operating parameters. The rise in friction factor of three sides roughened duct over one side roughened duct for varying relative roughness pitch and relative roughness height was found to be respectively as 11-34% and 15-41%. The maximum values of Nusselts number is obtained at relative roughness pitch of 12 and relative roughness height of 0.036. The maximum values of friction factor is obtained at relative roughness pitch of 8 and relative roughness height of 0.045.

Keywords: Solar air heater, Dimple shape, One side roughened duct, Relative roughness pitch, Relative roughness height, Three sides roughened duct.

1. Introduction

Solar air heaters follow a solar thermal technology in which the energy from the sun is captured by an absorbing medium and used to heat air. Solar air heating is a renewable energy heating technology used to heat or condition air for buildings or process heat applications. It is typically the most cost-effective out of all the solar technologies, especially in commercial and industrial applications, and it addresses the largest usage of building energy in heating climates as space heating and industrial process heating [1]. The value of heat transfer coefficient and heat capacity for air is low which reduces the heat transfer rate and thus increases the heat loss to the surroundings. Efforts have been made to improve the thermal efficiency by devising roughness in the form of wires, ribs, dimples baffles, fins, making the surface corrugated and packed bed and many more [2]. The low efficiency of SAH is due to less heat transfer between the collector and beneath flowing air, consequently raising collector's temperature, which leads in greater heat loss from the collector's surface to the nearby surroundings [3-7]. Literature of SAH reveals that the development of laminar sub-layer on heat exchanging surfaces and low heat capacity of air are responsible for less heat transfer resulting in lower thermal efficiency [8-9]. Even though the implication of roughness on heat exchanging surfaces results in an appreciable enhancement in heat transfer from the collector's surface to the under flowing air, the increase in frictional losses needs to be considered precisely. Heat transfer augmentation must be achieved at the cost of minimizing frictional losses. In order to dismantle the viscous sub-layer the core flow mustn't be disturbed. This is the reason the selection of roughness geometry and orientation is given utmost importance [10-14].

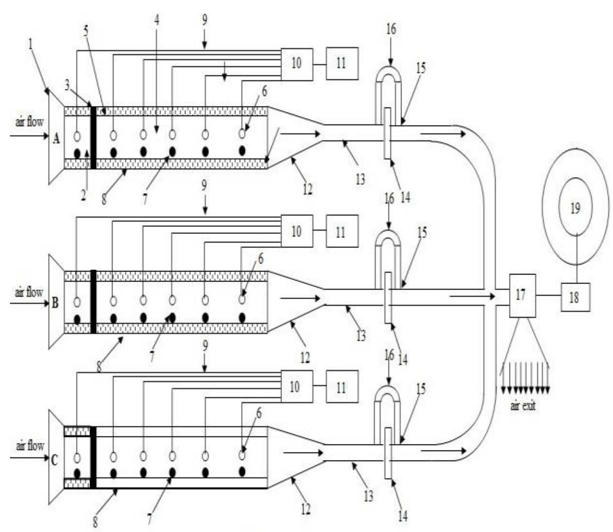
Prasad and Saini [9] provided roughness as transverse ribs of small height and explained the effects of p/e and e/D_h on heat transfer and friction factor. It was found that the maximum augmentation in Nu and f were 2.38 and 4.25 times respectively over the non roughened surface. Gupta et al. [15-16] studied the effect of, p/e, e/D_h, α and Re on the performance of an inclined wire roughened SAH and found that the maximum augmentation in Nu and f were at 60˚ angle of attack. Gupta et al also stated that in comparison to transverse wire, the inclined wire have more heat transfer augmentation due to development of the secondary flow accompanied with the destruction of the viscous sub-layer in the vicinity of the ribs attached to the roughened duct. Momin et al. [17] conducted experiments on V-shaped roughened ducts and discussed its effect on heat transfer and friction factor characteristics under a varying range of roughness geometries and flow parameters as Re between 2500-18000, e/D_h between 0.02-0.034 and constant p/e of 10. The maximum increase in Nusselts number and friction factor was found to be 2.30 and 2.83 times over non-roughened ducts. Varun et al. [18] provided roughness in the form of combination of transverse and inclined ribs and concluded that the geometry having p/e = 8 yielded maximum thermal efficiency.

There was a conclusive information in the literature that most of the roughness provided was limited to a single side (flow facing side) of the absorber plate. If roughness is provided to the side walls (2 nos) as well, they can participate in the heat transfer augmentation process accompanied by the slightest increase in pressure drop resulting in an appreciable enhancement in heat transfer [19-20]. The main objective of the present study is to provide roughness on the three sides (one top and two side walls) of the roughened duct and determine the augmentation in heat transfer and friction factor in compared to the single side roughened duct.

In the present investigation, roughness provided is in the form of concave dimple-shape on three sides and one side roughened duct under identical flow and geometrical conditions. The present paper projects the augmentation in Nusselts number and friction factor for three sides over one side roughened duct as a function of Reynolds number (Re), relative roughness pitch (p/e) and relative roughness height (e/D_h).

2. Experimental set-up and methodology

Experimentation under actual outdoor conditions has been performed on multiple sets of one and three sides roughened ducts containing dimple roughened absorber plates of varying roughness dimensions. The schematic diagram of the test setup is shown in Fig. 1. A 2130 mm x 630 mm wooden board of 25 mm thickness is used to formulate three ducts namely A, B and C. Each duct has a length of 2130 mm in which 630 mm serves as entry section and 1500 mm serves as test section. The entire duct design is based on ASHRAE Standard [21].



1. Trapezoidal shaped air inlet
2. Non-roughened duct section

3. Insulation between entry and test length
4. Insulation
5. Thermocouple
6. Thermometer
7. Glass covers
8. Copper wire
9. Selector switch
10. Digital voltmeter
11. Diverging section
12. Cylindrical pipe
13. Roughened duct section
14. Orifice-plate
15. Flange couplings
16. U-tube manometer
17. Blower
18. Motor
19. Variac

Fig.1. Schematic diagram of the experimental set-up

Once the flow is stabilized and the stagnation condition is achieved, readings for inlet and outlet air temperatures, plate temperatures, pressure drop across the duct and the orifice and solar insolation is recorded. Photograph of the experimental set-up is shown in Fig. 2

Fig.2. Photograph of the experimental set-up

The thermocouples were connected to a digital voltmeter indicating their output. The pressure drop across the orifice was measured with a U-tube manometer and along the duct was measured using multi-tube manometer having least count of 0.001 mm of H_2O. Pyranometer was used to measure the intensity of solar radiation and ambient temperature at different time intervals. The range of operating parameters is shown in Table. 1.

Table 1. Range of operating parameters

S. No.	Name of parameter	Symbolic representation	Range of operating parameter
1.	Reynolds number	Re	2500-12500
2.	Relative roughness pitch	p/e	8-15
3.	Relative roughness height	e/D_h	0.018-0.045
4.	Ambient temperature	T_∞	(24-44) ˚C
5.	Solar insolation	I	(720-960) W/m^2

3. Data Reduction

The recorded data from experimentation for plate and air temperatures under steady state conditions at a varying mass flow rate and heat flux was used to calculate the heat transfer from the absorber plate to the under flowing air. Using the pressure drop across the orifice plate, the prevailing mass flow rate across the duct is:

$$\dot{m} = C_d A_o \left[\frac{2\rho_a \Delta P_o \sin\theta}{1-\beta^4} \right]^{0.5} \qquad (1)$$

Heat lost by the collector = Heat gained by the under flowing air

i.e. $hA_p\left(T_{pm} - T_{fm}\right) = \dot{m}C_p\left(T_o - T_i\right)$ (2)

i.e. $h = \dfrac{\dot{m}C_p\left(T_o - T_i\right)}{A_p\left(T_{pm} - T_{fm}\right)}$ (3)

Nusselts number is determined using heat convective heat transfer co-efficient as:

$$Nu = \frac{hD_h}{K} \tag{5}$$

The friction factor was determined for the test length 1500 mm using Darcy Weisbach equation as:

$$f = \frac{(\Delta P_d)D_h}{2\rho L V^2} \tag{6}$$

The uncertainty prevailing in the measurement of various parameters has been calculated following a simple procedure suggested by Klein and McClintock [22], the values of uncertainty involved in the measurement of Nusselts number and friction factor were found to be $\pm 3.28\%$ and $\pm 4.16\%$ respectively.

4. Validation

The experimental values of Nusselts number and friction factor for one side concave dimple roughened duct have been compared with those of a similar duct model adopted by Saini and Verma [23].

The Nusselts number and friction factor for one side dimple roughened duct is given by the eq. no. 9 and 10.

$$Nu_{1r} = 5.2\times10^{-4}(\mathrm{Re})^{1.27}\left(\frac{p}{e}\right)^{1.15}\left(\frac{e}{D_h}\right)^{0.0333} \times\left[\exp(-2.12)\left(\log\left(\frac{p}{e}\right)\right)^2\right] \times\left[\exp(-1.30)\left(\log\left(\frac{e}{D_h}\right)\right)^2\right] \tag{9}$$

$$f_{1r} = 0.642\,\mathrm{Re}^{-0.423}(p/e)^{-0.465}\times(e/D_h)^{-0.0214} \times\left[\exp(0.054)(\log(p/e))^2\right] \times\left[\exp(0.840)(\log(e/D_h))^2\right] \tag{10}$$

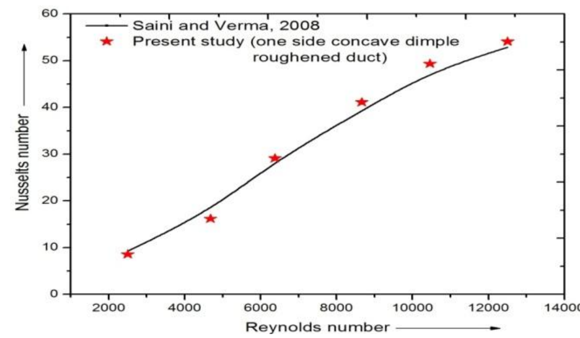


Fig. 7. Comparison of estimated and experimental values of 'Nu' for one side roughened duct

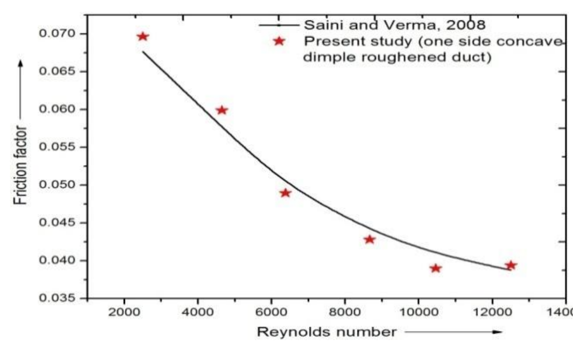


Fig. 8. Comparison of estimated and experimental values of 'f' for three sides roughened duct

The value for Nusselts number and friction factor for one side dimple roughened duct was found to compare well with the values so obtained from the correlations suggested by Saini and Verma. The mean deviation between the estimated and the experimental value was found to be following and the same has been shown in Fig.7 and 8:

- $\pm 3.7\%$ (Nusselts number)
- $\pm 4.5\%$ (Friction factor)

Based on the above comparison, reasonably good agreement between the experimental and estimated values of Nusselts number and friction factor guarantees the exactness of the information being gathered from this test setup.

5. Results and Discussions

The effects of dimple shape roughness element and the variation in the roughness parameter on heat transfer and fluid flow characteristics has been investigated and discussed. The values of the Nusselts number and friction factor for the three sides roughened ducts as a function of Reynolds number have been compared to those of one side roughened duct under identical experimental conditions.

5.1 Nusselts number

The augmentation in Nusselts number achieved as a result of providing artificial roughness in the form of concave dimple shape on the three sides roughened duct over one side roughened duct with an increasing Reynolds number for varying relative roughness pitch (p/e) and relative roughness height (e/D_h) is shown in Fig. 9 and Fig. 10.

Fig. 9 depicts that with an increase in p/e ratio, the Nusselts number increases, but only up to a relative roughness pitch of 12 beyond which it tends to decrease with an increase in relative roughness pitch. The maximum and minimum values of Nusselts number is obtained at the relative roughness pitch value of 12 and 8 respectively at a constant relative roughness height of 0.036 for the range of parameters investigated.

Fig. 10 shows the effect of relative roughness height on Nusselts number. The maximum heat transfer rate is achieved at the relative roughness height of 0.036.

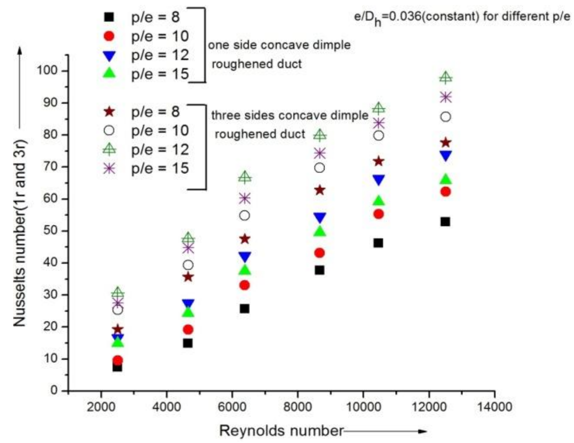


Fig.9. Effect of relative roughness pitch on 'Nu' for one and three sides roughened duct

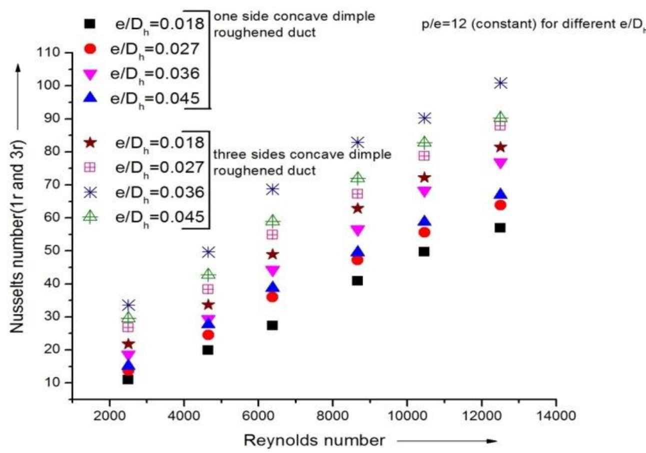


Fig.10. Effect of relative roughness height on 'Nu' for one and three sides roughened duct

5.2 Friction Factor

Artificially roughened SAHs are often characterized by the rise in pressure drop across the roughened duct, which results in an increment of friction co-efficient leading to a higher pumping power requirement. Numerous researchers have worked on different roughness geometry, trying to optimize the geometrical parameter to obtain a minimum rise in friction co-efficient. The effect of relative roughness pitch (p/e) and relative roughness height (e/D_h) on the friction factor with increasing Reynolds number is shown in Fig. 11 and 12 respectively.

Fig. 11 shows the variation in friction factor of three sides over one side roughened duct with Reynolds number with an increasing relative roughness pitch ratio. As the Reynolds number increases, the friction factor decreases monotonously. The three sides roughened duct requires more pumping power than one side roughened duct. For both the roughened ducts, friction factor decreases with increasing relative roughness pitch. The maximum and minimum friction factor for both three sides and one side roughened ducts is obtained at the relative roughness pitch values of 8 and 15 respectively.

Fig. 12 shows the variation in friction factor with Reynolds number with increasing relative roughness height ratio. It can be concluded that as the Reynolds number increases, the friction factor decreases with decreasing relative roughness pitch. Efforts should be made to optimize the geometrical parameter to achieve maximum heat transfer rate at a minimum rise in the co-efficient of friction. The maximum and minimum friction factor for both three sides and one side roughened duct is obtained at relative roughness height ratio of 0.045 and 0.018 respectively.

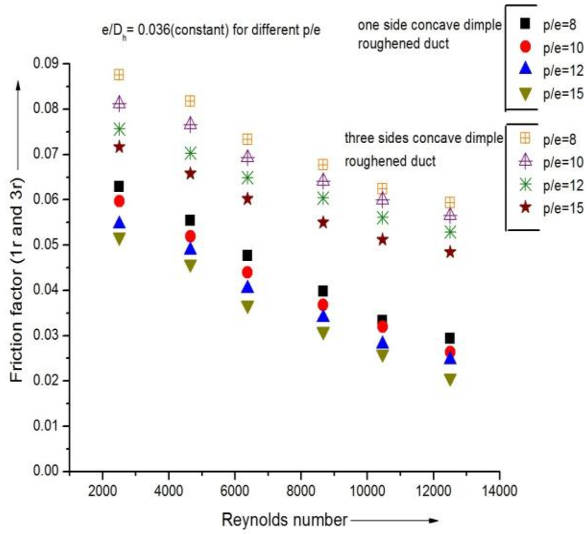


Fig.11. Effect of relative roughness pitch on 'f' for one and three sides roughened duct

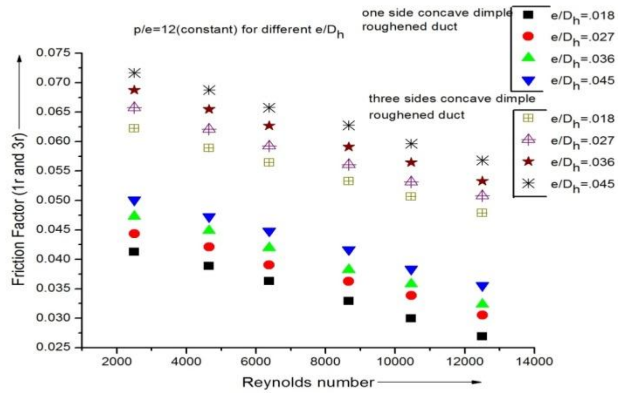


Fig.12. Effect of relative roughness height on 'f' for one and three sides roughened duct

6 Conclusions

Roughness geometry in the form of concave dimple resulted in an appreciable augmentation of heat transfer with an allowable rise in friction characteristics; following conclusions can be drawn:

- The maximum values of Nusselts number were obtained at relative roughness pitch of 12 and relative roughness height of 0.036.
- The augmentation in the Nusselts number of three sides roughened duct over one side roughened duct for varying relative roughness pitch in the range of parameters investigated was found to be 25-86%.
- The enhancement in the Nusselts number of three sides roughened duct over one side roughened duct for varying relative roughness height in the range of parameters investigated was found to be 21-81%.
- The maximum values of friction factor were obtained at relative

roughness pitch of 8 and relative roughness height of 0.045.

- ❖ The augmentation in friction factor of three sides roughened duct over one side roughened duct for varying relative roughness pitch and relative roughness height was found to be respectively as 11-34% and 15-41% in the range of parameters investigated.
- ❖ The average enhancement in the Nusselts number for three sides roughened duct over one side roughened duct for varying relative roughness pitch and relative roughness height was respectively found to be as 59% and 48% in the range of parameters investigated

References

1. Hsieh, J.S., Solar energy engineering. New Jersey: Prentice Hall; 1986.
2. Sharma, S.K., Kalamkar, V.R., Thermo-hydraulic performance analysis of solar air heaters having artificial roughness–A review Renewable and Sustainable Energy Reviews 41 (2015) 413–435.
3. Duffie, J.A., Beckman, W.A., Solar engineering thermal processes, 2nd ed. New York: John Wiley; 1991.
4. Sukhatme, S.P., Solar energy: Principles of thermal collection and storage, 9th Ed. New Delhi: Tata McGraw-Hill; 2003.
5. Dippreyy, D.F., Sabersky, R.H., Heat and momentum transfer in smooth and rough tubes at various Prandtl numbers. Int J Heat Mass Transf 1963; 6:329–53.
6. Webb, R.L., Eckort, E.R.G., Goldstein, K.J., Heat transfer and friction in tubes with repeated rib roughness. Int J Heat Mass Transf 1971; 14:601–17.
7. Firth, R.J., Meyer, L., A comparison of the heat transfer and friction factor performance of four different types of artificially roughened surface. Int J Heat Mass Transf 1983; 26(2):175–83.
8. Nguyen, T.M., Khodadadi, J.M., Vlachos, N.S., Laminar flow and conjugate heat transfer in rib roughened tubes. Numer Heat Transf Part A: Appl: Int J Comput Methodol 1989; 15:165–79.
9. Prasad, B.N., Saini, J.S., Effect of artificial roughness on heat transfer and friction factor in a solar air heater. Sol Energy 1988; 6:555–60.
10. Varun, Saini, R.P., Singal, S.K., A review of roughness geometry used in solar air heaters. Sol Energy 2007; 81:1340–50.
11. Ahn, S.W., The effect of roughness types on friction factor and heat transfer in roughened rectangular duct. Int Commun Heat Mass Transf 2001; 28:933–42.
12. Singh, A.P., Varun, Siddhartha, Effect of artificial roughness on heat transfer and friction characteristics having multiple arc shaped roughness element on the absorber plate. Sol Energy 2014; 105: 479–493.
13. Alam, T. and Kim, M.H., A critical review on artificial roughness provided in rectangular solar air. Renewable and Sustainable Energy Reviews 69 (2017) 387–400.
14. Hans, V.S., Saini, R.P. and. Saini, J.S. Performance of artificially roughened solar air heater-A review, Renewable and Sustainable Energy Reviews, 13, 2009, 1854-1869.
15. Gupta, D., Solanki, S.C., Saini, J.S., Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates. Sol Energy 1993; 51:31–37.
16. Gupta, D., Solanki, S.C., Saini, J.S., Thermohydraulic performance of solar air heaters with roughened absorber plates. Sol Energy 1997; 61:33–42.
17. Momin, A.M.E., Saini, J.S., Solanki, S.C., 2002. Heat transfer and friction in solar air heater duct with v-shaped rib roughness on absorber plate. International Journal of Heat Mass Transfer 45, 3383–3396.

18. Varun, Saini, R.P., Singal, S.K., Investigation of thermal performance of solar air heater having roughness elements as a combination of inclined and transverse ribs on the absorber plate. Renew Energy 2008; 33:1398–405.
19. Prasad, B.N., Behura, A.K. and Prasad, L., Fluid flow and heat transfer analysis for heat transfer enhancement in three sided artificially roughened solar air heater. Sol. Energy 105, 2014, 27–35.
20. Behura, A.K., Prasad, L. and Prasad, B.N., Heat transfer, friction factor and thermal performance of three sides artificially roughened solar air heaters. Solar Energy 130 (2016) 46–59.
21. ASHRAE, Standard, 93–97, 1977. Method of testing to determine the thermal performance of solar air heater. American society for heating, refrigeration and air conditioning engineering, New York, pp. 1–34.
22. Kline, S.J., McClintock FA. Describing uncertainties in single sample experiments. Mechanical Engineering 1953; 75: 3-8.
23. Saini, R.P., Verma, J., Heat transfer and friction factor correlations for a duct having dimple-shape artificial roughness for solar air heaters. Energy 2008; 33:1277–87.

Nomenclature

Symbol	Name of Parameter	Unit
L	Length of the roughened duct	[m]
W	Width of the roughened duct	[m]
H	Height of the roughened duct	[m]
$\dot{m}$	Mass flow rate	[kg/s]
C_p	Specific heat capacity	[J/kgK]
T_o	Air outlet temperature	[˚C]
T_i	Air inlet temperature	[˚C]
T_∞	Ambient air temperature	[˚C]
Q_u	Useful heat gain	[J]
C_d	Coefficient of discharge	
A_o	Area of orifice plate	[m^2]
A_p	Area of absorber plate	[m^2]
h	Convective heat transfer coefficient	[W/m^2K]
Nu	Nusselts number	
f	Friction factor	
T_{pm}	Plate mean temperature	[˚C]
T_{fm}	Fluid mean temperature	[˚C]
d	Pipe diameter	[m]
D	Orifice plate diameter	[m]
D_h	Hydraulic diameter	[m]
I	Solar insolation	[W/m^2]
p/e	Relative roughness pitch	
e/D_h	Relative roughness height	

Greek symbols

β	Ratio of pipe diameter to orifice diameter	
η	Thermal efficiency	[%]
θ	Inclination of U-tube manometer	[˚]
ρ	density	[Kg/m^3]

Demand of Plastic Fuel in India

First S. Afroz,, *M.Tech (Thermal)*, Second S. Ravi Sagar, *M.Tech (Thermal)*

***Abstract*—India Facing a bigger challenge today of recession in Public sector, Indian economy slowing down. The Major sector which is highly affected is FUEL. We all complain about the rising fuel prices, while the government is trying to promote biofuel across the country, many commuters have also started opting for electric mobility, which is costing them half the price of Petrol and Diesel Vehicle. At this point of time we should rethink about Fuel availability from non-biodegrable waste. If the plastic waste converted into energy our country will be able to solve the demand of power and plastic waste management. Plastic remain under the Ground for 500 years, which leads to the contamination of soil and thus pollute the Environment. Over the Years several waste to wealth mechanism have been adopted to recycle and reuse plastic in innovative ways. One such trends has been the conversion of Plastic waste to fuel and making it usable for both domestic and industrial purpose. Daily Generation of over 15,000 tonnes of plastic, the prospect to conversion to fuel are abundant, provided there is enough Infrastructure available. The current Union government also addresses the issues of plastic waste and ways with which the problem could be dealt with.**

***Index Terms*—Waste Management, Fuel, power, energy, wealth mechanism**

I. Introduction

We all are aware about the R's Recycle, Reuse and Regenerate. In the case of plastic which is non-biodegradable and very harmful for environment and major solid waste for municipal and Industries, due to lack of integrated solid waste management, most of the plastic waste neither collected properly nor disposed of in appropriate manner to avoid its negative impact on environment. Worldwide Plastic production is predicted to increase by 3.8% every year until 2030.Flexible non-recycle plastic material are used in an increasing range of application like packaging,3D printing and construction. We need to expand our range of option for keeping this plastic waste out of landfill. One potential approach is "plastic to energy", which unlock

chemical energy stored in waste plastic and uses it to create fuel. Countries like Japan, Germany and united states have already implemented the plastic to fuel conversion process with much success. These three is also been successful in creating business model out of the conversion process, resulting in a conversion model becoming a profitable business one. Though India still has long way to go in terms of adopting plastic to fuel as a business model, discoveries are being made to convert plastic to usable fuel.

II. Work Ahead for Fuel Generation from Plastic

Few Indian and company have taken initiative in this regard for plastic waste management and power generation to fulfil our demand of economic growth and to protect the environment from health hazard.

A. IIP(Indian Institute of Petrolium)

A constituent Laboratory of the council of scientific and Industrial Research (CSIR) in 2014, developed a unique process of converting plastic waste like polyethylene and polypropylene, both together accounting for 60 percent of plastic waste, can be converted to either gasoline or diesel. The technology can convert 1 kg of plastic to 750 ml of automotive grade gasoline. Due to nearly nil presence of Sulphur in the produced fuel, IIP'S plastic converted fuel is pure and meets the EURO-III standard. IIP also stated that a vehicle using this fuel would be able to run at least two Km more per liter. The technology was developed by IIP after nearly a decade of research in hope of commercializing it for Industrial usage.

B. Mechanical Engineer and Professor (Satish Kumar)

Mr. Satish Kumar has come up with a three-step process called plastic pyrolysis to convert plastic into fuel. Today, the fuel is being sold to local industries for as less as Rs. 40 per liter. He has registered the company with ministry of micro, small and medium Enterprises (MSME). Since 2016, he has converted 50 tonnes of unrecyclable plastic into fuel. At present, his company is recycling 200 200kg o plastic and producing 200 liters of petrol out of it every day. "The process helps recycle plastic into diesel, aviation fuel, and petrol. About 500 kg of non-recyclable plastic can produce 400 liters of fuel. It doesn't pollute the air as the process happens in a vacuum." However, using this fuel for vehicles is yet to be tested.

III. METHODOLOGY INVOLVED IN CONVERSION OF PLASTIC INTO FUEL

There are several processes of plastic conversion into fuels. Out of which two are Gasification and pyrolysis.

Gasification involves heating the waste plastic with air or steam, to produce a valuable industrial gas mixture called "synthesis gas", or syngas. This can then be used to produce diesel and petrol or burned directly in boilers to generate electricity.

Pyrolysis is generally defined as the controlled heating of a material in the absence of oxygen. In plastic Pyrolysis, the macromolecular structure of polymer is broken down into smaller molecules and sometimes monomer units. Further degradation of these subsequent molecule depends on a no. of different condition including temperature, residence time, presence of catalyst. Accordingly, the reaction will be thermal and catalytic pyrolysis. Plastic waste is continuously treated in a cylindrical chamber. The plastic is pyrolyzed at 300 C -500 C.

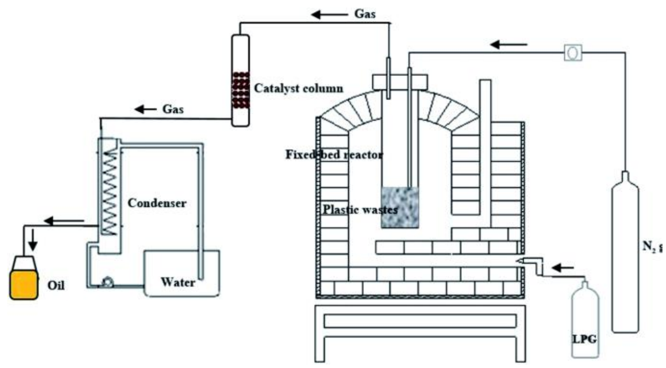


Fig. 1 Schematic diagram of bench scale pyrolysis unit.

A. Pyrolysis Process.

A bench scale fixed-bed pyrolysis stainless steel batch reactor (diameter 35 cm, length 60 cm, maximum capacity 5 kg) was used for production of oil from plastic packaging wastes (Fig. 1). 1 kg of plastic wastes was loaded into the reactor for each pyrolysis reaction. The reactor was heated externally by liquid propane gas to the required operating temperature at heating rates of 10, 15, 20 and 25 °C min−1. Reaction was carried out at 500 °C for 10 minute under a flow of nitrogen purge gas. The pyrolysis gas was passed through catalyst column at a ratio of 0.05, 0.1, 0.15 and 0.2 by weight of catalyst to plastic. The catalyst column made from glass (diameter 5 cm, length 100) and load with catalyst in pellet form. Pellets are formed on compression of the catalyst powder with hydraulic press at pressure of 50 bar, the resulting pellet have a diameter of 1 cm and length of 1.5 cm. No binder was used in the formation of the pellets. Fig. S1 in the ESI† demonstrates the bentonite clay, hydraulic press machine, palletization block and catalyst in pellet form. The resulting gas products were collected *via* water cooled condenser. The oil yield was determined based on the initial mass of plastic waste.

TABLE I

Resin	Conversion Rate
PET	30%
HDPE	Data not Available
PVC	30%
LDPE	70%
PP	50-60%
PS	80-85%

IV. Result and Discussion

The solution of plastics-to-fuel holds promise in not only curbing such pervasive pollution but also providing a significant economic benefit to regions. The American Chemistry Council estimates plastic-to-fuel facilities in the US alone would create nearly 39,000 jobs and almost $9bn in economic output, making the global market potential of such an industry huge.

Plastic-derived fuels are also capable of producing a cleaner burning fuel than traditional sources due to their low Sulphur content, considering the majority of developing nations use Sulphur-heavy diesel. Though slow, progress is being made on the waste to fuel conversion front in India. Unfortunately, despite the big stride in waste management, extensive setting up of waste to fuel plants across the country is still awaited. The technologies employed to convert plastic waste to fuel are not complicated to replicate, and if done so on a large scale, will only help in addressing the growing issue of India's plastic waste. Like other country such as Australia we could create direct job opportunities in plastic conversion plant, and indirect jobs around installation, maintenance and distribution of energy and fuels. We might even see jobs in R&D to explore other waste conversion technologies. This could be a better and upcoming job opportunities for Indians.

PTF technologies can address a critical fraction of the plastic waste stream that has been historically difficult to reutilize, such as LDPE, PP and PS, preventing littering and the landfill disposal of end-of-life plastics. When PTF technologies target end-of-life plastics that are not easily or economically absorbed in recycling markets, they are considered complementary to recycling and existing waste hierarchies. The development of PTF infrastructure can also:

- Create green indirect and direct jobs,
- Divert end-of-life plastics from landfill disposal, extending the lifespan of existing disposal sites and prolonging the siting and construction of new ones,
- Create local demand for low-value plastics that can find their way into streets, streams and the ocean,
- Produce a local source of synthetic crude oil and/or refined fuels to displace fossil fuel derived imports, and
- Reduce air pollution in many parts of the world by substituting low or ultra-low sulfur content fuels for high-sulfur content fuels.

References

[1] Matesh Basappa Khot. (2017, 09). Plastic Waste into Fuel Using Pyrolysis Process. *Journal.* [IRJET]. *volume (04),* 342. Available: www.irjet.net

[2] By Think Change Indiar. (2019,06). Think Change India.

[3] :Muxina Konarova, university of Queensland. (2014,05). If we can't recycle it , why not turn our waste plastic into fuel?

[4] Ocean Recovery Alliances.(2015,06). 2015 Plastic To Fuel Project Developer Guide. Available: www.oceanreco.org

[5] Saptrishi Dutta (2017, 08). Recycling Plastic In India: Converting Plastic Waste To Fuel, The Unrealized Potential.

Development of Automatic Gear Changing Mechanism

P.Nithin[1] E.Prashanth[2]
M.TECH
[1,2]Department of Mechanical Engineering MRCE

Abstract— In the current world of vehicle, gear changing system are manually and automatic. Gear changing system is main in automobile to vary the speed ,Manual gear changing system is less cost than automatic gear shifts system but manually gear shifting system is difficult to understand for people or it take physical effort to change gear. To beat this disadvantage we try to apply touch screen based automatic gear changing system. In automatic gear changing system by touch on touch screen panel gear is shift. By applying this gear changing system it gives cost reduction in compare of automatic gear shift system and flexible or simple then manually gear shift system, and by applying this automatic gear changing system on automobile everyone can drive car easily. The purpose of this research is to reduce physical effort of human being and they can think only in driving and avoid the accidents. In present report we studied literature review baised on this review we define transmission system and gear shifting mechanism to modify a manual gear changing mechanism.

Key words: Actuator, Micro-controller, - 4.3- Touch screen LCD, Four wheeler Engine, Manual gear box, Clutch plate

I. INTRODUCTION

A. Detailed Description Of Problem

Today use of automobile is growing rapidly and to survive in the market is quite difficult with previous technologies. So to survive in the market new technologies are mandatory. The technology must be flexible enough to undertake changes. The technology must continue to grow. During our study on the recent gear shifting mechanism we find out many problems in present system.

In the recent trend there are two existing gear shifting system consists 1) manual gear shifting 2) automatic gear shifting. Here are the problems of existing system are explained below:

***1)* Manual Gear Shifting Mechanism:**

The problems in manual gear shifting mechanism are described below

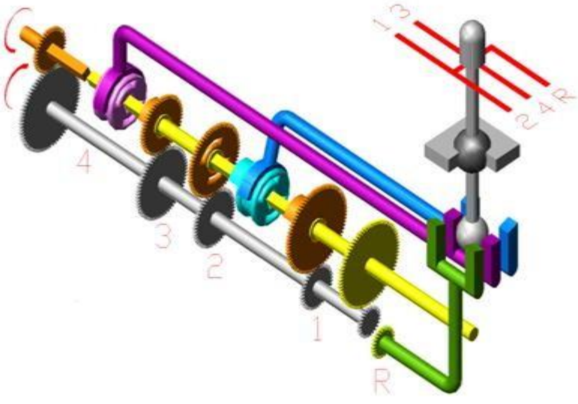


Fig. 1: Manual gear shift technology.

– Required More Human Effort

In manual gear shifting mechanism, the gear can be change with the help lever. To provide motion of lever human effort is required. Without applied human effort gear cannot be change, so sometime it's burdensome for car driver.

– Transmission Error

In manual shifting mechanism. The gear is changed by the driver so that it may be chance to shifting wrong gear instead of true gear. So in manual transmission system the chance of transmission error is high.

– Lack Of Response

We notice that while driving the car hesitates to refuse into gear. Manual transmissions can have the lack of response, so that car cannot run fast as per required in that gear.

– Humming Sound

Manual transmission will emit the sound while changing the gear. This sound is louder due to mechanical operation. The clucking sound is always done when we shift gear within transmission.

– Complex System

The manual transmission system is very complex because it is consist of complex mechanism like gear, lever operation, clutch engage & disengage mechanism.

– Gears Slipping

If the transmission system is immediately slipping in out of gear while driving, then chance of accident is high. So it is very risky. The reason of this problem due to hold the gears.

– Difficult To Understand

In manual transmission system the gear can be changed manually. For change gear we must be change lever operation so that it is difficult to remember &understand.

***2)* Automatic Gear changing Mechanism**

There are following problems which occur in automatic gear shifting mechanism.

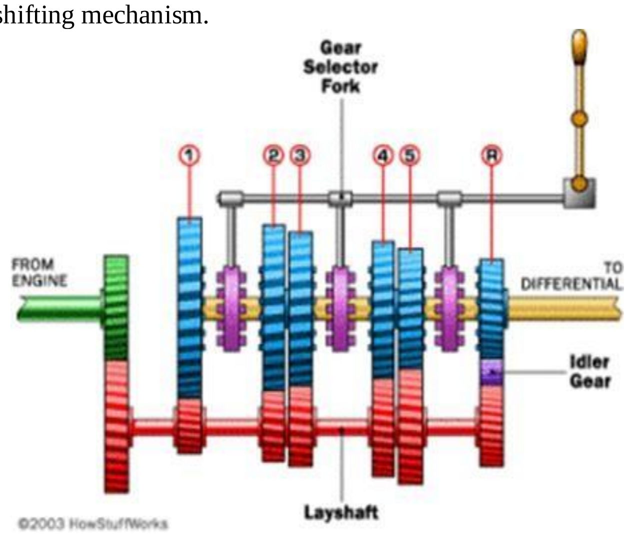


Fig. 2: Automatic gear shift technology.

– High Cost

The automatic transmission system is very costly due to its functioning.

– Less Efficiency

Automatic gear shifting mechanism have a low efficiency compare to manual gear shifting mechanism.

– Gear Slipping

As per mentioned in manual transmission problem gear slipping problem is also occurred in automatic transmission system.

– Grinding Or Shaking

Automatic transmission act a little differently, in automatic transmission system will take some time to wiggle into gear at first instead of typical smooth gear shifting.

– Delayed Engagement

Delayed engagement is one type of slip in this transmission. The clutch & brake do not operate instantly; due to this reason delayed engagement is occurred in automatic transmission system.

The purpose of this research is to reduce people physical effort and they present only in driving and prevent from accidents. In the present world of automobile, gear shifting system are manually and automatic.

Gear shifting system is important in automobile to vary the speed. So automatic gear shifting system is costly than manually gear shifts system and manually gear shifting system is difficult to understand for some people or it take physical effort to change gear.

To overcome this disadvantage we apply touch screen based automatic gear shifting system. In that system by touch on touch screen panel gear is shift. By applying this gear shifting system it gives cost reduction in compare of automatic gear shift system and flexible or simple then manually gear shift system. And by applying this system on automobile it's easier to drive car for everyone. It also reduces the possibility of transmission error of manual gear system. The following are the objective of touch screen based gear shifting mechanism.

1) Low Cost:

This system is less costly compared to automatic gear transmission system but it is costly compare to manual gear transmission system.

2) Flexibility:

This system is flexible compare to the manual transmission system so that it's beneficial to use this system compare to the manual gear shifting mechanism.

3) Easy to operate:

This is semi-automatic shifting mechanism in which touch screen are used so that gear can be shift with the help of pressing number so it is easy to operate.

4) Less human effort:

With the help of touch screen based gear shifting mechanism, no effort is required to change gear. Gear can change just with touch the appropriate number on the screen, so no effort is required

5) High efficiency:

The efficiency of semi-automatic gear shifting mechanism up to 85-90% ,whereas automatic gear shifting mechanism have a efficiency only 75%, so it is very reliable compare to other system.

6) Less transmission error:

Transmission error is occurring due to the improper gear at the certain speed. This is eliminating in this project because we provide indicator for wrong gear at specific speed.

7) Additional Outcomes:

– This project will save time and expenses of user.
– Easy to understand.
– Prevent from accident
– No emit sound while changing gear
– Applicable for all class

II. LITERATURE REVIEW

The literature studied provides so far insight information or new way to carry out this research. An attempt has been made to present the finding research paper on Gear Shift Mechanism. MT Gear Shifting Behavior indicated on manual transmission gear shift mechanism force required to shift gear he got result that max force required to shift gear back to one is 17.88N at shifting rod angle 12.86 °."A Brief Review of Transmission in Automobiles" revealed that the engine provides its highest torque outputs approximately in the middle of its range, while often the greatest torque is required when the vehicle is moving from rest or traveling slowly. "A Brief Review of Transmission in Automobiles" revealed that the engine provides its highest torque outputs approximately in the middle of its range, while often the greatest torque is required when the vehicle is moving from rest or traveling slowly. "Development of Actuator Control Strategy for DC Motor Controlled Automated Manual Transmission (AMT)" developed that Actuator control strategy for automated manual transmission (AMT) which uses electro mechanical Dc motor controlled linear actuators. Develop a strategy for deciding the operation of actuator. "Design Proposals for Low Cost Automated Manual Transmission (AMT)"analyzed that different automated manual transmission (AMT) system for automobile vehicle and developed a low cost design proposal for AMT system. In house designed actuators will require much time to prove the functionality according to our requirement and would demand increase in the development time of the project.) "A Literature Review on Automated Manual Transmission (AMT)" analyzed that transmission based actuator (TBA) uses multi-speed transmissions such that heavy, high-torque motors can be traded for high-speed, reduced mass motor-transmission combinations.) "A Literature Review on Automated Manual Transmission (AMT) analyzed that transmission based actuator (TBA) uses multi-speed transmissions such that heavy, high-torque motors can be traded for high-speed, reduced mass motor-transmission combinations.

III. CONCLUSION

The various research works shows that the system can be with hydraulic actuators or electrical based actuators. Actuators are basically used for gear shift actuations. It has the advantage of lower weight and higher efficiency with respect to other transmission system. Moreover, since AMT is directly derived from manual transmission with the integration of actuators into existing devices, the cost of this system is very less compare to Automatic transmission system.

IV. FUTURE SCOPE

With the help of touch screen people are easily applied gear without more effort. Touch screen can also be used in two wheel vehicle. Shifting of the gear system is more flexible and more reliable. An overall strategy aiming to the improvement of the gear shift quality should take into account the reduction of shifting time, the minimization of mean vehicle deceleration due to traction loss and the minimization of vehicle and driveline oscillations due to variation of transmitted torque. Existing transmission system can be converted into this transmission system. Here, the gear stick is to be replaced by touch screen for the driver to interact with the actuator, so that gear can be applied according to requirement. Future work should basically focus on low cost system with optimized control strategy.

REFERENCES

[1] Tian Shaopeng (2011) a study on "MT Gear Shifting Behavior" CISME Communications in Information Science and Management Engineering ,Vol.1 No.2 2011 PP.32-36 www.jc-isme.org

[2] Ranbir Singh (2012) "A Brief Review of Transmission in Automobiles" IJESRT- international journal of engineering sciences & research technology, Vol.1, issue 2: Page No.D222-D226, July-August (2012).

[3] Chinmay Kirtane (2013) "Gear Shift Schedule Optimization and Drive Line Modeling for Automatic Transmission" 1st International and 16th National Conference on Machines and Mechanisms (iNaCoMM2013), IIT Roorkee, India.

[4] Suraj S Raut (2014) "Automatic Transmission Gearbox with Centrifugal Clutches" IJESRT- international journal of engineering sciences & research technology, Volume: 03 Issue: 10 Oct-2014.

[5] Hiroshi Kuroiwa (2004) study on "Next-generation Fuel-efficient Automated Manual Transmission" Hitachi Review Vol.53 (2004).

EFFECT OF WELDING SPEED AND GROOVE ANGLE ON STRENGTH OF BUTT WELD JOINT USING TIG WELDING

MOHD ASLAM ALI
M Tech (Thermal Engineering)
Malla Reedy college of Engineering,Mysammaguda
(Affiliated to jntuh)
Hyderabad,Telangana

ABSTRACT

Welding is the metal joining process in which two or more metal having same material or different can be joined by heating to a plastic state .It is mostly used for joining metals in process industry, in fabrication, maintenance, repair of parts and structures. The metal plates and pipes used in process industry and they have welding strength as their important parameter.

In this thesis, the welding speed and geometry to find out tensile and impact strength in case of butt weld joint will be done. For V-groove geometry different models of plate with various included angles from 35^0, 45^0, 50^0will be made from structural steel (A633 Grade E).Currently different welding speeds are used in precision welding applications such as nuclear reactor pressure vessels, boilers etc. where welding accuracy as well as quality with strength is an important parameter. So in this project experimentation will be done on different welding speed such as 0.4 cm/sec, 0.8 cm/sec and 1.20cm/sec to prepare a V-groove butt weld joint. Generally the V-groove geometry with included angle up to 60^0 is in use.

Keywords: Welding Speed, Weld Joint, Tig Welding

CHAPTER-1

INTRODUCTION

Welding is, at its center, merely the way of bonding 2 objects of metallic. Whereas there are opportunity approaches in which to affix metallic (riveting, brazing and bonding, as an example), attachment has turn out to be the strategy of selection for its electricity, potency and flexibility.

There are loads of completely special attachment methods, and a number of ar being unreal all of the time. Some methods use warmth to generally melt 2 objects of metal along, commonly including a "filler steel" into the joint to behave as a binding agent. Opportunity methods deem pressure to bind metal along, and still others use a mixture of each heat and stress. No longer like bonding and brazing, anywhere the metal gadgets being joined stay unaltered, the approach of attachment continually adjustments the work items.

This may appear to be a trivial reason, however it is certainly vital to know-how why attachment produces such robust bonds. In the approaches of soldering and brazing, portions of steel arejoined with the aid of introducing a third material (with a decrease melting factor) into the mixture. Melting this 0.33 material among the surfaces of the unique portions binds the portions together. The bond, but, is handiest as robust as the becoming a member of material. Welding, then again, cuts out the intermediary and joins the original portions immediately to each other. The result is a strong, cohesive bond that's regularly as strong as the fabric itself.

As for substances, a few ar plenty of easier to weld than others. Steel may be an fantastic choice because of its power, affordability and weldability. As a rule, the more potent the metallic, the harder it's to weld. Consequently, many metallic alloys had been developed with attachment in thoughts. Of path, actually any metallic will be welded, in addition to forged iron, bronze, steel element and even metallic, although the latter wishes a extremely covered atmosphere as a result of the metal is consequently reactive.

Whatever you are welding, bear in mind: protection first. If you have ever seen welding in man or woman, you could testify to the blinding brightness the system creates. Looking directly at a weld web page without protection can produce what is referred to as arc eye, a painful irritation of the cornea that feels like getting sand on your eye. No surprise that an awesome welder's mask is a prerequisite for any welding outfit.

Welding mask are available many patterns. The handiest ones have a darkened panel that the welder seems thru even as welding. More superior masks automobile-darken as the welding website receives brighter. In addition to mind-blowing brightness, welding can generate temperatures of up 10,000 ranges F (five,538 ranges Celsius) and showers of sparks, making heavy-obligation gloves and a long-sleeve shirt essential.

Lastly, proper ventilation is vital, relying on the welding approach. Welders may be exposed to dangerous substances along with lead, mercury and carbon monoxide. Vent hoods can prevent fumes from collecting within the workspace.

THE PROCESS OF WELDING

Most welding carried out nowadays falls into one in all two classes: arc welding- and torch welding. Arc welding use-s an electrical arc to soften the paintings materials in addition to filler cloth (occasionally referred to as the welding rod) for welding joints. Arc welding includes attaching a grounding cord to the welding material or other steel surface. Another twine referred to as an electrode lead is located at the material to be welded. Once that lead is pulled away from the material, an electric powered arc is generated. It's a little like the sparks you see while pulling jumper cables off a car battery. The arc then melts the work portions together with the filler fabric that allows to sign up for the pieces.

CLASSIFIATION OF WELDING PROCESSES

Welding technique can be labeled into distinct classes depending upon the following

criteria :

(a) It may be classified as fussion welding or strain welding depending upon at the application of heat. If software of warmth isn't always required, it's far referred to as stress welding.
(b) In case of fusion welding it may categorized low temperature welding and high temperature welding. When heat is generated to expand low temperature it's far called low temperature welding like soldering and brazing. Other fusion welding methods are high temperature welding strategies.
(c) Fusion welding can also be categorised on the idea of method of warmth era like gasoline welding, electric powered arc welding, resistance welding, thermit welding, and many others.
(d) On the basis of the kind of joint produced it is able to be categorised as butt welding, seam welding, spot welding, lap joint welding, and so on.

Each of the above sort of welding can be similarly labeled relying on other micro stage traits.

Gas Welding

It is a fusion welding wherein sturdy gas flame is used to generate warmth and lift temperature of metal pieces localized on the place wherein joint is to be made. In this welding metallic pieces to be joined are heated. The metallic for this reason melted starts off evolved flowing along the rims in which joint is to be made. A filler metallic may also be added to the flowing molten steel to top off thehollow space at the edges. The hollow space filed with molten metal is allowed to solidify to get the strong joint. Different combos of gases may be used to obtain a heating flame.

The famous fuel mixtures are oxy-hydrogen combination, oxygen-acetylene, and so forth. Distinct blending proportion of two gases in a mixture can generate exceptional sorts of flames with distinct characteristics.

Oxy-Acetylene Welding

Oxy-acetylene welding can used for welding of wide variety of metals and alloys. Acetylene mixed with oxygen while burnt below a managed surroundings produces large quantity of warmth giving higher temperature upward thrust. This burning additionally produces carbon dioxide which facilitates in stopping oxidation of metals being welded. Highest temperature that may be produced by this welding is 3200oC. The chemical reaction worried in burning of acetylene is

$$2C_2H_2 + 5O_2 = 4CO_2 + 2H_2O + Heat$$

on the basis of supply strain of gases oxy-acetylene welding is labeled as high stress welding on this device both gases oxygen and acetylene furnished to welding quarter are excessive strain from their respective excessive stress cylinders. The different one is low stress welding in which oxygen is furnished from excessive pressure cylinder but acetylene is generated by means of the action of water on calcium carbide and provided at low stress. In this example high strain supply of oxygen pulls acetylene at the welding zone.

An evaluation can be drawn among low strain and excessive stress welding. High strain welding equipment is handy, substances pure acetylene at constant strain, with better control and coffee fees in comparison to low stress welding.

CHAPTER-2

LITERATURE REVEIW

1.Effect of welding geometry parameter on hardness for aisi304 tig.

Welding is an area wherein technological tendencies out match the tendencies in its technology base that's generally driven with the aid of the outstanding industrial demand for welded structure. Reliability, Reproducibility and Viability necessities are forcing Technologists to take a look at weld defects consisting of distortion, warm cracking, in a systematic and logical technique than on experimental basis.

2. A overview paper on impact of welding pace and groove angle on Strength of butt weld joint the usage of tig welding.

Welding is most critical operation in any enterprise. It is crucial to optimize the diverse parameters of welding process in order that we can achieve the reliability, productivity and great of the goods. So industries are forcing the engineers to take a look at the welding manner parameters including electrodes, inert fuel, present day, voltage and so forth. The objective of any industry is manufacturing of excessive quality merchandise at low fee and increase the manufacturing fee. TIG welding system is versatile and normally used operation for joining of materials with the software of warmth and /or strain or fillet material to increase the production with much less time and price.

METHODOLOGY

Objective of the work

In this thesis, materials V-groove geometry distinct models of plate with diverse included angles from 350, 450, 500will be crafted from structural metal (A633 Grade E).Currently special welding speeds are including zero.Four cm/sec, zero.Eight cm/sec and 1.20cm/sec to put together a V-groove butt weld joint.

CHAPTER-3

EXPERIMENTAL PROCEDURE

In this thesis, experiments are made to understand the effect of TIG welding parameters welding speed and groove angle on output parameters such as hardness of welding, tensile strength of welding.For the experiment, welding parameters selected are shown in table.

The welding current and electrodes considered are

PROCESS PARAMETERS	LE VEL1	LEVEL2	LEVEL3
WELDING SPEED (cm/s)	0.4	0.8	1.20
GROOVE ANGLE(^{0})	35	45	50

GROOVE ANGLE(^{0})	WELDING SPEED (cm/s)
35	0.4
35	0.8
35	1.2
45	0.4
45	0.8
45	1.2
50	0.4
50	0.8
50	1.2

CHAPTER-4

INTRODUCTION TO TAGUCHI TECHNIQUE

- Taguchi defines Quality Level of a product because the Total Loss incurred by means of society due to failure of a product to perform as desired when it deviates from the added goal overall performance degrees.
- This consists of fees associated with negative overall performance, operating prices (which adjustments as a product ages) and any added charges due to dangerous facet consequences of the product in use.

Taguchi Methods

- Help businesses to perform the Quality Fix!
- Quality troubles are because of Noises inside

the product or manner machine

- Noise is any unwanted impact that will

increase variability

- Conduct tremendous Problem Analyses
- Employ Inter-disciplinary Teams
- Perform Designed Experimental Analyses
- Evaluate Experiments using ANOVA and Signal-to noise strategies

Defining the Taguchi Approach

- Noise Factors Cause Functional Variation

- They Fall Into Three "Classes"

– 1. Outer Noise – Environmental Conditions

– 2. Inner Noise – Lifetime Deterioration

– three. Between Product Noise – Piece To Piece Variation

- The Point Then Is To Produce Processes Or Products The Are ROBUST AGAINST NOISES

- Don't spend the money to put off all noise, build designs (product and manner) that may perform as desired – low variability – within the presence of noise!

- WE SAY:

ROBUSTNESS = HIGH QUALITY

- TO RELIABLY MEET OUR DESIGN GOALS MEANS: DESIGNING QUALITY IN!

TAGUCHI PARAMETER DESIGN FOR TURNING PROCESS

In order to pick out the system parameters affecting the selected device exceptional characteristics of turning, the subsequent process parameters are selected for the present work: reducing pace (A), feed fee (B) and intensity of reduce (C). The selection of parameters of interest and their tiers is based totally on literature evaluate and a few initial experiments conducted.

Selection of Orthogonal Array

The procedure parameters and their values are given in table. It was also decided to look at the 2 – factor interaction outcomes of technique parameters on the selected traits at the same time as turning. These interactions had been taken into consideration among slicing speed and feed charge (AXB), feed price and depth of reduce (BXC), cutting velocity and depth of cut (AXC).

PROCESS PARAMETERS	LE VEL1	LEVEL2	LEVEL3
WELDING SPEED (cm/s)	0.4	0.8	1.20
GROOVE ANGLE(^{0})	35	45	50

Results: Using randomization technique, specimen was turned and cutting forces were measured with the three – dimensional dynamometer. The experimental data for the cutting forces have been reported in Tables. Feed and radial forces being 'lower the better' type of machining quality characteristics, the S/N ratio for this type of response was and is given below

$$\text{S/N ratio} = -10 \log\left[\frac{1}{n}(y_1^2 + y_2^2 + ... + y_n^2)\right] \quad ... (1)$$

Where $y_1, y_2, ..., y_n$ are the responses of the machining characteristics for each parameter at different levels

Results

Taguchi technique stresses the significance of reading the response variant the usage of the sign–to–noise (S/N) ratio, resulting in minimization of great characteristic variation due to uncontrollable parameter. The slicing pressure is taken into consideration because the quality feature with the idea of "the larger-the-better". The S/N ratio for the larger-the-higher is:

S/N = -10 *log(Σ(Y2)/n))

Where n is the range of measurements in a tribulation/row, in this case, n=1 and y is the measured value in a run/row. The S/N ratio values are calculated by means of taking into account above Eqn. With the help of software program Minitab 17.

5.CONCLUSION

The experiment designed by Taguchi method fulfills the desired objective. Fuzzy interference system has been used to find out the ultimate tensile strength .The all possible values of have been calculated by using MINITAB 17.0 software. Analysis of variance (ANOVA) helps to find out the significance level of the each parameter. The optimum value was predicted using MINITAB-17 software.

The welding parameters are Welding speed, and groove angle for TIG welding of work piece steel. In this work, the optimal parameters of welding speed are 0.4cm/s, 0.8 cm/s & 1.2 cm/s, groove angle 35,45 and 50 degrees. Experimental work is conducted by considering the above parameters. Ultimate tensile strength validated experimentally.

The experimental results confirmed the validity of the used Taguchi method for enhancing the welding performance and optimizing the welding parameters in TIG welding at welding speed 1.2 cm/s , and groove angle 35.

6.REFERENCES

1.Arpaci, V. S., 1995, "Buoyant Turbulent Flow Driven by Internal EnergyGeneration," Int. J. Heat Mass Transfer, Vol. 38, pp.

2.761-2770.Asfia, F. J., and Dhir, V. K., 1994, "An 2.Experimental Study ofNarural Convectionin a Volumetrically Heated Spherical Pool With Rigid Wall," ASME Paper 94-WA/H7--26.

3.Cheung, F. B., i980a, "HeatSource-Driven Thermal Convection at Arbirrary Prandtl Numbers," J. Fluid Mech., YoL 97, pp. 743-758.

4. Cheung, F. B., 1980b, "The Boundry Layer Behavior in Transient Turbulent Thermal Convection Flow," ASME JoURNAL oF HEAr TRANSFERV, ol. 102, pp.373-375.

5.Cheung, F. B., 1978, "Turbulent Thermal Convection in a Horizontal FluidLayer With Time Dependent Volumetric Energy Sources." AIAA/ASME Thermo physics and Heat Transfer Conf.,78-HT-6, Palo Alto.

6. Cheung, F.8., 1977, "Natural Convection in a Volumerrically Heared FluidLayer at High Rayleigh Numbers," Int. J. Heat Mass Transfer, Vol. 20, pp.499-506.

Modern Technology in Solar Energy Generation

Nimmala Mahesh[1],P. Manikya rao [2]
Dr. M.G.R Educational and Research Institute, [1,] Mallareddy college of Engineering[2]

Abstract:**Solar energy is playing a pivotal role in compensating the electrical energy as there is short fall in this engergy due to more demand and decline trends of conventional source of energies exhaustion of fuels like coal, petroleum, natural gases and constant of environmental and climatic changes to cope up this photovoltaic installation is being done in an electrical system to compensate and enhance the energy. an photovoltaic installation in an electrical system is made from the assembly of various photovoltaic units that uses solar energy to produce the electricity in a cheaper way from sun power. Till now the use and scope of solar energy is limited and has not reached upto masses Moreover the efficiency of the system is also low due to which the output is not sufficient as compared to input as in some installed case of solar panel it has been observed that efficiency is not more that 27%. To make it versatile and more useful for the masses newer trends and innovations will help. These have discussed in this paper.**

Keywords:**Solar panels, Efficiency, Namadd, renewable engergy resources, distributed generation**

I. Introduction

Now-a-days distributed generation (DG) is not a new concept. Without this the rectification of energy crises is not possible [1]. In most of the countries the electrical power demand is more than the electrical power generated. On the other hand there is a serious decline in the availability of natural resources, fuels, coal and gases etc. The generation of hydro power plant also varies due to variation in inflow of water from catchment area. When the capacity of hydro power plant decreases the power shortage arises. The solar power plant may be installed in such a fashion that these may work in unison for example when draught is more showering of sun is more. In this way shortage of power will be compensated by the energy governed by the solar power plant. Moreover this installation is to be done in such a way that solar panel will cover the rivers or reservoir reducing the evaporation which enhancing the capacity of dam. On the other hand the solar panel covering this area will generate electrical power which will the enhance the power generation of the system. Moreover by using some newer technologies this power generated may be integrated with the power grid to enhance the capacity of grid.

The development and uses of solar energy at large scale is not only reasonable method of energy resources utilization in the future but also effective frame to improving energy resource crises economically. There is different solar resource in different area, seasons, and weather conditions and so on because of so many influencing factors [2]. As this of kind of renewable energy is available in abundance in nature. The research and application of solar energy will be used to deal with alternative energy [3] – [4]. The advantages of renewable energy sources are enormous as they are free from gas emissions from few conventional energy resources which have impact on the global warming. If this generation of solar energy expedited rigorously can meet the most of the energy demand of the world. Use of Renewable energy will prove as panacea for solving the climatic and environmental problems as every sector of society is keen to solve these problems on the health ground problems. Currently, renewable energy sources install 15 percent to 20 percent of the world's total energy demand [5]. The solar energy is considered as the most promising and important renewable. It is envisaged that solar energy power plants would meet all human needs and would eventually replace the conventional power plants [6].

II. TYPES OF DISTRIBUTED GENERATION

There are different types of distributed generation according to the constructional and technical points of view as shown in Figure (1) [1].

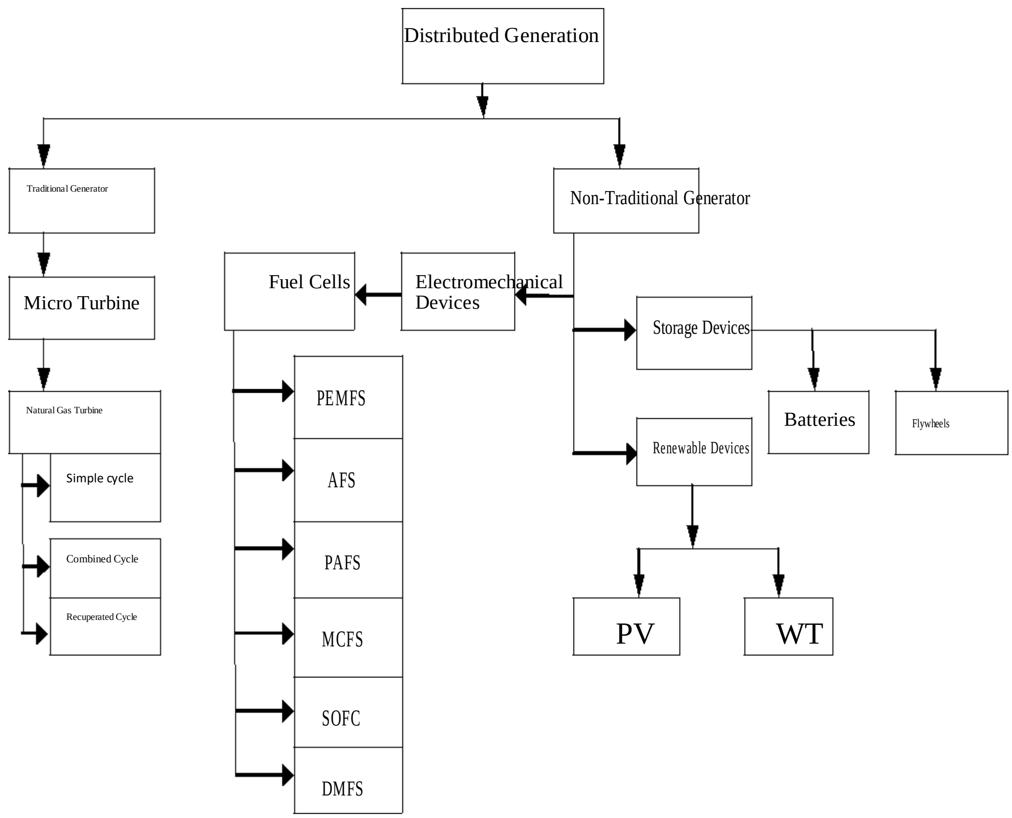


Figure 1 Types of Distributed Generation

III. The Principle Of Energy Storage

The circulation medium was heated by synchronous tracking and non-tracking solar collector and injected into the heat exchanger which was set in concrete pile with the help of circulation pump. After heat exchanging between the heat exchanger and concrete pile, circulation medium was pumped to synchronous tracking solar collector, so formed circulations.And then, the solar energy which was gathered by synchronous tracking and non-tracking solar collector was stored constantly in the underground concrete energy storage pile [7]. The circulating principle is shown in Figure(2). The solar nstal are used to generate steam which drives the steam turbines coupled with alternator as per figure (3).

IV. Solar Energy Policies Of India

The Government of India has increased its focus on developing alternativeresources of energies especially Solar Energy under the policies related to energydevelopment. The solar energy is available in abundance and almost free of cost as it is available from nature. Due to rapid economic expansion India is one of the

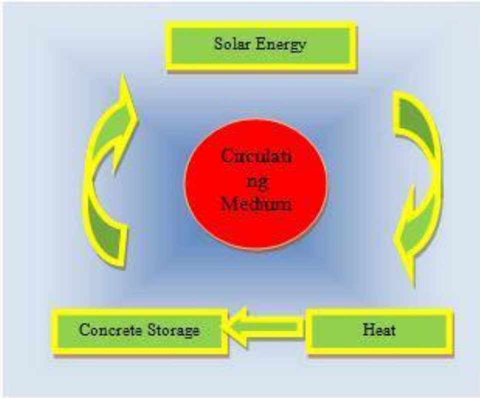


Figure 2 the circulating principle of solar energy

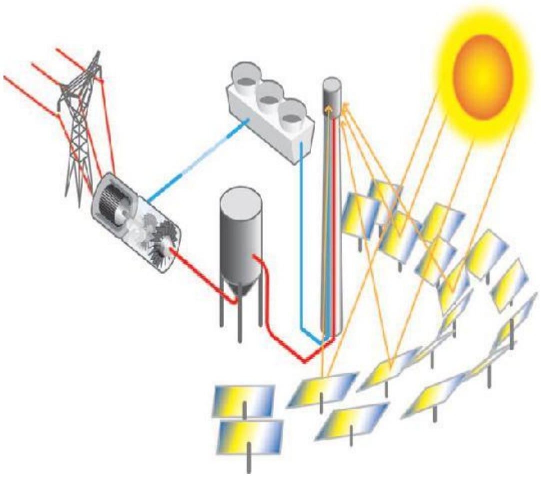


Figure 3 System of Power generation from solar energy

most growing markets and expected to be second largest energy contributor in energy market in the world by 2035. Due to limited domestic fossil fuels reserve, the India has strong planning to expand the renewable energy sources for power sector.

1. To supply the electricity to all the areas included the rural areas as mandated in section 6 of electricity act. Both the Central and State Government will jointly nstalled to achieve this objective at the earliest. Rural Electrification will be done for securing electricity access to the entire household in rural sector. Most of this requirement will be fulfilled by use of renewable energy sources.
2. Reliable rural electrification would be done either through conventional or non conventional methods of electricity whichever is more suitable and economical. Non conventional sources of energy especially Solar can be utilized even where Grid connectivity exists [8].
3. Particular attention is to be given to Dalit Bastis, Tribal areas and other weaker sections of the society the other newer resources.
4. Rural Electricity Corporation of India (REC) is the nodal agency at central govt. Level to implement these programs of electrification in rural areas. The REC will nstal all the goals set up by the National Common Minimum Programme ensuring timely implementation [9].
5. Responsibility of operation and maintenance & cost recovery could be discharged through appropriate arrangement with Panchayats, Local Authorities, BDO, and NGO etc [10].

6. This Great task of Rural Electrification requires cooperative efforts of all agencies like Govt. Of India, State Government and community education cell in rural areas.
7. The Electricity act 2003 has provision of restructuring the electricity industry which unbundled the vertically integrated electricity supply in each state. Now generation, transmission and distribution companies have been formed by the Regulatory Commission of state electricity board. Regulatory Commission will also specify the minimum percentage of electricity that each distribution utility must get from renewable energy sources [11].

V. Newer Methods Which Will Enhance the Use of Solar Energies

Day by day new trends and innovations are being developed throughout the world in R&D centers, automobile sectors and domestic use in institutions, hostels to reduce the energy wastage and to generate the power by solar devices. Many of them are explained given below:
1. Solar cells of higher efficiency have been developed having conversion efficiency more than 37% as compared to the previous solar cells having efficiency of 27 % made of two materials. Tata power is going to install the solar panels having 35% efficiency [12]. In These cells three photo absorption layers are stacked together. This has been developed by stacking Indium, Gallium and arsenide as the bottom layers. These cells have capability of absorbing the light from various wavelengths available in sunlight and convert into electrical energy. Through optimal process the active area has been increased. This breakthrough in technology has been done by new energy and industrial technology development organization.
1. The conversion efficiency of solar panels/plates is increased by newer devices of cleaning these panels. The device makes use of automated "dry-sweep'' to push dust and dirt away from the surface of these devices. In south Arabian language the device is known as nsta which is very rugged and have low maintenance cost. The device is powered by the lithium ion batteries. These batteries are charged by the array itself and have high efficiency. The device has moving parts. It is very interesting that this act like a robotic arm and automated work with scheduling. The device can jump the obstacle between the panels[13].
2. Throughout the world research and development is done to procure more and more energy from various devices and technologies. Under this concept Japan developed a fabric which is known as a solar cell fabric capable of harnessing the energy from sunlight while you are moving by wearing this fabric made cloth. This fabric is made from wafer thin solar cells woven in a stylish way. The electricity generated will be capable to charge the mobile and other portable electronic gadgets. The thread used will be stronger and which increases the life or durability of the fabric cloth. The same idea can be embedded/used in the blind makers and certain type of curtain will also generate power, when sun rayeson these. Various companies developing this type of fabric in association with solar cell maker. This will help the men to recharge these small gadgets while in sun [14].
3. A new trend of solar panel roofs have come in which most of the buildings the roof is covered with solar panel. In some advanced countries like china the roof of the max. Of the homes is made of solar panel by using aaluminum or strong alloy to support the weight of panels. In remote area where grid supply is not viable these panels generate power for themselves and supply electricity to the neighbours also who cannot afford the cost of installation. This will help nearby masses and community in that region where distribution of power is not feasible by other ways & transmission may not be possible due to heavy expenditure.
4. In some countries in urban area some hobbyist of solar energy generate electrical power this system for sufficient for their requirement and surplus generated power is supplied to the grid empowering the national grid. The solar panel should be installed on the vehicles where it is possible so that charging of batteries and other devices in the vehicle may be done with the help of solar energy. Whenever the solar rays fall on the panel this will improve the electrical efficiency of the vehicles. This type of experimentation and uses are already being done in Japan and in other advanced countries where conversion technologies from solar to electrical are being used frequently and sufficiently. In India also solar panels have been installed in metro railway service.
5. As we are aware of that electrical power demand is increasing and viewing the climatic concerns it is desired that renewable energy sources especially solar may be integrated to the utility grid. By using better flexibility in integration through power electronics. Harmonics can be reduced and the reactive power can be balanced.

6. In these days most of the power industries switchover side by side starting manufacturing of the solar inverters of high capacity. These may be utilized to get emergent power if not continues at remote locations where there is no grid supply.
7. CSP systems technology is used for power generation in the system large, flat, sunlight mirrors known as heliostats receive sun light at the top of the tower. A fluid for heat transfer is used to generation the steam which is used for production of electrical power .In some countries the capacity of these plant as high as up to 200 MW. These power tower are very popular in these days because of solar to electrical conversion efficiency is high [15].

VI. Distributed Solar Energy Generation

In [16] the authors has explained the use of Distributed solar photovoltaic (PV) systems is producing electricity onsite, so reducing the requirement to build up new transmission line and also avoiding line losses. Distributed generation also offer significant benefits to the consumers while providing resiliency to an electric grid that is based on the traditional and centralized model. These systems are used in applications ranging from small commercial to residential and for industrial use. Though this market is still primarily driven by government incentives, distributed solar PV will continue its steady march in future. Due to reduced market activity in Italy and Germany, global distributed solar photovoltaic market contracted slightly in 2012, However, growth in the United States, China, Japan, and other countries continued, driven by solar PV module price reductions, the growth of third-party financing models, and feed-in tariffs. Navigant Research forecasts that, from 2013 to 2018, 220 GW of distributed solar PV will be installed worldwide, representing $540.3 billion in revenue.

VII. Conclusion

Due to decline aviability of natural's fuels and viewing environmental changes causes due to conventional method of generation, the use ogf solar energy is becoming popular and urgency of the day. This will create healthy environment for the humain beings which are suffering from the various hazards due to pollution from the nstalled contents. Moreover the power generation due to hydro power plant is not also regular due to irrugalar flow of water from the catchment area. So it is concluded that solar power plant may be nstalled in such a way so these may work in unison with hydro and other methods of generation to enhance the clean and green energy.

References

[1] W El-Khattam and M.M.A Salama, "Distributed generation technologies, definitions and benefits", Electric Power Systems Research, vol. 71, Issue 2, p.p.,119–128, October 2004.
[2] Mingzhi Zhao, Zhizhang Liu and Mingjun Yu, "Data acquisition and analyzing of solar energy resource", Information and Automation (ICIA), Harbin, p.p., 2205-2208, 20-23 June 2010.
[3] Zhao Qingbo, Shan Baoguo, Situation and Outlook of Energy Demand in the World, Energy of China, 2002(02):34-36.
[4] Zehner and Ozzie, "Green Illusions", Lincoln and London: University of Nebraska Press. Pp., 331–42, 2012
[5] G.M. Shafiullah, A.M.T. Oo, D. Jarvis, .B.M.S. Ali, P. Wolfs, "Prospects of Solar Energy in Australia", Electrical and Computer Engineering (ICECE), 2010 International Conference, Dhaka p.p., 350 – 353, 18-20 Dec. 2010.
[6] S. Rao and B.B.Parulekar, "Energy Technology Non-Conventional, Renewable & Conventional", Khanna Publication, 3rd, 2012.
[7] Yan Zhao, Dajun Zhao, Yirong Yao, Xin Fang ,Shousheng Li, "Development of Solar Energy Underground Seasonal Storage Device and Its Parameters Measuring System", Information Engineering (ICIE), 2010 WASE International Conference, Beidaihe, Hebei, p.p., 231 – 234, 14-15 Aug. 2010.
[8] G. D. Kamalapur and R. Y. Udaykumar, "Rural Electrification in the Changing Paradigm of Power Sector Reforms in India", International Journal of Electrical and Computer Engineering, p.p., 147-154, vol. 2, No.2, April 2012.
[9] N. Sasidhar, "Electricity online trading in India", Rural Electricity Corporation of India, 2012.
[10] Giuseppe Buglione, Guido Cervigni, Eileen Fumagalli, Elena Fumagalli and Clara Poletti, "Integrating European Electricity Markets", Center for Research on energy and environmental economics policy, Report 2, Oct. 2009.
[11] MadhuKhanna, KusumMundra, AmanUllah, "Parametric and Semi-Parametric Estimation of the Effect of Firm Attributes on Efficiency: Electricity Generating Industry in India", Journal of International Trade and Economic Development. Pp. 419–436.vol. 8, No. 4, Sept. 2011.
[12] Electronics for You, "Most efficient Solar Cell", Technology news, January, 2013.

[13] GeorgeEitelhube, “Seed Fund Winners - NOMADD: The No-water Mechanical Automated Dusting Device”, King Abdullah University of Science and Technology (KAUST), energy and sustainability are primary research drivers, 2012.

[14] John Badding, “Flexible Silicon Solar-Cell Fabrics May Soon Become Possible”, journal Advanced Materials,Penn State University, 6 December 2012.

[15] DolfGielen, “Concentrating Solar Power technology”, Energy Technology Systems Analysis Programme, International Renewable Energy Agency, January 2013.

[16] G. D. Rai, “Non-Conventional Energy Sources”, Khanna Publication, 4th edition, 2012.

DESIGN AND STUDY OF A MULTI PLATE CLUTCH

B DILEEPKUMAR

M.Tech (cad/cam)

Malla Reddy college of engineering, maisammaguda

(affiliated to Jntuh)

Hyderabad,Telangana.

ABSTRACT:

Clutch is a mechanism which transfers the rotary motion of one shaft to the other shaft when desired. In automobiles friction clutches are widely used in power transmission applications. To transmit maximum torque in friction clutches selection of the friction material is one of the important tasks.

In this thesis, the general introduction to the arrangement, design and some basic concept of multi plate wet type clutch. Fluid plays an important role in this type of clutch so some of their properties are discussed. Some losses due to design parameters are also discussed. To meet the requirements of low fuel consumption, good driving performance and manufacturing feasibility. This paper will provide a design overview of the transmission architecture, main characteristics, key subsystems and control strategies. This gives a better understanding about working principle of clutch, material used for making the clutch plates. Effect of design consideration can be further studied during its application in various conditions.

In this paper, we design a multi plate clutch by using empirical formulas. A 2D drawing is drafted for multi plate clutch from the calculations & a 3D model is created in the modeling Software CREO for Automobile Applications. Analysis done in ANSYS with different materials.

Static analysis to determine the deformation, stress and strain for the single plate clutch. Modal analysis is to determine the deformations with respect to frequencies.

Keywords: 2d 3d design, materials, stress, strain, friction, power transmission.

CHAPTER:1

INTRODUCTION:

The clutch is a mechanical device, which is used to connect or disconnects the source of power from the remaining parts of the power transmission system at the will of operator. The clutch can connect or disconnect the driving shaft and driven shaft. An automotive clutch can permit the engine to run without driving the car. This is desirable when the engine is to be started or stopped, or when the gears to be shifted. Clutch is a mechanism for transmitting rotation, which can be engaged and disengaged. The clutch connects the two shafts so that they can either be locked together and spin at the same speed (engaged), or be decoupled and spin at different speeds (disengaged). Depending on the orientation, speeds, material, torque produced and finally the use of the whole device, different kinds of clutches are used. The clutch in itself is a mechanism, which employs different configurations. The friction clutch is an important component of any automotive machine. It is a link between engine and transmission system which conducts power, in form of torque, from engine to the gear assembly. When vehicle is started from standstill clutch is engaged to transfer torque to the transmission; and when vehicle is in motion clutch is first disengaged of the drive to allow for gear selection and then again engaged smoothly to power the vehicle. Generally there are two types of clutches based on type of contact Positive clutch– Friction clutch– Single plate comes under the category of friction clutch. Desirable properties for friction materials for clutches: The two materials in contact must have a high– coefficient of friction.

INTERLOCKING PARTS CLUTCHES:

This type of clutch has protruding circular edge and a hole for them that engages and disengages during operation. This type is less effective since human foot or hand power on clutching reaches about 10 KN or 1,000 kg.

FRICTION CLUTCHES :

A friction clutch The vast majority of clutches ultimately rely on frictional forces for their operation. The purpose of friction clutches is to connect a moving member to another that is moving at a different speed or stationary, often to synchronize the speeds, and/or to transmit power. Usually, as little slippage (difference in speeds) as possible between the two members is desired.

CHAPTER:2

APPLICATIONS:

MACHINERY:

This type of clutch is used in some lawnmowers, copy machines, and conveyor drives. Other applications include packaging machinery, printing machinery, food processing machinery, and factory automation.

AUTOMOBILES:

When the electromagnetic clutch is used in automobiles, there may be a clutch release switch inside the gear lever. The driver operates the switch by holding the gear lever to change the gear, thus cutting off current to the electromagnet and disengaging the clutch. With this mechanism, there is no need to depress the clutch pedal. Alternatively, the switch may be replaced by a touch sensor or proximity sensor which senses the presence of the hand near the lever and cuts off the current. The advantages of using this type of clutch for automobiles are that complicated linkages are not required to actuate the clutch, and the driver needs to apply a considerably reduced force to operate the clutch. It is a type of semi-automatic transmission.

LOCOMOTIVES :

Electromagnetic clutches have been used on diesel locomotives, e.g. by Hohenzollern Locomotive Works.

CHAPTER:3

OTHER TYPES OF ELECTROMAGNETIC CLUTCHES:

MULTIPLE DISK CLUTCHES:

Introduction – Multiple disk clutches are used to deliver extremely high torque in a relatively small space. These clutches can be used dry or wet (oil bath). Running the clutches in an oil bath also greatly increases the heat dissipation capability, which makes them ideally suited for multiple speed gear boxes and machine tool applications.How it works – Multiple disk clutches operate via an electrical actuation but transmit torque mechanically.

ELECTROMAGNETIC TOOTH CLUTCHES:

Of all the electromagnetic clutches, the tooth clutches provide the greatest amount of torque in the smallest overall size. Because torque is transmitted without any slippage, clutches are ideal for multi stage machines where timing is critical such as multi-stage printing presses. Sometimes, exact timing needs to be kept, so tooth clutches can be made with a single position option which means that they will only engage at a specific degree mark. They can be used in dry or wet (oil bath) applications, so they are very well suited for gearbox type drives.

ELECTROMAGNETIC PARTICLE CLUTCHES:

Introduction Magnetic particle clutches are unique in their design, from other electro-mechanical clutches because of the wide operating torque range available. Like a standard, single face clutch, torque to voltage is almost linear. However, in a magnetic particle clutch torque can be controlled very accurately. This makes these units ideally suited for tension control applications, such as wire winding, foil, film, and tape tension control. Because of their fast response, they can also be used in high cycle applications, such as card readers, sorting machines, and labeling equipment.

AUTOMOBILE POWERTRAIN:

This plastic pilot shaft guide tool is used to align the clutch disk as the spring-loaded pressure plate is installed. The transmission's drive splines and pilot shaft have a complementary shape. A number of such devices fit various makes and models of drivetrains.

MOTORCYCLES:

A BASKET CLUTCH:

Motorcycles typically employ a wet clutch with the clutch riding in the same oil as the transmission. These clutches are usually made up of a stack of alternating plain steel and friction plates. Some plates have lugs on their inner diameters that lock them to the engine crankshaft. Other plates have lugs on their outer diameters that lock them to a basket that turns the transmission input shaft. A set of coil springs or a diaphragm spring plate force the plates together when the clutch is engaged.

On motorcycles the clutch is operated by a hand lever on the left handlebar. No pressure on the lever means that the clutch plates are engaged (driving), while pulling the lever back towards the rider disengages the clutch plates through cable or hydraulic actuation, allowing the rider to shift gears or coast. Racing motorcycles often use slipper clutches to eliminate the effects of engine braking, which, being applied only to the rear wheel, can cause instability.

CHAPTER:4

OTHER CLUTCHES AND APPLICATIONS:

Belt clutch: Used on agricultural equipment, lawn mowers, tillers, and snow blowers. Engine power is transmitted via a set of belts that are slack when the engine is idling, but an idler pulley can tighten the belts to increase friction between the belts and pulleys.

Dog clutch: Utilized in automobile manual transmissions mentioned above. Positive engagement, non-slip. Typically used where slipping is not acceptable and space is limited. Partial engagement under any significant load can be destructive.

Hydraulic clutch: The driving and driven members are not in physical contact; coupling is hydrodynamic.

SPECIALTY CLUTCHES AND APPLICATIONS:

SINGLE-REVOLUTION CLUTCH:

Single-revolution clutches were developed in the 19th century to power machinery such as shears or presses where a single pull of the operating lever or (later) press of a button would trip the mechanism, engaging the clutch between the power source and the machine's crankshaft for exactly one revolution before disengaging the clutch. When the clutch is disengaged and the driven member is stationary. Early designs were typically dog clutches with a cam on the driven member used to disengage the dogs at the appropriate point.

CASCADED-PAWL SINGLE-REVOLUTION CLUTCHES:

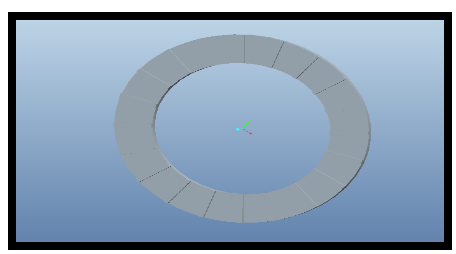
Cascaded-pawl single-revolution clutch driving the cam cluster in a Teletype Model 33 that performs fully mechanical conversion of incoming asynchronous serial data to parallel form. The clutch drum, lower left, has been removed to expose the pawls and trip projections.

These superseded wrap-spring single-revolution clutches in page printers, such as teleprinters, including the Teletype Model 28 and its successors, using the same design principles.

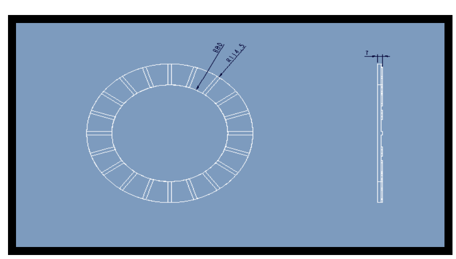
KICKBACK CLUTCH-BRAKES:

These mechanisms were found in some types of synchronous-motor-driven electric clocks. Many different types of synchronous clock motors were used, including the pre-World War II Hammond manual-start clocks. Some types of self-starting synchronous motors always started when power was applied, but in detail, their behaviour was chaotic and they were equally likely to start rotating in the wrong direction. Coupled to the rotor by one (or possibly two) stages of reduction gearing was a wrap-spring clutch-brake. The spring did not rotate. One end was fixed; the other was free. It rode freely but closely on the rotating member, part of the clock's gear train. The clutch-brake locked up when rotated backwards, but also had some spring action. The inertia of the rotor going backwards engaged the clutch and wound the spring. As it unwound, it restarted the motor in the correct direction. Some designs had no explicit spring as such—but were simply compliant mechanisms. The mechanism was lubricated and wear did not present a problem.

3D MODEL:



2D MODEL:



STATIC ANALYSIS OF SINGLE PLATE CLUTCH:

Materials used

Steel:

Young's modulus = 205000mpa

Poisson's ratio = 0.3

Density = 7850kg/mm^3

Cast iron:

Young's modulus = 110000 mpa
Poisson's ratio = 0.28

Density = 7200

Copper :

Young's modulus = 101000mpa

Poisson's ratio = 0.32

Density = 6800

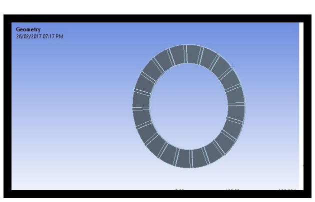
Save Pro-E Model as .iges format

→→Ansys → Workbench→ Select analysis system → static structural → double click

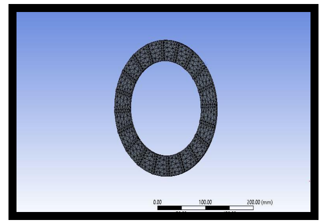
→→Select geometry → right click → import geometry → select browse →open part → ok

→→ Select mesh on work bench → right click →edit

Double click on geometry → select MSBR → edit material →



Select mesh on left side part tree → right click → generate mesh →



Select static structural right click → insert → select rotational velocity and fixed support →

5 . RESULTS TABLES:

STATIC ANALYSIS RESULTS

MATERIAL	DEFORMATION(mm)	STRESS (N/MM^2)	STRAIN
STEEL	1.0788E-5	0.3023	1.6012E-6
CAST IRON	1.92E-5	0.31251	2.84E-6
COPPER	1.54E-5	0.32894	2.452E-6

FATIGUE ANALYSIS RESULTS

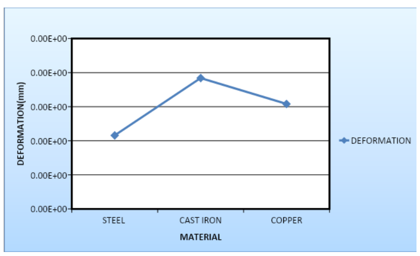
MATERIAL	LIFE	DAMAGE	SAFETY FACTOR
STEEL	1E10	0.1	4.3094
CAST IRON	1E10	0.1	4.4159
COPPER	1E10	0.1	4.1953

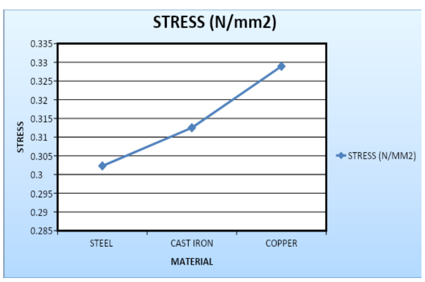
MODAL ANALYSIS RESULTS

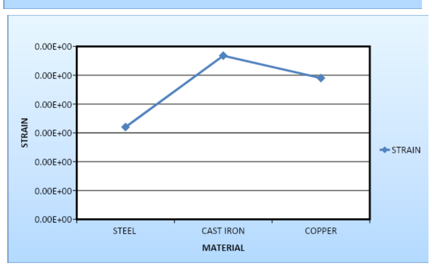
MATERIAL	frequency	deformation 1	frequency	Deformation2	frequency	Deformation 3
STEEL	1.3133E+05	589.73	1.314E+05	655.2	1.31E	626.29

					+ 05	
CAST IRON	1.0233E+005	616.95	1.023E+05	691.08	1.024E+05	659.93
COPPER	93596	569.86	93665	623.97	93696	597.78

GRAPHS







6 . CONCLUSION:

Static Structural, modal and fatigue analysis is done for multi plate clutch using the properties of the three materials. Outer material is steel and Materials used for friction plate are varied Cast iron, Cork and Asbestos. Comparison is done for above materials to validate better friction material for multi plate clutch. By observing static analysis results, the stress values for all materials are less than that of their respective yield stress values. The deformation and stress values are less for Cast Iron and Asbestos is used. By observing modal analysis results, the deformation is less for Cast Iron but the frequencies are less when Cork is used. Since the frequencies are lesser, the vibrations in the clutch will be reduced when cork is used. By observing fatigue analysis results, the life is more for Cast Iron but the damage is more for Cork and Asbestos. The clutch will be failed if the applied load is multiplied with the damage value. Since the damage value is more for Cork and Asbestos, the clutch when both the materials are used will fail at very larger loads. So it can be concluded that using Cork for friction plate is better.

6 . REFERENCES:

[1] Mr.P.V.Narasimha Rao.L, Ch.Chandrarao ―Static and Dynamic Analysis of Clutch Plate With Crack‖, IJRMET, Volume 4, Issue Spl - 1, Nov 2013

[2] Mr.Ganesh Raut, Anil Manjare, P Bhaskar ―Analysis of Multidisc Clutch Using FEA‖ IJETT, Volume 6, Dec 2013

[3] Prof.P.Baskar, Anil Jadhav, Gauri Salvi, Santosh Ukamnal, ―Static Structural Analysis of Multi plate Clutch with Different Friction Materials‖ IJERT, Volume 2, Issue 11, November 2013

[4] Dr. J. P. Modak, Shrikant V.Bhoyar, Dr. G. D. Mehta, ―Dynamic Analysis Of Single Plate Friction Clutch‖, IJERT, Vol. 2, Issue 7, July – 2013

[5] Mr.Prashil M. Mhaiskar, Nitin D. Bhusale, Mayur D. Pastapure ―Vibration Analysis of Dry Friction Clutch Disc by Using Finite Element Method‖, IJERT, Volume 3, Issue 1, January – 2014

[6] Mr.Rajesh Purohit, Pooja Khitoliya and Dinesh Kumar Koli, ―Design and Finite Element Analysis of an Automotive Clutch Assembly‖, ScienceDirect, Procedia Materials Science 6, 2014

[7] Mr.Monarch K. Warambhe, Gautam R. Jodh, Mamta G. Pawar, ―Design and Analysis of Clutch Using Sintered Iron as a Friction Material‖ ,IJITEE, Volume-3, Issue-7, December 2013

[8] Mr. P.Naga Karna, Tippa Bhimasankara Rao, ―Analysis of Friction Clutch plate using FEA‖ ,IJERD, Volume 6, Issue 2, March 2013

[9] Mr. B. Nivas, M. Nithiyanandam, S. Tharaknath, A. AvinashKumar, ―Design and Analysis of Clutch Plate Using Steel Material [En –Gjs-400 -15steel]‖, IOSR-JDMS, Volume 13, Issue 5, May. 2014

Design Modification of Heat Exchanger Using CFD Tools

[1]Prakash D Chavan, [2]Kandy Sandya

[1]*Assistant Professor, Mechanical Department, Malla Reddy college of engineering, Maisammaguda, secunderabad, India.*

[2]*Assistant Professor, Mechanical Department, Malla Reddy college of engineering, Maisammaguda, secunderabad, India.*

.

Abstract

Heat exchangers are used in the thermal system to maintain the temperature of the working liquid. Among the various types of heat exchanger, the shell and tube heat exchanger is the most commonly used heat exchanger based on its simple design and performance aspects. Even though these shell and tube heat exchangers operates at its designed point, it can be even effectively designed to achieve better heat transfer rate.

Since most of the shell and tube heat exchangers are designed based on the traditional design concepts, in this paper, we have planned to modify the design of the heat exchanger. For this purpose, we have selected a reference heat exchanger with its practical performance results. Using the design data from the existing heat exchanger, the CAD model was generated using solid works software and it was analysed using the CFD software under the actual operating conditions.

Then, the design modifications was carried out in the inner tubes and baffles accordingly. Initially we have changed the design of baffles and based on the analysis results, the cross section of the tube was modified. These modified design was analysed using the CFD tools under the same operating conditions and the results was compared with the actual and modified designs.

By this, we will be justifying the application of CFD tools in the design of heat exchangers to predict its performance in the early design stage. Due to this, the time and money invested on the man and equipments will be reduced to the industry for developing an efficient heat exchanger.

Keywords: Heat exchangers – CFD tools – performance analysis – design optimization.

INTRODUCTION

Heat exchangers are widely used in the thermal systems where there are requirement of maintaining the system temperature in order to get the quality product. The quality of the final product is directly associated with the efficiency and effectiveness of the heat exchanger. The efficiency of the heat exchanger is based on the quantitative values of the inlet and exit temperature difference.

The efficiency of this type of heat exchanger is determined by the amount of heat transferred from hot to cold mediums and in vice versa. Also in some peculiar cases, for the customized purposes, the flow inside the heat exchangers are changes from counter flow to parallel flow. This is purely based on the application where it is been used. In most of the cases, the type of fluid flowing in the shell and in the tubes are also changed to get better efficiency.

Based on the above, this project is based on the performance study of a shell and tube heat exchanger using a CFD tool. This study is carried out with respect to the available practical data. The practical data are taken from the reference paper and the design of the heat exchanger is also taken from the reference paper. The reference paper contains the practical experimental data. This study is to justify the application of CFD tools in the design of the heat exchanger system. The analysis will be carried under varied conditions as per the reference paper.

The first phase of this project deals with the study and analysis of the heat exchangers along with the selection of reference paper continued by the modelling of the heat exchanger system. The second phase of this project will be contained with the CFD analysis of the system using the solid works flow simulation tools. Then the experimental and CFD results will be compared to justify the effective application of the CFD tools in thermal system design.

METHODOLOGY

Methodology is the basic requirement for a project, because it defines the proper start and end conditions of the works to be done. Proper planning and execution of the workflow decides the successful completion of the project. The methodology of this project is as follows.

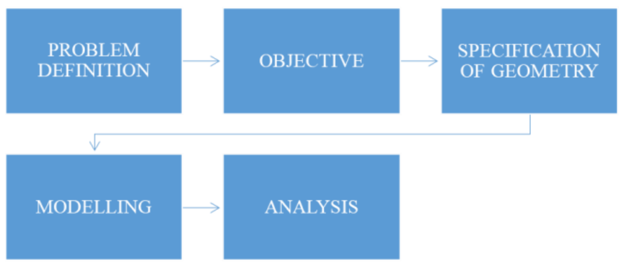


Figure 1: The strategic planning process

Problem Definition

The problem is defined as to model a heat exchanger as per the physical dimensions available in the industry and to conduct the CFD analysis for the existing operating conditions of 55 degree Celsius and 25 degree Celsius with hot water in shell and cold water in tube. This analysis have to be done under parallel flow conditions and verify the CFD results are same as actual experimental results with general limitations.

Also the analysis should also be conducted for various operating conditions like parallel flow and counter flows and varied baffle geometric conditions.

Objective

The objective of this project is to justify the applications of CFD tools in the thermal system design. Also the baffle conditions have to be optimized accordingly to find the best operating conditions.

Geometric Specifications

The geometry specifications are taken from the design data available in the reference journals [1]. The following are the details of the heat exchanger used for analysis.

Figure 2: Real time experimental setup

Shell Details

Outer diameter = 142mm

Inner diameter = 136mm

Length of the HE = 1500mm

No. of baffles = 5

Distance between baffles = 300mm

Baffle opening = 25% (except first and last)

Tube Details

Outer diameter = 23mm

Inner diameter = 20mm

Length = 1200mm

No. of tubes = 9

Material of construction

Shell = Stainless Steel

Tubes = Copper

Baffles = Copper

Modelling

The following are the CAD images of the Heat exchanger modelled using Solid Works and drafter using Auto CAD.

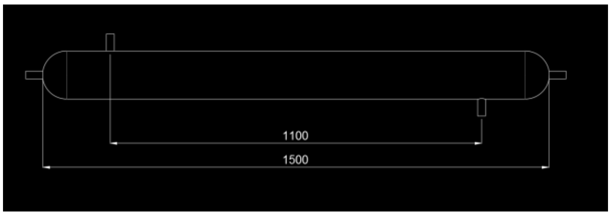


Figure 3: Dimensions of Shell and openings

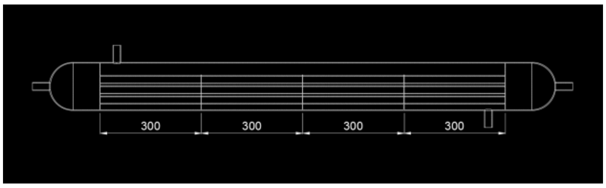


Figure 4: Dimensions of baffle arrangements

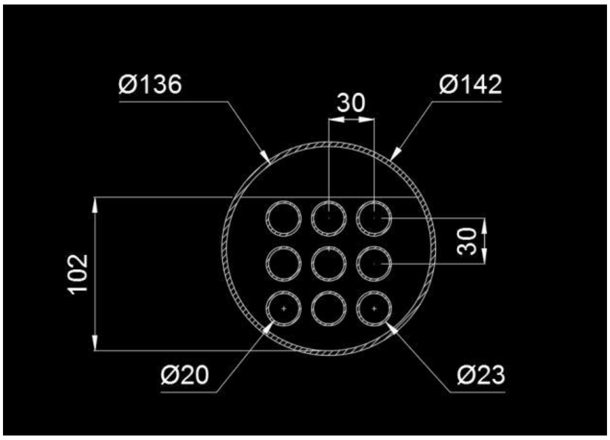


Figure 5: 8 Dimensions of shell, tube and baffle

Figure 6: Front view of the Heat Exchanger

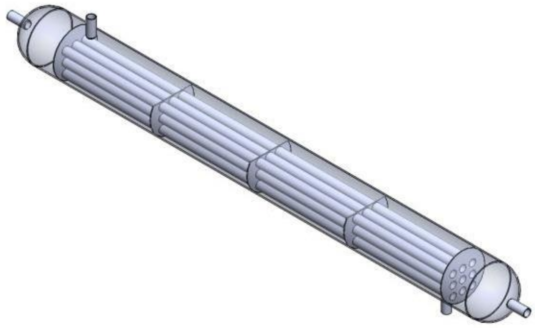


Figure 7: View of tubes and baffles

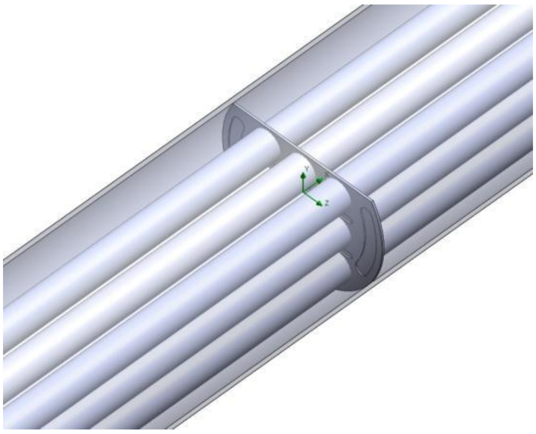


Figure 8: Slots in baffles (slots are of half the thickness)

Analysis

The performance analysis for the heat exchanger was done using Solid Works Fluid Flow Simulation. The analysis was conducted for the following operating conditions.

Case 1: Hot water in Shell and Cold water in Tube – Parallel flow (actual condition)

Case 2: Hot water in Shell and Cold water in Tube – Counter flow

Case 3: Hot water in Tube and Cold water in Shell – Parallel flow

Case 4: Hot water in Tube and Cold water in Shell – Counter flow

Among these analysis, the case 1 is the actual condition in which the system is been operated and tested in the industry. The following are the results of the experimental test.

Table 1: Ranked problems of the Russian road transport enterprises and options for solution of these problems

Dated- 11th & 12th of April 2014 (12:00 to 4:30 PM), ambient temperature: 33 (°C)

Serial number	Hot water inlet (°C)	Hot water outlet (°C)	Cold water inlet (°C)	Cold water outlet (°C)	Degree of cooling (°C)	Degree of heating (°C)	Effectiveness of heat exchanger
1.	55.3	47.5	25.5	31.1	7.8	5.6	0.26174
2.	55.4	47.9	24.2	30.3	7.5	6.1	0.24038
3.	55.0	47.5	25.0	30.0	7.5	5.0	0.25
4.	55.3	47.5	25.5	31.1	7.8	5.6	0.26174
5.	55.4	47.9	24.2	30.3	7.5	6.1	0.24038
6.	55.7	48.1	24.7	30.0	7.6	5.3	0.24516
7.	55.3	48.3	25.2	29.3	7.0	4.1	0.23255

Form the above set, the set 3 is taken for CFD analysis and for comparison purpose. The above set is for case 1 analysis. The analysis results of all the four cases will be compared and the best operating conditions will be selected for the further development activities. The following are the proposed development models.

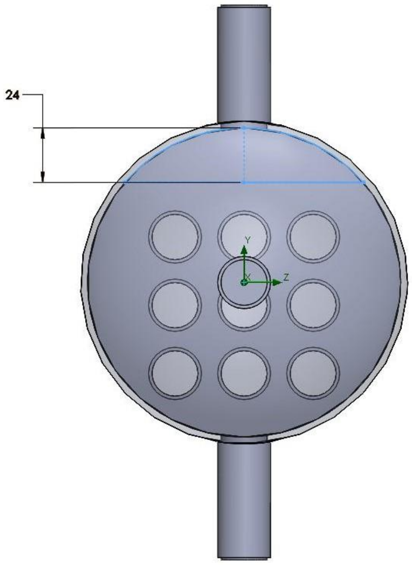


Figure 9: Baffle gap reduced from 34 mm to 24mm

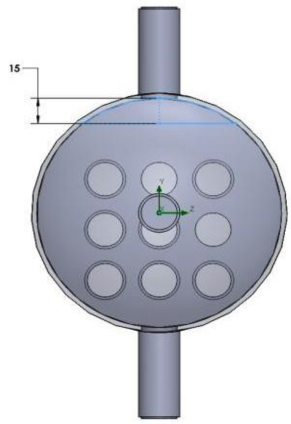


Figure 10: Baffle gap reduced from 34 mm to 15mm

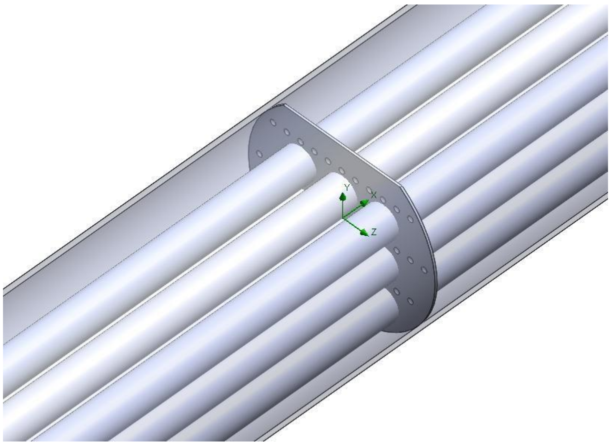


Figure 11: Baffle gap 15mm along with spherical dimples on baffles

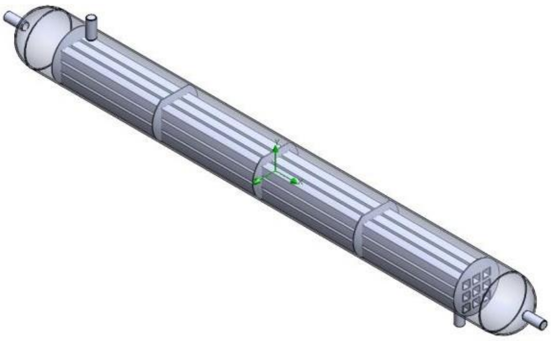


Figure 12: Proposed change in tube shape _ Rectangular shape tubes

This rectangular shaped tube design will be implemented in the design which is having better performance among those which have been mentioned earlier.

The CFD analysis was conducted under the following 3 steps.

- Pre-processing
- Solution
- Post-processing

The model importing and cleaning, meshing, boundary conditions and material property assigning are all done at the pre-processing stage. The solver settings and output settings and simulations are carried at the solution stage. The extraction of results from the saved database in the form of contour plots and tabulated values are done in the post-processing and this post-processing will be explained in the upcoming chapter.

The boundary conditions used for the analysis are as follows.

1. Hot liquid inlet = 55 0C
2. Cold liquid inlet = 25 0C
3. Mass flow rate (Hot) = 0.027 Kg/sec
4. Mass flow rate (Cold) = 0.014 Kg/sec
5. Ambient temperature = 30 0C
6. Hot and cold liquid = Water

Assumptions made are:

1. Flow is laminar and turbulent
2. U = 750 W/m2K (based on data available in net)

RESULTS AND DISCUSSION

The following are the typical CFD contour plot outputs. These outputs are retrieved using the post processing tools. In total 9 cases have been analyzed and its quantitative results are displayed as tabular columns and histogram charts following the below schematic CFD plots.

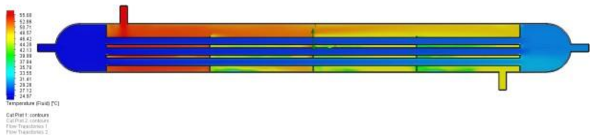


Figure 13: Interior fluid temperature (front plane cut section)

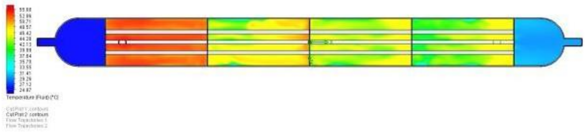


Figure 14: Interior fluid temperature (top plane cut section)

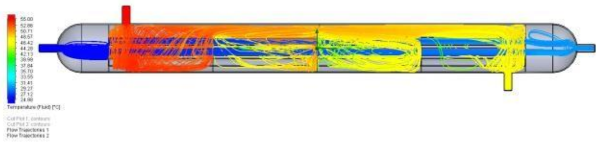


Figure 15: Interior fluid flow pattern (flow trajectories)

Table 2: Results of Case 1 to Case 4 (actual model with varied operating conditions)

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 1	46.845	29.393
CASE 2	46.689	29.594
CASE 3	48.522	28.913
CASE 4	48.473	29.201

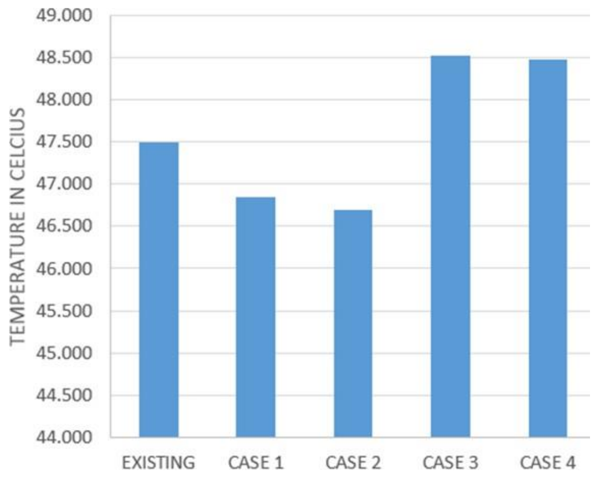


Figure 16: Hot water outlet temperatures – case 1-4

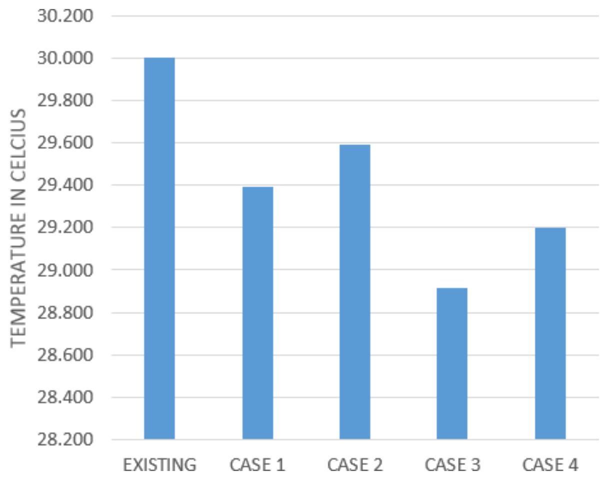


Figure 17: Cold water outlet temperatures case 1-4

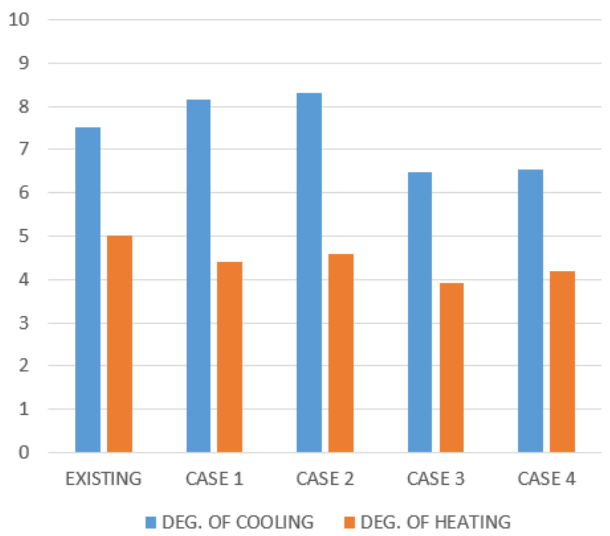


Figure 18: Degrees of cooling and heating – case 1-4

Table 3: Results of Case 5 to Case 8 (geometry modifications with case 2 condition)

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 5	46.838	29.600
CASE 6	46.414	29.675
CASE 7	45.913	29.869
CASE 8	45.745	29.885

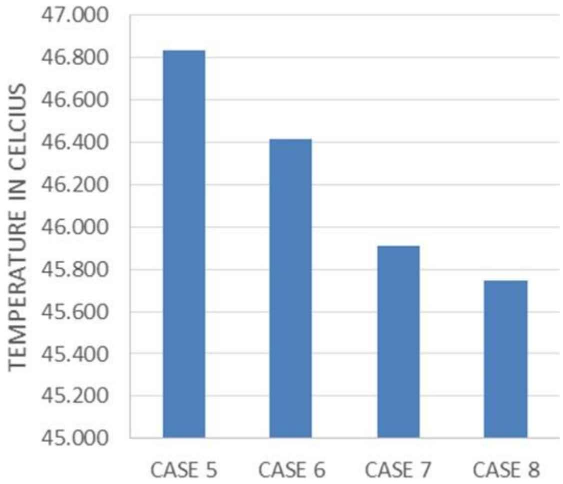


Figure 19: Hot water outlet temperatures – case 5-8

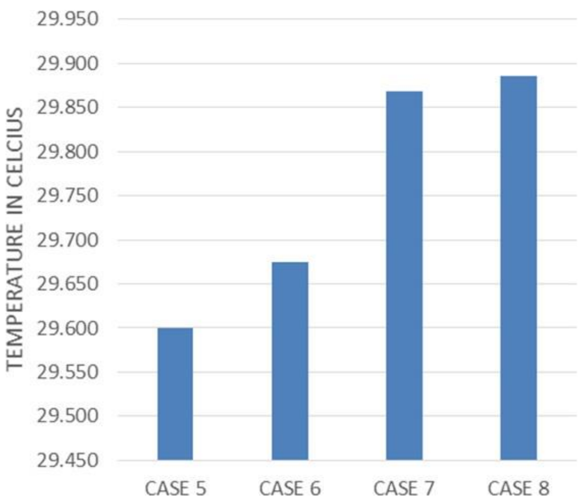


Figure 20: Cold water outlet temperatures case 5-8

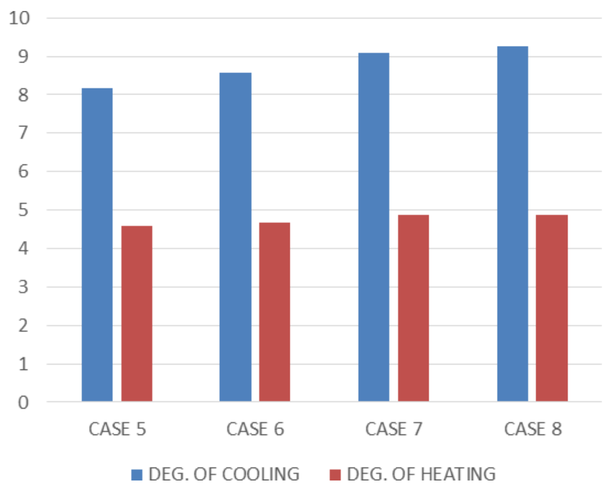


Figure 21: Degrees of cooling and heating – case 5-8

From the actual experimental data, the third serial data was taken for analysis and the obtained results are 47.5 and 30.0o C, whereas for the same data, the CFD results obtained are, 46.845 and 29.393o C (case 1). This proves that, the CFD results are comparatively matching with the experimental results. Hence on further analysis, it was found that the case 2 operating condition was having better results than the other three cases.

Using the case 2 operating conditions, the geometry of the heat exchanger was modified and analysed. On observation it was found that, the geometry with case 2 and slots is having slight better performance. On further improvement activity, the baffle gap was reduced and analysed. This shows that the performance increases as the volume of water inside the shell increases, but not beyond a limit as it affects the discharge issues. So the gap of 15mm was maintained and for this condition, the baffles are analysed for spherical dimples instead of slots. This condition results proves that the output is much better than the other operating conditions. The spherical dimples are used as an innovative idea.

Based on the above results, it was found that the modification of baffles with spherical dimples are having better performance than all other cases. In addition to this, an attempt was made to check the performance by changing the tube's cross section to rectangular from circular. The same was done and its performance results are plotted below.

Table 4: Results of Case 8 and Case 9

CONDITIONS	HOT OUTLET [°C]	COLD OUTLET [°C]
CASE 8	45.745	29.885
CASE 9	45.363	30.377

From the above it can be said that, the change in cross section of the tube is having considerable amount of impact in the performance of the system. On comparing the results, the proposed rectangular shape along with the spherical dimples is having better performance than the other cases.

CONCLUSION

The requirement of the industry was to analyse their heat exchanger for the using CFD tools and compare the results and if the results are satisfied, then the development of the system have to be carried out. This was taken as the problem statement and the actual setup was modelled and analysed under the actual boundary conditions to validate the CFD results. The CFD outputs have justified that the actual results and software results are mere co-incidence based on some assumptions.

Further the operating conditions of the system was varied and analysed to find the best effective condition. The case 2 condition was found to be the best ant it was suggested. Now based on this case 2 results, the modifications in baffle design were conducted and it was analysed under same boundary conditions. The baffle designs with spherical dimples was found to be having better performance compared to the other design suggestions.

Also, an initiative was made to check the performance by changing the cross section of the tubes for the design having better performance. This was implemented using the rectangular cross sections for the tube and the analysis was conducted and the results were found to be better than the all other proposed cases.

Hence, it can be concluded that, the rectangular tube is having better performance than the circular tube and the modifications in the baffle designs can be implemented to get the better performance.

REFERENCES

[1] Vindhya Vasiny Prasad Dubey, Raj Rajat Verma, Piyush Shanker And A.K.Srivastava, "Performance Analysis Of Shell & Tube Type Heat Exchanger Under The Effect Of Varied Operating Conditions", Journal Of Mechanical And Civil Engineering (Iosr-Jmce), Volume 11, Issue 3 Ver. Vi (May- Jun. 2014), Pp 08-17.

[2] Vikas Kumar, D. Gangacharyulu, Parlapalli Ms Rao And R. S. Barve, "Cfd Analysis Of Cross Flow Air To Air Tube Type Heat Exchanger".

[3] 3. Avinash D Jadhav, Tushar A Koli, "Cfd Analysis Of Shell And Tube Heat Exchanger To Study The Effect Of Baffle Cut On The Pressure Drop", International Journal Of Research In Aeronautical And Mechanical Engineering, 2321-3051i.

[4] 4. Hetal Kotwal, D.S Patel, "Cfd Analysis Of Shell And Tube Heat Exchanger- A Review", International

Journal Of Engineering Science And Innovative Technology, Volume 2, Issue 2, March 2013

[5] Alok Vyas, Mr. Prashant Sharma, "An Experimental Analysis Study To Improve Performance Of Tubular Heat Exchangers", Alok Vyas Et Al Int. Journal Of Engineering Research And Applications, Vol. 3, Issue 6, Nov-Dec 2013, Pp.1804-1809.

[6] Apu Roy, D.H.Das, "Cfd Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications", International Journal Of Mechanical & Industrial Engineering, Volume-1 Issue-1, 2011

Date: 18 | Dec-2019

"I.C. Engine Cylinder Fins Transient Thermal Analysis By Using ANSYS Software"

Satish Lingoji1, Nagaraju Dappu2

1Research Scholar, Mechanical Engineering Department MRCE Hyderabad (TG.)
2Research Scholar, Mechanical Engineering Department MRCE Hyderabad (TG.)

---***---

ABSTRACT - *The cooling mechanism of the air cooled engine is mostly dependent on the fin design of the cylinder head and block. Cooling fins are used to increase the heat transfer rate of specified surface. Engine life and effectiveness can be improved with effective cooling. The main aim of the project is to study and comparing with 100 cc Hero Honda Motorcycle fins and analyzes the thermal properties by varying geometry, material and thickness. Parametric models of cylinder with fins have been developed to predict the transient thermal behavior. Presently Material used for manufacturing the models is aluminum alloy 6063 which has thermal conductivity of 200W/mk. We are analyzing the designed models by taking the thermal temperature of 100 degrees centigrades. The energy transfers from the combustion chamber of an internal combustion engines are dissipate in three different ways. Transient thermal analyses were performed for actual and proposed design of engine cylinder in order to optimize geometrical parameters and enhanced heat transfer from the IC engine. Result revel that the proposed design of IC engine has better performance and heat transfer rate from the heating zone in the IC engine that is why the result of present work is more concentrate on it and also proposed replacement of new design by Using ANSYS 17.0 software.*

Key words: *Internal Combustion engine, transient thermal analysis, CATIA, ANSYS, Heat Transfer, Aluminum*

I. INTRODUCTION

Most combustion engines or internal combustion engines are fluid cooled using either air (aeriform fluid) or a liquid agent like water run through a device (radiator) cooled by air. In air cooling system, heat is carried out or driven away by the air flowing over and round the cylinder. Here fins are sew the plate and cylinder barrel which offer extra heat conductive and heat radiating surface. In water cooling system of cooling engines, the cylinder walls and heads are given jacket Cooling fins facilitate keep Chevrolet potential unit battery at ideal temperature we all know that just in case of internal combustion (IC) engines, combustion of air and fuel takes place within the engine.

FIG. 1

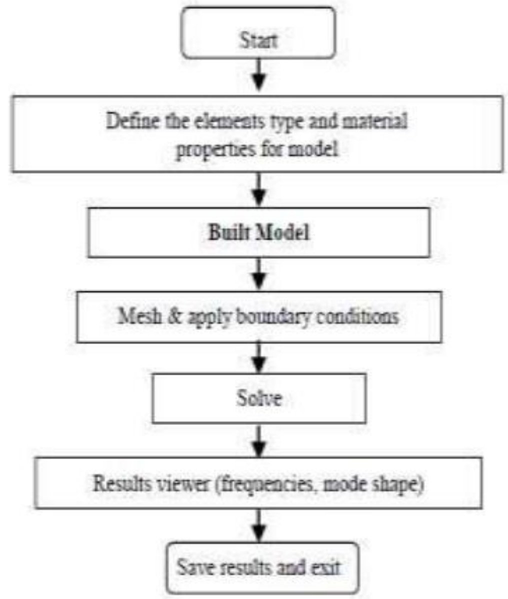


FIG. 2

Types of Cooling System

There are primarily Two types of cooling systems:

1. Air cooled system, and
2. Water cooled system.

II. METHODOLOGY

Step 1: Aggregation data and information associated with cooling fins of IC engines.
Step 2: A completely parametric model of the cylinder block with fin is formed in CATIA ver 5.0 software package.
Step 3: Model obtained in Step a pair of is analyzed using ANSYS17.0 (Workbench), to get the warmth or heat rate , thermal gradient and nodal temperatures.
Step 4: Manual calculations are done.
Step 5: Finally, we tend to compare the results obtained from ANSYS and manual calculations for completely different material, shapes and thickness.

FIN EQUATION

The purpose of fins in IC engine is to boost convective heat transfer from engine. The first purpose behind the operation of fins is to boost the effective heat transfer space from the surface. A balance of energy is performed on this component within which it's assumed that the component is at constant and uniform temperature of T.

Where C_1 and C_2 are constant that can be determined from the boundary conditions.

<u>Case I:</u> The fin is very long and the temperature at the end of the fin is approach to the temperature of the surrounding T_∞.

Conductive heat transfer at the base of fin, according to Fourier's law

$$Q_{Fin} = -kA\left(\frac{dT}{dx}\right)_{x=0}$$

$$Q_{Fin} = \sqrt{hPkA}(T_0 - T_\infty)$$

K = 200 w/mk = 0.2 w/mm k

h = 5 x $10^{-0.006}$ w/mm² k

T_0 = 1000 ⁰C

T_∞ = 25 ⁰C

$$Q_{Fin} = \sqrt{hPkA}(T_0 - T_\infty)$$

Width of fins = 1.5 mm

Thickness of fins = 1 mm

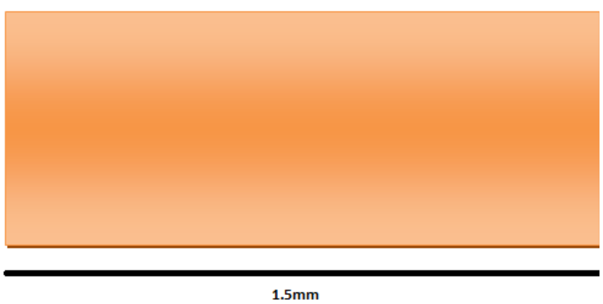


FIG. 3

Q = √ 5x10-0.006 x 2 (1.5 + 1) x 0.2 x 1.5 x 1 x 975

Q = 2670 watt

Material properties and Boundary Conditions

Engine: 100 cc Hero Honda

Aluminum Alloy 6063

Thermal conduction K = 200 W/m-K=0.2 W/mm-K

Specific heat Cp = 0.9 J/g0C = 900 J/Kg-K

Density = a pair of 2.7 g/cc = 2700 kg/m^3= 0.0000027 kg/mm^3

Boundary Condition:

Ambient Temperature: 250

Cylinder Internal temp. = 1000 0C

Heat Flux = 22 W/mm2

Film constant worth = 5x10-006 w/mm2 0C

III. SIMULATION

EXITING MODEL

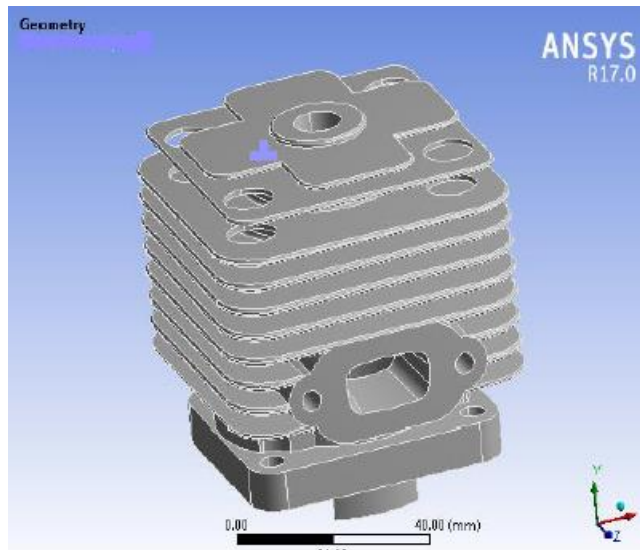


Fig.04

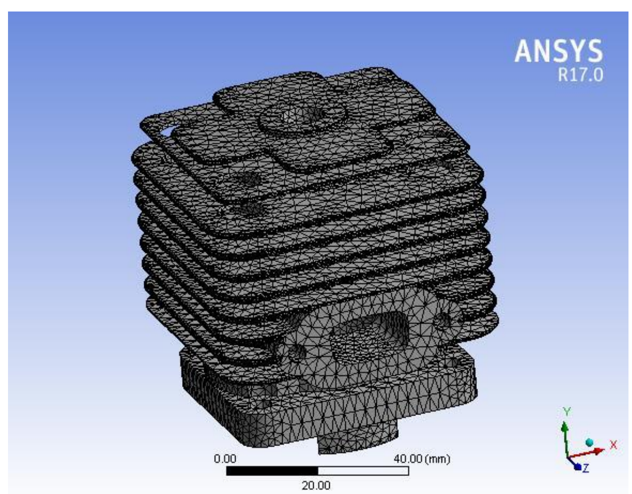


Fig.05

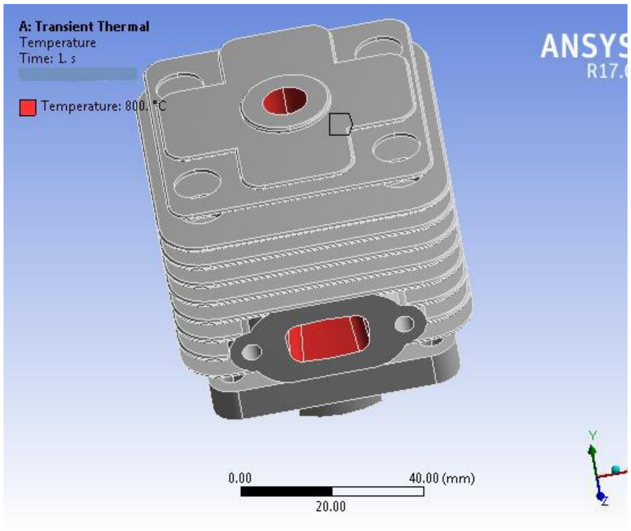


Fig.06

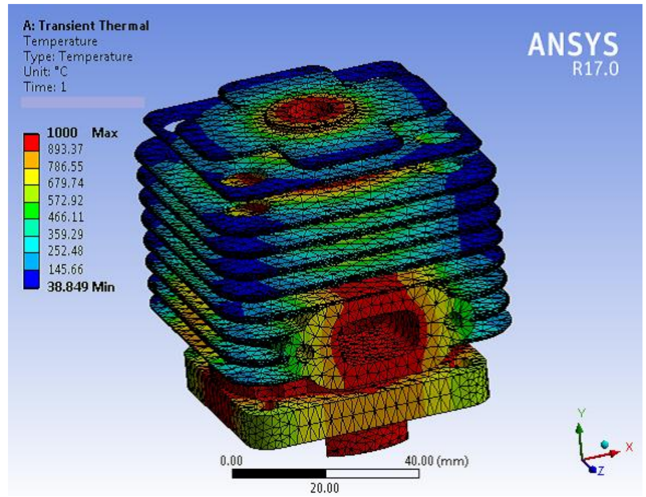


Fig.08

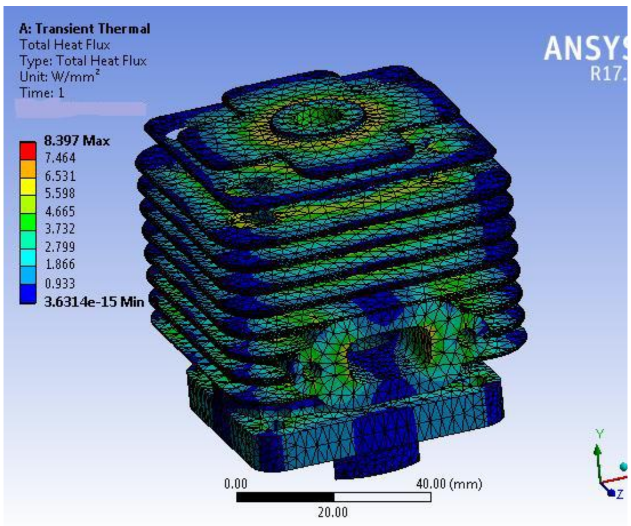


Fig.07

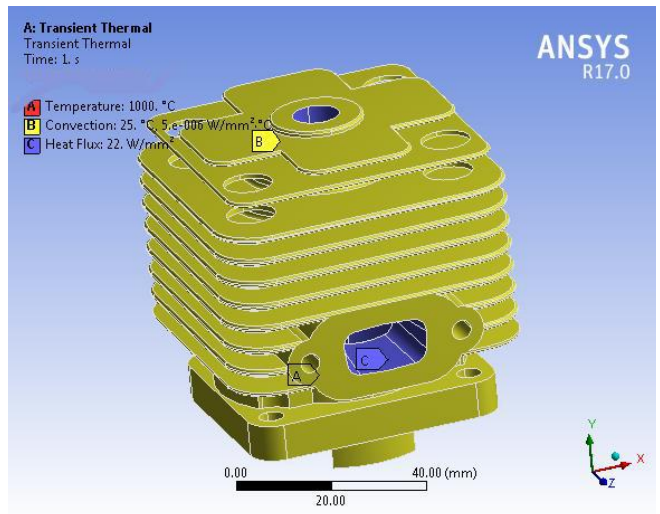


Fig.09

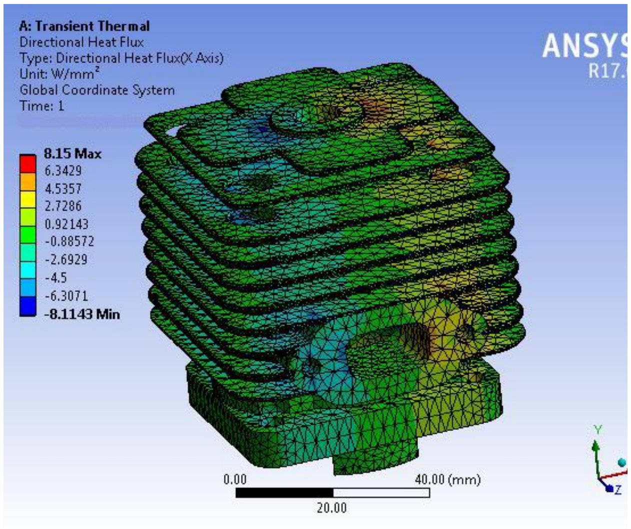


Fig.10

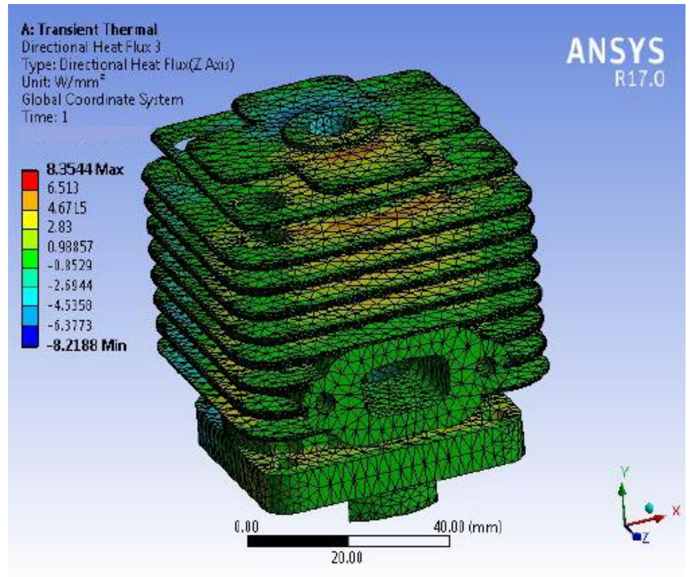


Fig.12

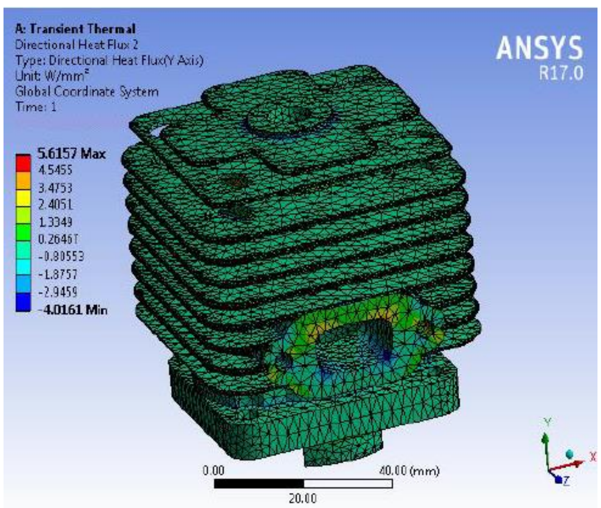


Fig.11

PROPOSED MODEL

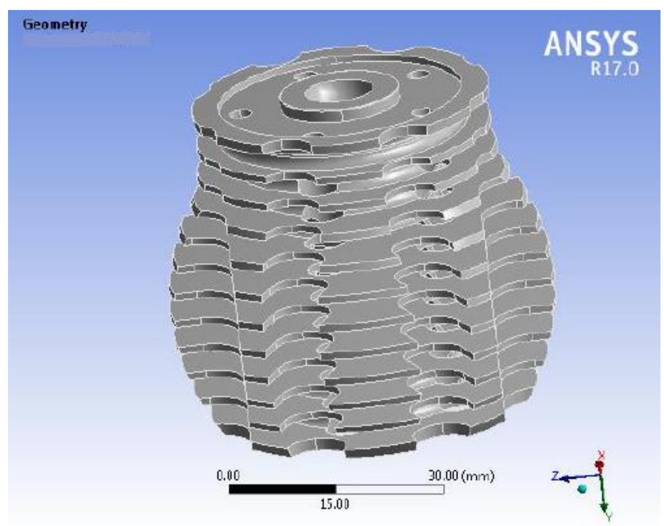


Fig.13

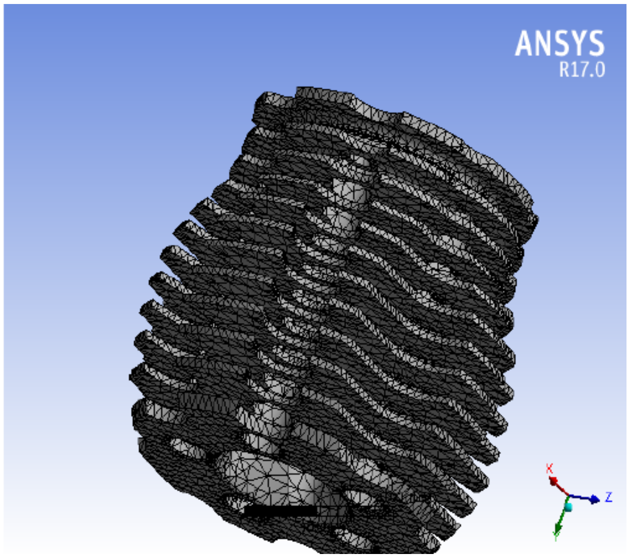


Fig.14

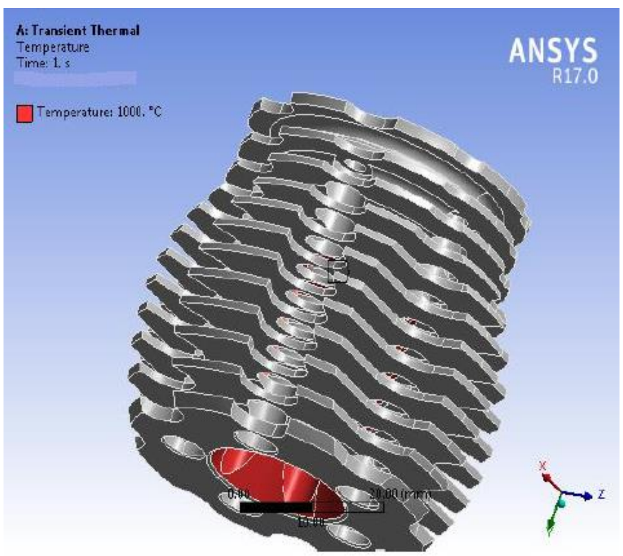


Fig.15

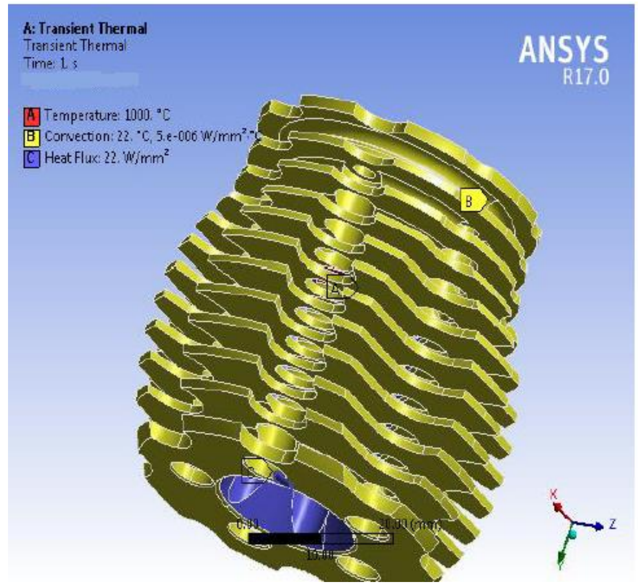


Fig.16

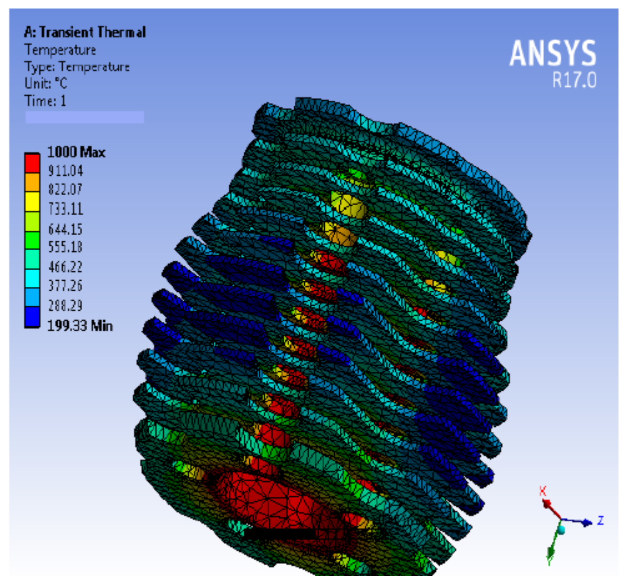


Fig.17

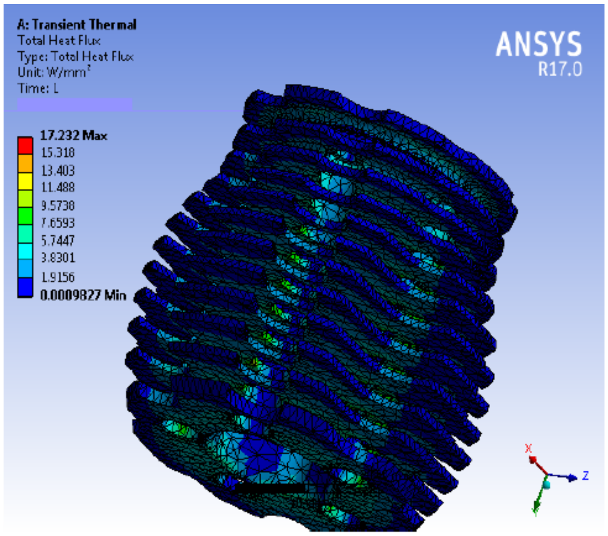


Fig.18

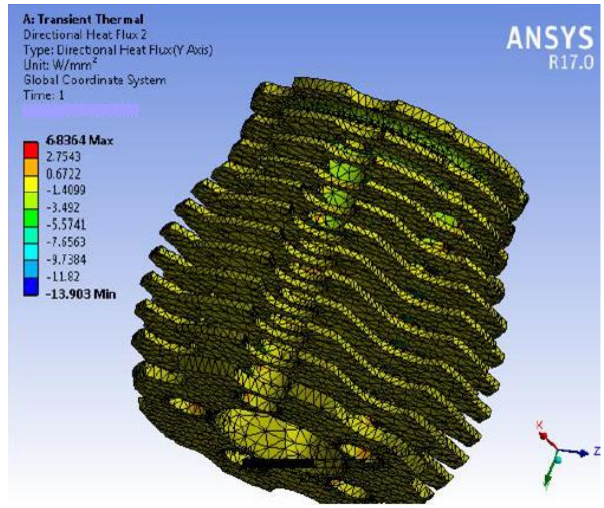


Fig.20

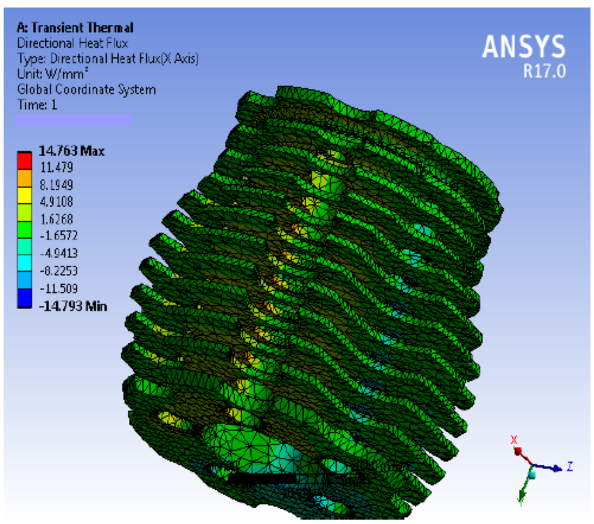


Fig.19

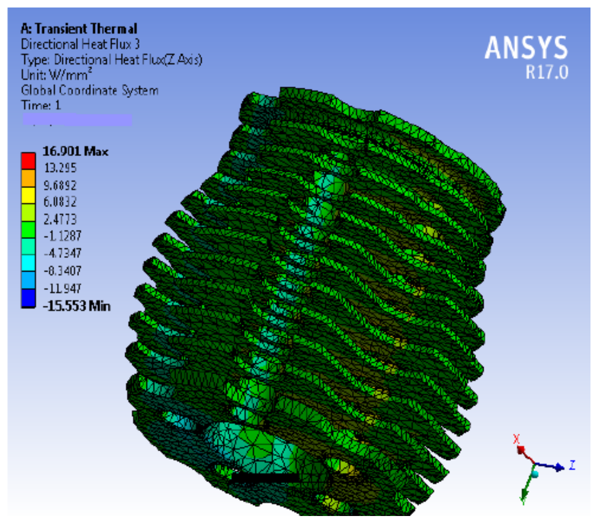


Fig.21

Table.1

S.NO	Temp(⁰C)	Existing Model	Proposed Model
Temp(^{0}C)	*Max*	*1000*	*1000*
	Min	*38.84*	*199.33*
Total Heat (w/mm2)Flux	*Max*	*8.397*	*17.2*
	Min	*3.63E-15*	*9.83E-04*
Directional Heat Flux(w/mm2) (X Direction)	*Max*	*8.15*	*14.76*
	Min	*-8.11*	*-14.79*
Directional Heat Flux(w/mm2) (Y Direction)	*Max*	*5.6*	*6.8*
	Min	*-4.0*	*-13.9*
Directional Heat Flux(w/mm2) (Z Direction)	*Max*	*8.3*	*16.9*
	Min	*-8.2*	*-15.5*

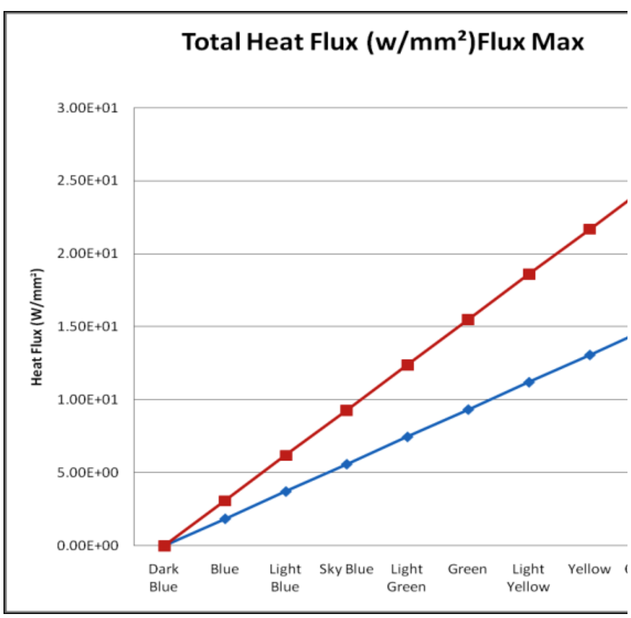


Fig.22

IV. RESULT & DISCUSSION

The transient thermal analysis ware performed using an analytical software package ANSYS worktable supported finite volume analysis. The consequences of various vital geometrical parameters for the transient natural convective heat transfer rate from each actual and projected style of engine.
Transient thermal analyses were performed for actual and projected design of engine cylinder so as to optimize geometrical parameters and increased heat transfer from the IC engine. Within the present work transient thermal analysis is performed on actual style and additionally on two completely different geometrical styles at close temperature 25 ^{0}C. The subsequent points are recognized within the variety of conclusive statements that are as follows.
1. The results of transient thermal analysis of actual design of engine cylinder at close temperature 26oC indicates the utmost temperature is 1000 oC and minimum temperature is 199.3 oC, most enthalpy flux generated is 17.2 W/mm2 and minimum heat flux generated is 9.83E-04 W/mm2, most Directional heat flux in Y-direction generated is 13.2 W/mm2 and minimum Directional heat flux generated is -7.46 W/mm2
2. The results of transient thermal analysis of proposed model of engine cylinder at close temperature 25 ^{0}C indicates the utmost temperature is 1000 ^{0}C and minimum temperature is 167.8 ^{0}C, most total heat flux generated is 27.8 W/mm2 and minimum heat flux generated is 0.002828 W/mm2, the utmost directional heat flux in X-direction generated is 14.76 W/mm2 and minimum

directional heat flux generated is -14.79 /mm2 , directional heat flux in Y-direction generated is 6.8 W/mm2 and minimum Directional heat flux generated is -13.9W/mm2 and directional heat flux in Z-direction generated is 16.9 W/mm2 and minimum Directional heat flux generated is -15.5 W/mm2.

V. CONCLUSION

During this paper we've got designed a cylinder fin body used n a 100cc Hero Honda motorbike and 3D modeling software package CATIA ver 5.0 and used material for fin body is metallic element alloy fins and internal core with gray forged iron. We have a tendency to are commutation with metallic element alloy 6063 for entire body. the form of the fin is rectangular; we've got modified the form with circular geometry formed. To summarize this conclusion, the projected style of IC engine has higher performance and heat transfer rate from the heating zone within the IC engine that's why the results of present work is additional focus on it and additionally projected replacement of recent style.

VI. FUTURE SCOPE:

The aim of gift work to extend heat transfer rate from the heating zone in IC engine, for that transient thermal analysis are performed on actual style of Hero Honda 110 CC single cylinder engine. There are some doable future works which can be doable for more analysis;

1. Radiation analysis also can be performed for constant work.
2. Altogether sorts of analysis within the present work the fabric used for casting is Al alloy; another material may used.
3. CFD analysis also can be done to grasp air flow round the casting.

REFERENCES

[1]**Shubham Shrivastava & Shikar Upadhyay** "Thermal Analysis of IC Engine Cylinder Block with Fins Perpendicular to the Axis of Piston Movement" International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online) Vol. 3, Issue 2, pp: (139-149), Month: October 2015 - March 2016.

[2] Vinay Kumar Attar & Himanshu Arora "Transient Thermal Analysis of Internal Combustion Engine Piston in Ansys Workbench By Finite Element Method" International Journal of Engineering Sciences & Research Technology, ISSN: 2277- 9655 Attar* et al., 5(6): June, 2016.

[3] Chidiebere Okeke-Richard & Sunny Sharma " Thermal Analysis and Comparison of Cylinder Blocks of 4S, SITwo Wheeler Engine Using ANSYS" International Journal of Innovative Science, Engineering & Technology, ISSN 2348 – 7968 Vol. 3 Issue 5, May 2016.

[4] G.Bahadur Vali & Krishna Veni "Design/Modeling and Thermal Analysis on Cylinder Head of I.C Engine" International Journal & Magazine of Engineering, Technology, Management and Research, ISSN No. 2348- 4845 Volume No.03 (2016), Issue No.05 (may).

[5] Abhishek Mote at el "Analysis of Heat transfer through fins of an IC Engine using CFD" International Research Journal of Engineering and Technology (IRJET) e- ISSN: 2395 -0056, p- ISSN: 2395-0072 , Volume: 03 Issue: 04 | Apr-2016.

[6] KM Sajesh, Neelesh Soni and Siddhartha Kosti "Design Modification and Heat Transfer Analysis of Air Cooled Rectangular Fin Engine" International Journal of Recent Scientific Research ISSN: 0976-3031 Volume: 7(3) March - 2016.

[7] Mr. Manir Alam & Mrs. M. Durga Sushmitha "Design and Analysis of Engine Cylinder Fins of Varying Geometry and Materials" International Journal of Computer Engineering In Research Trends, ISSN (O): 2349-7084 Volume 3, Issue 2, February- 2016, pp. 76-80.

[8] Swati Saini & Kagdi Dhruvin Nileshkumar "Comparison of CFD Analysis of Heat Transfer Augmentation & Surface Heat Transfer Co efficient of I. C. Engine Fins at Different wind Velocities with Experimental Investigation" International Journal for Scientific Research & Development| Vol. 4, Issue 03, 2016 | ISSN (online): 2321- 0613.

[9] P.T. Nitnaware & Prachi S. Giri "Design Optimization Of An Air Cooled Internal Combustion Engine Fin Using CFD" Journal of Multidisciplinary Engineering Science and Technology (JMEST) ISSN: 3159- 0040 Vol. 2 Issue 11, November - 2015.

[10] H.Sumithra & B. Sandhya Rani "Design Optimization and Transient Thermal Analysis of Four Stroke Petrol Engine Cylinder Head" International Journal of Emerging Technology in Computer Science & Electronics (IJETCSE) ISSN: 0976- 1353 Volume 18 Issue 2 – NOVEMBER 2015.

[11] M Syamala Devi1, E Venkateswara Rao & K Sunil Ratna Kumar"Optimization of Engine Cylinder Fin by Varying its Geometry and Material" International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064, Paper ID: SUB159073 , Volume 4 Issue 10, October 2015.

[12] Athirah Abdul Aziz, Adlansyah Abd Rahman and Abdul Aziz Hassan "Design of Customized Modular Cylinder Head for Si-Engine" ARPN Journal of Engineering and Applied Sciences ISSN 1819-6608, VOL. 10, NO. 17, SEPTEMBER 2015.

[13] Gokul Karthik "Design Modification and Analysis of Fins in Two Wheeler Engines by CFD" International Journal of Research in Electrical and Electronics Technology (IJREET) Volume 2 - Issue 1 August 2015.

[14] Shamim Alam & Dr. Gaurav Tiwari "Increase the Efficiency of Internal Combustion Engine Fin by Replacing the Conventional Fin with Parabolic Fin" International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 2, Vol. 2, June 2015. e-ISSN: 2394-5141, ISSN: 2394-5133, pp 80-84.

[15] Ashwin Shridhar, Asokan Ram Deepak, S Jayaraj “Int. Journal of Engineering Research and Applications “ Conjugate Heat transfer Analysis of helical fins with airfoil cross section and its comparison with existing circular fin design for air cooled engines employing constant rectangular cross-section, ISSN : 2248- 9622, Vol. 5, Issue 6, (Part -5) June 2015, pp.88

Optimization of MIG Welding Parameters for Improving Strength of Welded Joints

SRIKANTH MUTHAMALA
M.TECH (THERMAL ENGINEERING)
MECHANICAL ENGINEERING DEPARTMENT
MALLA REDDY COLLEGE OF ENGINEERING

Abstract: **The problem that has faced the manufacturer is the control of the process input parameters to obtain a good welded joint with the required weld quality. Traditionally, it has been necessary to study the weld input parameters for welded product to obtain a welded joint with the required quality. To do so, requires a time-consuming trial and error development method. Then welds are examined whether they meet the requirement or not. Finally, the weld parameters can be chosen to produce a welded joint that closely meets the joint qualities. Also, what is not achieved or often considered is an optimized welding parameters combination, since welds can often be formed with very different parameters. In other words, there is often a more ideal welding input parameters combination, which can be used. In this thesis, the influence of welding parameters like welding current, welding voltage, welding speed on ultimate tensile strength (UTS) of AISI 1050 mild steel material during welding. A plan of experiments based on Taguchi technique has been used. An Orthogonal array, signal to noise (S/N) ratio and analysis of variance (ANOVA) are employed to study the welding characteristics of material & optimize the welding parameters. The result computed is in form of contribution from each parameter, through which optimal parameters are identified for maximum tensile strength. From this study, it is observed that welding current and welding speed are major parameters which influence on the tensile strength of welded joint.**

Key words: **MIG Welding; Speed; Taguchi Method; Mini Tab; Tensile Test;**

I. INTRODUCTION

Welding is, at its core, simply a way of bonding two pieces of metal. While there are other ways to join metal (riveting, brazing and soldering, for instance), welding has become the method of choice for its strength, efficiency and versatility.

There are tons of different welding methods, and more are being invented all the time. Some methods use heat to essentially melt two pieces of metal together, often adding a "filler metal" into the joint to act as a binding agent. Other methods rely on pressure to bind metal together, and still others use a combination of both heat **and** pressure. Unlike soldering and brazing, where the metal pieces being joined remain unaltered, the process of welding always changes the work pieces.

WELDING TOOLS OF THE TRADE

The most basic welding rigs, for occasional use in a home workshop, can be had for under $100. Typically, these rigs are set up for **shielded metal arc welding**(**SMAW**), or **stick welding**. Many units only have an on/off switch in the way of controls, making them simple to operate. **Torch welding rigs** are small and easy to work with, which is part of why they're commonly used. These torches use oxyacetylene for the flame, along with a filler rod. But some rigs (like those used in laser-beam welding) are so expensive and complicated that they are only used in industrial applications.As for materials, some are much easier to weldthan others. Steel can be a great choice because of its strength, affordability and weld ability. As a rule, the stronger the steel, the harder it is to weld. Accordingly, several steel alloys were developed with welding in mind. Of course, almost any metal can be welded, including cast iron, bronze, aluminium and even titanium, although the latter requires a highly protected atmosphere because the metal is so reactive.

Whatever you're welding, remember: safety first. If you've ever seen welding in person, you can testify to the blinding brightness the process creates. Looking directly at a weld site without protection can produce what's known as **arc eye**, a painful inflammation of the cornea that feels like getting sand in your eye. No wonder that a good welder's mask is a prerequisite for any welding outfit.

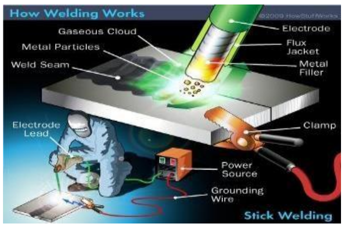


Fig1. welding process

THE PROCESS OF WELDING

Most welding done today falls into one of two categories: arc welding and torch welding.

Arc welding uses an electrical arc to melt the work materials as well as filler material (sometimes called the welding rod) for welding joints. Arc welding involves attaching a grounding wire to the welding material or other metal surface. Another wire known as an electrode lead is placed on the material to be welded. Once that lead is pulled away from the material, an electric arc is generated. It's a little like the sparks you see when pulling jumper cables off a car battery. The arc then melts the work pieces along with the filler material that helps to join the pieces.

Feeding the filler into the welding joint takes steady hands and an eye for detail. As the rod melts, the welder must continuously feed the filler into the joint using small, steady, back-and-forth motions. These motions are what gives welds their distinctive appearance. Going too fast or slow, or holding the arc too close or far away from the material can create poor welds.

Shielded metal arc welding (SMAW or stick welding), gas metal arc welding (more commonly known as metal inert gas, or MIG, welding) and gas tungsten arc welding (frequently called tungsten inert gas, or TIG, welding) all exemplify arc welding.

These three common methods each offer unique advantages and drawbacks. Stick welding, for instance, is inexpensive and easy to learn. It's also slower and less versatile than some other methods. Oppositely, TIG welding is difficult to learn and requires an elaborate welding rig. TIG welding produces high-quality welds, however, and can weld materials that other methods can't.

Torch welding represents another popular welding method. This process typically uses an oxyacetylene torch to melt the working material and welding rod. The welder controls the torch and rod simultaneously, giving him or her a lot of control over the weld. While torch welding has become less common industrially, it's still frequently used for maintenance and repair work, as well as in sculptures (more on that later).

II. LITERATURE REVIEW

Gas Metal Arc Welding (GMAW), sometimes referred to by its subtypes Metal Inert Gas (MIG) welding or Metal Active Gas (MAG) welding, is a semi-automatic or automatic 0020 Arc welding process in which a continuous and consumable wire electrode and a shielding gas are fed through a welding gun. A constant voltage, direct current power source is most commonly used with GMAW, but constant current systems, as well as alternating current, can be used. In this research work an attempt was made to develop a response surface model to predict tensile strength of inert gas metal arc welded AISI 1040 medium carbon steel joints. The process parameters such as welding voltage, current, wire speed and gas flow rate were studied. The experiments were conducted based on a four-factor, three-level, face centred composite design matrix. The empirical relationship can be used to predict the yield strength of inert gas metal arc welded AISI 1040 medium carbon steel. Response Surface Methodology (RSM) was applied to optimizing the MIG welding process parameters to attain the maximum yield strength of the joint.

III. PROBLEM DESCRIPTION

Objective of the work

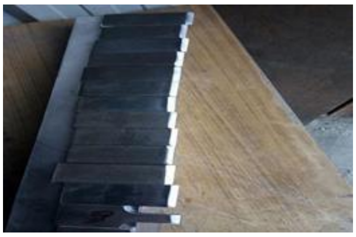
In this thesis, materials AISI 1050 Mild Steel are welded by varying process parameters welding speed, welding current and welding voltage. Effect of process current on the tensile strength of weld joint will be analysed.

EXPERIMENTAL PROCEDURE

In this thesis, experiments are made to understand the effect of MIG welding parameters welding speed, welding current and welding voltage on output parameters such as hardness of welding, tensile strength of welding.

MIG welding experimental images

MIG welding machine



Work pieces (AISI 1050 STEEL)

Dumbbell shape work pieces for tensile test

Work pieces' setup

Welding process

For the experiment, welding parameters selected are shown in table.

The welding current and electrodes considered are

PROCESS PARAMETERS	LE VEL1	LEVEL 2	LEVEL 3
WELDING CURRENT (AMP)	180	230	280
WELDING SPEED (m. m/s)	200	300	400
WELDING VOLTAGE (V)	22	24	26

WELDING CURRENT (AMP)	WELDING SPEED (m. m/s)	WELDING VOLTAGE (V)
180	200	22
180	300	24
180	400	26
230	200	22
230	300	24
230	400	26
280	200	22
280	300	24
280	400	26

WELDING CURRENT (AMP)	WELDING SPEED (m. m/s)	WELDING VOLTAGE (V)	ULTIMATE TENSILE STRENGTH (MPa)
180	200	22	375
180	300	24	410
180	400	26	451.197
230	200	22	403
230	300	24	440.581
230	400	26	372
280	200	22	375.287
280	300	24	369
280	400	26	378

IV. TENSILE TEST REPORTS

INTRODUCTION TO TAGUCHI TECHNIQUE

- Taguchi defines Quality Level of a product as the Total Loss incurred by society due to failure of a product to perform as desired when it deviates from the delivered target performance levels.
- This includes costs associated with poor performance, operating costs (which changes as a product ages) and any added expenses due to harmful side effects of the product in use.

Taguchi Methods

- Help companies to perform the Quality Fix!
- Quality problems are due to Noises in the product or process system
- Noise is any undesirable effect that increases variability

- Conduct extensive Problem Analyses
- Employ Inter-Disciplinary Teams
- Perform Designed Experimental Analyses
- Evaluate Experiments using ANOVA and Signal-to noise techniques

TAGUCHI PARAMETER DESIGN FOR TURNING PROCESS

In order to identify the process parameters affecting the selected machine quality characteristics of turning, the following process parameters are selected for the present work: cutting speed (A), feed rate (B) and depth of cut (C). the selection of parameters of interest and their ranges is based on literature review and some preliminary experiments conducted.

Selection of Orthogonal Array

The process parameters and their values are given in table. It was also decided to study the two – factor interaction effects of process parameters on the selected characteristics while turning. These interactions were considered between cutting speed and feed rate (AXB), feed rate and depth of cut (BXC), cutting speed and depth of cut (AXC).

PROCESS PARAMETERS	LE VEL1	LEVEL 2	LEVEL 3
WELDING CURRENT (AMP)	180	230	280
WELDING SPEED (m. m/s)	200	300	400
WELDING VOLTAGE (V)	22	24	26

Using randomization technique, specimen was turned and cutting forces were measured with the three – dimensional dynamometer. The experimental data for the cutting forces have been reported in Tables. Feed and radial forces being 'lower the better' type of machining quality characteristics, the S/N ratio for this type of response was and is given below:

$$\text{S/N ratio} = -10 \log \left[\frac{1}{n} (y_1^2 + y_2^2 + ... + y_n^2) \right] \quad ...(1)$$

Where $y_1, y_2, ..., y_n$ are the responses of the machining characteristics for each parameter at different levels.

TAGUCHI ORTHOGONAL ARRAY

WELDING CURRENT (AMP)	WELDING SPEED (m. m/s)	WELDING VOLTAGE (V)
180	200	22
180	300	24
180	400	26
230	200	22
230	300	24
230	400	26
280	200	22
280	300	24
280	400	26

OBSERVATION

The following are the observations made by running the experiments. The ultimate tensile strength observed.

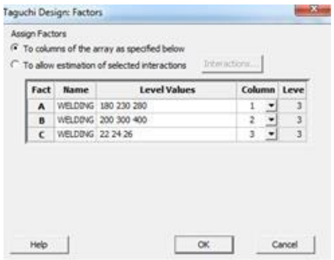
UTS (MPa)
375
410
451.917
403
440.581
372
375.287
369
378

OPTIMIZATION OF ULTIMATE TENSILE STRENGTH USING MINITAB SOFTWARE

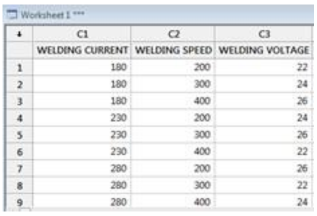
Design of Orthogonal Array

First Taguchi Orthogonal Array is designed in Minitab17 to calculate S/N ratio and Means which steps is given below:

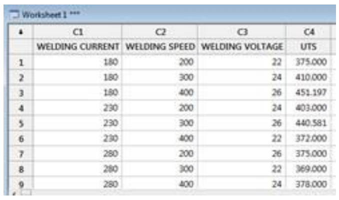
FACTORS



OPTIMIZATION OF PARAMETERS


Worksheet 1 ***

	C1	C2	C3
	WELDING CURRENT	WELDING SPEED	WELDING VOLTAGE
1	180	200	22
2	180	300	24
3	180	400	26
4	230	200	24
5	230	300	26
6	230	400	22
7	280	200	26
8	280	300	22
9	280	400	24

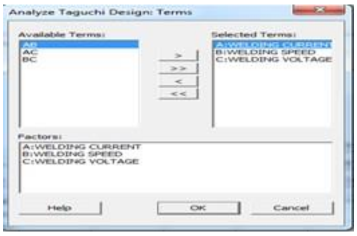

Worksheet 1 ***

	C1	C2	C3	C4
	WELDING CURRENT	WELDING SPEED	WELDING VOLTAGE	UTS
1	180	200	22	375.000
2	180	300	24	410.000
3	180	400	26	451.197
4	230	200	24	403.000
5	230	300	26	440.581
6	230	400	22	372.000
7	280	200	26	375.000
8	280	300	22	369.000
9	280	400	24	378.000

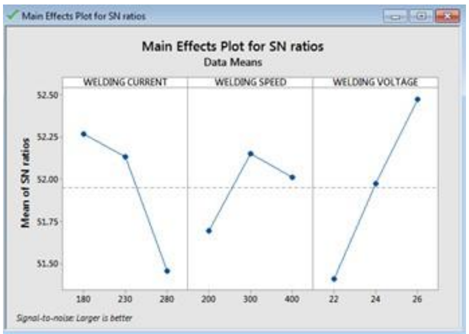
	C1	C2	C3	C4	C5
	WELDING CURRENT	WELDING SPEED	WELDING VOLTAGE	UTS	UTS 1
1	180	200	22	375.000	376
2	180	300	24	410.000	409
3	180	400	26	451.197	450
4	230	200	24	403.000	404
5	230	300	26	440.581	441
6	230	400	22	372.000	371
7	280	200	26	375.000	374
8	280	300	22	369.000	368
9	280	400	24	378.000	379

Analyse Taguchi Design – Select Responses

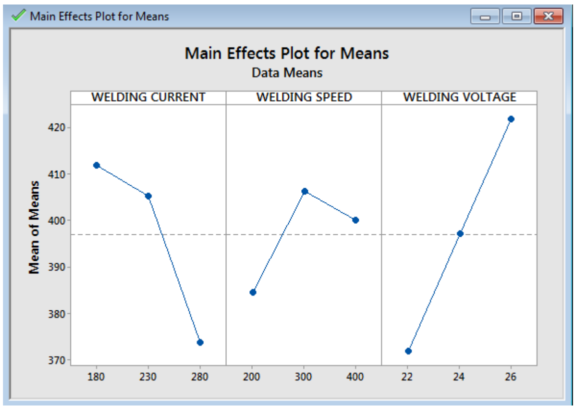
Terms



	C1	C2	C3	C4	C5	C6	C7
	WELDING CURRENT	WELDING SPEED	WELDING VOLTAGE	UTS	UTS 1	SNRA1	MEAN1
1	180	200	22	375.000	376	51.4922	375.500
2	180	300	24	410.000	409	52.2451	409.500
3	180	400	26	451.197	450	53.0758	450.599
4	230	200	24	403.000	404	52.1169	403.500
5	230	300	26	440.581	441	52.8846	440.791
6	230	400	22	372.000	371	51.3992	371.500
7	280	200	26	375.000	374	51.4690	374.500
8	280	300	22	369.000	368	51.3287	368.500
9	280	400	24	378.000	379	51.5613	378.500



S/N ratio plot



Means plot

V. RESULTS

Taguchi method stresses the importance of studying the response variation using the signal–to–noise (S/N) ratio, resulting in minimization of quality characteristic variation due to uncontrollable parameter. The cutting force is considered as the quality characteristic with the concept of "the larger-the-better". The S/N ratio for the larger-the-better is:

S/N = -10 *log(Σ(Y²)/n))

$$S/N = -10 * \log(\Sigma(Y^2)/n))$$

Where n is the number of measurements in a trial/row, in this case, n=1 and y is the measured value in a run/row. The S/N ratio values are calculated by taking into consideration above Eqn. with the help of software Minitab 17.

The force values measured from the experiments and their corresponding S/N ratio values are listed in Table

	C1	C2	C3	C4	C5	C6	C7
	WELDING CURRENT	WELDING SPEED	WELDING VOLTAGE	UTS	UTS 1	SNRA1	MEAN1
1	180	200	22	375.000	376	51.4922	375.500
2	180	300	24	410.000	409	52.2451	409.500
3	180	400	26	451.197	450	53.0758	450.599
4	230	200	24	403.000	404	52.1169	403.500
5	230	300	26	440.581	441	52.8846	440.791
6	230	400	22	372.000	371	51.3992	371.500
7	280	200	26	375.000	374	51.4690	374.500
8	280	300	22	369.000	368	51.3287	368.500
9	280	400	24	378.000	379	51.5613	378.500

VI. CONCLUSION

The experiment designed by Taguchi method fulfils the desired objective. Fuzzy interference system has been used to find out the ultimate tensile strength. The all possible values of have been calculated by using MINITAB 17.0 software. Analysis of variance (ANOVA) helps to find out the significance level of each parameter. The optimum value was predicted using MINITAB-17 software.

The welding parameters are Welding current, welding voltage and welding speed for MIG welding of work piece AISI1050 steel. In this work, the optimal parameters of welding speed are 200m.m/s, 300 m.m/s & 400 m.m/s, welding current are 180, 230 &280 amps, and welding voltage are 22, 24 &26 volts. Experimental work is conducted by considering the above parameters. Ultimate tensile strength validated experimentally.

The experimental results confirmed the validity of the used Taguchi method for enhancing the welding performance and optimizing the welding parameters in MIG welding at welding speed 400 m.m/s, welding voltage 26 volts and welding current 180 amps

VII. REFERENCES

[1]. Optimization of mig welding process parameters to predict maximum yield strength in aisi 1040 ajit hooda1*, ashwani dhingra2 and satpal sharma2

[2]. A Review on Optimization of MIG Welding

Parameters using Taguchi's DOE Method Satyaduttsinh P. Chavda1, Jayesh V.Desai2 , Tushar M.Patel3

[3]. Optimization of the process parameters for mig welding of aisi 304 and is 1079 using fuzzy logic method prasenjit mondal1, dipankar bose2

[4]. Optimization of MIG welding Process Parameter using Taguchi Techniques Kapil B. Pipavat1 , Dr. Divyang Pandya2 , Mr. Vivek Patel3

[5]. Optimization of MIG Welding Parameters for Hardness of Aluminium Alloys Using Taguchi Method Vineeta Kanwal1 , R S Jadoun2

[6]. Parameters Optimization for Tensile Strength & Hardness of MIG Welding Joint of HSS & Mild Steel by Using Taguchi Technique Er. Rahul Malik1 , Er. Surjeet Gahlot2 , Dr. S.K. Jarial3

[7]. A Review on Parametric Optimization of MIG Welding Parameters by using Various Optimization Techniques 1Pranesh B. Bamankar Asst. Professor Arvind Gavali College of Engg. Satara 2Amol Chavan Student Arvind Gavali College of Engg. Satara Tushar Phadtare Student Arvind Gavali College of Engg. Satara

Additive fabrication in Automotive sector

Abstract

Automotive sector faces new challenges every day, new design trends and technological deployments from research push companies to develop new models and facelifts in short term, requiring new tools or tool reshaping. The Automotive sector is one of the most competitive business areas where time-to-market decrease plays an important role. Additive fabrication is the solution which enables the flexible production of customized products without significant impact on costs and lead time. Automotive companies developing new models and facelifts every day, pushed by new design trends and technological evolution where aesthetics, aerodynamics, safety, and weight reduction of the vehicle are key issues. Therefore, new tools or tool reshaping for new components is required, for body panels or other technical components. Digitalization helps the Automotive sector to turn their ideas into successful vehicle faster and more efficiently. Automobile manufacturers can increase the efficiency of their research and development processes, enabling them to get their products on the market with less time and efficiently.. An important parameter of using metal AM processes in the automotive sector is fabricating complex lightweight structures which at the same time possess good rigidity. The weight of the automotive parts can be reduced significantly by using the ability of AM processes to maximum advantage to produce parts with complex geometries while maintaining relative strengths. This paper provides a review of today and future of additive fabrication in Automotive sector.

***Keywords**:* Automotive sector, Additive fabrication, 3D printing, STL, CAD.

1. Introduction

Additive fabrication commonly known as 3D printing allows the direct conversion of design construction files into fully functional objects. It is a process of joining materials to make object from 3D model data usually layer upon layer. In this the material is joined or solidified under computer control to create a three-dimensional object, with material being added together (such as liquid molecules or powder grains being fused together) typically layer by layer.
Vehicle manufacturers have been at the forefront of implementing additive fabrication technology. One of the most prolific additive fabrication applications in the automotive world is in rapid prototyping. This is also among the oldest uses of the technology, with some large auto manufacturers having prototyped parts with 3D printers for more than 20 years. Beyond prototyping, automotive manufacturers are now increasingly bringing it into use for actual production. In recent years, additive fabrication (AM) technologies have radically changed our way to design, develop and manufacture new products. In the Automotive sector, additive fabrication technologies have made wonders to bring new shapes to life, allowing for lighter and more intricate structures at the best possible cost.

Aim and Objectives:
- Innovative and without making use of tools fabrication of intricate shape and light weight components is possible.
- Maximum freedom for design, allowing the creation of complex yet light weight components with high level of rigidity.
- Additive fabrication can significantly reduce the material waste, reduce the amount of production steps, inventory being held and reduce the amount of distinct parts needed for an assembly work.
- Reducing the need for manual assembly
- Additive makes it possible to create internal complexities and precisely control microstructure.
- Enables production of components with integrated functionality without the need for tools, thereby cutting development and production costs.
- Time and cost reduction (shorter lead time).
- Leads to more market opportunities.

Firstly we'll explain the procedure i.e., how the additive fabrication is carried out and then requirements in Automotive sector, AM in the Automotive sector, common automotive applications, current and future uses of AM in Automotive sector, challenges for Automotive sector. But before that you should have the knowledge of different AM processes.

2. Additive fabrication Procedure

I. CAD: Producing a digital model is the first step in the additive fabrication process. The most common method for producing a digital model is computer-aided design (CAD). There are a large range of free and professional CAD programs that are compatible with additive manufacture. Reverse engineering can also be used to generate a digital model via 3D scanning.

II. STL conversion and file manipulation: A critical stage in the additive fabrication process that varies from traditional fabrication methodology is the requirement to convert a CAD model into an STL (stereolithography) file. STL uses triangles (polygons) to describe the surfaces of an object. A guide on how to convert a CAD model to an STL file can be found here. There are several model limitations that should be considered before converting a model to an STL file including physical size, watertightness and polygon count. Once a STL file has been generated the file is imported into a slicer program. This program takes the STL file and converts it into G-code. G-code is a numerical control (NC) programming language. It is used in computer-aided fabrication (CAM) to control automated machine tools (including CNC machines and 3D printers). The slicer program also allows the designer to customize the build parameters including support, layer height, and part orientation.

III. Printing: 3D printing machines often comprise of many small and intricate parts so correct maintenance and calibration is critical to producing accurate prints. At this stage, the print material is also loaded into the printer. The raw materials used in additive fabrication often have a limited shelf life and require careful handling. While some processes offer the ability to recycle excess build material, repeated reuse can result in a reduction in material properties if not replaced regularly. Most additive fabrication machines do not need to be monitored after the print has begun. The machine will follow an automated process and issues generally only arise when the machine runs out of material or there is an error in the software.

IV. Removal of prints: For some additive fabrication technologies removal of the print is as simple as separating the printed part from the build platform. For other more industrial 3D printing methods the removal of a print is a highly technical process involving precise extraction of the print while it is still encased in the build material or attached to the build plate. These methods require complicated removal procedures and highly skilled machine operators along with safety equipment and controlled environments.

V. Post processing: Post processing procedures again vary by printer technology. SLA requires a component to cure under UV before handling, metal parts often need to be stress relieved in an oven while FDM parts can be handled right away. For technologies that utilize support, this is also removed at the post-processing stage. Most 3D printing materials are able to be sanded and other post-processing techniques including tumbling, high-pressure air cleaning, polishing, and coloring are implemented to prepare a print for end use.

3. Requirements in Automotive sector

Weight - final parts: One of the most critical aspects relating to the Automotive sector is the weight reduction of components. Automotive applications make use of advanced engineering materials and complex geometries in an attempt to reduce weight and improve performance. AM is capable of producing parts from many of the lightweight polymers and metals that are common in the Automotive sector.

Complex geometries - prototypes and final parts: Affecting weight and aerodynamics (and therefore vehicle performance) is the geometry of a part. Automotive parts often require internal channels for conformal cooling, hidden features, thin walls, fine meshes and complex curved surfaces. AM allows for the manufacture of highly complex structures which can still be extremely light and stable. It provides a high degree of design freedom, the optimization, and integration of functional features, the manufacture of small batch sizes at reasonable unit costs and a high degree of product customization even in serial production.

Temperature - testing and final parts: Many automotive applications require significant heat deflection minimums. There are several AM processes that offer materials that withstand temperatures well above the average 105°C sustained engine compartment temperatures. SLS nylon, as well as some photo-cured polymers, are suitable for high-temperature applications.

Moisture - testing and final parts: Most components that go into the production of automobiles must be moisture resistant, if not moisture proof, entirely. One major benefit of additive fabrication is that all printed parts can be post-processed in order to create a watertight and moisture resistant barrier. Additionally, many materials, by their very nature, are suited for humidity and moisture plagued environments.

Part consolidation - prototyping and final parts: The number of items in an assembly can be reduced by redesigning as a single complex component. Part consolidation is a significant factor when considering how AM can benefit the reduction of material usage, thereby reducing weight and in the long run, cost. Part consolidation also reduces inventory and means that assemblies can be replaced with a single part should repairs or maintenance need to take place; another important consideration for the Automotive sector.

3. AM in the Automotive sector

Communication: Designs in the Automotive sector often begin as scale models showcasing the form of a vehicle. These are often also regularly used for aerodynamic testing. SLA and material jetting are used to produce high detail, smooth, scale models of automotive designs. Accurate models allow design intention to be clearly communicated and showcase the overall form of a concept.

Validation: Prototyping using AM is now commonplace in the Automotive sector. From a full size wing mirror printed quickly with low cost FDM to a high detail, full color dashboard, there is an AM technology suited to every prototyping need. Some AM engineering materials also allow for full testing and validation of prototype performance.

Pre-production: One of the areas AM has been most disruptive is the production of low cost rapid tooling for injection moulding, thermoforming and jig and fixtures. Within the Automotive sector this allows for tooling to be quickly manufactured at a low cost and then used to produce low to medium runs of parts. This validation mitigates the risk when investing in high cost tooling at the production stage.

Production: Since production volumes in the Automotive sector are generally very high (greater than 100,000 parts per year) AM has predominantly been used as a prototyping solution rather than for end part fabrication. Improvements in the size of industrial printers, the speed they are able to print at and the materials that are available mean that AM is now a viable option for many medium-sized production runs, particularly for higher-end automobile manufacturers that restrict production numbers to far fewer than the average.
With the possibility of producing multiple design iterations in a shorter amount of time (and at little additional cost), 3D printing is an effective tool for product development. Typically, a part must go through several design cycles before the final design is agreed upon. With 3D printing, this stage can be sped up dramatically. Additionally, cost-efficient design improvements can be made relatively quickly, since the technology does not require expensive tooling to produce a prototype.

Tooling: Tooling is used extensively within the Automotive sector to help produce high-quality products. Additive fabrication can complement this process by creating jigs, fixtures and other customized tooling equipment. The Volkswagen Europa assembly plant is already using AM to produce tooling equipment in-house, rather than sourcing tooling equipment from third party providers. With a 10-day turnaround for positioning and screw assembly (down from 56 days using a third party source), AM clearly shows itself to be a cost-effective return on investment for tooling production, enhancing the overall production process.[3,4]

End-Part Production: Although additive fabrication was originally adopted as a prototyping tool, recent advances in AM technology and materials make the production of small and medium-size production of end parts possible. This can range from exterior components to inner parts such as bellows, complex ducting, mounting brackets, and engine components. One example is Bugatti: only this year, the luxury car manufacturer announced it had produced a fully functional titanium brake caliper — entirely 3D printed. With such breakthroughs in end-part production, 3D printing is set to become a key technology for this application.

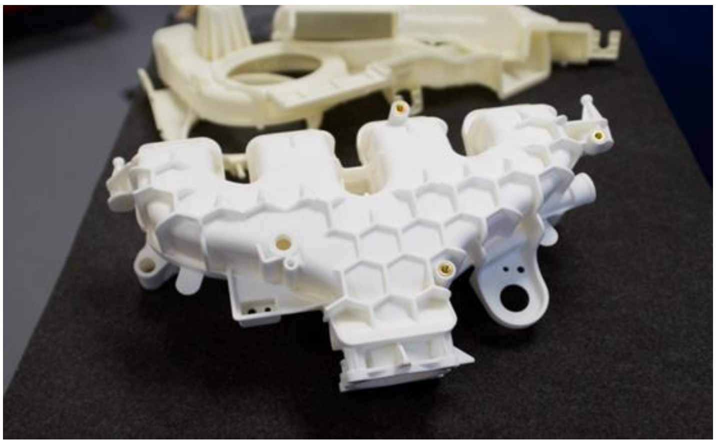
Bugattis 3D-printed brake caliper

Spare parts: Automakers can leverage the benefits of 3D printing to create spare parts on demand. With high inventory costs associated with storing spare (and often rarely ordered) parts, 3D printing provides a cost-effective means to produce parts needed on time and on demand, leading to improved delivery times, reduced inventory costs and a simplified supply chain.

4. Common Automotive Applications

Bellows: AM can be used to make semi-functional bellow pieces where some flexibility is required in assembly or mating. Generally, this material/process is best to consider for applications where the part will be exposed to very few repetitive flexing motions. For projects that require significant flexing, other Polyethylene based SLS materials such as Duraform “Flex” are better suited.

Complex Ducting: By using SLS to manufacture non-structural low volume ducting such as environmental control system (ECS) ducting for aerospace and performance racing, you can design highly optimized, very complex single piece structures.



A complex, functional ducting design printed in SLS nylon

With SLS it is possible to engineer in variable wall thicknesses and increase the strength to weight ratio through the application of structurally optimized surface webbing. This is a very costly detail to apply with traditional fabrication techniques. For SLS there is no cost for complexity, parts are printed without support and to a high level of accuracy.

High detail visual prototypes: Unlike traditional prototyping methods some AM processes are able to produce multicolor designs with a surface finish comparable to injection moulding. These models allow designers to get a greater understanding of the form and fit of a part. This highly accurate method of prototyping is also ideal for aerodynamic testing and analysis as the surface finish that is able to achieve is often representative of a final part. AM is used regularly to manufacture automotive components that rely on aesthetics over function resulting in everything from wing mirrors and light housings to steering wheels and full interior dashboard designs being produced. Material jetting and SLA printing are the two most common methods used for aesthetic prototypes producing parts from a photo-activated resin.

Full colour, textured visual prototypes like this centre console can be produced via AM

Functional mounting brackets: Being able to rapidly manufacture a complex, lightweight bracket overnight is a trademark of the AM industry. Not only does AM allow for organic shapes and designs to be manufactured but AM also requires very little input from an operator meaning that engineers are able to quickly take a design from a computer to assembly in a very short amount of time. This is not possible with traditional fabrication techniques like CNC machining where a highly skilled machine operator is needed to produce parts. Powder bed fusion technologies like SLS nylon and metal printing are best suited for functional parts and offer a range of materials (from PA12 nylon to titanium).

A functional alternator bracket printed using SLS nylon

5. Current Uses of Additive fabrication

Exhausts and emissions:
melting to create cooling vents.

Fluid handling: Selective laser melting and electron beam melting are utilised with aluminium alloys. These techniques can be used to make pumps and valves within the fluid handling system.

Exterior: Using selective laser sintering, polymers are currently used to manufacture wind breakers and bumpers.

fabrication process: Hot work steels and polymers can be used together with a variety of additive fabrication processes such as selective laser sintering, selective laser melting and fused deposition modelling for prototypes, casting and customised tooling.
The above uses in the Automotive sector are used from both small companies and large international conglomerates.

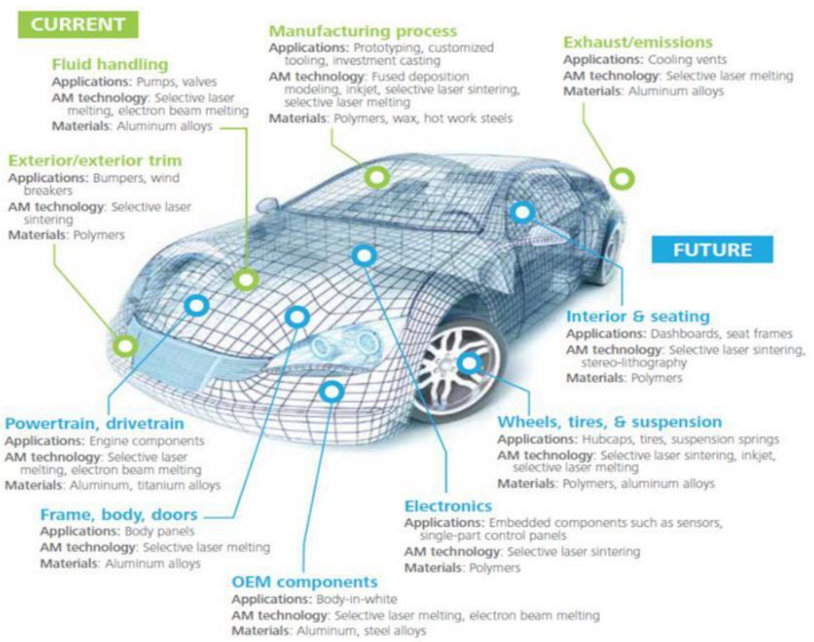
Interior and seating: Using polymers and the techniques of stereo-lithography and selective Laser sintering, dashboards and seat frames could be manufactured.

Tyres, wheels and suspension: Aluminium alloys and polymers can be manipulated with the aid of selective Laser sintering, selective laser melting and inkjet technology to create suspension springs, tyres and hubcaps.

Electronics: Selective laser sintering can be used on polymers to manufacture a range of delicate components including parts which have to be embedded, such as sensors, and single part control panels.

Framework and doors: Selective laser melting can be used on metal compounds such as aluminium alloys to create body panels, including framework and doors.

Engine components: Various functional parts of the engine can be made from metals such titanium and aluminium allows when techniques such as electron beam melting and selective Laser melting are used. [1,2]



6. Challenges for the Automotive sector

Mass production: While automotive OEMs are increasingly incorporating AM systems into development and production, one challenge to wider adoption is a production volume. With over 80 million cars produced in 2017 alone, the Automotive sector is heavily reliant on mass, series production. 3D printing should therefore not be seen as a replacement for traditional fabrication methods, well-suited to mass volumes, but as a complementary tool for lower volume, customised end parts.

Build sizes: Another challenge faced by automakers is the limited build size of many AM systems. Although larger parts can be produced with 3D printing technology, this must be done in the form of modular parts. These in turn currently have to be assembled or attached together through other processes, such as welding. However, large-scale additive fabrication is an important and growing area of research, with technologies that can support larger build sizes, such as Wire Arc Additive fabrication (WAAM) and Big Area Additive fabrication (BAAM), actively being researched and developed.

AM skills gap: Additional investment into developing AM-specific skills must also be addressed if the technology is to take off more widely. Design for additive fabrication as well as the operation and maintenance of AM systems, materials and post-processing are all vital skills that must be developed and nurtured. While much has been said on the current skills gap for AM, partnerships with universities and internal training programmes are one way of ensuring a skilled talent pool able to work with the particularities of AM technology.

Conclusion

This paper aimed to figure out the scenario of additive fabrication in Automotive sector. Improvements in additive fabrication materials and product quality are expanding the usefulness of 3D printing in auto fabrication. Some parts produced using AM technologies occasionally have tiny "voids" or pores that can weaken overall strength. In others, dimensional accuracy is not always on par with parts made with conventional fabrication processes. These and other quality issues can diminish product uniformity and consistency, a challenge for high-volume industries such as automotive in which quality and reliability are critical.

Profitability in the Automotive sector is driven by volume. Given these enormous volumes, the low production speed of AM is a significant impediment to its wider adoption for direct part fabrication. This has made high-speed AM an important area of research.

Despite of this, AM offers a versatile set of technologies that can support auto companies as they pursue performance, growth, and innovation. Traditional fabrication techniques are deeply entrenched and will likely hold a dominant position in the Automotive sector for the foreseeable future. Yet the breadth of AM capabilities—and the success of on-going efforts to broaden their application—suggest that going forward, additive fabrication will also play an important role in shaping the global automotive landscape.

Although, the doors for conventional fabrication are still open and will play a dominant role in automotive fabrication, additive fabrication is making inroads and is obvious to change the global shape of the industry.

References

[1] D. Cooper, J. Thornby, N. Blundell, R. Henrys, M. A. Williams, and G. Gibbons, "Design and Manufacture of High Performance Hollow Engine Valves by Additive Layer fabrication,"Materials & Design, vol. 69, pp. 44-55, 2015.

[2] S. Biamino, B. Klöden, T. Weißgärber, B. Kieback, and U. Ackelid, "Properties of a TiAl Turbocharger Wheel Produced by Electron Beam Melting," in Fraunhofer Direct Digital fabrication Conference, Berlin, Germany, 2014.

[3] M. Cotteleer, M. Neier, and J. Crane, "3D Opportunity in Tooling: Additive fabrication Shapes the Future," Deloitte University Press 2014.

[4] A. V. Villalon, "Electron Beam Fabrication of Injection Mold Tooling with Conformal Cooling Channels," M.Sc. Thesis Industrial Engineering North Carolina State University, Raleigh, North Carolina, USA 2005.

[5] https://www.sculpteo.com/blog/2016/01/20/3d-printing-transforms-the-automotive-industry/

Design and Analysis of Hollow and Solid Shaft

GOVINDOLA VEERANDER GOUD

veerander85@gmail.com

M Tech Student, Department of MECH,

Malla Reddy college of Engineering, Hyderabad, India.

Abstract

The Roadway vehicles like vehicle, buses, trucks and land movers goods many mechanic ability in common like Engine parts, Propeller shafts, Gearbox, Brakes, Clutches, Wheels, etc., To make the vehicle kindling capable which in result mate the transportation economical, the weight of that vehicle should be reduced. Since the composite materials are publicity weight with more puissance & hardness, inclusion of composite materials to conventional steel materials custom in auto parts will lessen the weight and better the machinelike properties of those components. In this thesis, deals with shaft of MARUTHI OMNI to design the shaft for its minimum dimensions to suffice authentic question specification and then replace accepted steel material with composite material. The design of the propeller shaft is first

theoretically designed for steel, aluminum allay ,lose iron and kelvar composite essential for its safe dimensions. Then they can be created as a part shape for respective dimensions in CATIA software. After modeling, static analysis and Modal analysis can be carried out in the propeller shafts worn

Keywords: **Shaft; Coupling; ANSYS; CATIA;**

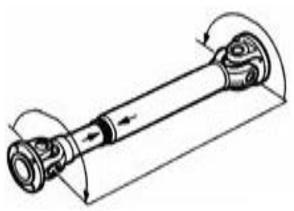
1. INTRODUCTION

SHAFT: Propeller Shaft is the shaft that transmits Command from the gear case to the differential gear in a motor vehicle from the engine to the propeller in a sail or flying machine.

Fig 1.1.1 Propeller shaft

Propeller shaft, sometimes exhort a card an well,

- Used in vehicles with a short distance between the engine and axles, and MR based four- wheel-drive vehicles.
- The friction wielding adopted at the junction contributes to an



ANSYS software.

Fig 1.1.2 hallow shaft

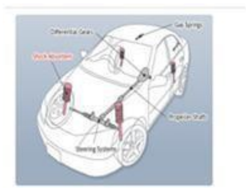
Types of Propeller Shaft: Single–

Piece–Type Propeller shaft :

Improvement in the strength,quality and durability of the junction

2-piece-type/3-piece-type

- Utilized as a part of vehicles with a long distance between the engine and axles and front engine front drive base foure-wheel-drive vehicles
- The division of the propeller shaft into two- or three- parts allows the critical number of revolutions to lowered preventing vibration issue from occurring, when the overall length of the shaft increased



- A tubular-section propeller shaft is normally used as it has (i) Hill weight, (ii)provides low resistance to misalignment, especially sag, (Hi) has serviceable torsional Strength , and (iv) provides low resistance (low inertia) to exchange in sharp quickness, which arise when a hookes type coupling is used to drift the shaft. Since a propeller shaft often revolve at high speed,

2. METHODOLOGY

I. Material modeling for the properties of nano composite, this has been Modeled using CATIA parametric software

ii. Modeling of propeller Shaft Model by using CATIA parametric software.

iii. Determination of stress, strain, deformation and frequencies.

iv. Modal analysis of the propeller shaft model.

3. LITERATURE REVIEW

Functions: Propeller shaft is a automatic constituting for transmitting torque and rotation, usually custom to connect other components of a drive discipline that cannot be connected expressly long of distance or the need to allow for relative movement between them. Propeller shaft make any machine move. Drive well is carriers of torque: they are subject to torsion and fleece urge, analogous to the difference between the input torque and the freight. They must therefore be strong enough to bear the stress, whilst avoiding too much additional load as that would in turn increase their inertia. The longer the shaft, the more bound it is to incline and bending is further promoted when rotation is address reason vibrations and resulting in an augment in noise. For this ground, the propeller shaft has been show to overwhelm vibrations proceed from a remote range of source.
Propeller Shaft Vibration: Small auto and short vans and trucks embodied a single propeller well with a omit-joint at the front end without having any undesired vibration. Vehicles with longer wheelbase need longer propeller shaft, which has a disposition to sag and to harry under certain operating circumstances (Fig. 26.3). As a result resonant vibrations are put up in the body of the vehicle, so that the body vibrates as the shaft whirls. The might agent responsible for the rebellant frequency of the propeller shaft causing the vibration may be grouped as syn Propeller Shaft Intermediate Support Bearings: Intermediate bearing-and-mount assemblies are incorporated to position and support the divided propeller shafts. These assemblies are either of (i) self-aligning bearing verify represent of («) obsequious-mounted influence assist semblance. Self-aligning intermediate-bearing supports are mostly employment on heavy-respect trucks. One type of this bearing support is a double-line courage-conduct with a deep-fossulate inner race and an internally semicircular external race (Fig. 26.7A). This arrangement compensates any shaft inflection through the inner line and nonsense, which tilt about the fixed dispossession-race sphere-shaped post.

4. RELEATED STUDY

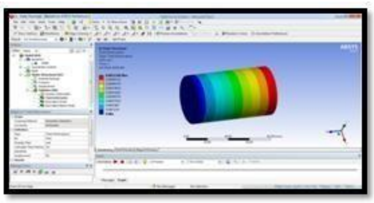
INTRODUCTION TO CATIA: (an acronym of computer aided three-dimensional interactive application) is a multi-plan software suite for electronic computer-aided purpose (CAD), computer-assist manufacturing (CAM), computer-relieve engineering (CAE), PLM and 3D, developed by the French party Dassault Systems. CATIA started as an in-tenement unraveling in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that measure purchaser of the CADAM software to develop Dassault's Mirage fighter jet. It was later adopted by the aerospace, self-propelled, shipbuilding, and other industries. Initially denominate CATI (conception assisted tridimensionally interactive – French for interactive aided three-dimensional project), it was renamed CATIA in 1981 when Dassault appoint a assisting to develop and sell the software and type a non-exclusive distribution agreement with IBM.

HALLO SHAFT

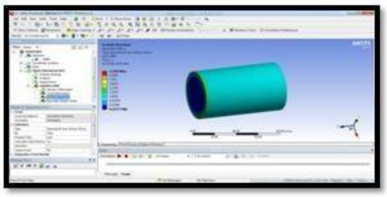
SOLID SHAFT

ANSYS Software: ANSYS is an Engineering Simulation Software (computer relieve Engineering). Its bowl shelter Thermal, Static, Dynamic, and Fatigue finite element analysis along with other use all designed to help with the development of the fruit. The party was founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI. Its primary view was to develop and nundinal finite element analysis software for stextural physics that could simulate static (motionless), functioning (drifting) and ardor transfer (thermal column) problems. SASI developed its business in analogue with the growth in computer technology and engineering needs.

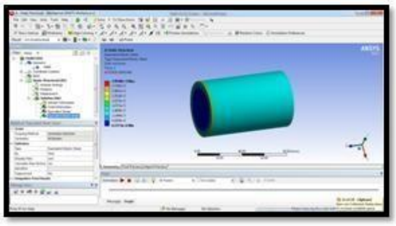
Stress



Strain



Total deformation

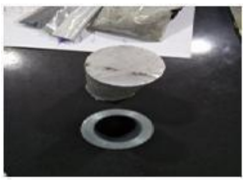


5. RESULT TABLES

STATIC ANALYSIS RESULTS

Models	Material	Deformation (mm)	Stress (N/m ^{2})	Strain
Solid	Aluminum	0.0011106	2.1263	2.9948 e-5
	Aluminum with BLF	0.00011475	2.0117	2.9193 e-5
Hollow	Aluminum	0.0012224	1.7055	2.639e -5
	Aluminum with BLF	0.0013249	1.6791	2.7984 e-5

PRACTICLE EXPERIMENT



6. CONCLUSION

The aluminum with BLF composite solid and hollw shafts are intend to meet safe design requirements as the stipulated steel shaft. From the static analysis the deformation, ,VonMises distress and weight are determined. In overall similitude aluminum with BLF composite hollw pit is correct only in weight curtailment and that too only 1.16% lesser weight than aluminum fineness with BLF compounded shaft.

7. REFERENCES

[1] Branislav Duleba, Ludmila Dulebova and Emil Spisak (2014), ‘Simulation and evaluation of carbon/ epoxy compound system second-hand FEM and tensile test’, Science Direct – Procedia Engineering 96(2014), pp.70-74.

[2] Srinivasa Moorthy R, Yonas Mitiku and Sridhar K (2013), ‘Design of Automobile Driveshaft using Carbon/Epoxy and Kevlar/Epoxy Composites’, American Journal of Engineering Reasearch (AJER) – Vol.-02, Issue-10, pp-173-179.

[3] Vicente Sanchez-Galvez, Laura Sanchez Paradela and Francisco Galvez (2014), ‘Analytical feint of dear speed impact onto lurcher glass/carbon epoxy composites targets’, Science Direct – Procedia Engineering 88(2014), pp.101-108.

[4] Bhushan K Suryawanshi and Prajitsen G Damle (2013), ‘Review of Design of Hybrid Aluminum/Composite Drive shaft for Automobile’, International Journal of Innovative Technology and Exploring Engineering (IJITEE) – Vol.-2, Issue-4, pp-259-266.

[5] Dr Andrew Pollard (1989), ‘Polymer Matrix Composites in Driveline Applications’, Journal of Composite Structures, Vol.25, pp- 165-175.

[6] Agarwal B D and Broutman L J (1990), ‘Analysis and deed of fiber composites’, John Wiley and Sons Inc.

[7] Rajanish M, Nanjundaradhaya N V and Ramesh S Sharma (2014), ‘An investigation on ILSS Properties of Unidiresctional; Glass Fibre Alumina Nanoparticles Filled Epoxy Nanocomposite at Different angles of Fibre Orientations’, Science Direct-Procedia Materials Science 10(2015), pp.555-562.

[8] Palanisamy P, Sureshkumar D and Balamurali S (2014), ‘Hybrid Design and Analysis of Advanced Propeller shaft’, International Journal of Engineering Sciences and Research Technology – Vol.-3, Issue-5, pp.123-127.

[9] Murugan R, Ramesh R and Padmanabhan K (2014), ‘Investigation on Static and Dynamic Mechanical Properties of Epoxy based Woven fabric Glass/Carbon Hybrid Composite Laminates’, Science Direct-Procedia Engineering 97(2014), pp.459-468.

SUPERIOR EQUIPMENT AND VARNISH OUTLOOK FOR GAS TURBINE APPLICATIONS

B.SAI VENKATA KRISHNA[1], Asst.Prof, Department of Mechanical Engineering.
E-mail: sai331988@gmail.com , Megha Institute of Engineering and Technology for Women, Hyderabad.

M.A.AZEEM[2], Asst.Prof, Department of Civil Engineering.
E-mail: azeem26jan@gmail.com , Megha Institute of Engineering and Technology for Women,Hyderabad

ABSTRACT:

The need for very specific hardness / hardness materials can only be satisfied by design concepts using reinforced compounds. Carbon fiber made of high-strength and durable silicon in a high-temperature titanium matrix is one of the main candidates whose development will be described. Design concepts for high-pressure turbines that incorporate thermal protection layers for ceramics, i.e. heat-insulating coatings, will go beyond the natural limits provided by the melting point of Ni-based super-turbine blades. The sophisticated design of the aerodynamic engine will focus on reducing specific fuel consumption and increasing the weight-to-weight ratio. In the end, this requires an increase in pressure ratios, as well as higher operating temperatures, and certainly poses a major challenge to the structural design and materials used. High-capacity materials for high temperatures are required, as are very light structures. Reducing the weight of the pneumatic dynamic motor requires a new compact compressor design with a few phases. Gas turbine blades are designed for cooling methods, as well as for cooling films in external cooling and thermal cooling in internal cooling. The turbine blade is designed with four-hole and six-hole heat cooling. The film cools the air in the blade through several small holes in the chassis. The current material used for the blade is chrome-plated steel. Here, it is replaced by composite materials of ceramic matrix and silicon carbide. Advanced evaporation treatment using electron beam technology is the preferred choice for manufacturing these coatings in highly rotating parts. However, considerable efforts are still needed to improve these coatings, make them more reliable, and thus achieve a philosophy designed to fully exploit their potential.

Keywords: *gas turbine, rotor blade, steady state thermal analysis, fiber-reinforced composites, thermal barrier coating.*

1. INTRODUCTION

During the design of the new aero engine, consideration should be given to the impact on key engine characteristics such as engine weight, specified fuel consumption (SFC), manufacturing costs and serviceability. For airlines, the direct operating costs of the aircraft are an initial qualifying parameter. Regardless of the specific fuel consumption, the propulsion / weight ratio is of great importance for military engines. The pay-to-weight ratio has been significantly improved over the years achieved by increased operating temperature and structural efficiency. It is very clear that advanced materials play a key role here. In fact, modern aircraft engines represent some of the most demanding and sophisticated building material applications in any engineering system manufactured today [1]. The first addresses recent developments in high pressure turbines / low pressure turbines and focuses on light titanium alloys, titanium aluminates and especially titanium matrix compounds (TMCs). The second covers the thermal barrier coatings (TBCs) of high pressure turbine blades that try to get rid of the bottleneck in developing improved performance engines. The excellent properties of titanium alloys include high specific strength and excellent corrosion resistance. Therefore, titanium alloys are found in aerospace applications where a combination of weight, strength, corrosion resistance and / or high temperature stability of light metal aluminum, high strength steel or nickel based super alloys is insufficient. In air motors, titanium alloys represent the most important class of engine compressor materials. The compressor blades were the first engine components made of titanium, and the titanium compressor discs are then introduced. The large front fan blades of modern jet engines are also often made of titanium alloy. Due to the constant increase in engine deviation rates, the latest blade designs exceed one meter in length. In these dimensions, the fan blade blade can become a serious problem because the blade edges can reach the speed of sound and cause acoustic / subsonic flow areas and associated shock waves. Advanced fan designs have improved blade stiffness by increasing column width and reducing the number of blades by about one third. Today, these "wide propeller blades" are used in the latest jet aircraft engines [2]. The new Benz (G 800) and GE / Pratt & Whitney Engine Alliance (GP7200) Airbus A380 engines will be about three meters in diameter and will include hollow titanium blades. Blisk technology is now standard in low and medium sized compressors for commercial and military engines. In the Eurofighter EJ200 engine, for example, the three stages of the fan section are of excellent design; the first two are manufactured by linear friction welding and the third by ECM.

Fig 1: Three stage blisk compressor

2. METHADOLOGY

This temperature limit for titanium alloys means that the hottest parts of the compressor, i.e., the discs and the blade in the later stages of the compressor, must be made of twice the weight of nickel-based super alloys. In addition, problems arise related to different thermal expansion behaviors and bonding techniques in the two alloy systems. Therefore, huge efforts are being made to develop a compressor made entirely of titanium. Titanium alloys that can be used at temperatures around 600 ° C or higher are required. This has been the motivation for intensive research and development in the field of titanium aluminize. These substances, based on intermetallic compounds α2 (Ti3Al) and γ (TiAl), have been studied for their ability to raise the application temperatures of titanium alloys to 650 ° C and 800 ° C, respectively. Excellent creep resistance is due to the organized nature of the crystalline structure. However, this structure also makes the intervals relatively fragile and difficult to distort. Alloys with Nb, Cr, V, Mn or Mo and microstructure optimization are two ways to increase ductility. Adequate tolerance for damage, pathological oxidation behavior and productivity (cost) are key factors that will determine the use of titanium acuminate in the aviation industry [3]. Due to the high reactivity of titanium alloys with SiC granules, manufacturing processes that occur with the least possible thermal load on the compound during manufacturing are preferred. Thus, processes based on vapor deposition and solid-state formability is considered. Today, the preferred route is fiber-coated matrix technology. The primary product is homogeneously coated matrix fibers allowing the manufacture of composite materials with excellent fiber coordination and precise matrix structure. Deposition of the magnetron spray layer from the vapor phase. Due to the high deposition rate, the electron beam vapor deposition (EB-PVD) is also used, but is limited to individual composition matrix alloys. In the second step, the matrix fibers are assembled or arranged using matrix, for example, winding techniques to achieve the desired geometry of components, encapsulated and then compressed at an even temperature at a constant temperature of about 950 ° C and pressures of about 2000. Tape. In the last step, the component is formed to its final geometry.

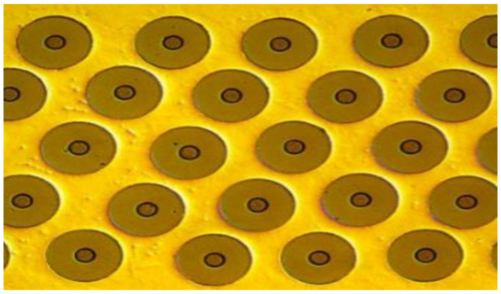


Fig 2: Titanium matrix composite

3. LITERATURE SURVEY

PAPER 1- Design and Analysis of Gas Turbine Blade by Theju V, Uday P S , PLV Gopinath Reddy , C.J.Manjunath

The purpose of this project is to design and analyze turbine pressure. Research is required to use new materials. In the present work, the turbine blade is designed with different materials, Inconel 718 and Titanium D-6. Try to check the effect of temperature and pressure caused by the turbine code. Thermal analysis was performed to verify the direction of temperature flow due to convection. Structural analysis was performed to investigate the effect of the combination of thermal loads and centrifugal effects and shear forces and turbine plate displacement. We have tried to suggest the best fit for the turbine index by comparing the results obtained from two different substances (Inconel 718 and T6 Titanium). Based on the designs and results, Inconel 718 can be considered the best economical material and has good physical properties at higher temperatures compared to titanium T6.

PAPER 2- Heat Transfer Analysis of Gas Turbine Blade Through Cooling Holes by K Hari Brahmaiah , M.Lava Kumar

In advanced gas turbines, the operating temperature of the turbocharger operates above the melting point of the blade material. An advanced cooling system should be developed to ensure the safe and continuous operation of high performance gas turbines. Different ways of cooling the blades are offered. One of these techniques is the presence of radial holes at high air cooling speeds along the plate. Heat transfer analysis of gas turbines was performed on four different sample blades without holes (5, 9 and 13) with holes. Reinforced wall). For heat transfer graphics and overall distribution, the code is optimized for 13 holes. A consistent and consistent analysis is performed using ANSYS with different chrome-plated steel blades and Inconel 718. Comparison of these materials shows that although Inconel 718 has excellent thermal properties, the induced stress is lower than chromium steels.

PAPER 3 - Film Cooling of the Gas Turbine Endwall by Discrete-Hole Injection by M. Y. Jabbari, K. C. Marston, E. R. G. Eckert and R. J. Goldstein

The film's cooling performance is tested for injection through separate openings at the end of the turbine blade. Efficiency is measured at about 60 sites in the injection area. Three nominal hit rates, two density rates and two Reynolds flow numbers are examined. Data analysis reveals that up to 60 locations are insufficient to determine the extent of the film's cooling effect with its strong local variations. Viewing the effects of cooling jets on the perimeter wall using ammonium diazo paper provides useful quality information for interpreting measurements, pathways and jet responses that change at fast speed and flow rate. intensity.

4. RELEATED STUDY

In general, composite materials are highly dependent on the properties of the solvent. Understanding the interaction between them forms the basis for physical evolution. The specificity of embedded properties always depends on the stability of these reactions. Long-term durability and robustness to the highest service temperatures are definitely a highlight of the enhanced TMCs. Energy, or more energy, energy associated with mass, is an important factor for structural simplicity and is a valuable property of great importance in design [4]. Dynamic aerospace and TMC data are shown in Figure 6 for temperatures up to 700 ° C. Although they are strong (or medium), the aluminum alloys, α2Ti3Al or mushrooms (Ti2AlNb), al-alumate, and suparalloysi (here 718) cover 10 to 25 km at room temperature and from 10 to 15 km at 800 ° C, The maximum TMC strength ranges from 40 to 60 km at room temperature and up to 50 km at 800 ° C. Maximum strength depends on the fiber part. A maximum of 40% electrical fracture was obtained. For SiC / TIMETAL 834, the maximum heat resistance of 2400 MPa is recoverable, which is well consistent with the calculations according to chemical law. The use of TMCs in advanced applications for advanced jet engines requires less energy; if the behavior is weak under heavy load, it is one of the most important criteria. Figure 7 shows the nonlinear force with reinforced SiC fibers 5 Christoph Leyens TIMETAL 834 under friction jump at temperatures up to 700 ° C. The TMCs have a positive effect, particularly at 700 ° C where the fiber strength is high in conductivity. The periodic pressure levels of TMC antagonists in LCF (low fatigue state) and HCF (high fatigue condition) were more than 100% greater than nonlinear ones. Although the maximum continuous closure is about 400 MPa, SiC / Ti reaches a fixed limit of more than 1000 MPa at 700 ° C.

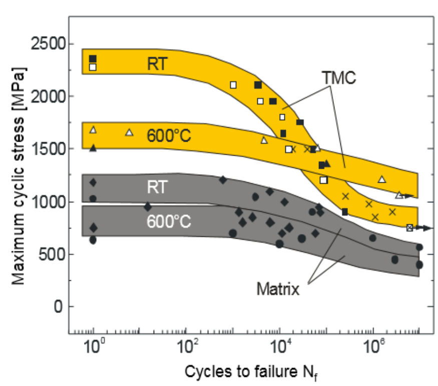


Fig. 3: Fatigue strength of TMCs in comparison

5. APPLICATIONS

The future development of natural gas is clearly aimed at raising the temperature of the turbine marine to more than 1700 ° C. There is no doubt that this ambitious goal can be achieved through by using non-economical televisions or by high temperature applications, in particular by expanding the use of heat exchangers (EB-PVD) using electromagnetic fields (TBCs).). Chlorofluorocarbons (CFCs) consisting of hot-tempered layers with reduced thermal conductivity - usually zirconia-stabilized particles - are used in the air and only in powder form skin. The paint showed good consistency of cement in the material [5] [6]. The TBCs application allows for increased engine / performance by increasing the temperature or reducing air conditioning. On the other hand, it is possible to extend the life of turtles by reducing the heat of iron as shown. During the process of EB-PVD, the strong electric field is melted and the cement material is transferred into a vacuum chamber. The cells are inserted internally during the healing process to ensure continued TBC. To achieve the measurement of specific elements of zirconium, the amount of oxygen is compromised in the discharge chamber. Intravenous reservoirs are placed on a storage tank at a rate of 3 to 30 microns / min. Specialized and polished microspheres provide smooth surfaces without the need for final drying or cooling. Due to the microscopic structure, the life of TBCs is prolonged and impairs tolerance. Characteristics and many advantages of TBCs compared to heat exchanger and evaporator featuring EB-PVD TBC on a steam engine powered by DLR using 150kW von Ardenne EB-PVD.

6. CONCLUSION

Mechanical devices, including high-strength, high-strength and high-strength fiber-reinforced titanium matrix materials, are ideal for high-tech applications, for example in turbine engines. Due to high costs and lack of knowledge of real estate, their use is severely restricted by applications. The following procedures should enhance TMC's ability to increase acceptance in the manufacturing market. Although most TMC applications today focus on temperature, the future of TMC is likely to be a future at high temperatures. For large-scale heat exchangers such as power plants, Eb-PVD shows the highest potential for increasing turbine power. TBCs represent a complex process involving (at least) attachments, bandages, heat transfer heaters and top cover. Each species can affect the life cycle of tuberculosis through a strong immune system. Complex and commonly used conditions, thermal, mechanical and mechanical conditions of the equipment, including heat transfer and cyclic effect, aggravate the situation.

7. REFERENCES

[1] J.C. Williams, The Development of Advanced Gas Turbines: the Technical and Economic Environment, Proc. Materials for Advanced Power Engineering 1994, ed. D. Coutsouradis, et al., Kluwer Academic Publishers (1994) 1831-1846

[2] M. Peters, J. Kumpfert, C.H. Ward, C. Leyens, Titanium Alloys for Aerospace Applications, in: Titanium and Titanium Alloys, Advanced Engineering Materials 5 (2003) 419-427

[3] F. Appel, M. Oehring, Gamma-Titanium Aluminide Alloys: Alloy Design and Properties, in: Titanium and Titanium Alloys, C. Leyens, M. Peters (eds.), 89152, Wiley-VCH, Weinheim, Germany, 2003

[4] H. Kestler, H. Clemens, Production, Processing and Applications of γ(TiAl)-Based Alloys, in: Titanium and Titanium Alloys, C. Leyens, M. Peters (eds.), 351-392, Wiley-VCH, Weinheim, Germany, 2003

[5] R. Leucht, H.J. Dudek, Properties of SiC-fibre reinforced titanium alloys processed by fibre coating and hot isostatic pressing, Materials Science & Engineering A188 (1994) 201-210

[6] C. Leyens, M. Peters, W.A. Kaysser, Oxidation Resistant Coatings for Application on High Temperature Titanium Alloys in Aeroengines, Advanced Engineering Materials 2 (2000) 1-5

THERMAL ANALYSIS OF MODIFIED COMBUSTION CHAMBER OF SPARK IGNITION ENGINE

BANOTH RAJENDRA PRASAD
MallaReddy College of Engineering, Masimmaguda, Hyderabad 500014, Telangana state

N. KISHORE KUMAR
MallaReddy College of Engineering, Masimmaguda, Hyderabad 500014, Telangana state

V.SRINU
MallaReddy college of Engineering, Masimmaguda, Hyderabad 500014, Telangana state

ABSTRACT

Efficiency of an internal combustion engine can be increased by increasing the heat energy generation inside the combustion chamber without effecting the performance of lubricating oil and lessening the strength of the piston. The main aim of this paper is to determine temperature distribution in a four-stroke, single-cylinder, water cooled, variable compression ratio (3-9), variable speed (2200-3000 rpm) spark ignition engine with brake power of 2.2 kW at a speed of 3000 rpm with copper coated combustion chamber (CCE) [copper-(thickness, 300 µ) was coated on piston crown, inner side of liner and cylinder head] and compared the engine, with conventional combustion chamber (CE) with neat gasoline operation. Copper coated power piston, copp.

Keywords: Copper Coating, Piston, Liner, Solid Works, Ansys Workbench, Thermal Analysis.

1. INTRODUCTION

With advent of urbanization, energy consumption is increasing drastically, out of which gasoline energy utilization is foremost as transportation development is fast. Though alcohols are important substitute for gasoline to overcome the fossil fuel crisis, it plays the major contribution for the exhaust pollutants in SI engine due to incomplete combustion of fuel [1]

Engine modification with copper coating on crown of the piston and inner side of cylinder head improves the engine performance as the copper is a good conductor of heat, stabilizes flame, improves pre–flame reactions and turbulence [2-5]. Temperatures inside the combustion chamber are very high and the heat generated is transferred to different components of power piston ,liner and cylinder head such that the need for the study of temperature distribution across the components is a must, especially to provide proper lubrication between liner and power piston and cooling to the walls of combustion temperature [6-10]. Finite Element Analysis is applied in such situations and thereby temperature distribution across the components is determined by using ANSYS programme which employs finite element-based software

1. MATERIALS AND METHODS

1.1 Making of copper coated combustion chamber

In the copper coated engine, top surface of the power piston, top surface of auxiliary piston an inside of liner are coated with copper using Twin wire spray gun. For 100µ thickness, nickel-cobalt-chromium bond coating was sprayed. On this coating, for another 300µ thickness, an alloy of copper (89.5%), aluminum (9.5%) and iron (1%) was coated with a MEC (Trade name of the company) flame spray gun. The bond strength of the coating was so high that it does not wear off even after operating it for 50hrs continuously.

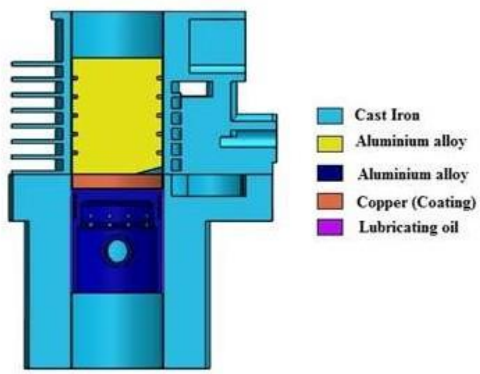
The specifications of the engine are given below

ENGINE SPECIFICATIONS

1 Type: Four-Stroke, Single Cylinder, Variable

2 Make: Greaves Limited

3 Rated Power:2.2 KW at 3000rpm



4 Bore and stroke:70mm x 66.7mm

5 Speed:2200rpm-3000rpm

6 Compression Ratio: 3:1 to 9:1

7 Spark Plug: Make: MICOBOSCH

8 Spark Plug Gap:0.6mm

9 Type of Ignition: Battery

10 Specific fuel consumption:500 gm/h KW

11 Lubricating Oil: SAE-40

12 Dynamometer: Eddy Current Dynamometer loading Rheostat

13 Temperature and Speed: By Digital Indicators

14 Starting: Auto Start by DC Motor

15 Cylinder Pressure: By Sensor, range :5000PSI

16 Exhaust Gas Calorimeter: IND-LAB Make

17 Torque arm distance:200mm

18 Orifice Diameter:20mm

19 Recommended Spark Ignition Timing: 25

2.2. Measurement of Temperature distribution of engine components

In the present scenario, steady state thermal analysis is done on the assembly of four stroke variable compression spark ignition engine to calculate temperature distribution, heat fluxes, temperature gradients and amount of heat lost or gained in engine components for different versions of the engine. Steady state thermal analysis is done in two main steps. First Geometric Modelling is done then Finite Element analysis is done.

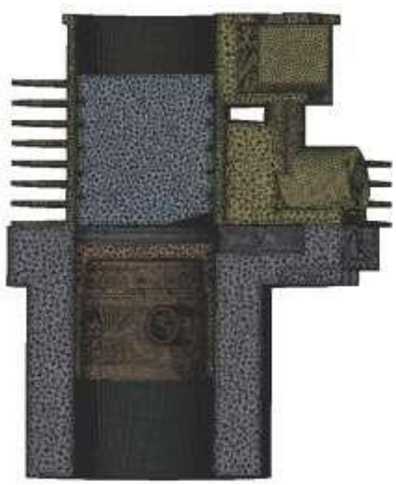
2.2.1. Solid Model Creation: In Geometric Modelling, 3-D geometry of power piston, auxiliary piston, liner and cylinder head are created. The models are generated using SOLID WORKS 16**.** The components are assembled by giving constraints to obtain the final assembly. The assembly is saved as. IGS file. The file is imported to ANSYS workbench15.

Figure 1 shows the Configuration of the assembly with cu coating piston, liner and cylinder head and showing different materials in various zones

Figure 1 Configuration of the assembly witcu coating piston, liner and cylinder head showing different materials in various zones

2.2.1. Meshing the model: In finite element modelling, mesh was generated with tet10 element type using ANSYS WORKBENCH 15. Since the geometry is complex, free style of meshing is employed. Fine mesh size is considered at small and critical components, while coarse mesh is considered for the remaining components.

Figure-2 Mesh employed in the thermal analysis for the assembly piston, liner and cylinder head with cu coating.



2.2.3 Boundary Conditions Application and Thermal Analysis Solution: ***The boundary conditions for the present problem are obtained from [11]***

The top surface of the power piston is applied with a convective heat transfer (h_c) of 250 w/m^2 k and a bulk temperature (T) of 920 ^{0} C and on the water jacket side of the liner, $h_{c=}$ 1800 w/m^2 k and T=60 ^{0} C and on the cylinder head fins of h_c =120 w/m^{2}k, T = 60 ^{0} C.

Steady state thermal analysis was solved after application of boundary conditions

3. RESULTS AND DISCUSSION

3.1 Thermal Analysis Results

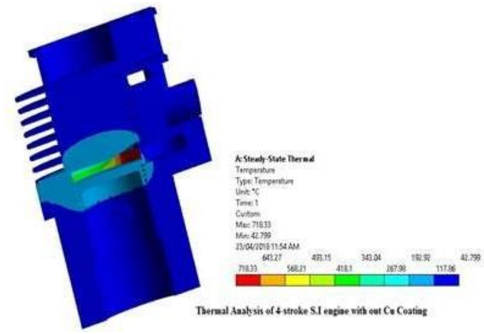


Figure 3 Isotherms of thermal analysis for the assembly of piston, liner and cylinder head of CE

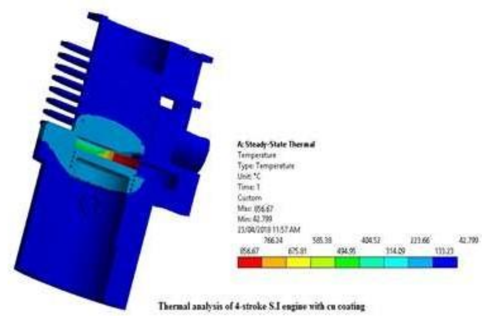
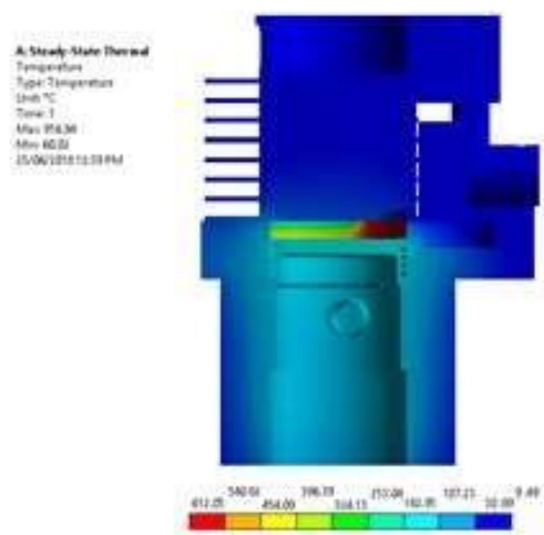


Figure 4 Isotherms of thermal analysis for the assembly of piston, liner and cylinder head of CCE

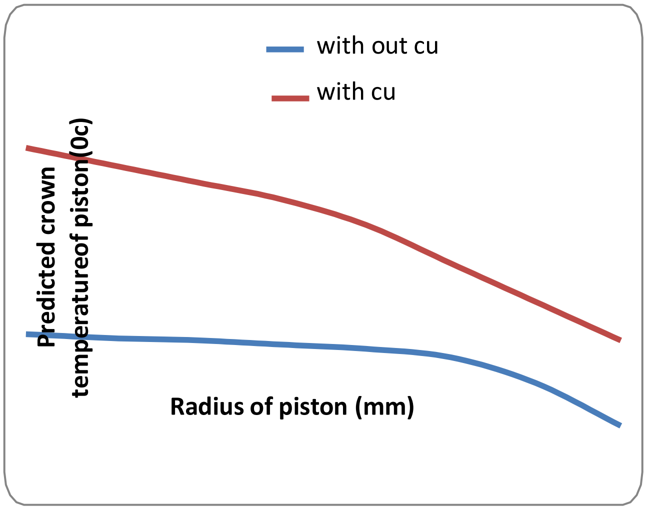


Prediction of the Temperature along the Piston Radius for the Base Engine and Catalytic Coated

Engine

The fluctuation in the crown temperature of the piston predicted by FEM analysis with radius of the piston for CE and CCE was shown in Figure-5

Figure 5 Variation of the predicted crown temperature with radius of the piston for CE and CCE.



From the Figure 5 it was noticed that, as the radius of piston increases, crown temperature of the piston decreases marginally for both CE and CCE. The temperature decreases at the outer periphery of the piston, as it is cooled by means of lubricating oil and also with the presence of fins. The temperature of the piston of CE is 181^0C while it is 225^0C for the piston of CCE at the crown surface. At the outer periphery of the piston, the temperature decreases to160^0C for CE and 180^0C for CCE. The temperature drop for the piston of CCE from the crown to the outer periphery was less as the piston is coated with copper, which has high thermal conductivity and hence thermal resistance was less.

Figure 6 Heat flux in the assembly of piston, liner and cylinder head of CE

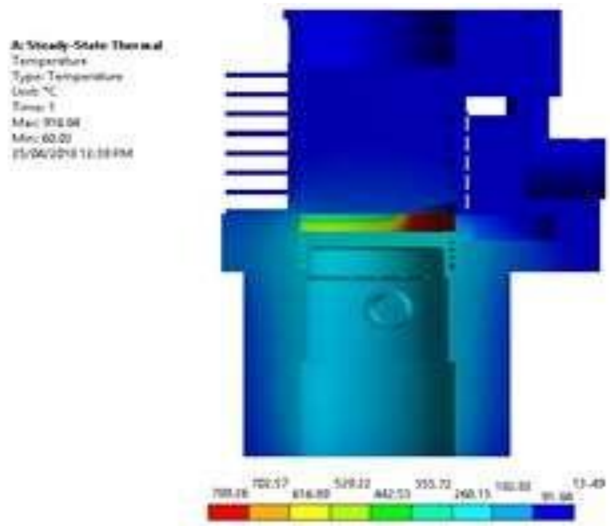


Figure 7 Heat flux in the assembly of piston, liner and cylinder head of CCE

Prediction of the Heat Flux along the Radius of the Piston in Base Engine and Catalytic Coated Engine

The variation of the percentage (%) increase in the heat flux in the piston of CCE over that of CE with its radius was shown in the Figure-8.

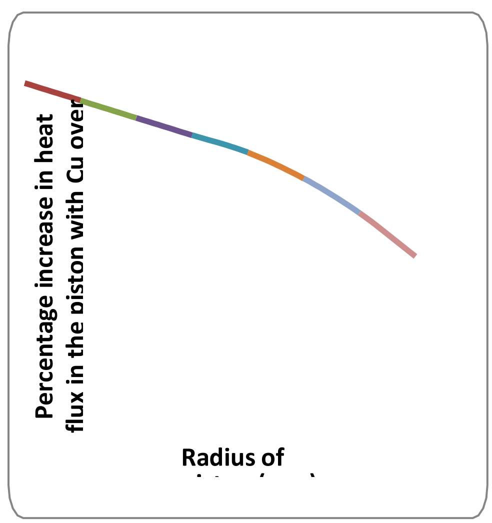


Figure 8 Variation of the % increase in the heat flux with radius of piston in CCE over CE.

As the radius of the piston increase, a marginal decrease in the heat flux(from19.6 % to 17.6 %) was noticed in the Figure-8. Since copper coating was done on the piston crown, maximum heat flux was concentrated at the centre of the piston crown. However, at the outer radius of the piston, heat flux was marginally lower, as the outer periphery of the piston is subjected to cooling by means of lubricating oil and fins.

The temperature at the inner side of the liner and cylinder head determined by FEM analysis were compared with the results obtained by experimentation, in order to ascertain the deviation of FEM results from experimental results.

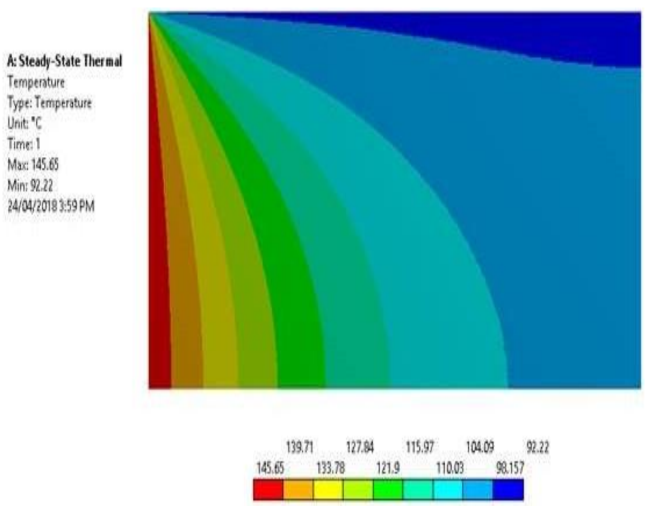


Figure-9,10 shows the isotherms of the lubricating oil between the piston and inner surface of liner of CE and CCE respectively

Figure 9 Isotherms from the finite element analysis in the lubricating oil between piston and inner surface of liner for CE

Figure 10 Isotherms from the finite element analysis in the lubricating oil between piston and inner surface of liner for CCE

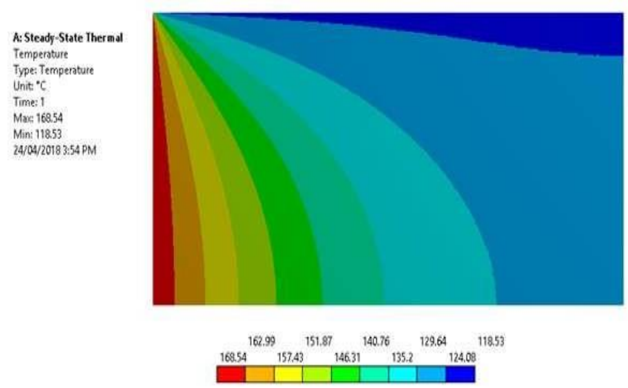
From the Figures-9,10 the temperatures of the lubricating oil varied from 92^{0}C to 145^{0}C for CE, while it varied between 118^{0}C to 168^{0}C for CCE respectively and found that temperatures are at safe limits for which lube oil does not deteriorate. The temperature limit for which lube oil does not deteriorate is 180^{0}C [12].

4. CONCLUSIONS

- Temperature at the top surface of the copper coated piston was 225^{0}C which is higher than 181^{0}C for the conventional engine.
- Heat flux is increased by17-19% along the radius of piston for the copper coated engine compared to conventional engine.
- Copper coating on the piston will not deteriorate the lubricating oil temperature as the lube oil temperature was in the safe limits between 118^{0}C to 168^{0}C,while it varied between 92^{0}C to 145^{0}C for conventional engine.

ACKNOWLEDGEMENTS

Authors thank authorities of Chaitanya Bharathi Institute of Technology, Hyderabad for facilities provided.



REFERENCES

[1] Khopkar, S.M., 2010, “Environmental Pollution Analysis,” New Age International (P) Limited, Publishers, New Delhi, pp.180-190.

[2] Murali Krishna, M.V.S., and Kishor,K., 2008, “Performance Of Copper Coated Spark Ignition Engine With Methanol–Blended Gasoline With Catalytic Converter, “Indian Journal (CSIR) of Scientific and Industrial Research, 67,July,543–548.

[3] Kishor, K., Murali Krishna, M.V.S., Gupta, A.V.S.S.K.S., Narasimha Kumar, S., and Reddy, D.N, 2010, “Emissions From Copper Coated Spark Ignition Engine With Methanol Blended Gasoline With Catalytic Converter,” Indian Journal of Environmental Protection,

30(3), pp.177–183

[4] Murali Krishna, M.V.S., Kishor, K., Murthy, P.V.K., Gupta, A.V.S.S.K.S., and Narasimha Kumar, S, 2010, ".Performance Evaluation Of Copper Coated Four Stroke Spark Ignition Engine With Gasohol With Catalytic Converter," International Journal of Engineering Studies, 2(4),pp.465–473.

[5] Murthy, P.V.K., Narasimha Kumar, S., Kishor,K., and Murali Krishna,M.V.S., 2011, "Aldehyde Emissions From Two-Stroke And Four-Stroke Spark Ignition Engines With Catalytic Converter Running On Gasohol," International Journal of Fuels and Lubricants, 50(2), pp.137-156.

[6] Murali Krishna,M.V.S., Kishor, K., Murthy, P.V.K., Gupta, A. V S. S. K .S. ,and Narasimha Kumar, S.,2011, "Comparative Studies On Performance Evaluation Of Four Stroke Copper Coated Spark Ignition Engine With Catalytic Converter With Alcohols,". International Journal of Advances in Engineering Research, 2(6), pp.*1–11.*

[7] V.R.Rajagopalan., S.Anand., D.Nagendra Kumar., V.Karunaharan., V.Lakshmi Narsimham.,"Investigations of In-Cylinder Heat Flux in a Single Cylinder, 4 Stroke, Air cooled, Spark Ignition Engine for Motor Cycle Application", SAE paper 320804,2015.

[8] Thet T.Mon, Rizalman Mamat, Nazri kamsah, "Thermal Analysis of SI-Engine using Simplified Finite Element Method", Proceedings of the world congress on engineering, Volume III,WCE- 2011.

[9] Murali Krishna,M.V.S., Kishor, K., Murthy, P.V.K., Gupta, A. V S. S. K .S. , and Narasimha Kumar, S.,2011, "Comparative Studies On Performance Evaluation Of Four Stroke Copper Coated Spark Ignition Engine With Catalytic Converter With Alcohols,". International Journal of Advances in Engineering Research, 2(6), pp.*1–11*

[10] Muhammet Cerit,Hakan S.Soyhan.,"Thermal analysis of a combustion chamber surrounded by deposits in an HCCI engine",Applied Thermal Engineering,volume-50,81-88,2013

[11] Kishor, k., "Investigations on Catalytically Activated Two Stroke Spark Ignition engine with alcohol blended gasoline", Ph.D. Thesis,J.N.T.university,Hyderabad,2014

[12] Heywood,J.B., "Internal Combustion Engine fundamentals", Thermo-chemistry of fuel air mixtures, Properties of working fluids ,Mc Graw-Hill Book Company, New York,1988.

ABOUT MRGI

Malla Reddy group of Institutions is one of the biggest conglomerates of hi-tech professional educational institutions in the state of Telangana, established in 2001 sprawling over 200 acres of land. The group is dedicated to impart quality professional education like pharmacy, Engineering & Technology, MCA, MBA courses. Our sole objective is to turn out high caliber professionals from those students who join us.

ABOUT MRCE

Malla Reddy group of Engineering (Formerly CM Engineering College) has been established under the aegis of the Malla Reddy Group of Institutions in the year 2005, a majestic empire, founded by chairman Sri Ch.Malla Reddy Garu. He has been in the field of education for the last 23 years with the intention of spearheading quality education among children from the school level itself. Malla Reddy College of Engineering has been laid upon a very strong foundation and has ever since been excelling in every aspect. The bricks of this able institute are certainly the adept management, the experienced faculty, the selfless non-teaching staff and of course the students.

ABOUT ICTIMES

ICTIMES started long back with its banner to promote the vision of future technologies that change the trends of life on this planet earth. Under this banner, the Department of Mechanical Engineering at MRCE organizes the ICFTME – International Conference on Future Technologies in Mechanical Engineering to provide a scholarly platform to ignite the spirit of Research and bring out the latent potential in teaching fraternity and student community. ICFTME accommodates major areas like, Applied Mechanics, Tribology, Advanced Manufacturing Process, Fluid Dynamics etc.,

ABOUT ICFTME

International Conference on Future Technologies in Mechanical Engineering (ICFTME-2019) will bring together innovative academicians, researchers and industrial experts in the field of Mechanical Engineering to a common forum. The idea of the conference is, for the scientists, scholars, engineers and students from the Universities across the world and the industry as well, to present ongoing research activities, and hence of foster research relations between the Universities and the industry with the rapid development of trends and studies in the fields concerned. ICFTME-2019 will provide a heartwarming platform to researchers, scholars, faculty and students to exchange their novel ideas face to face together.

Estd :2005

MALLA REDDY COLLEGE OF ENGINEERING

Approved by AICTE - New Delhi, Affiliated to JNTU - Hyderabad,Accredited by NBA & Accredited by NAAC.
ISO 9001:2015 Certified Institution, Recognition of College under Section 2(f) & 12 (B) of the UGC Act,1956.

Address: Maisammaguda, Dhulapally,(Post Via Kompally),Secunderabad - 500 100.
Ph: 040-64632248, 9348161222 ,9346162620. Email: principal@mrce.in
Website : www.mrce.in