© August 2018 | IJIRT | Volume 5 Issue 3 | ISSN: 2349-6002

Modelling and Analysis of Helical Gear

Ch.Suresh[1], Dr.M.Mahesh[2]
[1]*Student, Dept. of Mechanical Engineering, Guntur Engineering College, Guntur, A.P, India*
[2]*Professor, Dept. of Mechanical Engineering, Guntur Engineering College, Guntur, A.P, India*

Abstract- **Marine engines are among heavy duty machineries, which need to be taken care of in the best way during proto type development stages. These engines are operated at very high speeds which induce a large stresses and deflections in the gears as well as in other rotating components. For the safe function of the engine, these stresses and deflections have to be minimized in this project. Static structural analysis on a high speed helical gear used in a machine engines, have been performed. The dimensions of the model have been arrived at by the theoretical methods. The stresses generated and deflections of the tooth have been analyzed for different materials. Finally the results obtained by theoretical analysis and finite element analysis are compared to check correctness Conclusions have been arrive on the material which is best suited for the machine engines based on the result.**

Index Terms- **Helical gear, Manufacturing Methods of Gear, Analysis of gear, and Modelling of helical gear.**

I. INTRODUCTION

Gears are most commonly used for power transmission in all the modern devices. These toothed wheels are used to change the speed or power between input and output. They have gained wide range of acceptance in all kinds of applications and have been used extensively in the high-speed marine engines.

In the present era of sophisticated technology, gear design has evolved to a high degree of perfection. The design and manufacture of precision cut gears, made from materials of high strength, have made it possible to produce gears which are capable of transmitting extremely large loads at extremely high circumferential speeds with very little noise, vibration and other undesirable aspects of gear drives.

A gear is a toothed wheel having a special tooth space of profile enabling it to mesh smoothly with other gears and power transmission takes place from one shaft to other by means of successive engagement of teeth.

Gears operate in pairs, the smallest of the pair being called “pinion” and the larger one “gear”. Usually the pinion drives the gear and the system acts as a speed reducer and torque converter.

Helical gears offer a refinement over spur gears. The leading edges of the teeth are not parallel to the axis of rotation, but are set at an angle. Since the gear is curved, this angling causes the tooth shape to be a segment of a helix. Helical gears can be meshed in a parallel or crossed orientations. The former refers to when the shafts are parallel to each other; this is the most common orientation. In the latter, the shafts are non-parallel.

The angled teeth engage more gradually than do spur gear teeth causing them to run more smoothly and quietly. With parallel helical gears, each pair of teeth first make contact at a single point at one side of the gear wheel; a moving curve of contact then grows gradually across the tooth face to a maximum then recedes until the teeth break contact at a single point on the opposite side. In spur gears teeth suddenly meet at a line contact across their entire width causing stress and noise. Spur gears make a characteristic whine at high speeds and can not take as much torque as helical gears. Whereas spur gears are used for low speed applications and those situations where noise control is not a problem, the use of helical gears is indicated when the application involves high speeds, large power transmission, or where noise abatement is important. The speed is considered to be high when the pitch line velocity exceeds 25 m/s

Quite commonly helical gears are used with the helix angle of one having the negative of the helix angle of the other; such a pair might also be referred to as having a right-handed helix and a left-handed helix of equal angles. The two equal but opposite angles add to zero: the angle between shafts is zero – that is, the shafts are *parallel*. Where the sum or the difference (as described in the equations above) is not zero the shafts are *crossed*. For shafts *crossed* at right angles the helix angles are of the same hand because they must add to 90 degrees.

II. GENERAL DESCRIPTION OF FEM

In the finite element method, the actual continuum of body of matter like solid, liquid or gas is represented as an assemblage of sub divisions called Finite elements. These elements are considered to be inter connected at specified points known as nodes or nodal points. These nodes usually lie on the element boundaries where an adjacent element is considered to be connected. Since the actual variation of the field variables (like Displacement, stress, temperature, pressure and velocity) inside the continuum are is not know, we assume that the variation of the field variable inside a finite element can be approximated by a simple function.

These approximating functions (also called interpolation models) are defined in terms of the values at the nodes. When the field equations (like equilibrium equations) for the whole continuum are written, the new unknown will be the nodal values of the field variable. By solving the field equations, which are generally in the form of the matrix equations, the nodal values of the field variables will be known. Once these are known, the approximating function defines the field variable throughout the assemblage of elements.

The solution of a general continuum by the finite element method always follows as orderly step-by-step process. The step-by-step procedure for static structural problem can be stated as follows:

Step 1:- Discritization of Structure (Domain).

The first step in the finite element method is to divide the structure of solution region in to sub divisions or elements.

Step 2:- Selection of proper interpolation model.

Since the displacement (field variable) solution of a complex structure under any specified load conditions cannot be predicted exactly, we assume some suitable solution, within an element to approximate the unknown solution. The assumed solution must be simple and it should satisfy certain convergence requirements.

Step 3:- Derivation of element stiffness matrices (characteristic matrices) and load vectors.

From the assumed displacement model the stiffness matrix [K(e)] and the load vector P(e) of element 'e' are to be derived by using either equilibrium conditions or a suitable Variation principle.

Step 4:- Assemblage of element equations to obtain the equilibrium equations.

Since the structure is composed of several finite elements, the individual element stiffness matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equation has to be formulated as

$[K]\varphi = P$

Where [K] is called assembled stiffness matrix,

Φ is called the vector of nodal displacement and

P is the vector or nodal force for the complete structure.

Step 5:- Solution of system equation to find nodal values of displacement (field variable)

The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. After the incorporation of the boundary conditions, the equilibrium equations can be expressed as,

$[K]\varphi = P$

For linear problems, the vector 'φ' can be solved very easily. But for non-linear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness matrix [K] and 'φ' or the load vector P.

Step 6:- Computation of element strains and stresses.

From the known nodal displacements, if required, the element strains and stresses can be computed by using the necessary equations of solid or structural mechanics.

In the above steps, the words indicated in brackets implement the general FEM step-by-step procedure.

A. ADVANTAGES OF FEM

The FEM is based on the concept of discretization. Nevertheless as either a variational or residual approach, the technique recognizes the multi dimensional continuity of the body not only does the idealizations portray the body as continuous but it also requires no separate interpolation process to extend the approximate solution to every point within the continuum. Despite the fact that the solution is obtained at a finite number of discrete node points, the formation of field variable models inherently provides a solution at all other locations in the body. In contrast to other variational and residual approaches, the FEM does not require trail solutions, which must all, apply to the entire multi dimensional continuum. The use of separate sub-regions or the finite elements for the separate trial solutions thus permits a greater flexibility in considering continuum of the shape.

Some of the most important advantages of the FEM derive from the techniques of introducing boundary conditions. This is another area in which the method differs from other variational or residual approaches. Rather than requiring every trial solution to satisfy the boundary conditions, one prescribes the conditions after obtaining the algebraic equations for assemblage.

No special techniques or artificial devices are necessary, such as the non-cantered difference equations or factious external points often employed in the finite difference method.

The FEM not only accommodates complex geometry and boundary conditions, but it also has proved successful in representing various types of complicated material properties that are difficult to incorporate in to other numerical methods. For example, formulations in solid mechanics have been devised for anisotropic, nonlinear, hysteretic, time dependant or temperature dependant material behavior.

One of the most difficult problems encountered in applying numerical procedures of engineering analysis is the representation of non-homogeneous continua. Nevertheless the FEM readily accounts for non-homogeneity by the simple tactic of assigning different properties within an element according to the pre selected polynomial pattern. For instance it is possible to accommodate continuous or discontinuous variations of the constitutive parameters or of the thickness of a two-dimensional body.

The systematic generality of the finite element procedure makes it a powerful and versatile tool for a wide range of problems. As a result, flexible general-purpose computer programs can be constructed. Primary examples of these programs are several structural analysis packages which include a variety of element configurations and which can be applied to several categories of structural problems. Among these packages are ASKA, STRUDL, SAP and NASTRAN & SAFE. Another indicator of the generality of the method is that programs developed for one field of engineering have been applied successfully to problems in different field with a little or no modification.

Finally an engineer may develop a concept of the FEM at different levels. It is possible to interpret the method in physical terms. On the other hand the method may be explained entirely in mathematical terms. The physical or intuitive nature of the procedure is particularly useful to the engineering student and practicing engineer.

B. INTRODUCTION TO ANSYS

ANSYS Stands for Analysis System Product. Dr. John Swanson founded ANSYS. Inc in 1970 with a vision to commercialize the concept of computer simulated engineering, establishing himself as one of the pioneers of Finite Element Analysis (FEA). ANSYS inc. supports the ongoing development of innovative technology and delivers flexible, enterprise wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware. ANSYS Inc. continues its role as a technical innovator.

It also supports a process-centric approach to design and manufacturing, allowing the users to avoid expensive and time-consuming "built and break" cycles. ANSYS analysis and simulation tools give customers ease-of-use, data compatibility, multi platform support and coupled field multi-physics capabilities.

III. PROBLEM Definition

Marine engines are among heavy-duty machineries, which need to be taken care of in the best way during prototype development stages. These engines are operated at very high speeds which induce large stresses and deflections in the gears as well as in other rotating components. For the safe functioning of the engine, these stresses and deflections have to be minimized.

In this project, we have performed static-structural analysis on a high speed helical gear used in marine engines, using different materials. The results obtained in the static analysis are compared with those obtained in theoretical and a conclusion has been drawn on the material to be used for the gear.

A. Element considered for static analysis:

According to the given specifications the element type chosen is SOLID 45. SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

The element is defined by eight nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

The following Figure shows the schematic diagram of the 8-noded static solid element.

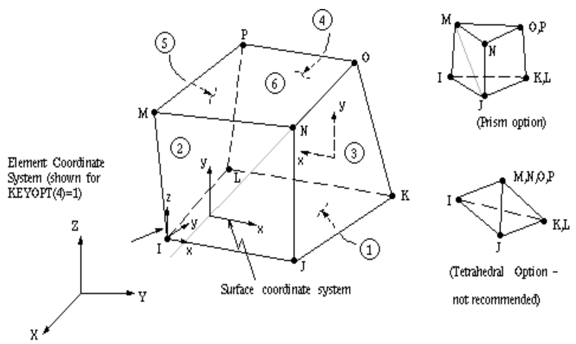

Fig: 1 SOLID-45, 8-NODE STATIC ELEMENT

B. Assumptions and Restrictions

Zero volume elements are not allowed. Elements may be numbered or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.

All elements must have eight nodes. A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers. A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

Mesh Generation

Before building the model, it is important to think about whether a free mesh or a mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes and has no specified pattern applied to it.

Compared to the free mesh, a mapped mesh is restricted in terms of the element shape it contains and pattern of the mesh. A mapped mesh contains either only quadrilateral or only triangular element, while a mapped volume mesh contains only hexahedral elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements.

For mapped mesh, we must build the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh. The type of mesh generation considered here is a free mesh since the 3D figure is not a regular shape. Solid 45 is used to model in ANSYS.

C. Boundary conditions

Geometry boundary conditions

The shaft is fixed at the centre along with its key.

Loads applied:

F_T=137572.60 N

F_R =55248.70 N

Finally the boundary conditions are verified before going for a solution.

IV ANSYS RESULTS

Contours of GEAR in ANSYS:

A. RESULTS FOR STEEL AS A MATERIAL:

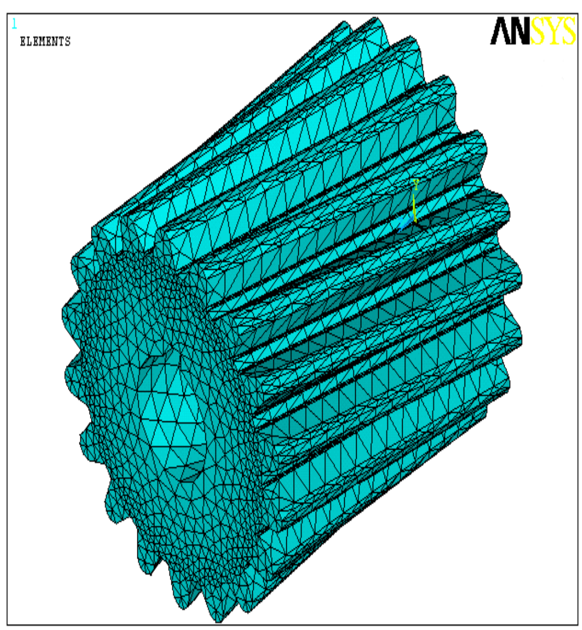

Fig.2: meshed gear model

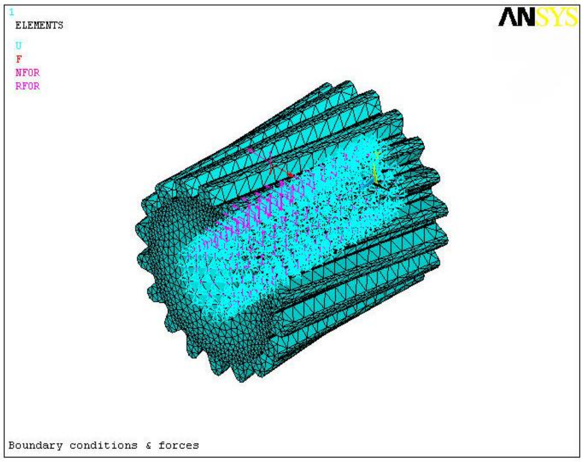

Fig.3: boundary conditions & Forces

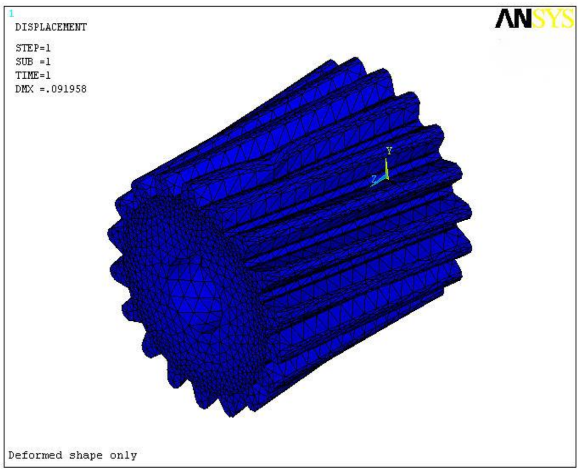

Fig.4: Deformed model of helical gear with steel as a material

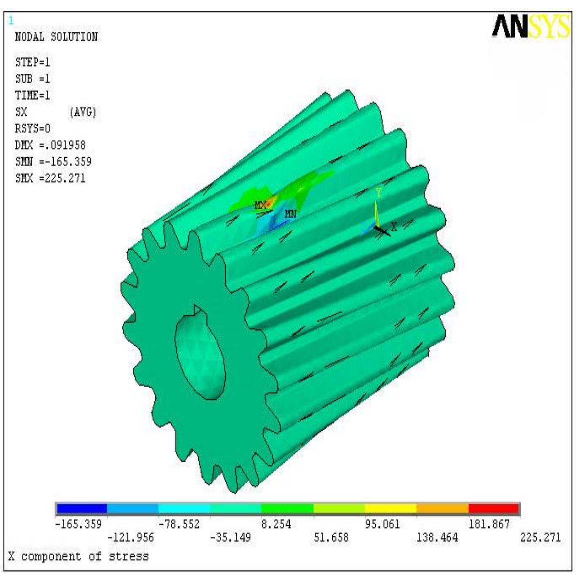

Fig.4: Stresses in X-direction of model of helical gear with steel as a material

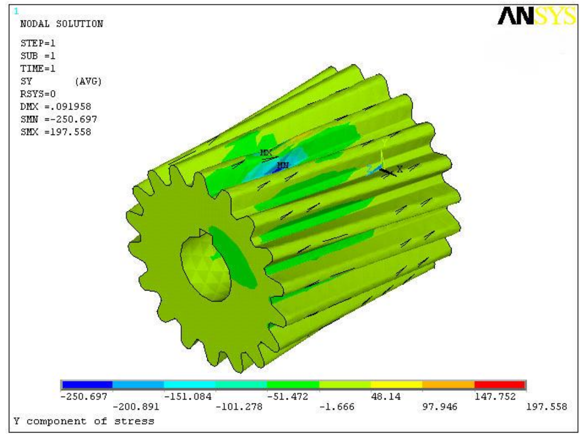

Fig.5: stresses in Y-direction of model of helical gear with steel as a material

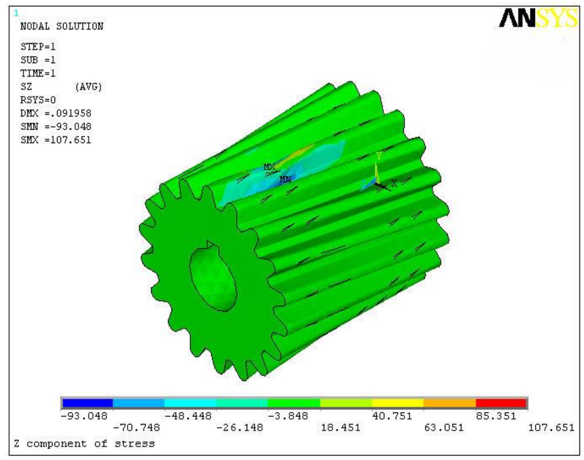

Fig.6: stresses in Z-direction of model of helical gear with steel as a material

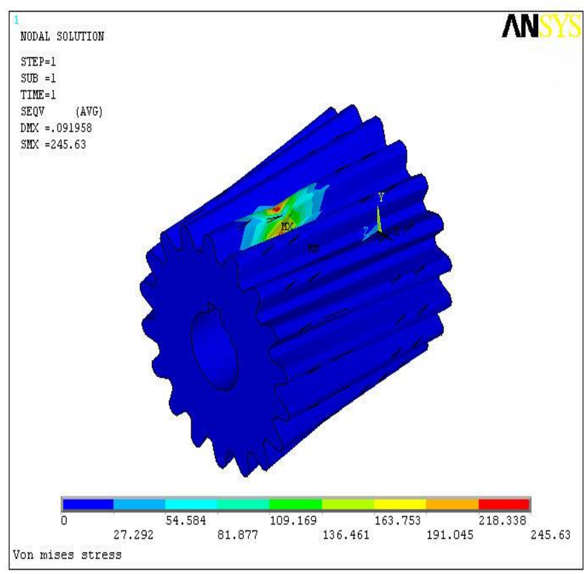

Fig.7: Von Misses stresses of model of helical gear with steel as a material

A. RESULTS FOR CERAMICS

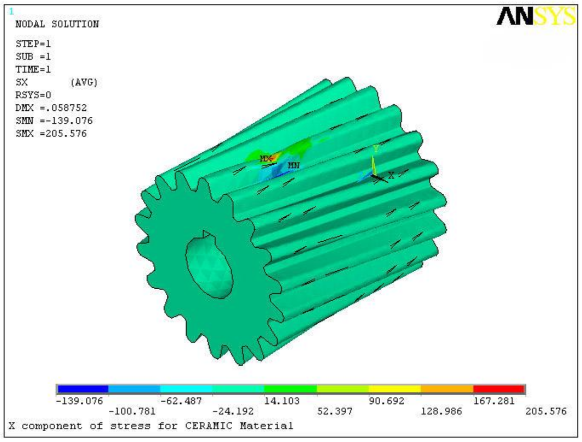

Fig.8: The stresses in X-direction of model of helical gear with ceramics as a material

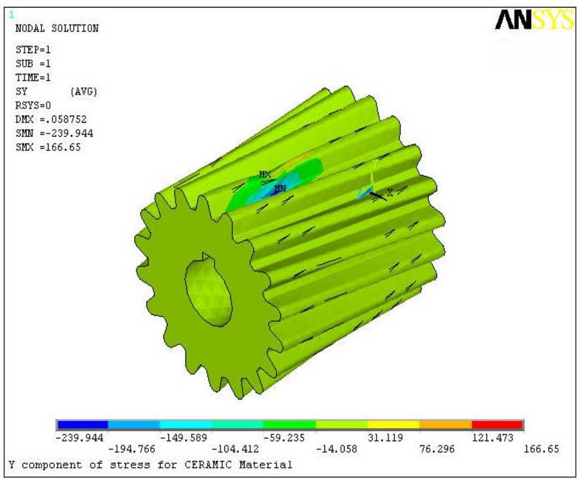

Fig.9: Stresses in Y-direction of model of helical gear with ceramics as a material.

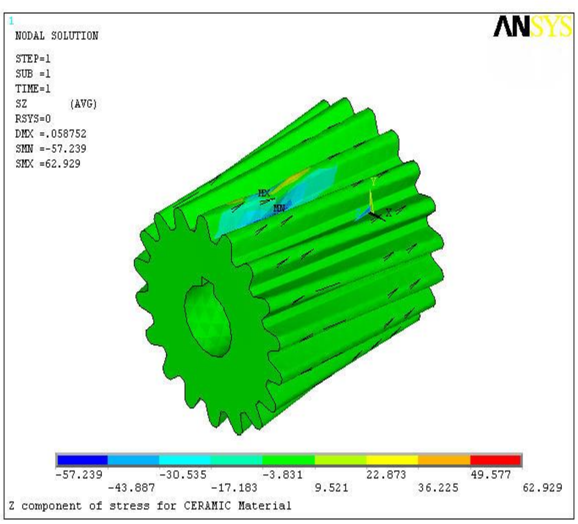

Fig.10: stresses in Z-direction of model of helical gear with ceramics as a material

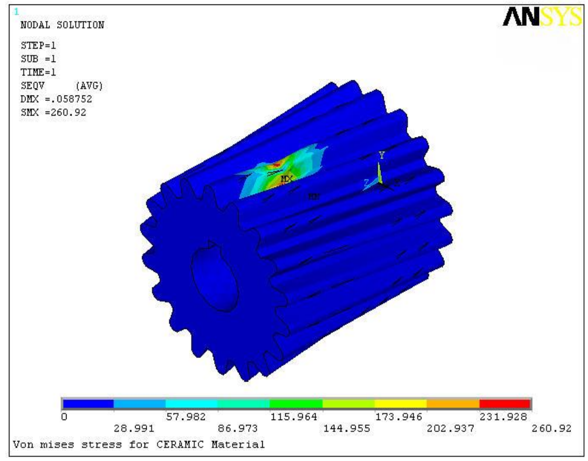

Fig.11: Von Misses stresses of model of helical gear with ceramics as a material

V. RESULTS & DISCUSSION

A. RESULTS FOR STEEL AS MATERIAL

The following table shows the comparison between theoretical results & experimental results.

PARAMETER	DESIGN STRESSES	INDUCED STRESSES
BENDING STRESS	400 N/mm^2	228.35 N/mm^2
COMPRESSIVE STRESS	1100 N/mm^2	178.59 N/mm^2

TABLE.1: comparison between theoretical results & experimental results

PARAMETER	THEORETICAL RESULTS	ANSYS RESULTS
DEFLECTION		0.091958 mm
BENDING STRESS	228.35 N/mm^2	225.271 N/mm^2
COMPRESSIVE STRESS	178.59 N/mm^2	165.359 N/mm^2
VON MISES STRESSES	σ_{yield}=972 N/mm^2	245.63N/mm^2

TABLE.2. comparison between theoretical results & experimental results

From the table 1 we observe that the bending & compressive stresses obtained practically from the ANSYS are much lower than those of the results obtained theoretically. Thus the design is safe from the structural point of view.

From the table.2 we observe that the induced bending & compressive stresses are much lower than the design stresses. Thus the design is safe from the structural point of view.

The maximum deflection is found to be 0.091958mm which is well within the permissible limits. Thus the design is safe based on rigidity point of view.

The induced von mises stresses with magnitude of 245.63N/mm^2 are much lower than the yield stress i.e. 972 N/mm^2 according to the manufacturer's specifications.

B. RESULTS FOR CERAMICS [98% Al_2O_3] AS MATERIAL

The following table shows the comparison between theoretical results & experimental results.

PARAMETER	DESIGN STRESSES	INDUCED STRESSES
BENDING STRESS	350 N/mm^2	220.35 N/mm^2
COMPRESSIVE STRESS	2500 N/mm^2	150.30 N/mm^2

TABLE 3: Comparison between theoretical results & experimental results.

PARAMETER	THEORETICAL RESULTS	ANSYS RESULTS
DEFLECTION		0.058752 mm
BENDING STRESS	228.35 N/mm^2	225.271 N/mm^2
COMPRESSIVE STRESS	178.59 N/mm^2	165.359 N/mm^2
VON MISES STRESSES	σ_{yield} = 972 N/mm^2	245.63N/mm^2

TABLE 4: Comparison between theoretical results & experimental results.

From the table 3 we observe that the bending & compressive stresses obtained practically from the ANSYS are much lower than those of the results obtained theoretically. Thus the design is safe from the structural point of view.

From the table .4 we observe that the induced bending & compressive stresses obtained are much lower than those of the design stresses. Thus the design is safe from the structural point of view.

The maximum deflection is found to be 0.058752mm which is well within the permissible limits. Thus the design is safe based on rigidity point of view.

The induced von mises stresses with magnitude of 260.92/mm^2 is far below the yield stress i.e. 2000 N/mm^2 according to the manufacturer's specifications.

Thus the helical gear parameters that constitute the design are in turn safe based on the strength and rigidity.

COMPARISION OF RESULTS:

PARAMETERS	STEEL	CERAMICS
DEFLECTIONS	0.091958 mm	0.058752 mm
BENDING STRESS	225.271 N/mm^2	205.576 N/mm^2
Compressive stress	165.359 N/mm^2	139.076 N/mm^2

Table 5 Comparison of Results

The results obtained above are less than material properties as mentioned before. Hence the design is safe and optimum.

VI CONCLUSION

- Bending and compressive stresses were obtained theoretically & by using Ansys software for both ceramic& steel.
- From the table 5, it is observed that bending and compressive stresses of ceramics are less than that of the steel.
- Weight reduction is a very important criterion in the marine gears.
- Hence ceramic material is preferred.

REFERENCES

[1] Thirupathi R. Chandrupatla & Ashok D.Belegundu., Introduction To Finite Eleement In Engineering, Pearson ,2003

[2] Joseph Shigley, Charles Mischike., Mechanical Engineering Design, Tmh,2003

[3] Maithra ., Handbook Of Gear Design,2000

[4] V.B.Bhandari., Design Of Machine Elements,Tmh,2003

[5] R.S.Khurmi ., Machine Design, Schand,2005

[6] Darle W Dudley.,Hand Book Of Practical Gear Design,1954

[7] Alec Strokes., High Performance Gear Design,1970

[8] Www.Matweb.Com

[9] Mirza Grebovic, "Investigation Of The Effects On Braking Performance Of Different Brake Rotor Designs".2013

[10] Limpert, Rudolf "Brake Design And Safety", Society Of Automotive Engineers., Inc, Pa, Usa, 2012

[11] Warren Chan, "Analysis Of Heat Dissipation In Mechanical Braking Systems". 2012

[12] .David A. Johnson, Bryan A. Sperandei, Et.Al., "Analysis Of The Flow Through A Vented Automotive Brake Rotor,2011
[13] David Antanaitis And Anthony Rifici, "The Effect Of Rotor Crossdrilling On Brake Performance". Sae 2006-01-0691
[14] Brake Design and Safety, 2nd Ed., Rudolf Limpert, 2010.
[15] Dr.N.K.Giri, Automobile Mechanics, Khannapublishers, 2010.
[16] Standards in spur and helical gears with FEM based verification," ASME Journal of Mechanical Design,Vol 128/114
[17] J.O.Nordiana, S.O.Ogbeide, N.N.Ehigiamusoe and F.I.Anyasi., 2007,"Computer aided design of a spur gear, "Journal of Engineering and Applied Sciences 2 (12); pp 17431747.
[18] Darle w dudley.,hand book of practical gear design,1954