Investigation of Stress Distribution on Dragline Bucket in Opencast Coal Mines

Sathish Kumar Mittapally, A. Akhila Sahithi, and Kairamkonda Harish(✉)

Department of Mining Engineering, University College of Engineering, Osmania University, Hyderabad, India
mskou@osmania.ac.in, kairamkondaharish2433@gmail.com

Abstract. Opencast mining is overwhelming in India due to its higher rate of extraction and the higher extension for applying current innovation for augmented production. Opencast mining is the obvious choice for a property with a large area of mineralization exposed or existing close to the surface and continuing to greater depths. The overburden must be removed in opencast mining to mine the ore beneath. Overburden varies from topsoil to hard rock. Overburden removal accounts for the majority of the costs associated with opencast mining. Draglines are the most cost-effective and popularly used of all overburden removal equipment. Draglines are designed to work around the clock, 365 days a year. Fracture, wear, tear, and fatigue failures in the dragline components occur as a result of the demanding working environment, resulting in more frequent maintenance, longer downtimes, and loss of production. The bucket strength and design seem to be the reason for the dragline downtime. The excavator's main working part is the cutting, loading, and unloading bucket. It is the primary source of external loads on the machinery because interactions with ground materials occur. This paper aims to understand the stresses acting on the dragline buckets and the factor of safety of the dragline bucket.

Keywords: Ansys · Deformation · Dragline bucket · Factor of safety · Stress distribution · Solid works

1 Introduction

Opencast mining is the best choice for a property with a large area of mineralization exposed or existing close to the surface and continuing to greater depths. The overburden must be removed in opencast mining to mine the ore beneath. Overburden varies from topsoil to hard rock. Overburden removal accounts for the majority of the costs associated with opencast mining. Draglines are the most cost-effective and popularly used of all overburden removal equipment. A dragline consists of a large bucket of 24 m^3- 120 m^3 which is suspended from a large truss like structure called a boom with wire ropes. The bucket is moved using ropes and chains. Large diesel or electric motors are used to power the hoist and drag ropes. Drag rope is used to drag the bucket horizontally to fill the blasted muck. The bucket is filled when the drag force overcomes the cutting

B. Raj et al. (Eds.): ICETE 2023, AER 223, pp. 1367–1376, 2023.
https://doi.org/10.2991/978-94-6463-252-1_136

Table 1. Laboratory test results

Material	Density (Kg/m^3)	Cohesion (KPa)	Internal friction angle (°)
Sandstone	2404	255.06	34.39

resistance of the ground at the teeth. The bucket is controlled for numerous operations by the hoist and the drag ropes as shown in the below Fig. 1. During dragline operation, the working components are exposed to sudden changes in stress and strain. This causes fractures, wear and fatigue in the parts of the dragline. To estimate the diggability of the dragline and the failures in bucket components, investigation of the interaction between the formation and the bucket tooth and determination of stress distribution on the bucket and its components during penetration and dragging processes are essential [3]. A factor of safety close to 1 means the component is operating close to its elastic limit, while a factor less than 1 signifies exceeding the elastic limit. A factor greater than 1 indicates a trouble-free condition [1].

Therefore, focus on the whole study and analysis is required towards the interaction between formation and bucket teeth to find out the resistive forces, the factor of safety, and Von Mises stress when the bucket is in static condition. Numerical simulation techniques are required to predict the stress distribution on the dragline bucket and its components. Finite Element Method (FEM) analysis is essential for the strength calculation of structures operating under specified boundary and loading conditions. FEM analysis aids in anticipating the future of any structural elements in the manner in which they will behave under stress. Finite elements are used in the simulation and analysis. In this study, Ansys workbench and Solidworks software is used for analysis.

2 Laboratory Tests

The soil samples of Opencast mine - I at Ramagundem area is collected and tested using the compaction test and direct shear test to know the material properties of soil sample such as dry density, moisture content, cohesion, and angle of internal friction as shown in Table 1. Also, the particle size distribution analysis is performed for the determination of the uniformity coefficient and coefficient of gradation.

3 Finite Element Analysis Analysis on Dragline Bucket

Finite element analysis is performed for very compacted soil using static structural in Ansys workbench and Solidworks software to analyze total deformation, equivalent Von Mises stress, and safety factor under static loading conditions such as self-weight conditions and loaded conditions.

3.1 Designing the Dragline Bucket Model Using Solidworks Software

The dragline bucket model was designed using Solidworks software. The bucket parameters used in the bucket design are given in Table 2. The bucket is considered to be made with cast iron. Bucket material properties used in the analysis are shown in Table 3.

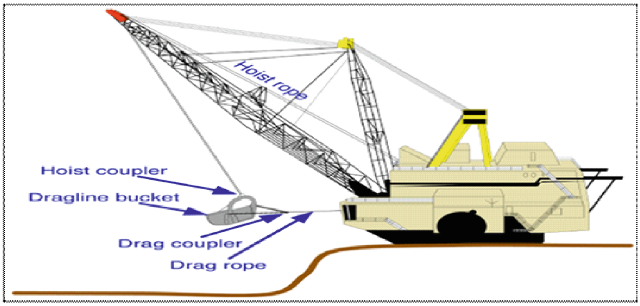

Fig. 1. Schematic view of dragline [2]

Table 2. Dragline bucket parameters

Bucket parameters	Value
Bucket width	3300 mm
Blackwall height	2100 mm
Sidewall height	2200 mm
Sidewall width	3400 mm
Number of teeth	5 units
Distance between the teeth	376 mm
Width of the teeth	350 mm

Table 3. Material properties of the bucket

Cast iron	Values
Young's Modulus	200 GPa
Poissons ratio	0.3
Tensile yield strength	230 MPa
Ultimate tensile strength	460 MPa
Density	7850 kg/m^3

3.2 Discretizing the Model Using Ansys Workbench and Solidworks Software in Static Condition

Using the tetrahedron elements, the model is discretized into a number of finite elements to obtain an appropriate solution. After discretizing the model, the number of nodes is 26634, and the number of elements is 14846. The discretized model and mesh generated by discretizing the model into a number of nodes and elements in the Ansys and Solidworks software as shown in Fig. 2.

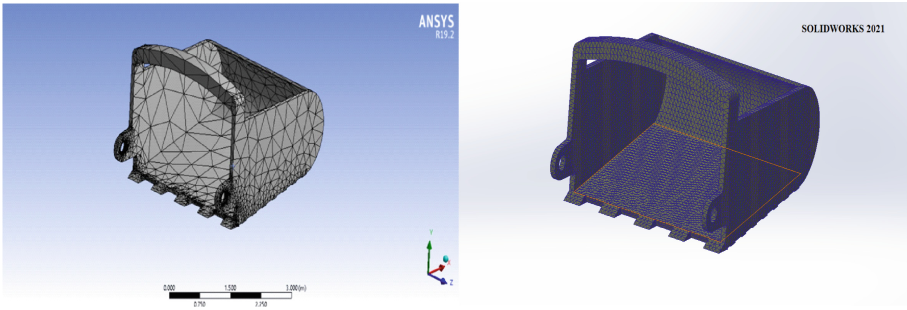

Fig. 2. Discretizing the model using tetrahedron elements in Ansys Workbench

3.3 Boundary Conditions

The analysis is done in the static loading condition in which the hitch element and arc anchors are considered to be fixed. The calculated load has been applied at the base of the bucket. For calculating load, two conditions are considered: (i) self-weight condition and (ii) loaded condition. Boundary conditions were applied to the bucket in self-weight and loaded conditions using Ansys workbench and Solidworks software, as shown in Fig. 3.

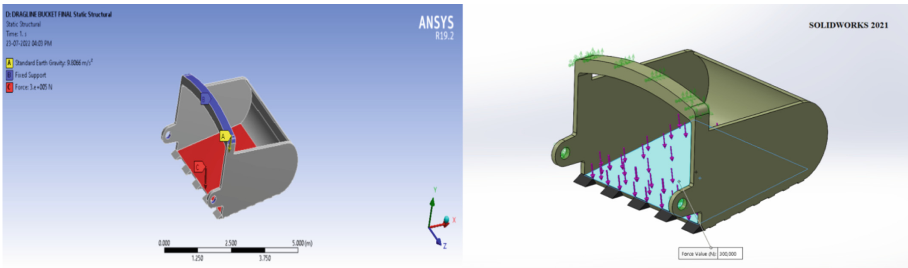

Fig. 3. Boundary conditions of static analysis of dragline bucket in self-weight condition in Ansys workbench and Solidworks software

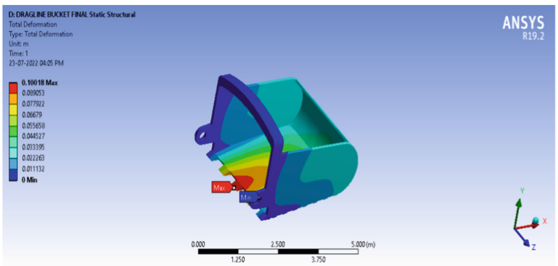

Fig. 4. Total deformation bucket under static and self – weight condition using Ansys workbench

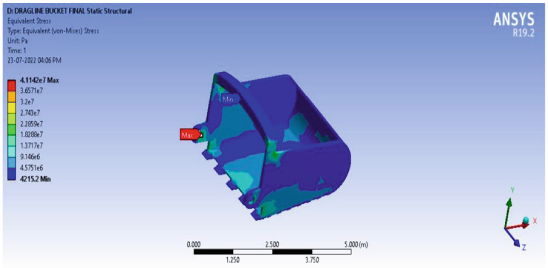

Fig. 5. Equivalent Von Mises stress bucket under static and self – weight condition using Ansys workbench

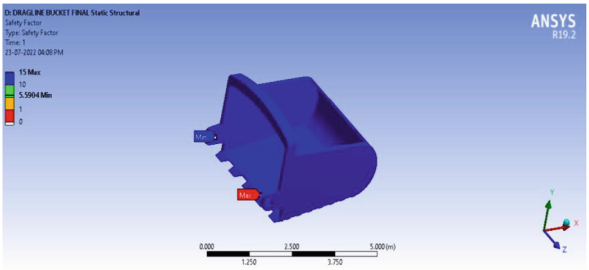

Fig. 6. Safety factor for bucket under static and self – weight condition using Ansys workbench

4 Bucket in Self-weight Condition in Ansys Workbench

Under static loading conditions, finite element analysis is performed to determine the total deformation, equivalent Von Mises stress, Minimum principal stress and safety factor of the dragline bucket using Ansys Workbench. A bucket in self-weight condition means that the bucket is empty, and only the dead load of the bucket is applied in the analysis. The estimated dead load of the bucket is 3×10^5N, based on this load, after the static analysis obtained results are total deformation varies from 0 to 0.10018 m as shown in the Fig. 4, equivalent is Von Mises stress varying from 4215.2 to $4.1142e^7$ Pa as shown in Fig. 5, and the minimum safety factor is 5.59 is shown in Fig. 6.

5 Bucket in Loaded Condition in Ansys Workbench

A bucket in loaded condition means that dead load plus payload is used in the analysis, and the value of the load is computed as 7.45416×10^5 N for broken sandstone. This load is applied to the bucket. By fixing the hitch elements and arc anchors, the results obtained are total deformation is varying from 0 to 0.21877 m as shown in Fig. 7, equivalent Von Mises stress varies from 4193.1 to $8.5925e^7$ Pa as shown in Fig. 8, and the minimum Safety factor is 2.67 is shown in Fig. 9.

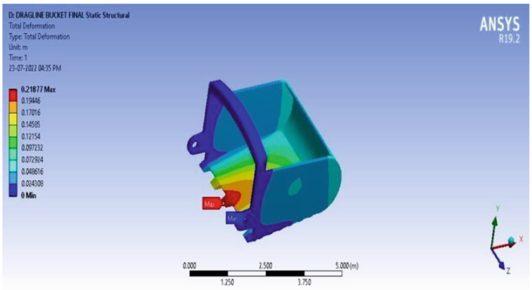

Fig. 7. Total deformation for sandstone under Static loading condition - Bucket in loaded condition using Ansys workbench

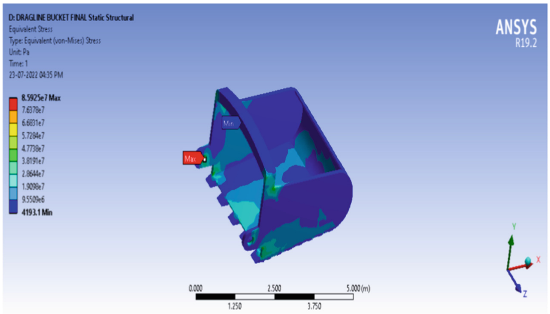

Fig. 8. Equivalent Von Mises Stress for sandstone under Static loading condition - Bucket in loaded condition using Ansys workbench

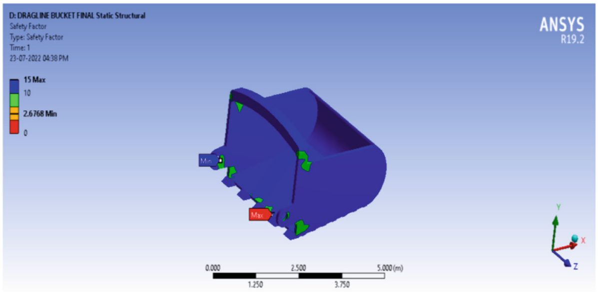

Fig. 9. Safety factor for sandstone under Static loading condition - Bucket in loaded condition using Ansys workbench

6 Bucket in Self-weight Condition in Ansys Workbench in Soildworks

A bucket in self-weight condition means that the bucket is empty, and only the dead load of the bucket is applied in the analysis. The estimated dead load of the bucket is 3×10^5 N; based on this data, the static analysis obtained results are total deformation varying

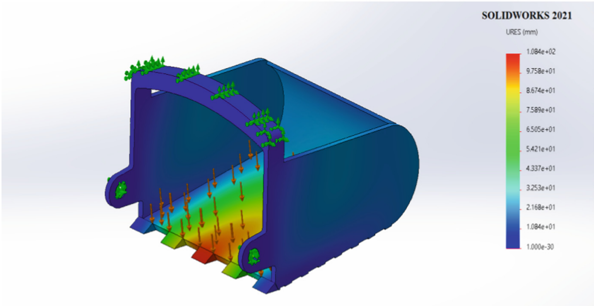

Fig. 10. Total deformation bucket under static and self–weight conditions using Solidworks

from 0 to 0.001084 m as shown in Fig. 10, equivalent Von Mises stress is varying from 56.76 to 3.544 e^7 Pa as shown in Fig. 11, the minimum Safety factor is 6.5 is shown in Fig. 12.

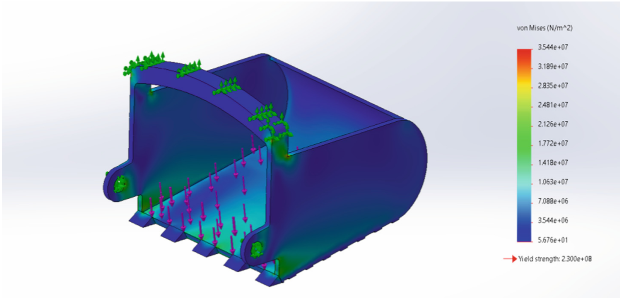

Fig. 11. Equivalent Von Mises Stress bucket under static and self–weight conditions using Solidworks

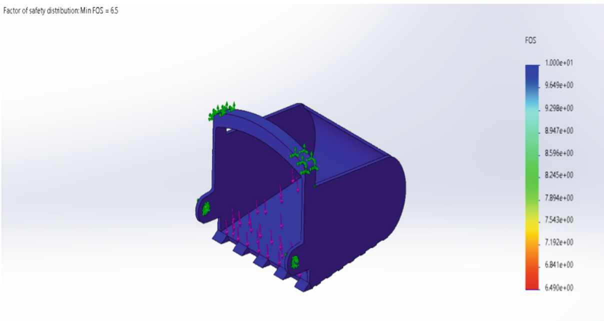

Fig. 12. Safety factor for bucket under static and self–weight conditions using Solidworks

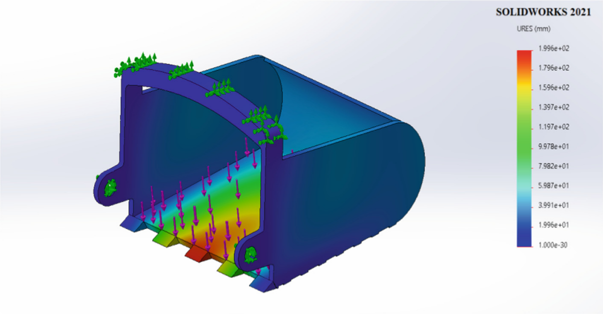

Fig. 13. Total deformation for sandstone under Static loading condition - Bucket in loaded condition using Solidworks software

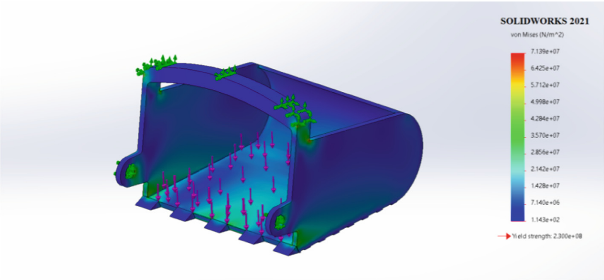

Fig. 14. Equivalent Von Mises Stress for sandstone under Static loading condition - Bucket in loaded condition using Solidworks software

7 Bucket in Loaded condition in Solidworks.

A bucket in loaded condition means that dead load plus payload is used in the analysis, and the value of the load computed as 7.45416×10^5 N for broken sandstone. This load is applied on the bucket. By fixing the hitch elements and arc anchors, the results obtained are total deformation varying from 0 to 0.1996 m, as shown in Fig. 13; equivalent Von Mises stress varies from 1143 to $7.13e^7$ Pa, as shown in Fig. 14; minimum Safety factor is 3.2 as shown in Fig. 15.

8 Result and Discussion

The Total deformation, equivalent Von Mises stress, and safety factors static loading conditions such as self-weight condition and loaded conditions obtained by using Ansys workbench and Solidworks software are represented in Table 4.

The graph representing a comparison between the safety factor of Ansys workbench and Solidworks under static loading conditions is shown below in Figs. 16 and 17.

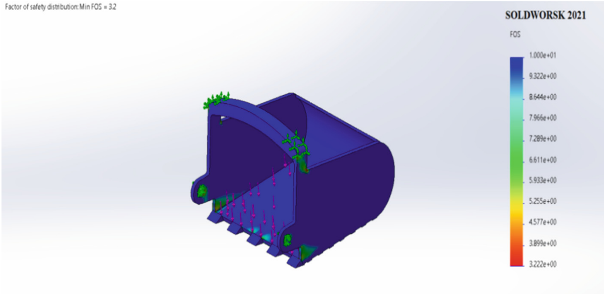

Fig. 15. Safety factor for sandstone under Static loading condition - Bucket in loaded condition using Solidworks software

Table 4. Static Loading condition results using Ansys workbench and Solidworks

Bucket in self-weight condition		
Analysis	**Ansys workbench**	**Solidworks software**
Total deformation (m)	0 -0.10018	0- 0.1084
Equivalent Von Mises stress (Pa)	4215 – 4.1142e^{7}	5676 – 3.544e^{7}
Safety factor	5.59	6.5
Bucket in Loaded condition		
Analysis	**Ansys workbench**	**Solidworks software**
Total deformation (m)	0–0.21877	0–0.1996
Equivalent Von Mises stress (Pa)	41931–8.5925e^{7}	1143–7.139 e^{7}
Safety factor	2.67	3.2

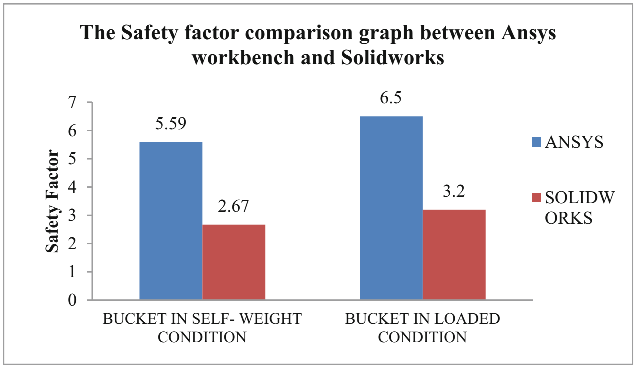

Fig. 16. The safety factor comparison graph between Ansys workbench and Solidworks

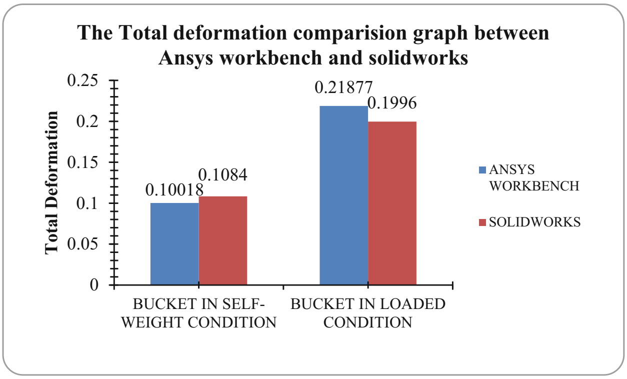

Fig. 17. The total deformation comparison graph between Ansys workbench and Solidworks

9 Conclusions

A factor of safety greater than 1 indicates proper functioning without issues. The analysis shows that stress concentration is maximum near the hitch element when the bucket is in a self-weight condition. In the loaded condition, it has been found that the magnitude of stress increased at the hitch element. As this maximum stress may not cause the failure of the whole body of the dragline bucket, overloading the bucket may cause fatigue failures to the critical components of the bucket.

References

1. Azam, S. F., Rai, P.: Stress analysis of dragline bucket in operating condition. Modelling, Measurement and Control C. 80(1), 10-15 (2019).
2. Bewley, A., Shekhar, R., Leonard, S., Upcroft, B., Lever, P.: Real-time volume estimation of dragline payload. 1571 – 1576 (2011).
3. Gölbaşi, O., Demirel, N.: Investigation of stress in an earthmover bucket using finite element analysis: A generic model for draglines. Journal of the Southern African Institute of Mining and Metallurgy, 115(7), 623–628 (2015).