iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** VI **Month of publication:** June 2017

DOI:

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Contact Stress Analysis and Stress Optimization of Spur Gear

Rahul Kamble[1], Prof. S.U. Ratnaparkhi[2]
[1,2]*Department of Mechanical Engineering*, RMD Sinhgad School of Engineering

Abstract: ***The gears for transmission develop stresses at the mating positions over the teeth. A pair of spur gear teeth is generally subjected to two types of cyclic stresses as bending stresses and contact stresses. These stresses are proportional to the amount of power transmitted while the design could offer favourable or adverse conditions for generations of the same. This dissertation work would identify the magnitude of the stresses for a given configuration of a gear transmitting power while trying to find ways for reducing stress and weight of the gear. Magnitude of mass and stress optimization could be the key parameters of assessing this work.***
Keywords: Spur gear, bending stress, contact stress, weight*.*

I. INTRODUCTION

Spur gears are widely used for Metal cutting machines, power plants, mechanical clock and watches, washing machines, Automobile gear boxes, Steel mills, Rolling mills etc. The literature survey concludes that study has been done to increase the performance of the spur gear by increasing the strength of the gear by using various materials like Structural steel, Grey cast Iron, Aluminium Alloy, Epoxy E Glass Unidirectional (UD). It also gives the idea to reduce stress distribution, deformation and weight of spur gear.

When the gears while transmitting the power develop stresses at the mating positions over the teeth. These stresses are proportional to the amount of power transmitted while the design could offer favorable or adverse conditions for generation of the same. Toothed gears are used to transmit the power with high velocity ratio. During this phase, they encounter high stress at the point of contact. This dissertation work identifies the magnitude of the stresses for a given configuration of a spur gear while transmitting power also trying to find ways for reducing stress, deformation and weight of the gear.

II. LITERATURE REVIEW

In the research paper "Study of Stress Relief Features of tooth of spur gear" by Dhavale A. S. , Abhay Utpat, explained that the stresses can be minimized by introducing stress relief features at stress zone. Stress redistribution is highly sensitive to the change in size, location and number to select size, Location and Number very carefully. Using two holes as a stress relieving features gives more stress reduction

In the research paper "Study of stress relieving features in Spur gear" by Sujit R. Gavhane, Prof. S.B. Naik explained stress relieving features for spur gear have been studied with single hole stress relieving technique by using FEA. Results of this study provides how to minimize stresses developed in spur gear. The stress features like circular holes are being used for many years to reduce the stresses in the components. .In the research paper "Spur gear tooth stress analysis and stress reduction using geometrical features" by M. Pramod Reddy, M. Santhosh. They used holes drilled across the entire tooth as a function of size and location. The ultimate objective of this work was to find the overall effect of hole size and location on the critical stresses in the gear.

III. METHODOLOGY

The objective of this paper Review the existing design and consider improvement for reducing stress concentration at the affected areas of the gear. Analyze/ optimize further for realization of the objective.

A. Modeling of Geometry

To design geometry of Spur Gear CATIA v5 R20 software is used with specification is considered for study. These all design is done with command pad and pocket. All this design of Gear and pinion are converted to iges format so as it will be export to Hypermesh software. After design iges files are imported to Hyper mesh workbench.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

B. Material Properties

Material properties may be orthographic or an isotropic. In this case material of the Gear is EN 9 having Young's Modulus 210 MPA, Density 7850Kg/m3 and Poisson's Ratio is 0.3 the analysis being linear modal and complete subassembly is made of the same material. It is chosen as constant and the same is entered through the menu path

C. FEM mesh generation

Fine meshing is key step in finite element analysis .In this processes the element is subdivided into many elements and network of nodes and elements are formed in current there are many different types of element each element has its own features & degrees of freedom for three-dimensional solid geometry software generates tetrahedron shaped (using auto mesh more accurate than quad mesh).quadratic element (manual meshing more accurate than tetrahedral meshing)due to fine meshing we get more accurate result (the accuracy of mesh depends on how close or how smaller it is)but it takes longer time. in coarse mesh size is generally taken large so the result obtained quickly but with less accuracy

D. Experimental Work

With the use of experimental set-up we can identify the deflection & stiffness for the component. Typically, UTM machine is used for determining the same. The software analysis can yield the stresses, Stiffness which in turn, can be verified further using the testing equipments for this purpose (future scope). The computational approach will give results more close to practical values through simulation/ analyses. The technique would deploy any of the following software tools: Patran/Nastran, ANSYS, Abaqus(3D) or any compatible CAE software in the Structural/ NVH domain.

E. Theorotical Calculation

Lewis introduced an equation for estimating the bending stress in gear teeth in which the tooth form entered into formulation. The beam strength of gear teeth is determined from an equation known as Lewis Equation and load carrying ability of toothed gears as determined by this equation gives satisfactory results. The load is transmitted from one gear to another it is all given and taken by one tooth because it is not always safe to assume that the load is distributed among several tooth when contact begins the load is assumed to be at the end of driving teeth. Because the load may be distributed among several teeth but it is almost certain that at some time during the contact of teeth the proper distribution of load does not exist and one tooth must transmit the full load.

Lewis Equation is given as

$\sigma_w=(W_T*h*6)/bt^2$

where,

6_w= Bending stress in N/MM^2

W_T=Tangential load acting at tooth in N

h= Height of tooth in MM

b= Width of gear face in MM

t=Thickness of gear tooth in MM

Contact stress Analysis in Pinion.

Power = 3.7 kw , N= 540rpm , W= 2111.32 N , h=5.046 b=21.771 mm

t=2.29 mm

$$\sigma_w = (W_T*h*6)/bt^2$$

$$\sigma_w = \frac{2111.32 * 5.046 *6}{21.771*2.29^2}$$

$$\sigma_w = 244.49\ N/mm^2$$

Contact stress Analysis in Gear

Power = 3.7 kw , N= 540rpm , W= 2215.32 N , h=5.046 b=21.771mm t=2.29 mm

$$\sigma_w = (W_T*h*6)/bt^2$$

$$\sigma_w = \frac{(2215.32*5.046*6)}{(21.771*2.29^2)}$$

$$\sigma_w = 256.62\ N/mm^2$$

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

IV. FEA STUDY

A. Finite Element Method

FEM is a numerical technique for finding approximate solutions to boundary value problems. A boundary value problem is a differential equation together with a set of additional restraints, called boundary conditions. The FEM uses different method to minimize an error and produce a stable solution. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses all the methods for connecting many simple element equations over many small sub domains, named finite elements, to approximate a more complex equation over a larger domain. The subdivision of a whole domain into simpler parts has several advantages:

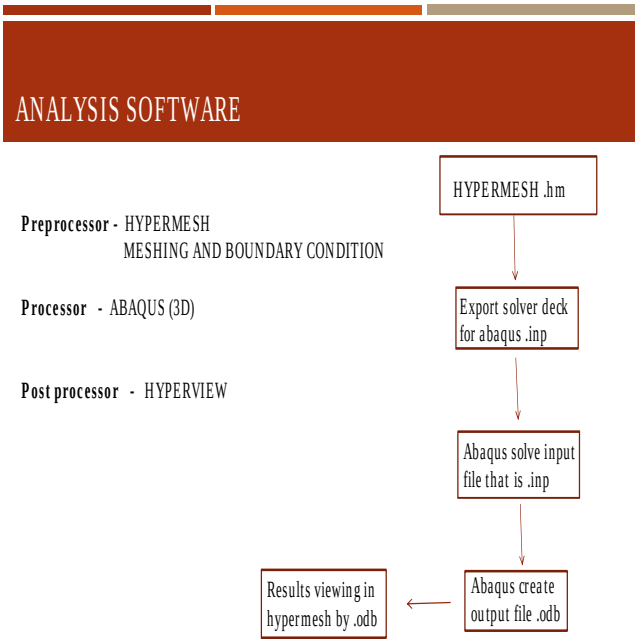
Accurate representation of complex geometry. Inclusion of dissimilar material properties .Easy representation of the total solution. Capture of local effects.

A perticular work out of the method involves dividing the domain of the problem into a collection of sub-domains, with each sub-domain represented by a set of element equations to the original problem, followed by systematically recombining all sets of element equations into a global system of equations for the final calculation. The unique system of equations has known solution techniques and can be calculated from initial values of the original problem to obtain a numerical answer.

A feature of FEM is that it is been proven to be numerically stable for linear static analysis, meaning that errors in the input and intermediate calculations do not match and cause the resulting output to be meaningless. Moreover, it can be easily applied on complex geometries. The element equations are simple equations that locally approximate the original complex equations to be studied, where the original equations are partial differential equations (PDE). To explain the approximate solution in this process, FEM is commonly used as a special case of Galerkin Method. In simple terms, it is a procedure that minimizes the error of approximation by fitting into the PDE.

B. Analysis Software

Import CAD generated geometry or finite element model information, Import multiple Hypermesh databases, models to existing display, Import connector files, Export CAD geometry or finite element information for specific analysis codes, Run a Hypermesh command file & export solver deck for ABAQUS (3D) (Processor) so that solve input file of Abaqus after that it creates an output file and rsults viewing in Hyperview.



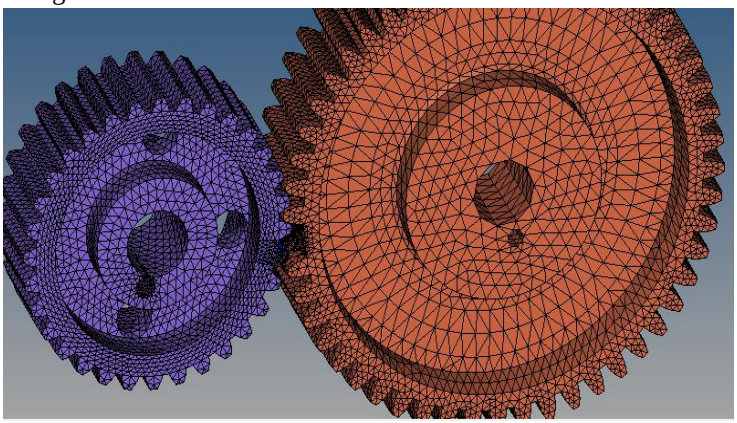
C. Analysis software

1) Abaqus (3D): It is a software suite for the finite element analysis & computer aided engineering. It is a software application used

Volume 5 Issue VI, June 2017
ISSN: 2321-9653

for both modeling & analysis of mechanical components & assemblies (pre-processing) & visualizing Finite element modeling result. A general purpose finite element analyser that employs implicit integration scheme(Traditional).A special purpose finite element analyser that employs explicit integration scheme to solve highly non-linear systems with many complex contacts under transient load. It is used in automotive, aerospace & industrial products industries the product is popular with academic and research industry due to positive wide material modeling capability and program ability to be customized it is also provide a good collection of multiphysics capabilities such as coupled acoustic-structure, making it attractive for production level simulators where multiple fields need to be coupled. In Hypermesh meshing of that part takes place after that export deck for abaqus after that abaqus solve input file and it creates output file and viewing results in Hyperview.

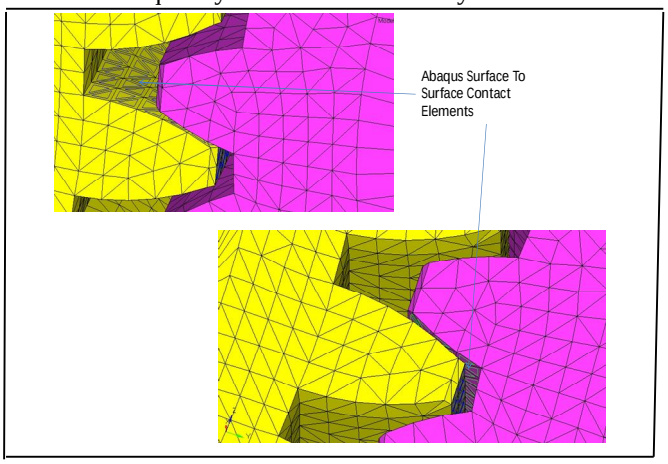
2) Hypermesh: Hypermesh is a three-dimensional finite element mesh generator with a build-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities. Hypermesh is built around four modules: geometry, mesh, solver and post-processing. Geometry module is used to define geometrical objects such as points, lines, surfaces and volumes while mesh module is used to create mesh (nodes and element topology). All instructions of the modules are prescribed either interactively using the graphical user interface (GUI) or in text files using Hypermesh own scripting language. Different types of meshing techniques apply for meshing but in this case type of meshing is Tetra meshing.



Meshing of Gear & Pinion

D. Fine meshing of Gear & pinion

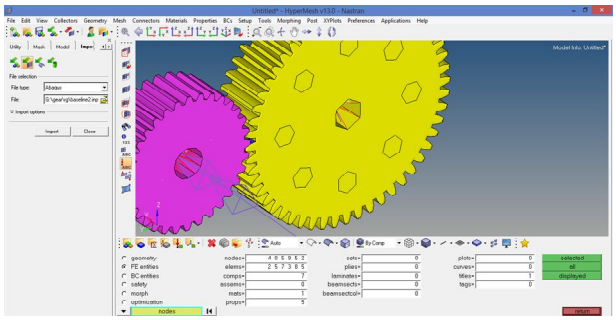
In below figure shows that fine meshing of Gear & Pinion tooth here we consider one tooth gear & pinion it will produce same result of all teeth. Fine meshing is key step in finite element analysis .In this processes the element is subdivided into many elements and network of nodes and elements are formed in current there are many different types of element each element has its own features & degrees of freedom for three-dimensional solid geometry software generates tetrahedron shaped (using auto mesh more accurate than quad mesh).quadratic element (manual meshing more accurate than tetrahedral meshing)due to fine meshing we get more accurate result (the accuracy of mesh depends on how close or how smaller it is)but it takes longer time. in coarse mesh mesh size is generally taken large so the result obtained quickly but with less accuracy.



Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

E. Contact stress Analysis-Element Count

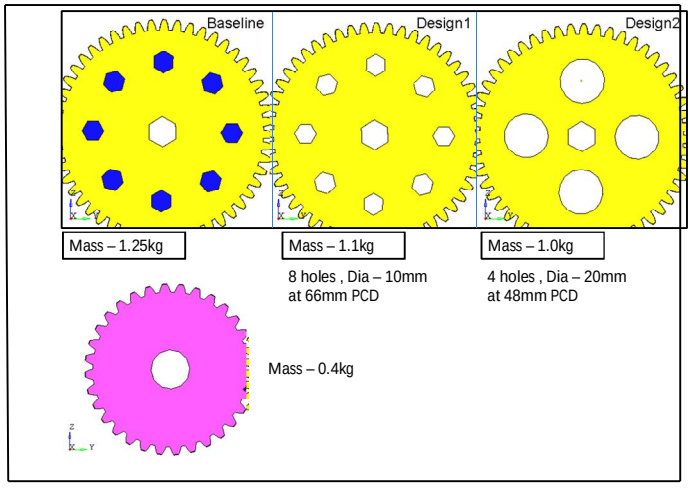
In FEA modeling element quality greatly effects the accuracy of analysis results. many modern finite element analyser solvers have routines to compensate for some measure of poor quality element but it is not a good practise to rely on these compensations. the FEA modeler must take into consideration element quality and thereby judge whether the analysis results are meaningful. The ideal four node (quad) plate element is a planner square two types of error can result from translating a single node. if one of the nodes is translated in the plane remaining nodes, interior angles change & edge lengths vary between sides introducing skew and aspect ratio into the element if one of the nodes is translated out of the planes of others result is warpage with first order tria elements warpage is not possible but aspect ratio & skew remain valid it measures element quality. The element checks in Hypermesh test their properties and provide feedback as to quality of element. Number of elements on Gear is 54079 and Number of nodes on gear 14128 and types of meshing is Tetra Meshing similarly number elements on pinion is 45687 and number of nodes on pinion is 11513 and type of meshing is Tetra meshing. RBE2(rigid beam element 2)which is used for the transmit torque uniformly so Total number Elements on Gear and pinion is 99768 and Total number of nodes on gear and Pinion is 25641 that is shown in Deck



Total nodes – 4,05952

Total Elements – 2,57,385

V. WEIGHT OPTIMIZATION OF GEAR



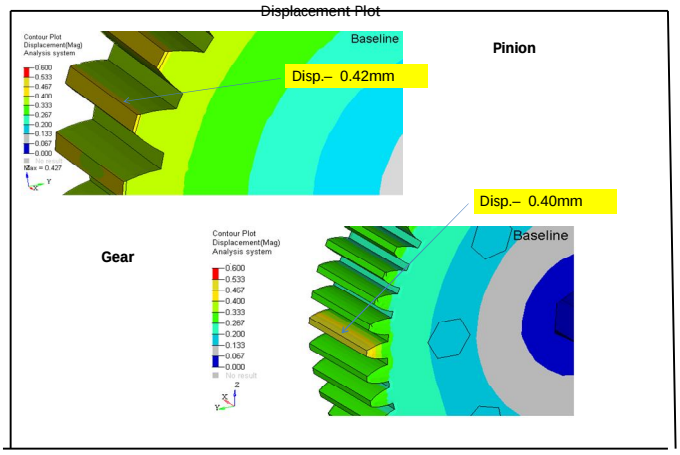
We used the various location to drill Circular holes. Similarly we used Number of holes and trying to find stress reduction, deformation and weight reduction.

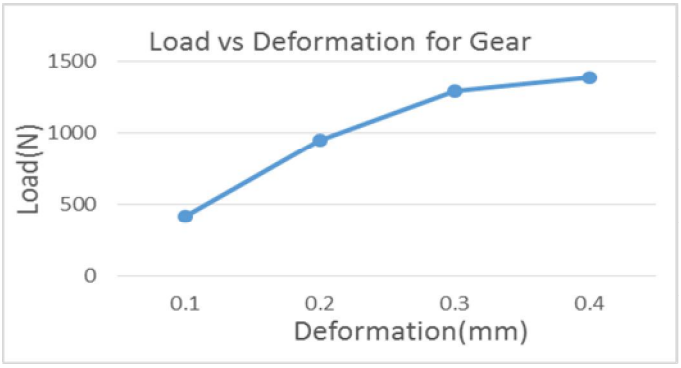
VI. EXPERIMENTAL RESULTS AND DISCUSSION

Experimentation is carried out with Compression loading on respective tooth of pinion and gear for Deformation test. This test is to be carried out on UTM machine, Strain Guages and Gear and Plunger Mechanism.

Volume 5 Issue VI, June 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Deformation testing is carried out at Praj mettalurgical lab Pune on Universal testing machine for gear and pinion respectively.





Graph: Load Vs Deformation for gear

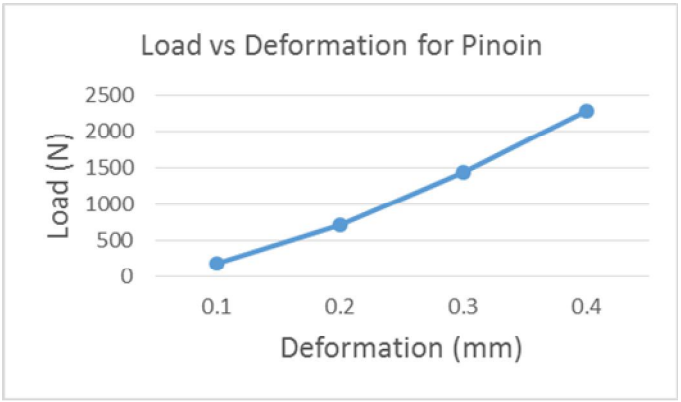
Deformation indicates indirect measurement of stress, so is very important to calculate deformation with respect to different loading conditions for gear and pinion respectively.

When gear and pinion are contacted ,the concentration of stress developed, plays vital role for failure analysis of both the parts. Failure is not only depend on stress developed, but also material accompanied with it.

Above graph shows ,as the load in also increases. Experimental readings are carried out for different deformation conditions(0.1 mm, 0.2mm, 0.3mm and 0.4mm)

On the same path Experimental readings are carried out for different loading conditions.The graph shows same family of crve as that of pinion.

Similarly Displacement plot of Pinion and Gear is also shown in CAD module which gives the displacement of pinion and gear of 0.42mm and 0.40mm respectively.



Graph: Load Vs Deformation for pinion

For obtaining 1mm deformation gear requires 419 N on the other hand for same deformation pinion requires 171 N.

VII. CONCLUSION

Studied the Spur gear profile and types of stresses which are particularly a) Bending stress and b) Contact stress.

When pinion and gear are engaged at a point, establishment of stress is unavoidable parameter .In order to avoid the failure of teeth, one has to study the stress concentration at engaged point.

Above study shows, for same amount of deformation i.e. 0.1mm pinion requires less load. As deformation increases beyond 0.2mm case becomes vice versa.

Studied the various stress relieving features: circular holes, elliptical holes or combination of both, oval shaped, aero fin holes etc.Also observed reduction of the magnitude of the mass from the baseline(1.25kg) to the design 1 (1.1kg) and design 2 (1.0kg).

REFERENCES

[1] Dhavale A.S. , Abhay Utpat ,2013, “Study of Stress Relief Features at Root of Teeth of Spur Gear”, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 3, Issue 3, May-Jun 2013, pp.895-899

[2] Mr. Bharat Gupta, Mr. Abhishek choubey,2012," Contact Stress Analysis of Spur Gear " , International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181, Vol. 1 Issue 4, June -2012

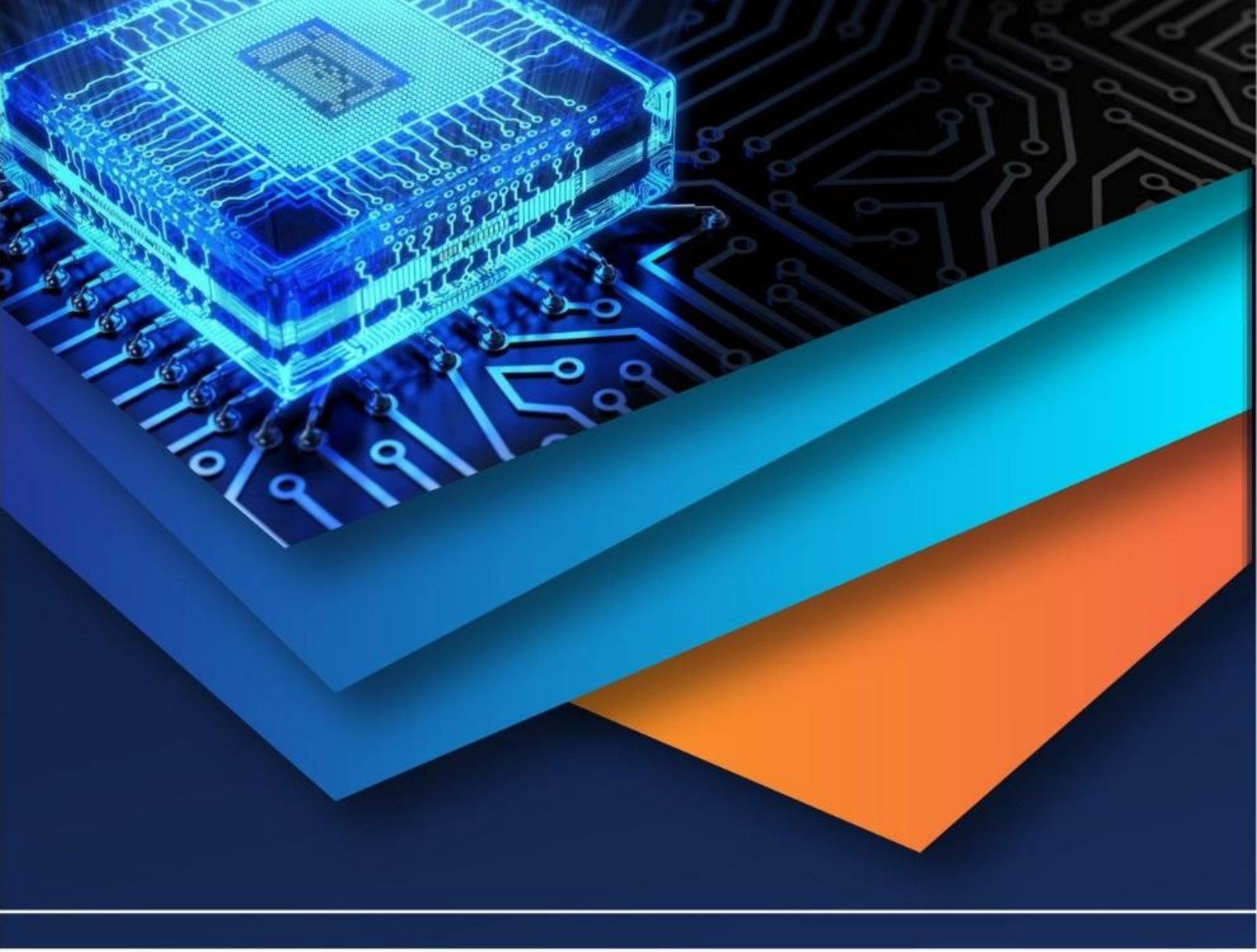
[3] M.Pramod Reddy, M. Santhosh, 2015 “Spur Gear tooth stress analysis and stress reduction using stress reducing geometrical features”. International Journal of Mechanical Engineering and Technology (IJMET) ISSN: 0976-6340, Vol.6, Issue 9, Sep 2015, pp17-29

[4] Deep Singh Vishwakarma, Dr. Rohit Rajvaidya 2014, “Modelling and reduction of root fillet stress in a spur gear using stress relieving feature”. International Journal of Modern Engineering Research (IJMER) ISSN: 2249-6645, Vol.4, Issue 7, July 2014 pp.41-48

[5] Mr.Sujit R.Gavhane, Prof. S.B. Naik,2014, “Study of stress relieving features in a spur gear”. International Journal of emerging Engineering Research and Technology (IJEERT) ISSN: 2349-4395, Vol.2, Issue 4, July 2014, pp 209-222

[6] M.Keerthi, K.Sandya ,K.Srinivas 2016, “Static and dynamic analysis of spur gear gear using different materials”. International research Journal of Engineering and Technology (IRJET) ISSN: 2695-0056, Vol.3, Issue 01, Jan 2016, pp 694-699

[7] Nidal H. Abu-Hamdeh, Mohammad A. Alharty " A Study on the Influence of using Stress Relieving Feature on Reducing the Root Fillet Stress in Spur Gear ", Proceedings of the 2014 International Conference on Mathematical Methods,



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY