International Journal for Ignited Minds (IJIMIINDS)

Finite Element Analysis and Optimization of Loading Bin and Mining Platform

Bipin J[a], Hanumanthlal S[b] & Harish H[c]

[a]*PG Student, Machine Design, V.V.I.T, Bengaluru, India.*
[b]*Asst. Prof, Dept. of Mechanical Engineering, V.V.I.T, Bengaluru, India.*
[c]*Asst. Prof, Dept. of Mechanical Engineering, V.V.I.T, Bengaluru, India.*

ABSTRACT

In the present work, a frame structure and bin structure used in the mining applications are analyzed for structural safety conditions. Initially theoretical calculations are carried out to find the minimum section dimensions for both frame and the bin structures. The geometries are built using Ansys preprocessor. Due to regularity of the members, shell and beam elements are used after creating the models using areas and lines. For the first frame structure, lines are meshed with beam188 element with appropriate section properties and ribs are meshed with shell elements. The analysis results shows complete safety of the frame structure for the given 35 tons load. Similarly the bin structure is built for the given specifications. The geometry is built using areas and lines. Here areas are meshed with shell elements and lines are meshed with beam188 elements. Appropriate properties are assigned and analysis is carried out for 16 ton ore structure. The result shows complete safety of the problem. But higher factor of safety is observed in the problem. So design iterations are carried out by varying thickness of ribs and shell until the design reaches to the allowable limits of deformation. Overall 48% reduction in the bin structure is observed. So finite element analysis can be effectively used to reduce the overall weight and in turn cost can be reduced. Similarly maintenance also comes down due to lesser weight of the structures.

Keywords *– Mining, Frame, Loading bin, Analysis, Optimization*.

1. INTRODUCTION

Here we mainly concern about weight optimization. The load of 35 ton is applied on mining frame and load of 16 ton is applied on loading bin. Here we have to do weight optimization for both the structures so that it should be safe for given loads. It is a method of extraction of raw materials from inside and surface of earth. Mining is important for production of materials which we use in our daily life. For example materials like iron, gold, silver, tin etc. Mining is foundation for modernization of industries.

In mining process the platform is used as supporting member to lift the ore from inside surface of the earth. Due to this the large amount of stress are acting on the platform. We should design the structure in such a way that, it should be capable to withstand that stress without deforming. In this project we are concerned about optimization so that we can reduce the wait and made the structure in less cost.

Bin is used to collect the ore which is lifted from earth which is approximately 16tons; this load of 16ton is applied on bin surface. Loading bin members are designed by using modeling software. In this project, surface members are meshed with shell element and section are modeled by beam element. The bin is made by creating rib structure.

Optimization is a popular subject in finite element analysis, and is becoming more important goal in the product development process analysis. This trend is facilitated by the ever-increasing computing power used to solve analysis problems. For the design engineer, it is often the real end goal.

During initial stages of civilization the people use stones and ceramics, later they found the metal in earth's surface. They used these metals to create tools and weapons which they used for purpose of hunting and cutting.

According to archaeological department records the oldest place of mining is Lions cave which is now present in Swaziland. According radiocarbon dating results it is about 43,000 years old; hematite mineral is mined by Paleolithic humans which contain iron ore.

Romans invented the mining engineering technology and are the one to do mining in large scale. They used fire and water for mining. Here first they heat the rock by firing and then quench it by stream of water. Due to thermal shock the rock gets cracked and later thy separate the cracked rocks.

2. METHODOLOGY

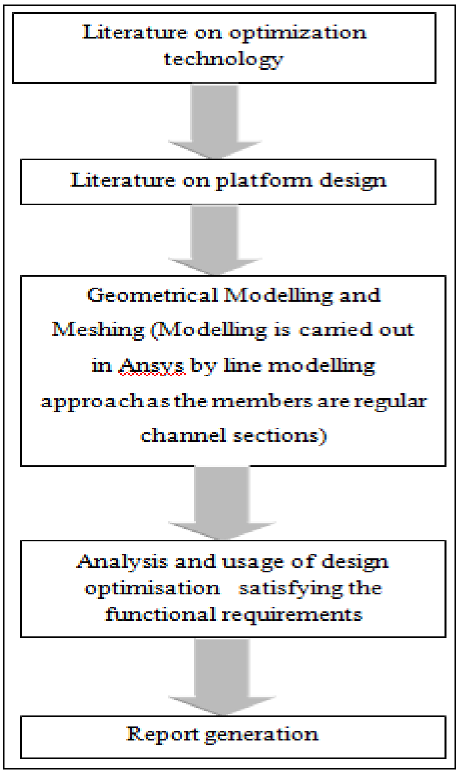


Fig 1: Methodology Flow Chart.

Here the data and required optimization techniques which are used in optimization process are collected. This project is based on mining structure and its safety, the survey is done on mining process and type of equipments which are used in mining and which type of structures comes under mining process. The dimensions required for the structure and the amount of load is acting on structure is calculated using available data and standards. After collecting all required necessary data geometrical model is done by using ANSYS (line modeling approach as the members are regular channel sections). After creating model analysis is done on model and finally report is generated.

3. GEOMETRICAL MODEL BUILT UP

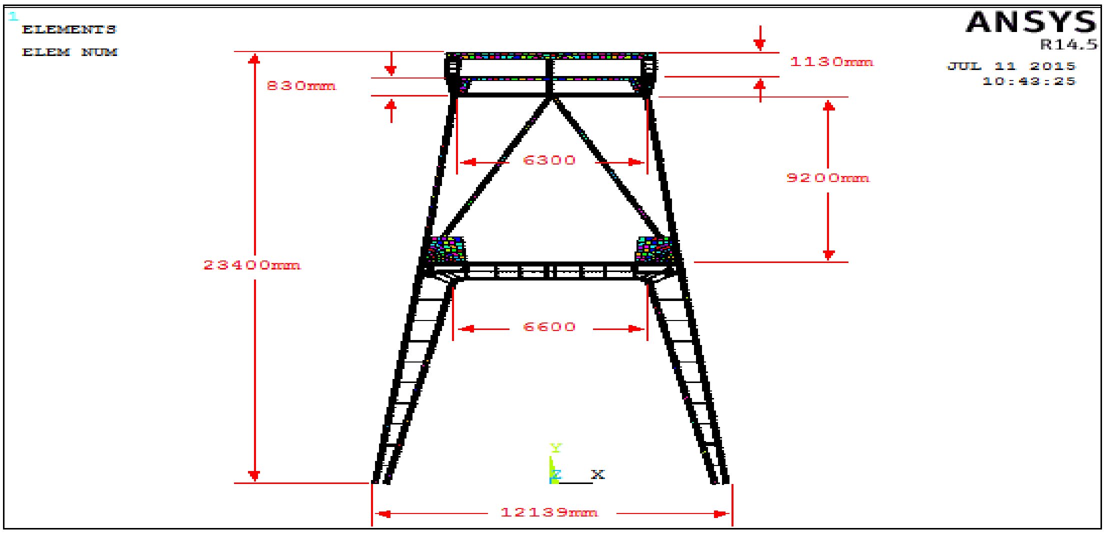


Fig 2: FE Mesh and Major Dimension of Frame Model.

The above fig 2 shows the frame model. The total height of the frame is 23400mm, lower width of the frame is 12139mm, and Top width is 6300 mm. The frame is designed by using four types of sections. The four sections are ‘L’ Section, ‘I’ Section, Rectangular section and Box section.

4. THEORETICAL CALCULATIONS

The beam dimensions required to take a load of 35 tons as distributed load.

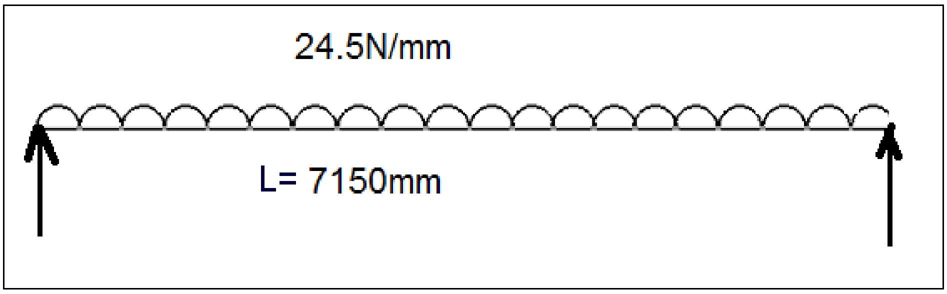


Fig 3: Top Beam Loading.

Maximum bending moment for uniformly distributed load $M_{max} = \frac{w*l^2}{8}$

$M_{max} = \frac{24.5*7150^2}{8}$ =156562656 N-mm

Section required for the problem for an allowable stress of 140

$140 = \frac{M*\left(\frac{h}{2}\right)}{I}$

Assuming the height of the beam as 200mm.

Minimum moment of inertia required is $I = \frac{M*\left(\frac{h}{2}\right)}{140}$

$=111830469 mm^4$

$= 0.111e9 mm^4$

Bin Thickness calculations:

Width of the bin: 654mm (at the bottom)

Width of the bin: 3500mm (at the top)

Height of the bin: 5180mm

Thickness of the bin: 8mm

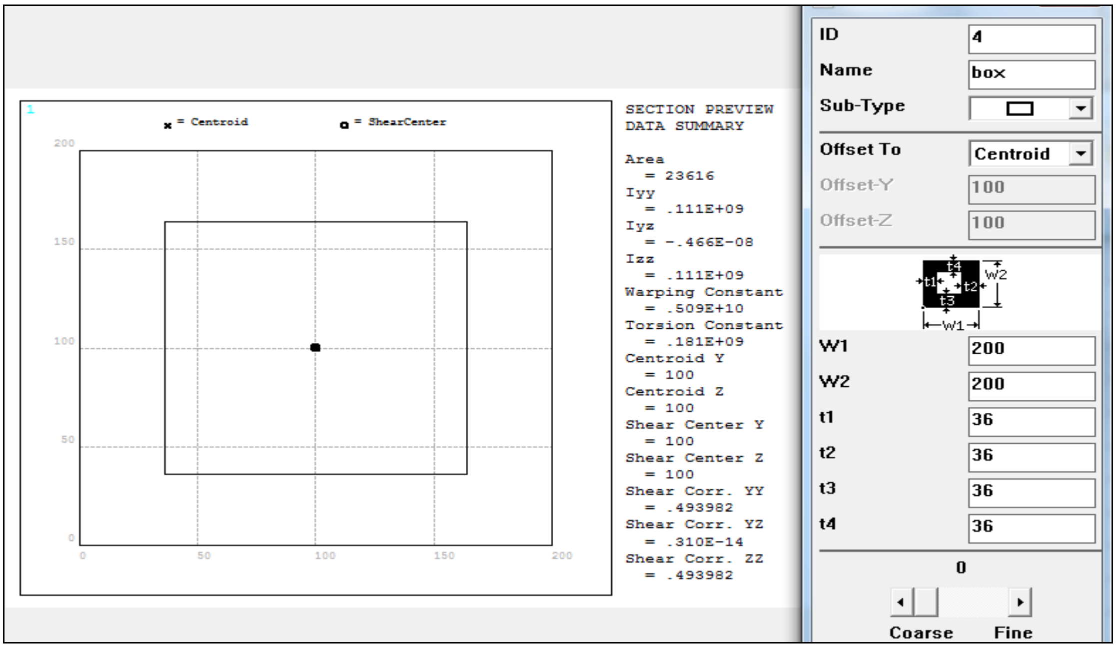


Fig 4: Section Considered for the Problem (0.111e9 mm^4).

Allowable shear stress of the bin τ =70 N/mm^2.
Shear load acting on the bin: 160000N
Shear stress developed =160000/ (8*654) =30.58N/mm^2.

Since developed shear stress is less than allowable critical shear, the geometry is safe for the given loading. The section considered for the problem is as shown in the above fig 4.

The Fig 4 shows the cross section of the beam with moment of inertia of 0.111e9mm^4 which is equal to the required section of 0.111e9mm^4. Since additional ribbing members will be added the structure will be safe for the given loads.

5. RESULTS AND DISCUSSION

5.1 Platform Analysis

5.1.1 Maximum Stress Location in Frame due to Self Weight

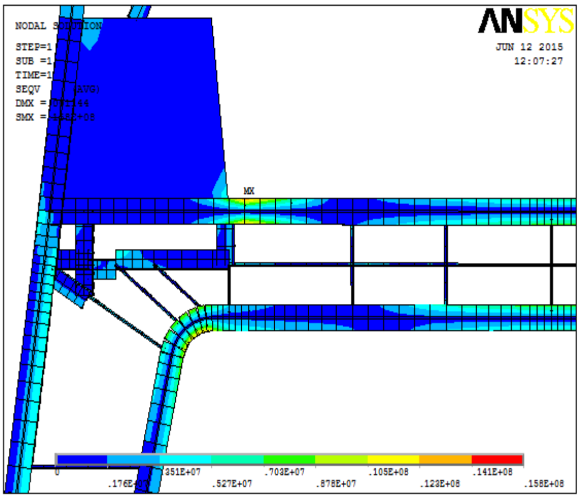


Fig 5: Maximum Stress Location

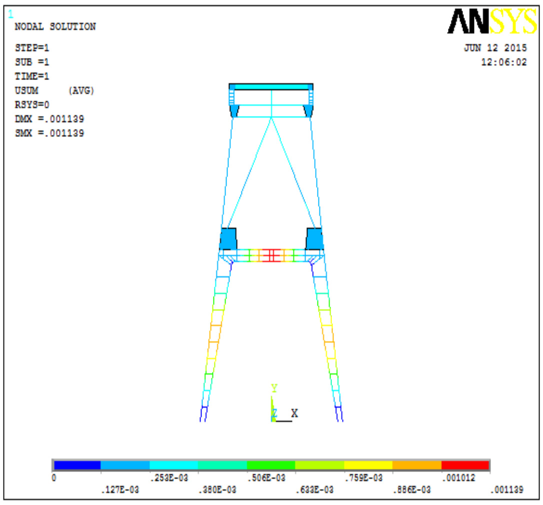


Fig 6: Displacement plot

The above figure 5 shows the region where section 1 is fixed with section 3 which interns covered with the plate which acts as supporting member when the structure is under loading condition. Maximum stress of 15.8Mpa is developed in this region.

The figure 6 shows the displacement of 1.13mm due to self-weight of the structure.

5.1.2 Maximum Stress and Displacement due to External Load

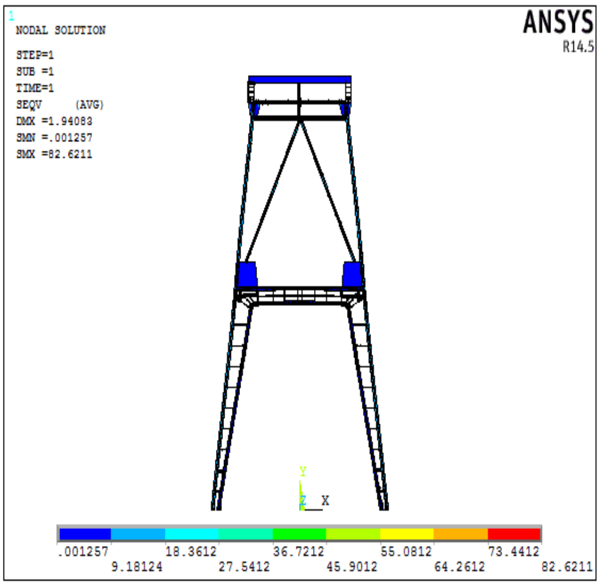


Fig 7: Vonmises stress plot

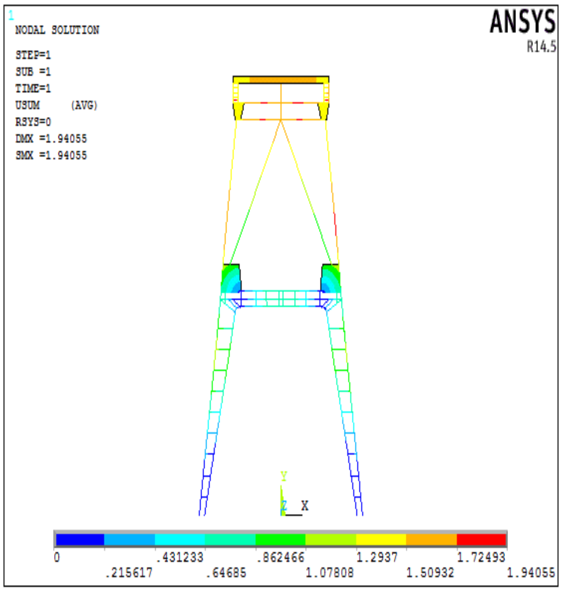


Fig 8: Displacement plot

The figure 7 shows the vonmises stress of 82.4MPa due to application of external load of 35000 kg on the plotform member.

The figure 8 shows the dispplacement of 1.94mm due to application of external load of 35000 kg on the plotform member.

5.1.3 Modal Analysis

The modal anlsysis is carried out to determine the natural frequency of the system to prevent it from resonance. The modal anlysis of the plotform is carried out and first 10 non zero values are as shown in the table 1 and the first mode shape is captured and represnted in the figure 10.

Set No	Frequency(Hz)
1	1.004
2	4.2378
3	4.2975
4	5.2379
5	7.08
6	7.13
7	7.49
8	7.57
9	8.11
10	10.17

Table 1: Modal Frequencies in the Frame

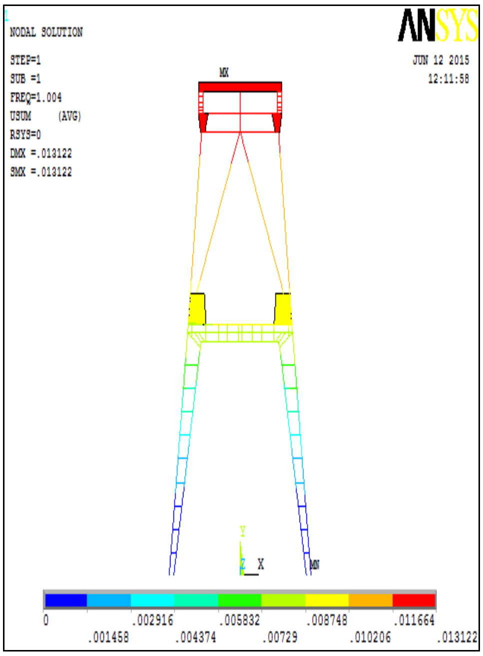


Fig 9: Mode Shape 1

5.2 Bin analysis

5.2.1 Self Weight Analysis

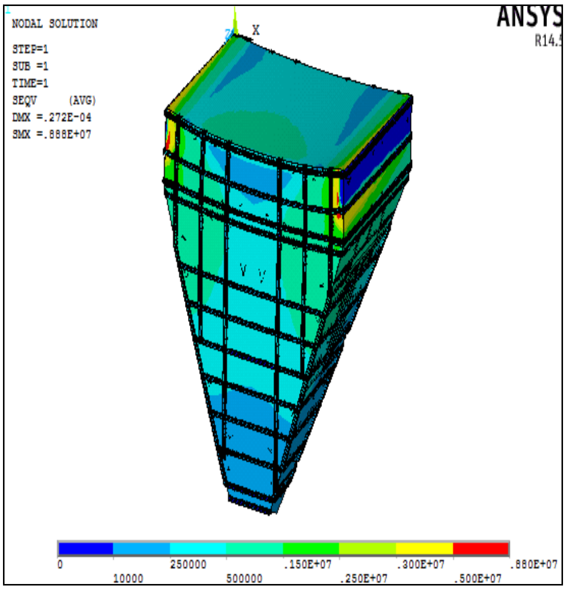


Fig 10: von-Mises stress plot

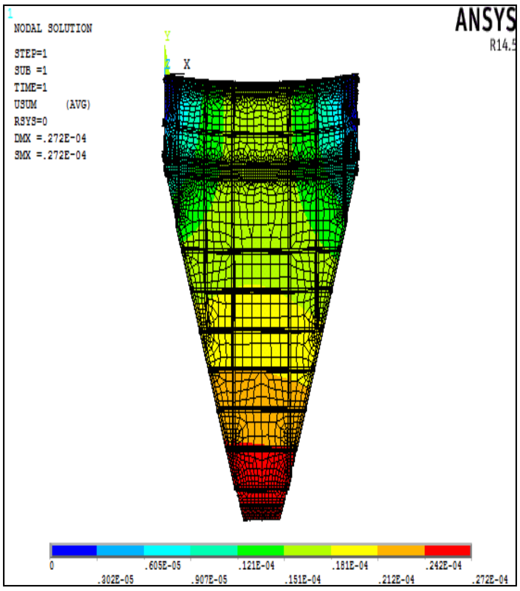
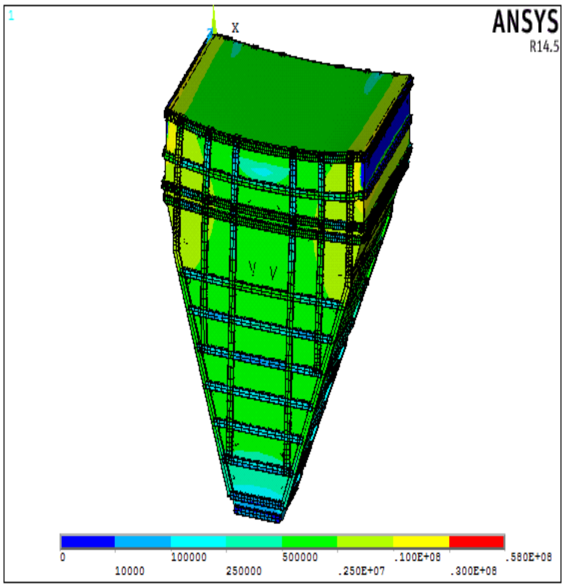


Fig 11: Displacement Plot in the Problem

The figure 10 gives deviation of stress in structure. Maximum von-Mises stress observed is 9Mpa. The stress is mainly concentrated in the region of supports. Other then the support region, stress is observed to be minimum.

The figure 11 shows maximum deformation of 0.02mm (0.272e-4m). Maximum deformation is observed at the bottom. This is mainly attributed to unsupported geometrical condition of the problem. The supports are given only at the top. The colors show variation of deformation in the structure.

5.2.2 Bin Analysis due to External Weight



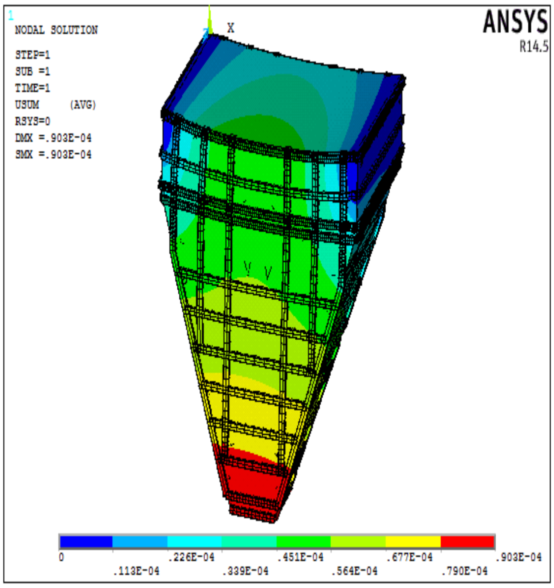


Fig 12: Final von-Mises Stress　　　　Fig 13: Final Displacement

Above figure 12 shows the von-Mises stress of 58MPa for the bin after doing design optimization.
Above figure 13 shows the final displacement of 0.09mm of the bin structure after application of load.

5.3 Discussion

Design analysis of both the frame and the bin structures are analyzed for stress and deformation and results are presented. The frame structure is built using beam188 and shell elements due to regularity of the cross sections. Many times the structures fail by self weight and so self weight analysis of heavy structures are required. Since the weight of the frame is considerable, self weight analysis is carried out by specifying density and acceleration loads. The result shows 1.1mm and 15.8Mpa stress in the members. Both the stress and deflections are less then allowable stress of 140Mpa and deflection limit of 8.4mm. So the frame structure is safe for the self weight. Maximum stress location is taking place near the curved region. So this can be improved if the stress exceeds the limiting stresses. The symbol 'Mx' represents maximum stress location. Further analysis with the external load of 35 tons (350KN), the structural results for stress and deformations are represented. The deflection has been increased from 1.1mm to 1.9mm which is less than the allowable deflection limits. Similarly the stress value of 82.62Mpa is also less than the allowable limit of 140Mpa.So the structure is safe. To find safety (factor of safety) in the components of assembly individual member results are presented. This helps in identifying the components under higher stress conditions. Further dynamic (Modal) analysis is carried out to find the resonant condition of the problem. The motor frequency should not match with the natural frequencies of the frame structure to prevent any possible resonance. So this helps in motor selection for operation. Further Bin analysis for 16 tons (160KN) shows very little deflection of 27.2microns and stress level of 8.8Mpa which shows undesirable factor of safety in the problem. So the structure is optimized by changing the bin thickness along with rib thickness. The result shows a reduction of 48% weight reduction maintaining the structural safety of the loading bin.

Details	Thickness		Deformation (mm)	von-Mises Stress (MPa)	Weight of the Structure (N)
	Shell Thickness (mm)	Rib Thickness (mm)			
1	8	10	0.0272	8.8	69760
2	7	8	0.0352	14.6	58510
3	6	6	0.562	28.2	47360
4	5	5	0.0829	42.6	39160
5	4.5	5	0.09	58	36250

Table 2: Design Optimization Results of Bin.

6. CONCLUSION

The Frame Structure and the bin structure used in the mining applications are analyzed using finite element software called Ansys and is validated with theoretical calculations. The overall summary is as follows.

- The results are analysed for structural deformation and stress conditions. The developed stresses and deformations are well within the allowable limits of the problem.
- For 16 ton capacity. The initial result shows the development of stress and deformations are very low compared to the allowable limits. So design optimization is carried out and the members are optimized (both bin shell and ribs). The design optimization process shows almost material saving of 48% representing finite element process can be effectively used to reduce the weight of the structure. But care should be taken to provide weld at the joints and proper welding techniques should be employed. In the analysis, the members are assumed to have complete contact. But in reality it is affected by the process of welding and preparation of members for weld.
- The optimizations are carried out till the maximum limits are reached. Here the maximum limit is specified for the bin structure as 100 microns (Design Specification). Even though stress is limited by 58Mpa(Less then allowable stress of 140Mpa), the problem is limited by the deflections. The deflection is also very important for maintaining the structural stability of the bin structure. Both bin thickness and rib thickness are playing the role in reducing the weight. Weight reduction is very important in the industrial manufacturing to reduce the cost and along maintenance.

REFERENCE

1. R.J.Duffin, E.L.Peterson and C.Zener, "*Geometric programming: theory and applications*", Wiley, New York.
2. G.B.Dantzig "*Linear Programming and Extensions*", Princeton University Press.
3. R. A. Gettatly and R. H. Gallagher, "*A procedure for automated minimum weight structural design*", Part
4. "*I - Theoretical basis, Part II - Applications*", Aero. Quart. Part I, Vol. 17, pp. 216-230 and pp.332-342.
5. M.M. Denn, "*Optimization by variational methods*", McGraw-Hill, New York.
6. G.S.G. Beveridge and R.S. Schechter, "*Optimization : theory and practice*", McGraw-Hill, New York.
7. J.L. Kuester and J.H. Mize, "*Optimization techniques with FORTRAN*", McGraw-Hill, New York.
8. M.J. Panik, "*Classical Optimization: foundations and extensions*", North-Holland Publishing Co., Amsterdam.
9. D.G. Carmichael, "*Structural modeling and Optimization*", Ellis Horwood Chichester.
10. Joe Metrisin (Florida Turbine Technologies, Inc.) 2002.