AMSE JOURNALS-AMSE IIETA publication-2017-Series: Modelling C; Vol. 78; N°3; pp 392-402
Submitted Dec. 26, 2017; Revised Jan. 04, 2018; Accepted Jan. 08, 2018
https://doi.org/10.18280/mmc_c.780310

Modelling of Dragline Bucket for Determination of Stress

*Shah Fateh Azam, **Piyush Rai

*Mining Enggering Department, Phd.Scholar IIT (BHU), Varanasi, (UP) 221005, India

(shahfateh.azam29@gmail.com)

**Mining Enggering Department, Professor, IIT, (BHU), Varanasi, (UP) 221005, India

(prai.min@itbhu.ac.in)

Abstract

Dragline is a substantial and costly machine. It is used in mining industries for removal of overburden, and the bucket is the crucial part of dragline which is applied for the removal of overburden. In this paper, an attempt has been made to analyze the stress analysis in a dragline bucket under the static condition using finite element method. For this purpose, a 3D robust model of the bucket has been developed with the help of auto cad and solid work. This 3D solid model was then imported into ANSYS 14.5 and analyzed under different loading condition. This study aims to find out the equivalent stress, damage, safety factor and fatigue life under static loading condition. Simulation results have highlighted those bucket hitch elements, and arc anchors have high-stress concentration and less safety factor.

Key words

Dragline bucket, FEM, Static condition.

1. Introduction

Draglines are used to remove the overburden (up to 50 m depth) for exposing the minerals in surface mine [4]. These draglines can be more than 4000-ton in overall weight, with bucket capacities ranging from 24m^3- 120m^3.The buckets are dragged against the blasted muck to fill the blasted overburden. The capital cost of dragline is Rs 500 crore approx for bucket capacity of 62m^3. A dragline system consists of a large bucket which is suspended from a boom (a large truss-like structure) with wire ropes. The bucket is maneuvered using some ropes and chains. The hoist rope, powered by large diesel or electric motors, and the bucket is supported to hoist-coupler assembly from the boom. The drag rope is used to draw the bucket assembly horizontally. By skillful maneuver of the hoist and the drag ropes, the bucket is controlled for various operations. A schematic diagram of a large dragline bucket system is shown in fig1.The bucket is an essential part of a dragline, which involves a capital cost of around Rs 13crore approx for 62m^3 of bucket size. The productivity of a dragline bucket is influenced by a host of varying conditions arising from operational environmental and human-based issues, irregularities and inhomogeneities in the working conditions. It has been observed that operational and resultant stress variations are essential issues that cause unsteady stress to induce damage to the bucket and its functional life [6].

Fig. 1. Typically, Dragline Bucket (62m^3)

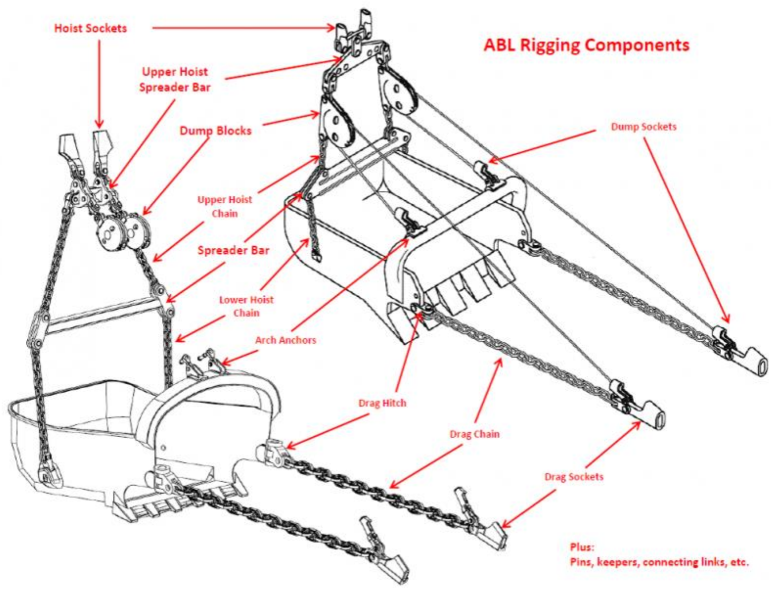


Fig. 2. Schematic Diagram of Bucket [11]

In this study, dragline bucket analysis includes examination and understanding of the stress distribution, life, damage, safety factor, etc. of the bucket body under different loading and working conditions. Numerical modeling technique has been used for examining all the critical parameters.

Finite element analysis (FEA) is the most powerful technique in the strength assessment of the structures working under known loads and boundary conditions. The FEM analysis of any structural component assists in predicting the structural design and masses in which it will behave under stress conditions [10].

Previous Studies: FEA has been invariably used for simulation and analysis under different loading conditions of dragline since a long time. Mouazen and Nemenyl (1999) developed an FEA model to simulate the cutting process of a sub formation with various geometries [1]. Abo-Elnor, Hamilton, and Boyle (2004) a 3D finite element analysis of soil–blade interaction were carried out based on predefined horizontal and vertical failure surfaces, to determine the characteristics of the soil–blade interface surface and study the effect of blade-cutting width and lateral boundary width on predicted forces [8]. Costello and Kyle (2004) developed a numerical technique to obtain the static equilibrium state of a conventional dragline excavation system, including static pose of the bucket, as well as internal loads acting on an element of the excavation system [2]. Coetzee and ELS (2009,2010) used the numerical modeling of excavation bucket filling in a complicated granular flow and optimized the filling process using DEM [3,7]. Manisa Tupkar and Zaveri (2015) designed the excavator bucket with force applied at the tip of teeth of excavator bucket to find the stresses [5]. Golbasi and Demirel (2015) developed a 3D model of the bucket and found out the stresses using Abacus software [6].

2. Methodology

Numerical modeling: The finite element method (FEM) is a relatively new and efficient numerical method [9]. In this paper finite element method has been used for simulating and analysis purpose. Finite elements are used in design improvement and optimization purpose for any mechanical parts. In this paper, ANSYS software has been used for analysis. The simulation includes equivalent stress, deformation, fatigue life and safety factor under different loading conditions. The subsequent process of simulation has been expressed in below paragraph.

The geometry of a Dragline Bucket: A solid model of a dragline bucket is created in the auto cad. The real-time dimension of a 62 m^3 dragline bucket is taken for study as depicted in Fig. 3.

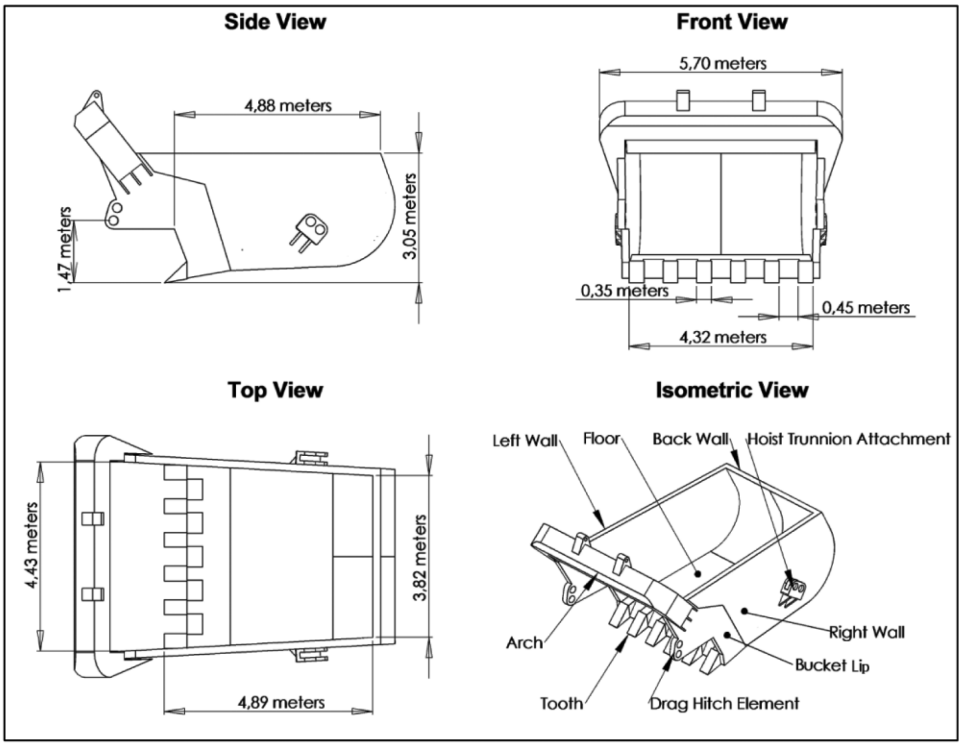


Fig. 3. Dragline Bucket Views from Different Perspectives [6]

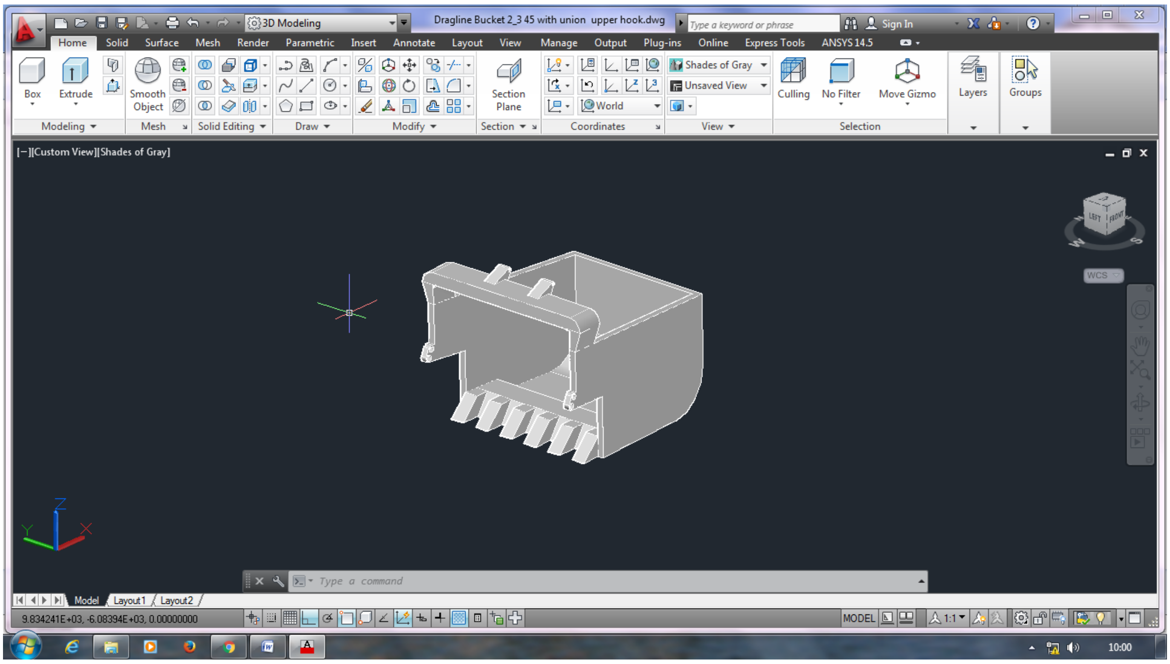


Fig. 4. 3D Solid Bucket Model

The auto cad generated solid model file was imported to solid works and converted to IGES file, which was then introduced to ANSYS 14.5 to apply loading and boundary condition for simulating the dragline bucket and the existing stress environment on the dragline bucket.

Material property

Table 1. Material Properties of Dragline Bucket

Material	**Steel**
Density	7.85e-006 kg mm^-3
Tensile Ultimate Strength	460 MPa
Tensile Yield Strength	250 MPa
Poisson's Ratio	0.3
Young's Modulus	2.e+005

According to the solid model, the capacity of the bucket is $62m^3$, and the mass bucket is 70 ton. When overburden material is filled in the bucket, then the total mass of the bucket is increased naturally. Considering sandstone as full content and density of sandstone varying between 1.8 to 2.40 ton/m^3, the analysis is done under different loading conditions.

Simulation and Analysis: ANSYS is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. Ansys simulation process provides finite elements to model behavior, supports material models and equation solvers for a wide range of mechanical design problems [10]. The meshing of the solid bodies is carried out using tetrahedron method and element size is taken at 50mm.Meshing pattern is illustrated in fig 5. The resultant meshing body incorporates 115982solid elements and 200292 nodes.

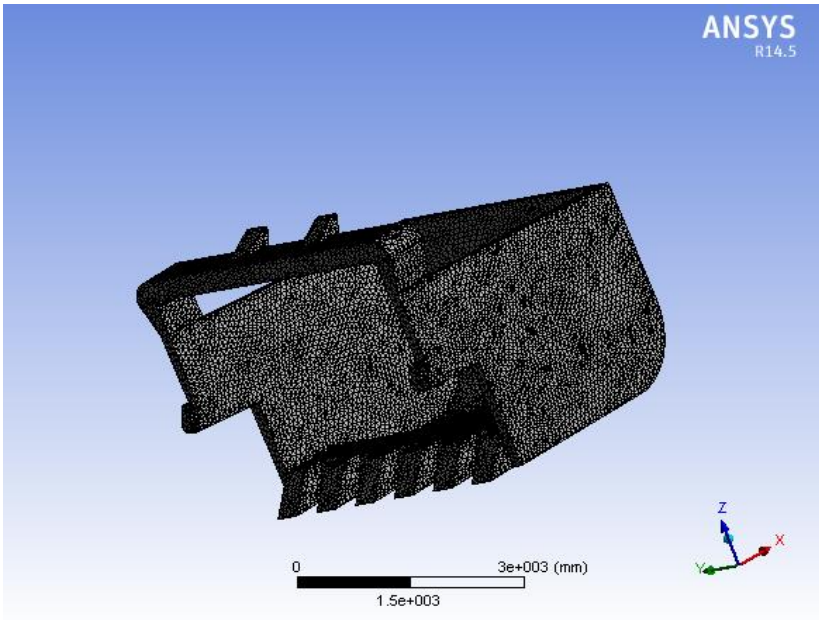


Fig. 5. Meshing Body of Bucket

a b

Fig. 6. (a, b) Empty and Loaded Bucket in Static Conditions

Boundary and loading condition: Analysis is done under the static situation in which hitch element and arc anchors have been considering to be fixed. The load has been applied to the base of the bucket. Two conditions have been taken into consideration, to simulate the empty and filled bucket situations. Different loads have been applied to the model in both the cases. The load computation is given as:

When the bucket is empty: The self-weight of the bucket for $62m^3$ size in mine A is 70 ton approx.

F_1= 70 x 10^3x g N

F_1=7x10^5 N

Where,

F_1= Applied load

g= acceleration due to gravity taken as 10 m/s^2

When the bucket is loaded: which represent bucket is filled with material. In this study, Material has been taken as sandstone having density 2.40 ton/m^3.

m =ρxV

m= 2400x62

m =148800kg

Total mass =mass of bucket+ mass of sand stone

=70000+148800

=218800 kg

Total load = mxg xf

F_2 =218800x10x0.85

F_2 =18.59800x10^5N

Where,

m= mass in kg

ρ= density of sand stone ton/m^3

V= volume of bucket in m^3

f= bucket fill factor

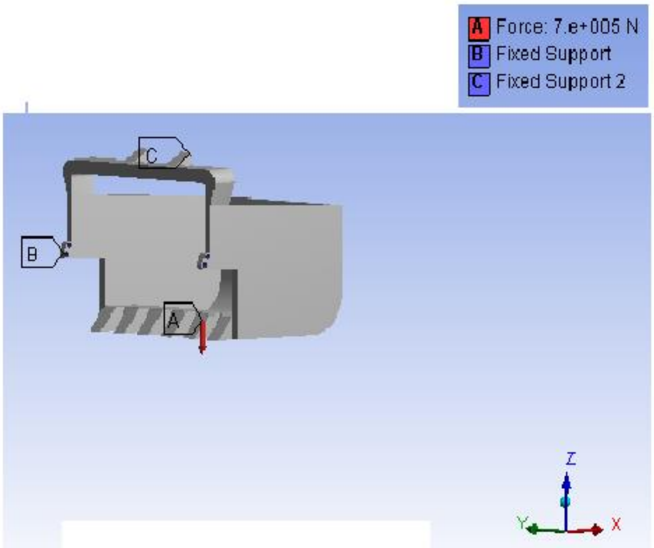


Fig. 7. Model Boundary and Loading Conditions

Result and Analysis: By using finite element method, analysis has been done with the help of ANSYS software. The outcome from the results has been expressed regarding equivalent stress, deformation and fatigue life under different loading conditions of the dragline bucket. Furthermore, the fatigue life has been estimated with the help of life, damage and safety factor of dragline bucket.

In this study, two different loading conditions have been taken into consideration in the analysis.

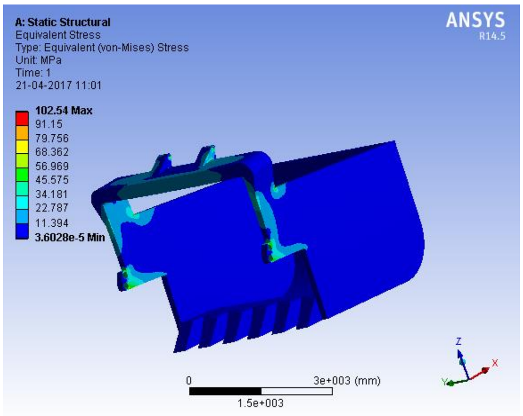
1. **Bucket in self- weight condition:** Which means that bucket is empty and only the dead load of the bucket is used in the analysis. A dead load of the bucket is $7x10^5$N is applied for simulation.

2. **Bucket in loaded condition:** This means that dead load plus payload are used in analysis and the value of load is$18.59800x10^5$ N applied for simulation.

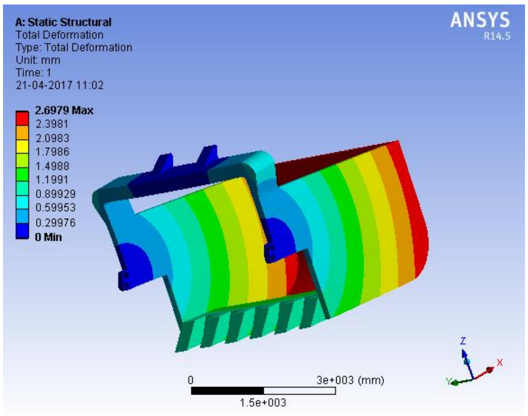
By Applying the boundary conditions and bucket load in ANSYS, following results have been obtained.

Condition 1. When bucket is self-weight condition

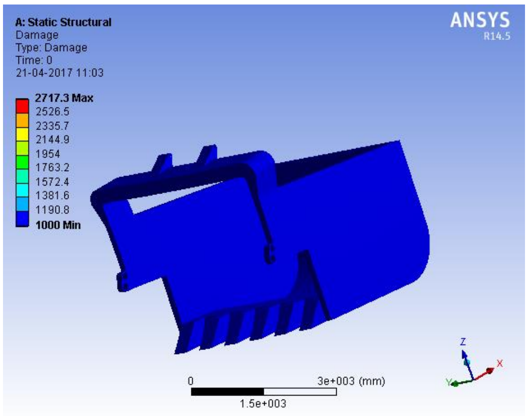
a. **Equivalent Stress (von-mises)**



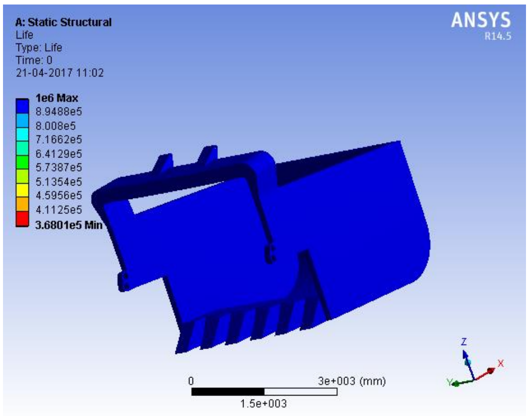
b. Total deformation



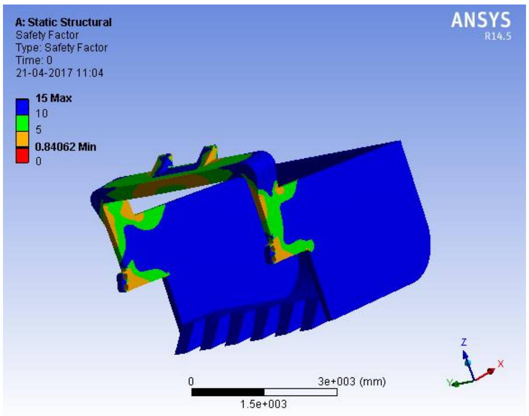
c. Damage



d. life



e. Safety Factor



f. Fatigue Sensitivity

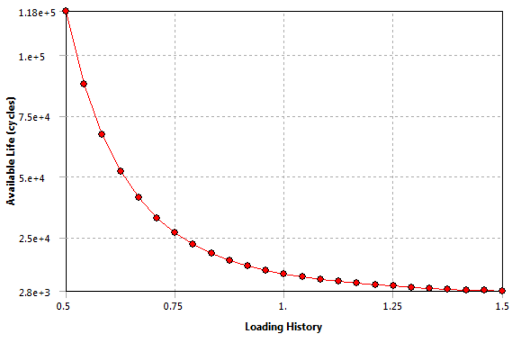


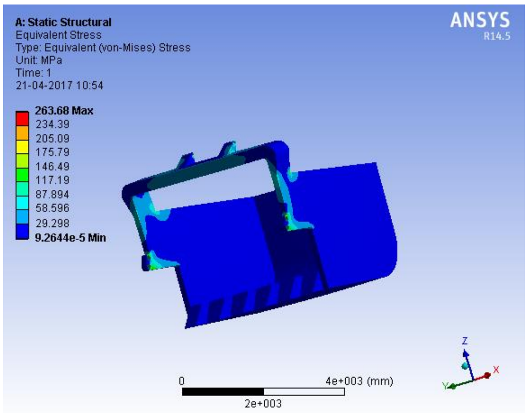
Fig. 8. (a, b, c, d, e, f) Shows the Analysis Results in Bucket Dead Load Condition

For estimation of equivalent stress in ANSYS software von-mises theory has been used for the study. The **von-mises** yield criterion (also known as the Maximum Distortion Energy Theory of Failure) suggests that yielding of a ductile material begins when the second deviatoric **stress** invariant reaches a critical value. Von-mises stress is a haven for design engineers. Using this information an engineer can say his design will fail if the maximum value of Von-mises stress induced in the material is more than the strength of the material. It works well for most cases, especially when the material is ductile [10].

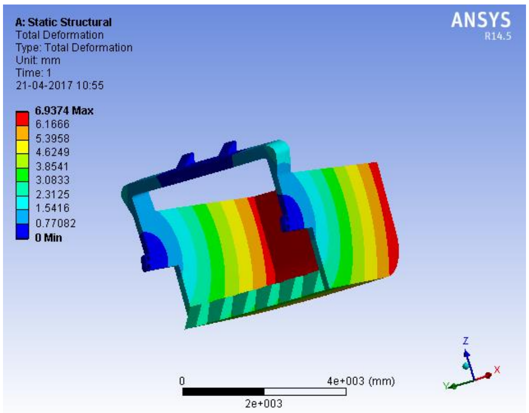
In this analysis, it is found that due to self-weight of the bucket the equivalent stress varies between $3.6x10^{-5}$-102.5 MPa in the hitch element and arc anchors of the bucket. Bucket fatigue life varies from $0.37xe^{6}$- $1xe^{6}$cycle. It has been observed from analysis safety factor is minimum near hitch element and arc anchors shown in fig.8 (e). Fatigue sensitivity analysis shown in fig.8 (f) indicates that if loading condition lowers 50% of its value life of the bucket will increase and if its value increases by 150% then we get decrease bucket life in the cycle.

Condition2. When the bucket is loaded and apply the load is $20x10^{5}$N on the bucket in a static condition.

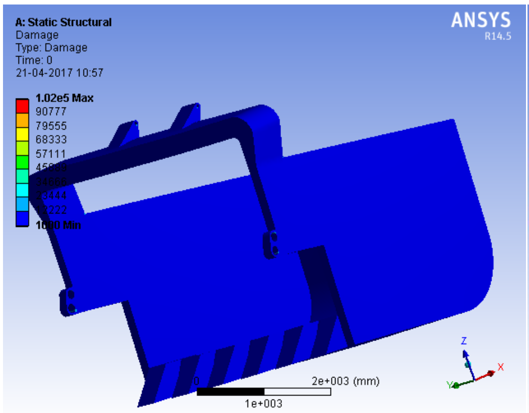
a. Equivalent Stress (von-mises)



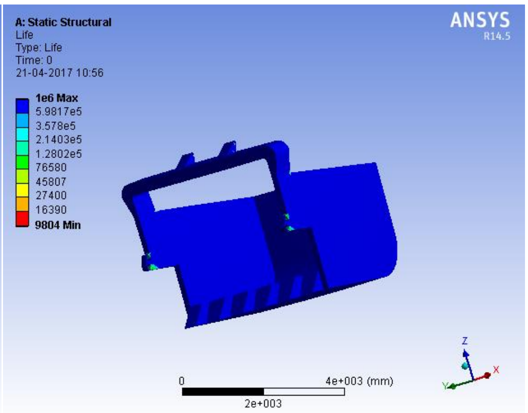
b. Total Deformation



c. Damage



d. Life



e. Safety Factor

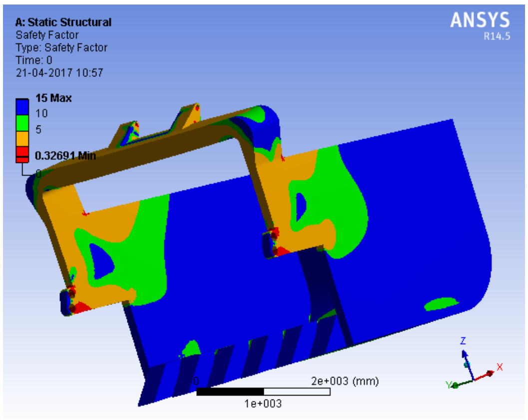


Fig. 9. (a, b, c, d, e,) Shows the Analysis Results in Bucket Loaded Condition

In this analysis, it is found that due to self-weight plus payload applied to the bucket then the equivalent stress varies between 9.264xe^{-5}-263.68 MPa in the hitch element and arc anchors of the bucket. Bucket fatigue life ranges from 9804 - 1xe^{6}cycle. It has been observed from analysis; factory of safety is minimum near hitch element and arc anchors shown in fig.9 (e)

Fig. 10. Shape Damage of Hitch Element Mine A

Conclusion

In both loading conditions, it has been observed from the analysis that the stress concentration was found near the hitch element and arc anchors mean the chances of failure is more in this location. In the case when the bucket is empty, the only dead load is applied on the bucket and after simulation, it has been observed that the stress variation is shown near the hitch element and arc anchors of the bucket and safety factor also decrease in this area. In the loaded bucket condition (filled with overburden), a similar observation has been found, but the magnitude of stress increased and hence, a factor of safety decreased. In the mine A, it has also been noted from the maintenance logs that the failure occurs near the hitch element of the bucket.

Reference

1. A.M. Mouazen, M. Nemenyi, Finite element analysis of subsoiler cutting in non-homogeneous, 1999, Soil and Tillage Research, vol. 51, pp. 1-15.
2. M.Costello, J. Kyle. A method for calculating the static condition of a dragline excavation system using dynamic simulation, 2004, Mathematical and Computer Modeling, vol. 40, pp. 233-247.
3. C.J Coetzee, D.N.J. Els, G.F. Dymond, Discrete element parameter calibration and the modeling of a dragline bucket filling, 2010, Journal of Terramechanics, vol. 47, pp. 33-44.
4. Design and loading of dragline buckets. A CSIRO DMS report.
5. M.P. Tupkar, S.R. Zaveri, Design and analysis of an excavator bucket (IJSRET), 2015, vol. 4, ISSN: 2278-0882.
6. Golbasi, N. Demirel. Investigation of stress in an earthmover bucket using finite element analysis, 2015, The Southern African Institute of Mining and Metallurgy, ISSN: 2225-6253.
7. C.J Coetzee, D.N.J. Els. The numerical modeling of excavator bucket filling using DEM, 2009, Journal of terramechanics, vol. 46, pp. 217-227.
8. Mootaz Abo-Elnor, R. Hamilton, J.T. Boyle. Simulation of soil–blade interaction for sandy soil using advanced 3D finite element analysis, 2004, Soil & Tillage Research, vol. 75, pp. 61–73.
9. Mootaz Abo-Elnor, R. Hamilton, J.T. Boyle. 3D Dynamic analysis of soil-tool interaction using the finite element method, 2003, Journal of Terramechanics, vol. 40, pp. 51–62.
10. https://en.wikipedia.org.
11. www.abl- services.com.