International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 2, April 2013

Research Paper

COMPUTER AIDED DESIGN, MODELLING AND ANALYSIS OF BULL GEAR OF A DRAG LINE

Shrikant Awatade[1]* and Umesh Bokade[1]

*Corresponding Author: **Shrikant Awatade,** ✉ sm_awatade@rediffmail.com*

In this project we have done computer aided design, modeling and analysis of bull gear of a drag line which specifically deals with drag system of dragline. Dragline is a heavy duty mechanical hardware used for removal of overburden in opencast mine. It is similar to a crane in which hoisting, dragging and swinging operation of buckets are possible over and above these, a walking mechanism is provided with the help of which entire platform of dragline can be shifted from one location to another. Design of heavy earth moving machinery requires accurate method and design should result in reliable construction and withstanding the required load while being economical. This project work contains stress analysis of BULL GEAR, part of the Dragline, in which the stress developed in gear is tried to reduce with the help of software approach. In this project detailed study of drag system is done than modeling of bull gear is done in PRO-E software and analysis is done in ANSYS software. BULL GEAR is once manufactured lifetime equipment so it is considered that once the part is manufactured, it should work lifelong but sometimes crack appears in Bull gear part. In this project, with the help of software, development of crack in bull gear is tried to reduce by using proper material properties and applying software results to the part.

Keywords: Bull gear, Drag line, Drafting, Discretization, FEA

INRODUCTION

Drag line is a heavy duty mechanical hardware used for removal of overburden in opencast mine. it is similar to a crane in which hoisting, dragging and swinging operation of buckets are possible over and above these, a walking mechanism is provided with the help of which entire platform of dragline can be shifted from one location to another.

Out of 31 open cast mines of W.C.L only 3 mines are having drag line.

These are

- Umred opencast mine (Nagpur district).

[1] Department of Mech. Engg., Priyadarshini Indira Gandhi College of Engg., Nagpur, India.

- Sasti opencast mine (Chandrapur district).
- Ghugus opencast mine (Chandrapur district).

Specification of Drag Line 15/90 Umred

- Machine commissioned in -1978
- Bucket capacity -15 m^3
- Boom length -90 m
- Motor supply -1900 kw
- Machine weight -1600 tones
- Walking speed -60 m/hr
- Dragging radius -80 m

Computer Aided Modeling of Bull Gear

Modelling Procedure of Bull Gear

Modeling and drafting of bull gear is done in pro-e software. The various tools used for modeling of bull gear is revolve, extrude, chamfer, round, pattern.

Various steps in modelling are as follows:

- Sketch dedendum circle and then extrude.
- Sketch the one teeth profile and then extrude.
- Pattern the teeth.
- Sketch one arm of gear and then extrude.
- Pattern the arm.
- Sketch the one hole.
- Pattern the holes.

Figure 1: Schematic Diagram of the Layout of Drag System

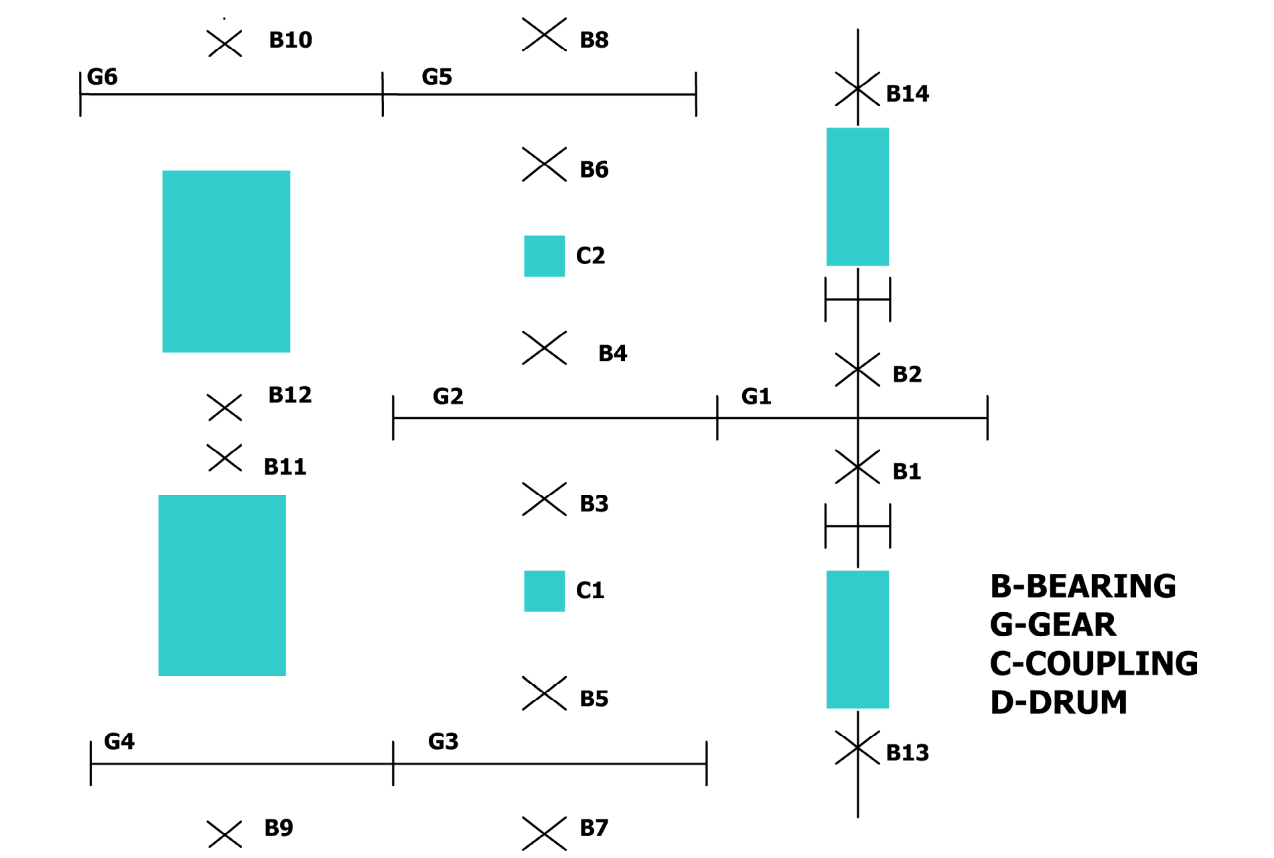
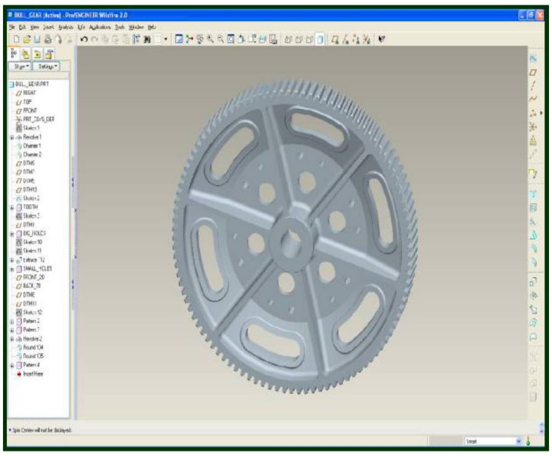


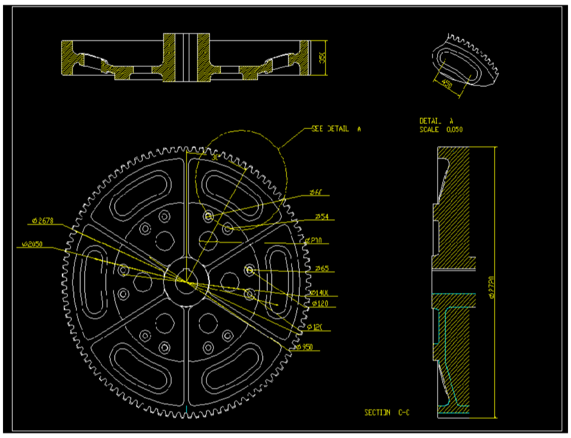
Figure 2: Modelling of Bull Gear



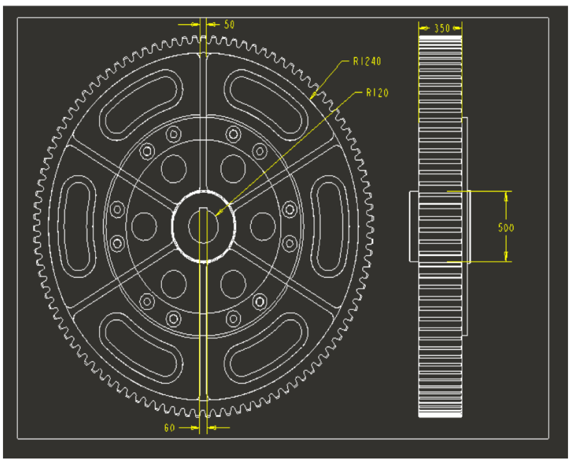
DRAFTING OF BULL GEAR

Figure 3: Drafting of Bull Gear

(a)



(b)



FINITE ELEMENT APPROACH

Finite Element Methodology

The solution of a general continuum problem by the finite element method always follows an orderly step by step process. With reference to static structural problems, the step by step procedure can be stated as follows:

Step 1: Discretization of the Structure

The first step in the finite element method is to divide the structure or solution region into subdivisions or elements. Hence the structure is to be modeled with suitable finite elements. The number, type, size and arrangements of the elements are to be decided.

Step 2: Selection of Proper Interpolation or Displacement Model

Since the displacement 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 form a computational point of view, but it should satisfy certain convergence requirements. In general, the solution or the interpolation model is taken in the form of a polynomial.

To satisfy the convergence requirements, the polynomial functions,

- Must be continuous within the element
- Must contain rigid body displacement or field variables
- Must contain constant strain states

Step 3: Derivation of Element Stiffness Matrices and Load Vectors

For the assumed displacement model, the stiffness matrix k^e and the load vector F^e of element “*e*” are to be derived by using either equilibrium conditions or a suitable variational principle.

Step 4: Assemblage of Elemental Equations to Obtain the Overall Equilibrium Equations

Since the structure is composed of several finite elements, the individual elemental stiffness matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equations have to be formulated as $[k]u = F$. Where $[k]$ is assembled stiffness matrix, u the vector of nodal displacements and P is called the vector nodal forces for the complex structure.

Step 5: Solution for the Unknown Nodal Displacements

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 $[k]u = F$.

For linear problems, the vector u is can be solved very easily. But for nonlinear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness matrix $[k]$ and/or load vector F.

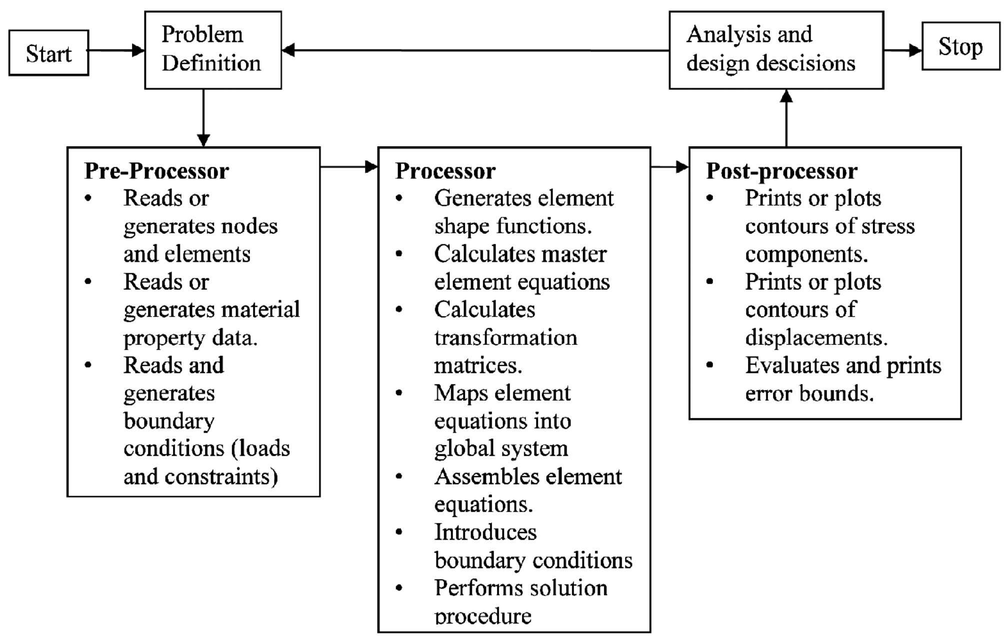
Step 6: Computational of Elemental Stress and Strains

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

The terminology used in the above six steps has to be modified if we want to extend the concept to other fields.

Process flow of Finite Element Analysis: The general steps followed in a finite element analysis with a commercial FEM package is as shown below:

Figure 4: Process Flow of FEA



METHOD OF ANALYSIS

Element type = Tet 10node 187

Number of nodes = 203252

Number of elements = 144664

Material property:

Young's modulus of Elasticity = 202 Gpa

Poisson's ratio = 0.3

Material of gear-CAST STEEL grade-I

Analysis is done in Ansys software. First step in analysis is to import bull gear teeth from pro-e software, then defining the various stages of Ansys. Which are as follows.

Preferences

Type of analysis-structural

P-method

Preprocessor

- Material property-
  - Modulus of elasticity-200 GPa
  - Posssion's ratio-0.3
- Element
  - Solid (Tet 10 node 187)
- Meshing
  - Mesh tool-free mesh-by volume

Solution

In this we are apply boundary condition

Force (*Ft*) = 87394.43 N

Displacement $Fx = Fy = Fz = 0$

Post Processor

Obtaining the solution, such as nodal solution, Element solution, vonmisses stresses, vonmisses strain, nodal displacement.

- Plot result – Nodal solution-deformation result
- Plot result – Element solution-deformation result
- Plot result – Element solution-stress-vonmisses stress
- Plot result – Element solution- stress-vonmisses strain

SOLID187 ELEMENT DESCRIPTION

Solid187 element is a higher order 3-D, 10-node element. Solid187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal *x*, *y*, and *z* directions.

The element has plasticity, hyperelasticity, creep, Viscoelasticity, Viscoplasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

Figure 5: Solid187 Geometry

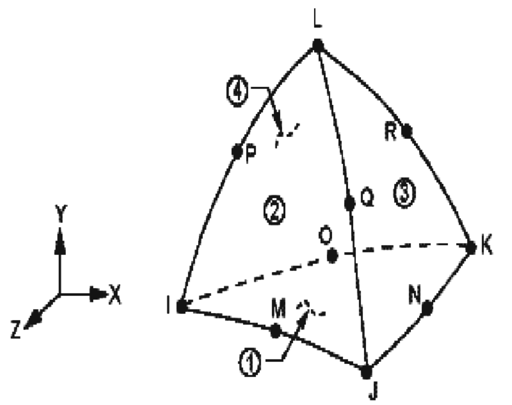


Figure 6: Meshing *Ft* = 87394.43 N

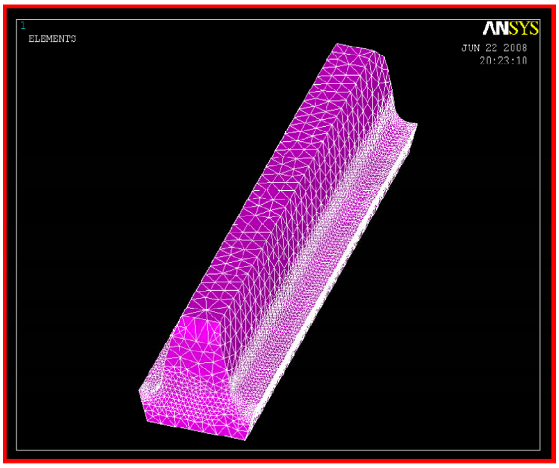


Figure 7: Boundary Condition

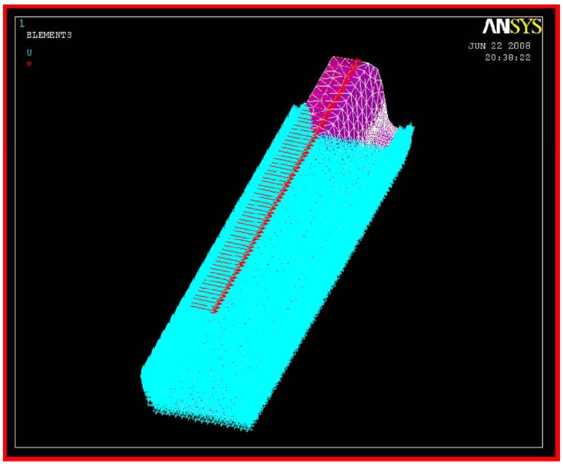


Figure 8: Degree of Freedom Degree of Freedom (*fx* = *fy* = *fz* = 0)

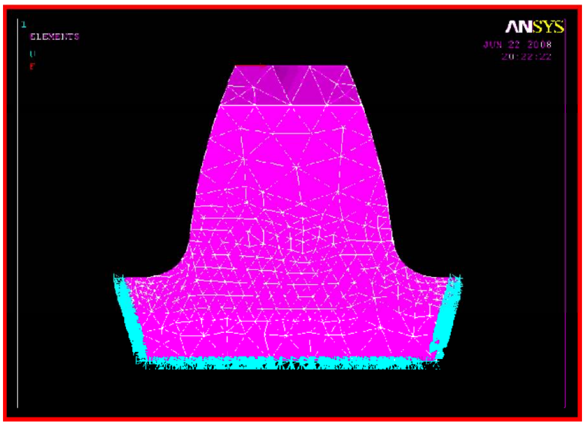


Figure 9: Deflection = 0.019181 mm for Force (*f*) = 87394.43 *n*

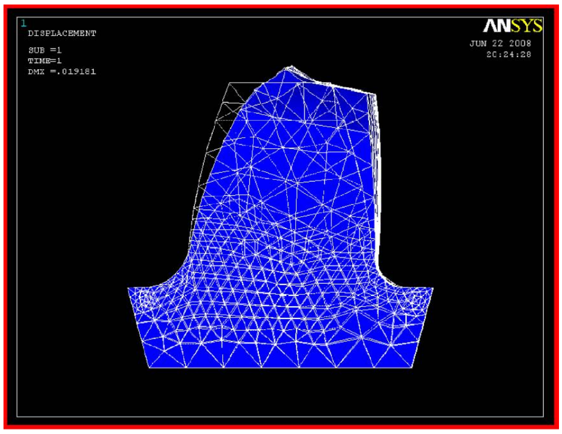


Figure 10: *X* Component of Stress = 102.034 mpa for Force (*f*) = 87394.43 *n*

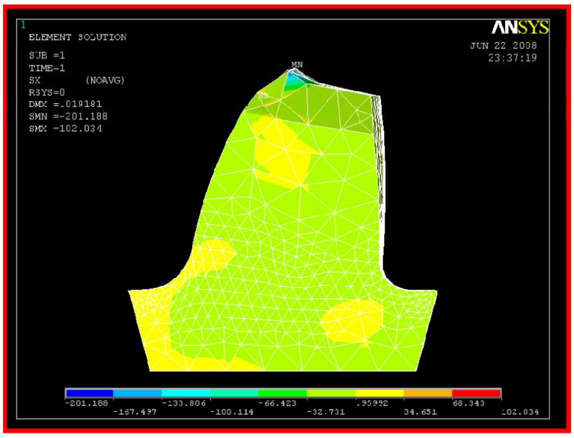


Figure 11: *Y* Component of Stress = 99.702 mpa for Force (*F*) = 87394.43 *n*

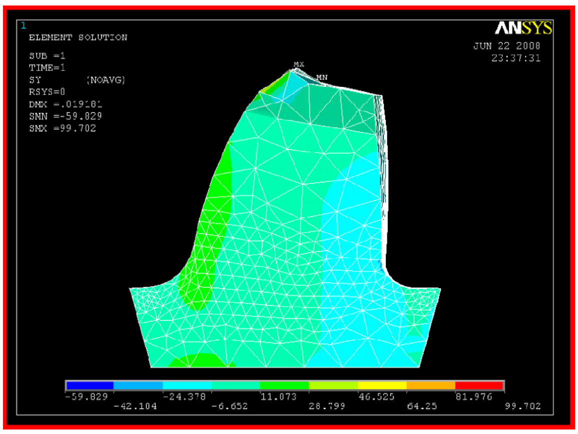


Figure 12: *Z* Component of Stress = 66.551 mpa for Force (*f*) = 87394.43 *n*

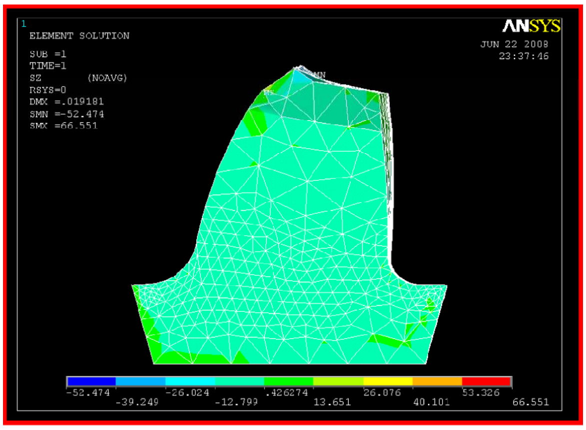


Figure 13: *XY* Shear Stress = 112.655 mpa for Force (*f*) = 87394.43 *n*

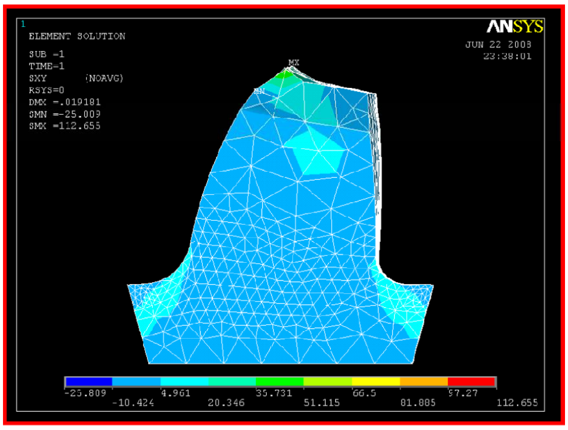


Figure 14: *X* Component of Stress = 102.034 mpa for Force (*f*) = 87394.43 *n*

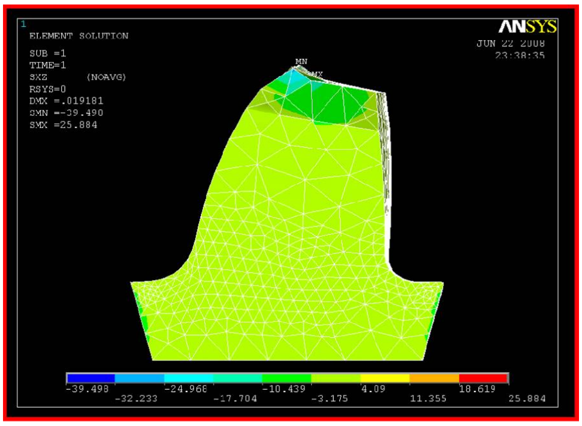


Figure 15: *XZ* Shear Stress = 25.899 mpa for Force (*f*) = 87394.43 *n*

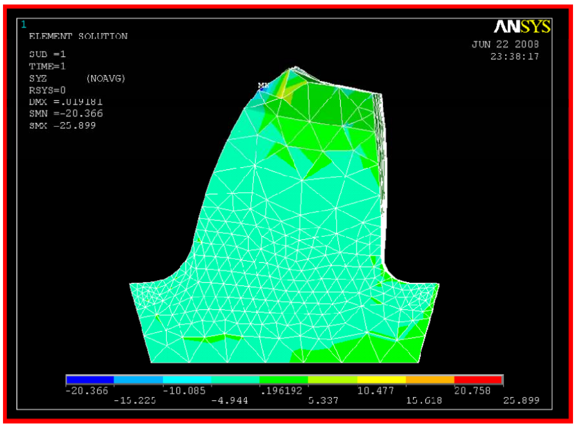


Figure 16: Equivalent von Mises Stress = 286.495 mpa for Force (*f*) = 87394.43 *n*

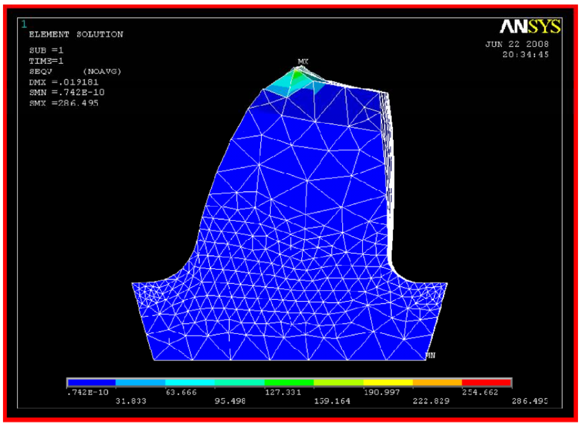


Figure 17: Equivalent von Mises Strain = 0.001418 for Force (*f*) = 87394.43 *n*

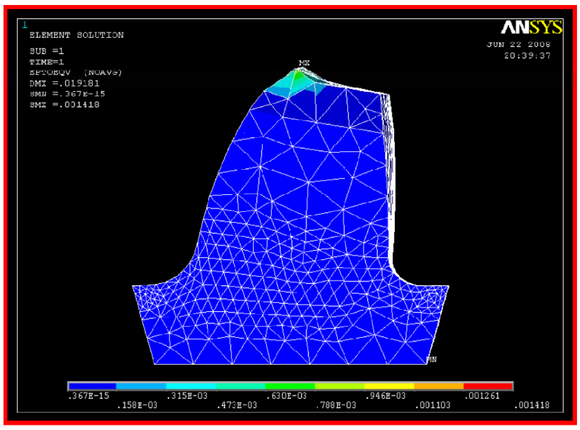
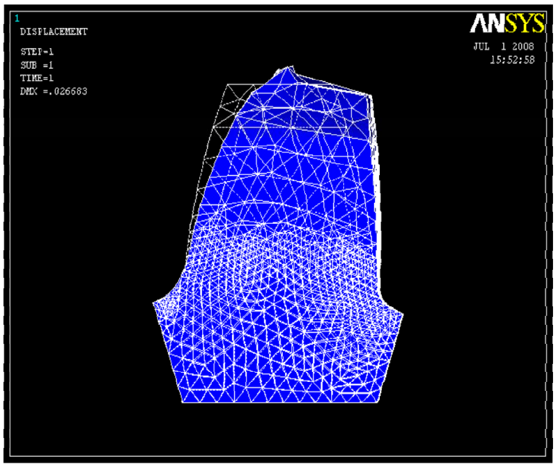


Figure 18: Deflection = 0.0266 mm for Force (*f*) = 115000 *n*



DYNAMIC LOAD ANALYSIS

$Fd = Ft + [\{21vp\ (ceb + Ft)\}/\{21vp + \sqrt{(ceb + Ft)}\}]$

Where, c = deformation factor

c = 8100 (gray cast iron and steel) 20° full depth

e = error in profile

e = 0.05 mm (commercially cut gear and class 1)

$Fd = Ft + [\{21vp\ (ceb + Ft)\}/\{21vp + \sqrt{(ceb + Ft)}\}]$

$Fd = 87394.43 + [\{21 * 6.3518\ (8100 * 0.05 * 350 + 87394.43)\}/\{21 * 6.3518 + \sqrt{(8100 * 0.05 * 350 + 87394.43)}\}]$

$Fd = 137330.9927\ N$

(Dynamic load at pitch circle radius) x (Pitch circle radius) = (Equivalent tip load) x (Addendum circle radius)

137330.9 * 1339 = Equivalent tip load * 1365

Equivalent tip load = 134715 *N*

Figure 19: For Force = 134715 *N* Deflection = 0.0306 mm

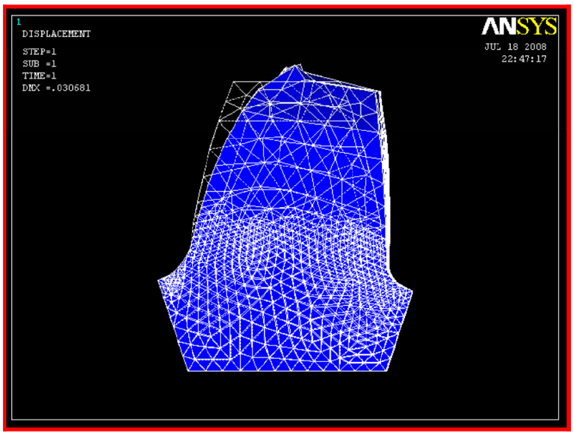


Figure 20: Vonmisses Stress 515.19 mpa

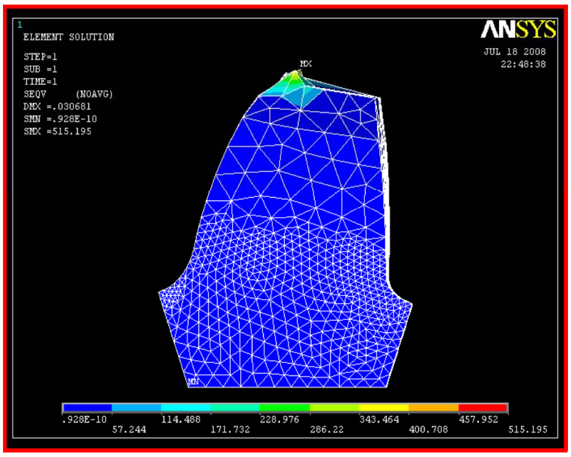


Figure 21: Vonmisses Strain 0.00255 mpa

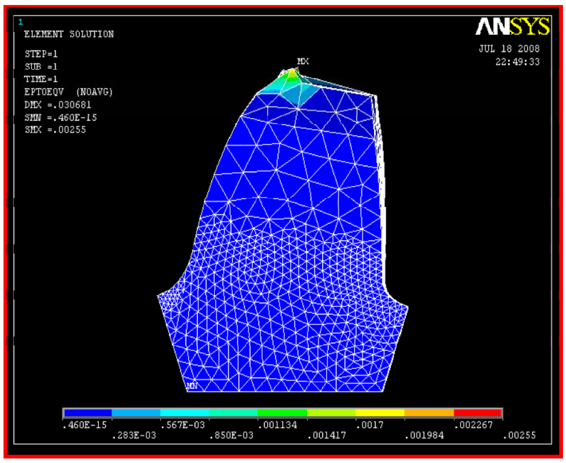


Figure 22: *X* Component of Stress 218.055 mpa

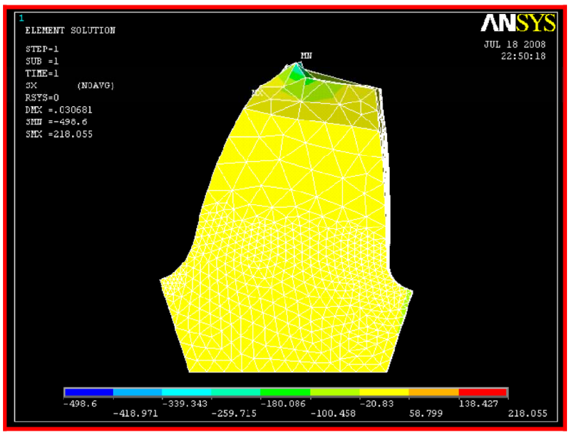


Figure 23: *XY* Shear Stress 216.454 mpa

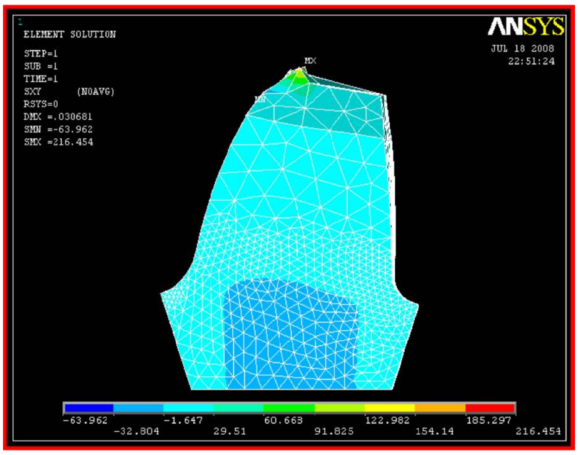


Figure 24: *Y* Component of Stress 55.116 mpa

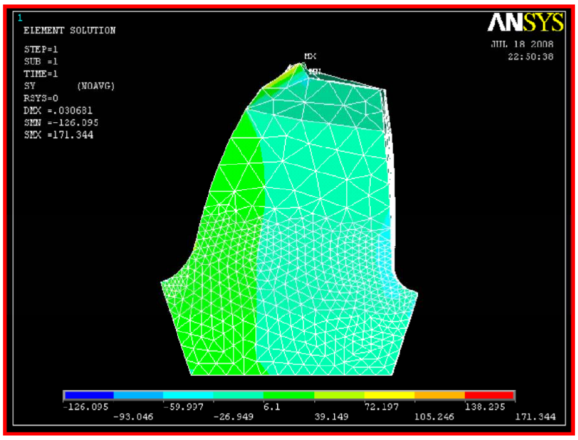


Figure 25: *YZ* Shear Stress 171.344 mpa

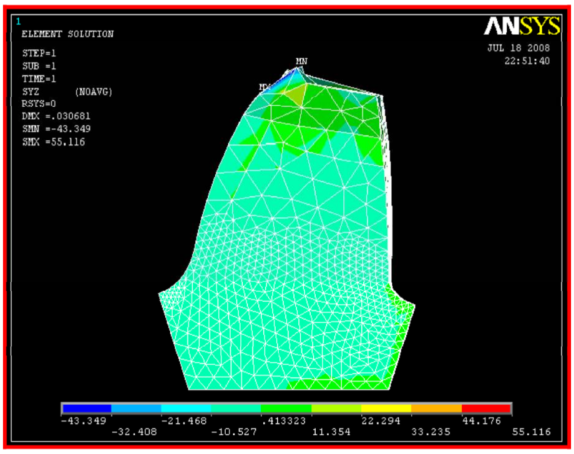


Figure 26: *Z* Component of Stress 118.524 mpa

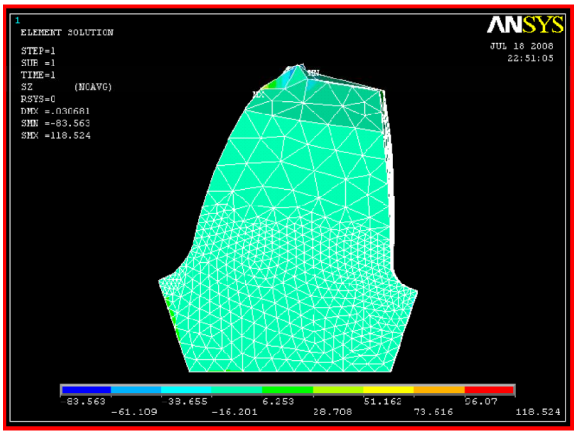
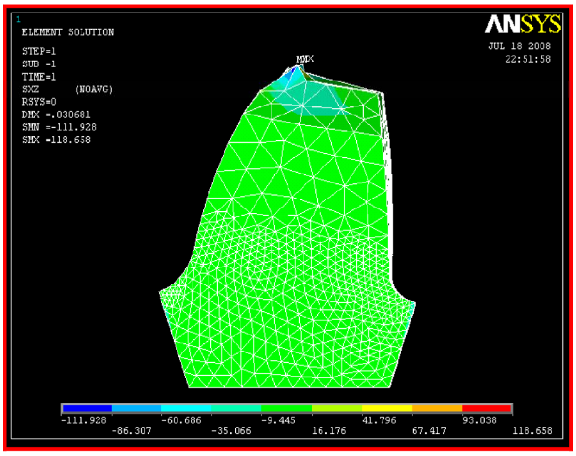


Figure 27: *XZ* Shear Stress 118.658 mpa



RESULTS AND DISCUSSION

Table 1: Comparison of Results

Result	*Ft* = 87394.43 *N*	*Ft* = 134715 *N*	% Increase
Deflection	0.019181 mm	0.0306 mm	59%
X Component of Stress	102.034 mpa	218.055 mpa	113%
Y Component of Stress	99.702 mpa	171.344 mpa	71%
Z Component of Stress	66.551 mpa	118.524 mpa	78%
XY Shear Stress	112.655 mpa	216.454 mpa	92%
YZ Shear Stress	25.884 mpa	55.116 mpa	113%
XZ Shear Stress	25.899 mpa	118.658 mpa	358%
Equivalent Von Mises Stress	286.495 mpa	515.319 mpa	79%
Von Mises Strain	0.001418	0.00255	79%

CONCLUSION

In the present study, effective methods to estimate the root bending stresses by the finite element analysis.

It was found that for tooth load 87394.43 *N* maximum deflection is 0.19181 mm and the stresses is 286.495 mpa which is less than the maximum design stress, i.e., 433.33 mpa for material under consideration and also if the force exceed 115000 *N* maximum deflection is found to be 0.266 mm and stresses is 443.623 mpa, which will break the tooth.

REFERENCES

1. Joseph E Shingley (2003), *Standard Book of Machine Design*, TaTa McGraw-Hill International Book Co.
2. Krishnamurti C S (1991), *FEA Theory & Programming*, 3rd Reprint, Tata McGraw Hill Publishing Company Limited, New Delhi.
3. Modak J P (2006), *Vibration Based Condition Monitoring of Drag Line*, July.
4. Peter Ridley (2001), "Dragline Bucket and Rigging Dynamics", Australian Conference on Robotics and Automation Sydney, November 14-15.
5. Reddy J N (1977), *Finite Elements Method*, TaTa McGraw-Hill Edition.
6. Robert D Cook (1989), *Concept & Application of Finite Elements Method*, 3rd Edition, John Wiley & Sons.
7. Robert I Norton (1996), *Machine Design an Integrated Approach*, Upper Sddle River, Prentice Hall.
8. Shiwalkar B D (1990), *Design of Machine Elements*, Central Techno Publication, Nagpur.
9. Sonpimple M K Phirke (2007), *CAD Modeling & Analysis of Dam Gate Lifting Hoist Mechanism*, Proceeding of IFToMM.
10. Tirupathi R Chandrupatla and Ashok D Belegunda (2007), *Introduction to Finite Elements in Engineering*, Pearson Education.
11. Zeping Wei (2004), *Stresses and Deformations in Involute Spur Gears by Finite Element Method*.