ASIT SHUKLA
and
PIYUSH RAI

Finite element based modelling and analysis of dragline boom structure

Dragline is the most widely used machine in opencast mining. It is used for removal of overburden in coal mining. During the digging process machine experiences various loadings on its front end structure, which typically consist of boom, bucket and wire ropes. These loads tend to develop stresses in each cycle of the dragline and causing the structural damage.The bucket is connected to the boom by hoist ropes on the top of the boom using boom point sheaves. As bucket start to dig, the forces on the boom start to increase and reached their maximum value at the end of digging cycle. In the current work, only the dragline boom is considered for investigation of stresses. Due to the complexity of the structure, only overall structural behaviour is predicted, and also a single joint is separately analysed for better understating of stresses acting near the joint. ANSYS 18.0 software is used for analysis purpose, and Solidworks software is used for three- dimensional model creation. Beam-to-solid sub-modelling is implemented inside the ANSYS software to minimize the computation time and requirements.

Keywords: *Finite element analysis (FEM), submodelling, boom cluster, fatigue life, ANSYS, solidworks.*

Introduction

Draglines are the giant machines used in the coal mining industry for excavation purposes. Their ability to dig deep down the earth make them even more powerful machines against their counterparts such as shovels and dumpers. Dragline performs overburden removal operation with their large earth digging buckets, which can remove soil more than 108 m^3 (140 yd^3) in a single cycle [1]. The only limitation of these machines is their digging depth and height where they can dump the waste material. Draglines can only dig up to a maximum of 50 to 80 m of overburden due to reach and dump height limitations [2]. Despite this limitation, these machines are still an integral part of opencast mining due to their efficiency and production rate. They provide the lowest overburden removal cost per tonne [3]. A dragline cycle typically consists of the positioning of the bucket to the digging site, dragging the bucket towards the machine to dig and then dump the material at a certain height by hoisting the bucket with the use of hoist rope. Fig.1 shows the schematic diagram of a dragline.

Dragline boom is a truss-like the structure of circular or rectangular hollow members. It consists of three or four main chords and bracing members which are connected to the main chord forming joints, which are known as clusters. A cluster is a very complex joint, and possibilities of failures at these joints are very high as compared to the other components. The welding process is used to connect these bracing members to the main chords for the formation of joint. During the digging cycle, forces on these joint started to increase and reach up to a maximum value.

Currently, only two types of design for dragline booms are in use by the different manufacturers. Bucyrus dragline boom: It has a tubular pipe member design with a triangular cross-section. This type of boom consists of three (3) main chords with lacings members connecting chords to form a

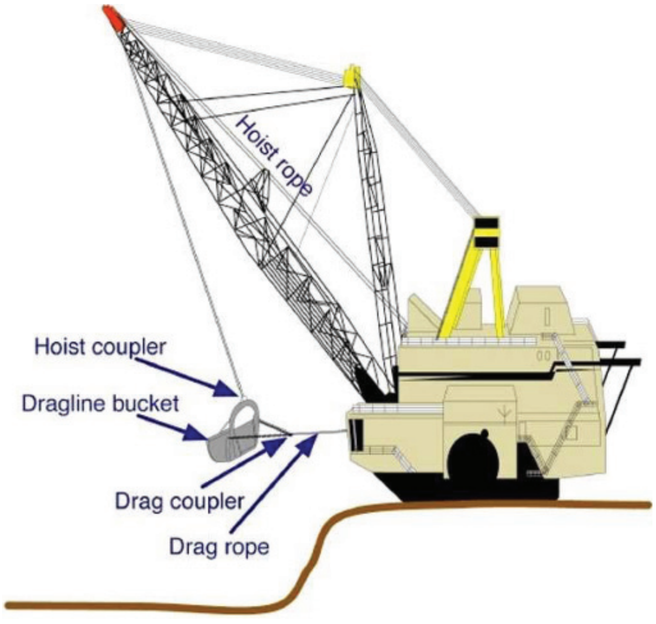


Fig.1 Basic diagram of the dragline

Messrs. Asit Shukla, Department of Mining Engineering, Ph.D. Scholar IIT (BHU), Varanasi, (UP) and Piyush Rai, Department of Mining Engineering, Professor, IIT, (BHU), Varanasi, India E-mail: asitshukla.stp34@gmail.com / prai.min@itbhu.ac.in

lattice type structure. The chord nodes where lacings intersect are identified as “clusters”.

The Marion and P&H booms: These are similar in structure with four (4) main chord design and having a rectangular cross-section. The main chords are I beam wide flange sections; lacings connect the chords forming the lattice. To minimise the possibility of structural failure in chords or cross lacing members, all welds and chord surfaces should be visible for rapid inspection.

Depending on the design, clusters may be overlapping or non-overlapping, as shown in Fig.2. In the current research article only non-overlapping type of joint design is selected for the prediction of fatigue life.

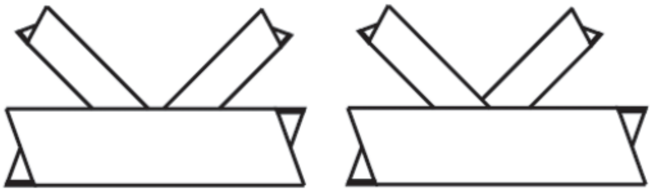


Fig.2 Non-overlap and overlap k joint (CIDECT design guide)

Background literature

For calculating the strength of the structures working under known boundary and loading conditions, finite element analysis is the most significant method. The FEM analysis of any structural element helps in forecasting the structural mass and design, when subjected to stress conditions [1].

FEM is an efficient numerical method [5]. It is used for simulation and analysis. FEMs are used in the design development and optimization of mechanical parts.

Dragline can be modelled as 2D vector loop representation with its front end assembly to calculate loads experienced on the parts and stresses of the boom [6, 7]. Discrete element method and finite element method can be used to formulate the kinematics of the dragline [8, 9]. There is always a regular cracking phenomenon at the welds, which means the cracking must be remotely detected, exactly located and repaired [7]. Various maintenance strategies are suggested to maintain the boom in operating condition. Various researchers worked on the simulation of welding-induced residual stresses in a CHS T joint. They concluded that at some point of the welded area may have higher stresses than the yield stress, but they are not capable of inducing cracks [8]. The on field measurement of stresses can be recorded with the use of strain gauges by installing them on the cluster location. These strain gauges are installed at some distance from the welded toe to measure stresses value during working cycle of the dragline [10].

The components such as boom along with boom foot, boom point sheave and suspension point attachments suffer a significant loading and unloading cycles. It causes them to develop cracks in these components at various locations, which ultimately lead to the complete failure of components if not properly cared and maintained. Although scheduled maintenance of these components is performed on a daily or weekly basis, yet, these failures seem to happen over a specified period. One of the recent failures in the Indian mining industry was the collapse of the dragline boom in a large mine.

Based on the above literature, it can be concluded that very less work is carried out on the design part of the dragline boom. In current research work, non-overlapping k joint is considered for the investigation of stresses and analysis.

Methodology

FINITE ELEMENT METHOD (FEM)

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimisation. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretised finite element problem with unknown nodal values. For a linear problem, a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values.

The steps involved in the finite element methods are as follows.

1. Discretise the continuum.
2. Select interpolation functions.
3. Find the element properties.
4. Assemble the element equations.
5. Solve the global equation system.
6. Compute additional results.

MODELLING

A drawing layout is prepared in Solidworks with all the important dimensions which are recorded from the field. Fig 3 shows the dragline boom design of a typical dragline. As we can see, there are four main chords and some bracing members which form several complex joints. Welding is neglected in the following design as the accurate design of the welded joint is not possible. Main chord diameter is 406 mm with 20 mm thickness and bracing members are of 207 mm with 8 mm thickness.

The design of the dragline boom is created as a multibody part in Solidworks. The multibody part is a top to the bottom approach of creating complex models. In this approach first, we create a wireframe diagram of our structure and then create solid structural members by providing proper dimension to them. Member profiles are saved in the design library of Solidworks and can be used again and again as and when required. Fig.4 shows the boom head and boom foot weldment.

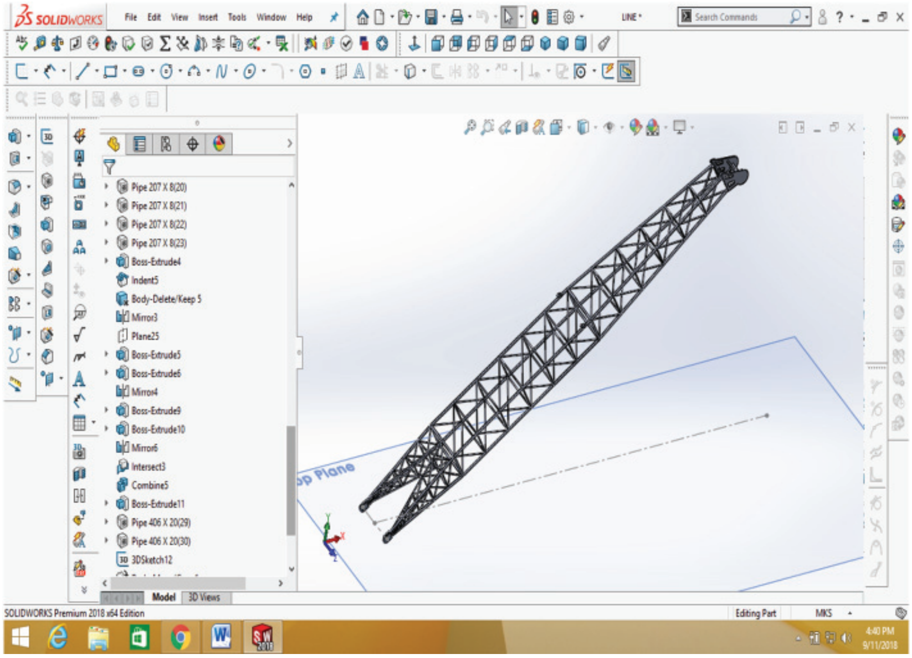


Fig.3 Dragline boom model created in Solidworks

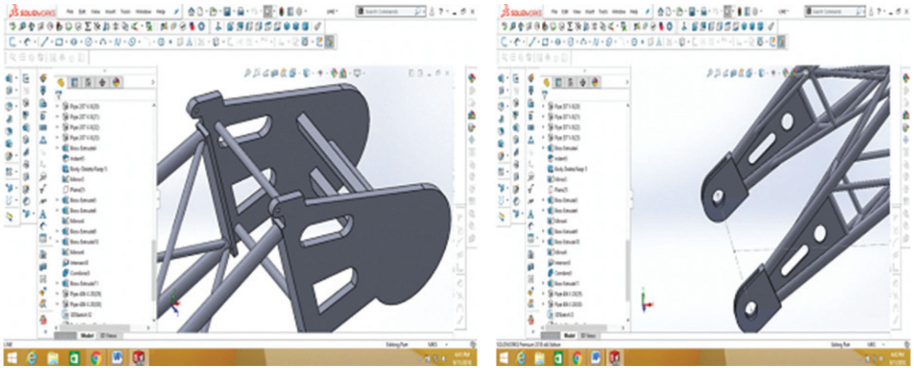


Fig.4 Boom head and Boom foot weldment

Meshing

In FEA, a 3D model is divided into various small segments known as an element based on the size and complexity of the model. Due to the size and complexity of the model, beam elements are considered as they significantly reduce the number of nodes and elements. Beam elements are quicker to solve, and deformation results are assumed to be pretty good for finite element analysis.

The number of nodes and elements are found to be 62674 and 31382, respectively, as shown in Fig.5.

Boundary conditions

Fixed conditions

Boundary conditions are the most crucial part of finite element analysis. To accurately predict the behaviour of the structural component, boundary conditions must be defined precisely so that they represent the real world conditions. Without defining the correct boundary conditions, analysis cannot be carried out accurately, or results may not be of any use.

As shown in Fig.6. A, B, and C are fixed boundary conditions as we cannot accurately model ropes or wires, so at these points, we keep it fixed by using fixed boundary conditions.

Material properties

For the selected design, structural steel is considered as material. Table 1 shows the basic material properties with all the necessary parameters.

Table 1: Material properties for structural steel.

Material	Structural steel
Density	7850 kg/m^3
Tensile strength	460 MPa
Yield strength	250 MPa
Poisson's ratio	0.30
Young's modulus	2.0 e+005 MPa

Depending upon the material selection, the weight of the boom is automatically calculated by the ANSYS software module. The total weight of the dragline boom is calculated as 146 tonnes.

Loading conditions

In the present research work, three loading conditions were observed which are as follows

- A dead load of the boom (self-weight).
- Bucket self-weight is acting on boom point in a vertical direction.
- Bucket payload is acting on the boom in the vertical direction (Dragline dictionary 2014).

Thus total load acting on the boom is the sum of the above three loads. Table 2 indicates the values of the above three loads.

Simulations

Beam-solid submodelling

Accurate prediction of fatigue life and fracture of complex structural connections is significant for the structural

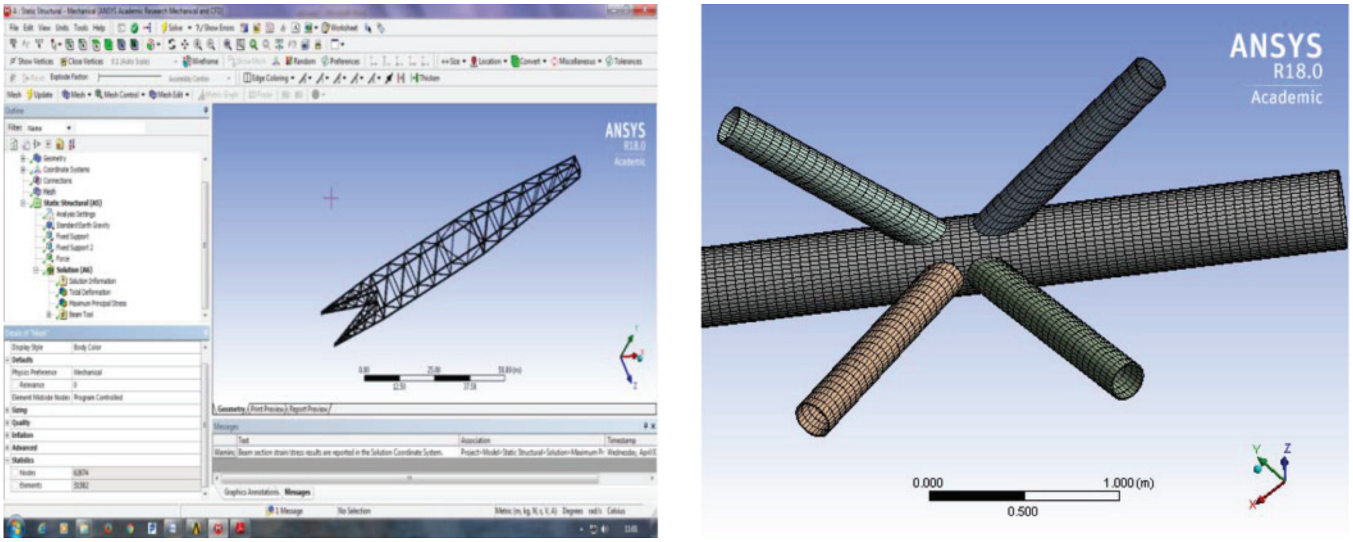
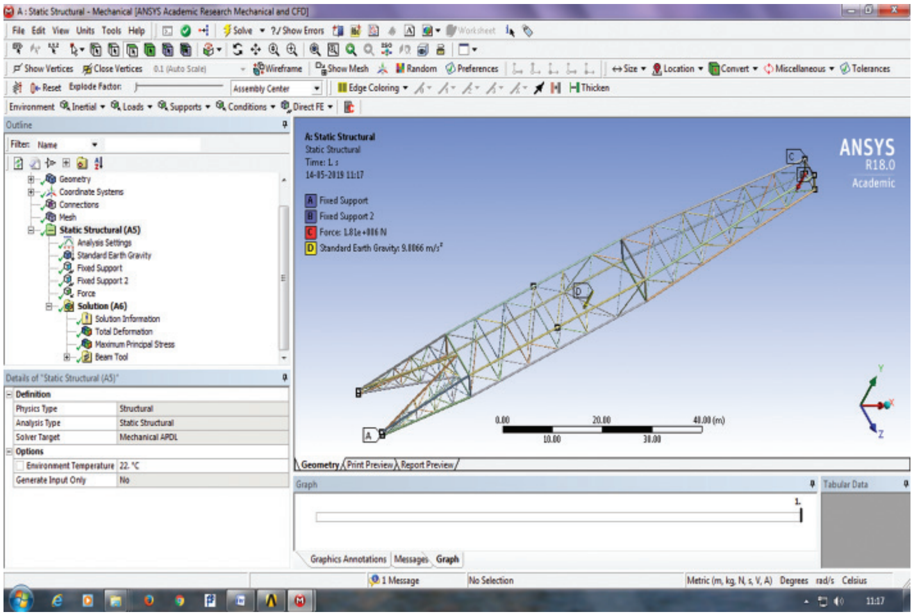


Fig.5 Meshed model of dragline boom



A: Static Structural
Static Structural
Time: 1. s
14-05-2019 11:17

A Fixed Support
B Fixed Support 2
C Force: 1.81e+006 N
D Standard Earth Gravity: 9.8066 m/s²

Details of "Static Structural (A5)"

Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Fig.6 fixed and loading boundary conditions

analysis. It can only be achieved with the highly refined local mesh area. However, for very large structures such as offshore platforms, it is almost impossible to perform this type of modelling resolution for the entire strcture model. For the reason described above, beam finite element models are very effective in isolating the peak forces and moments at the critical design joints. Detailed local 3D models are very advantageous in evaluating the detailed stress, and stress intensities in the local connections, where empirical solutions typically used are not always practical. Beam–to-solid submodel is based on two approaches: (ANSYS 17.2 user mannual).

(a) Cut boundary remote force,

(b) Cut boundary remote constraints.

Fig.7 illustrates the necessary procedure for beam-solid submodelling inside the ANSYS 18.0 software.

Global model analysis

The global model is the beam model of the entire structure. For analysing the global model in ANSYS workbench first, the model of dragline boom is imported to ANSYS workbench from Solidworks. Various boundary conditions are incorporated, and results are obtained. The essential results are deformation of the structure and combined stresses, which may give an impression of the entire structural behaviour. A local joint is shown in the figure below for the deformation values to be used for the further analysis of solid submodel. Fig.8 shows various results based on the beam model implementation.

Submodel analysis

To achieve accurate results for cluster area, a more refined mesh is required, which is not possible for a large complex structure. Also, more refined mesh increases the number of

TABLE 2: LOAD BREAKUPOF THE BOOM

	Type of load	Value of the load (tonne)
1.	Self-weight of boom	146
2.	Bucket self-weight	70
3.	Bucket payload	69
	Total weight (1+2+3)	285

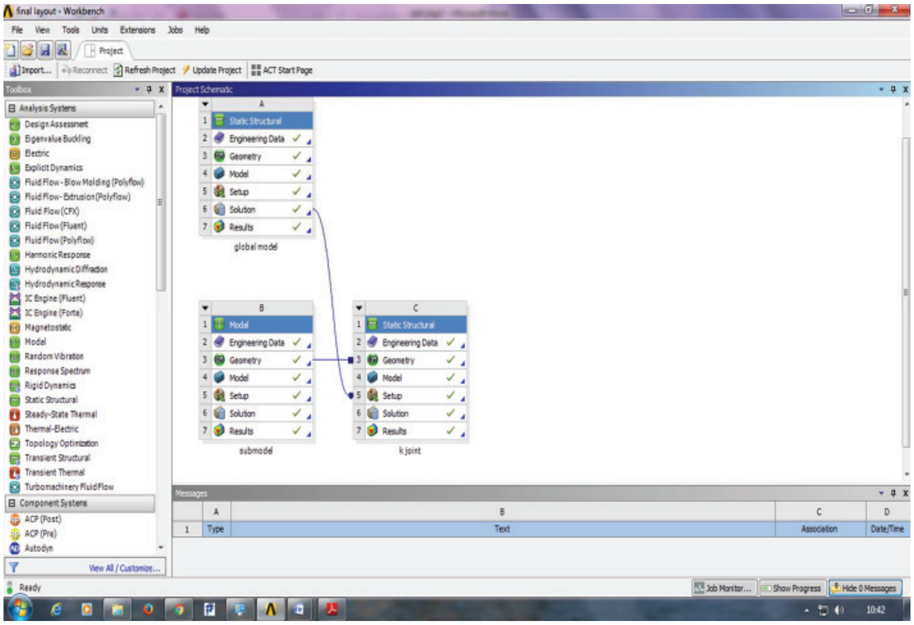


Fig.7 Global beam to solid submodelling implementation in ANSYS

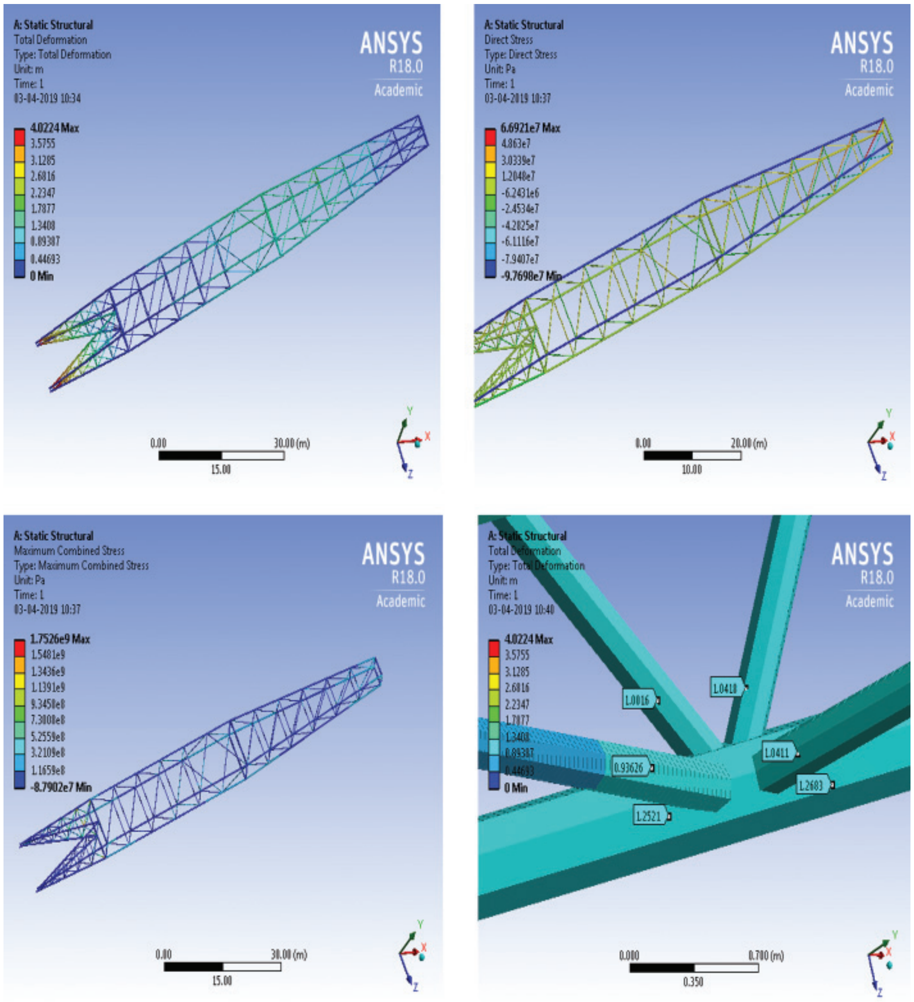


Fig.8 Global beam model results.

elements and nodes for the meshed model. It ultimately requires a large computation power and sometimes even with high computation set up that may not be able to solve the problem. So a solid submodel is created, and boundary conditions are applied from the global beam model result section. Submodel is the geometrically constrained model of the area of interest.

In the Fig.9, it can be seen that atone face of the model displacement boundary condition is applied while on rest remote force is applied. Displacement boundary conditions on one face of the model are incorporated to make the model stable.

After successful implementation of the beam to solid submodel in the software module, simulation is performed. Based on the different simulation results are obtained, which are summarised in the Fig.10.

Von mises stresses and deformation are the main results as depicted in Fig.10. To verify that our boundary conditions are correct deformation values are compared for global model and solid submodel at cut boundaries, which are found to be almost same and hence the result for von mises stress can be considered correct for the further calculation of fatigue life.

FATIGUE LIFE

Based on the results obtained from the von mises values obtained above ANSYS fatigue tool is used for calculating the fatigue life of the joint. ANSYS fatigue tool is available inside the static structural module of the workbench itself. It calculates fatigue life based on the values of von mises stress. Fig.11 represents the fatigue life of the main chord and one of the bracing members of the structural joint.

Results and discussion

After successfully running the

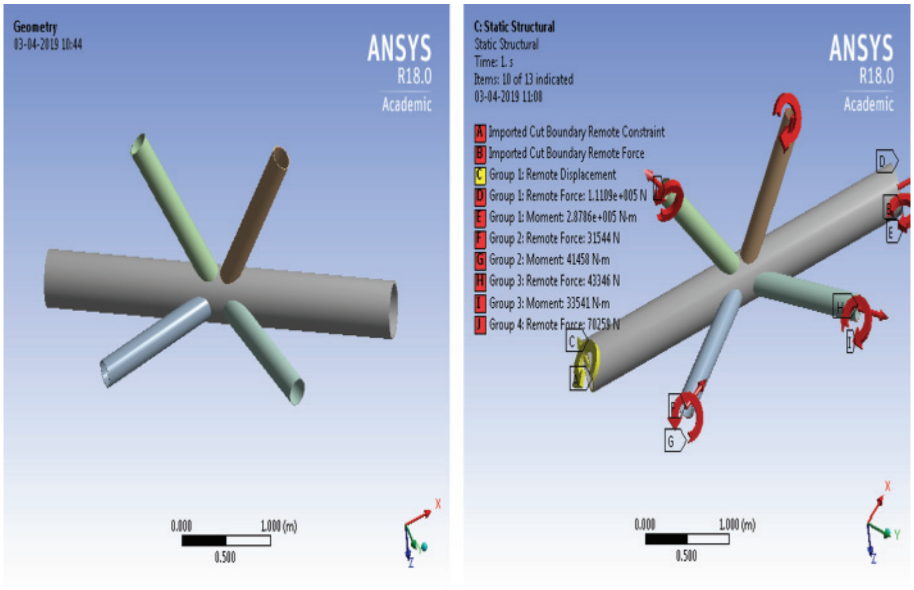


Fig.9 Solid submodel of the cluster with imported boundary conditions

simulations various results are listed above. Boom cluster selected above has 4 brace members and one main chord which together forms chs gap multiplaner *k* joint. Global beam model shows various results such as deformation, axial stresses and maximum combined stresses. Axial stresses varies from 97 MPa in compression to 69 MPa in tension. By separately analyzing a single joint fatigue life results are listed as shown in Fig.11. Fatigue life for each brace member and main chord is calculated. The stresses are concentrated near to the connection in welding, which obviously reduces the fatigue life of the joint. Life of the main chord is 29748 cycles and that for one of the brace member is 11826 cycles. These results confirm the findings that life of the main chord is much higher than the brace member life. Beam to solid sub-modelling approach is handy for analysing such type of truss-like structures. It also provides an easy way of analysing the local area with a more refined mesh inside the structure without not too much worry about the computation requirements. It is also an advantageous approach to define the overall behaviour of the structural components.

Conclusions

Based on the peer review of the literature 5, 6, 7, 8, 9 and successfully running the simulations based on different boundary conditions, several conclusion is obtained. Some significant findings are listed below:

1. Boom structure is one of the most critical components of the dragline. For the considered loading case boom design is safe as the values obtained for the stresses are well within the yield strength of the material.
2. Design and configuration of the boom are such that the main chords remain in compression during the loading while bracing members prevent the structure from bending effect of the applied load.

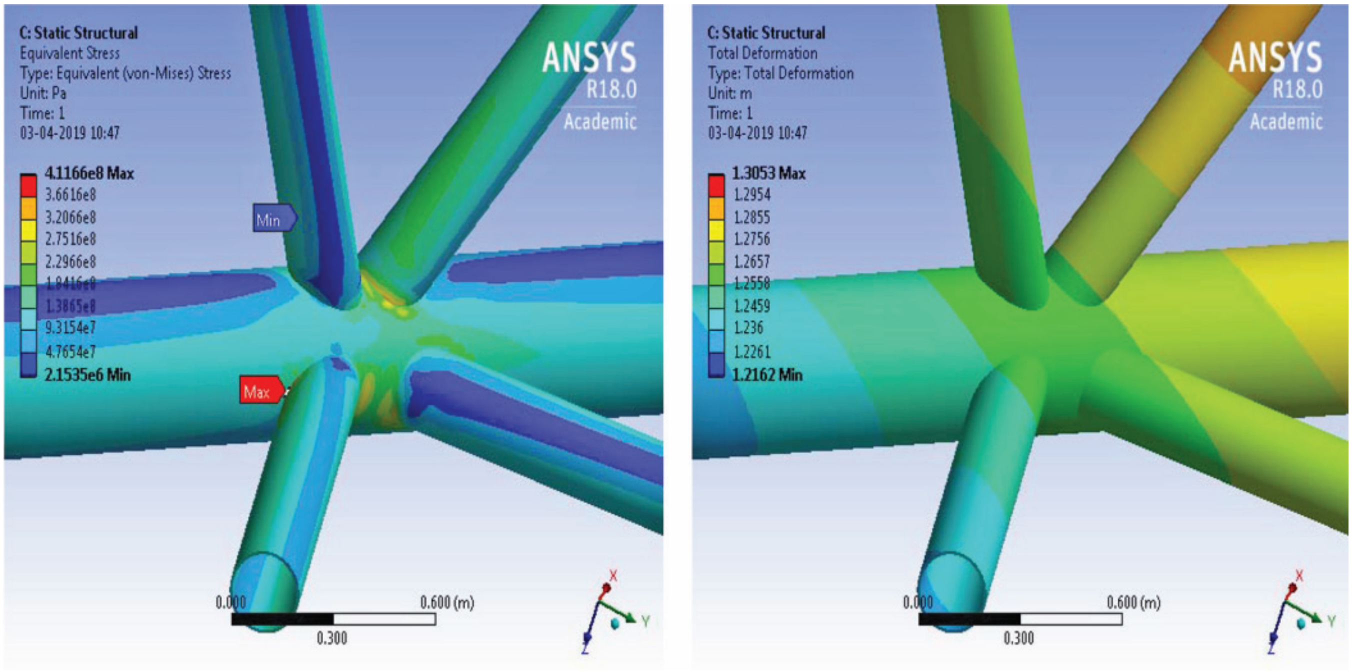


Fig.10 Equivalent von mises stresses, and total deformation results of boom joint

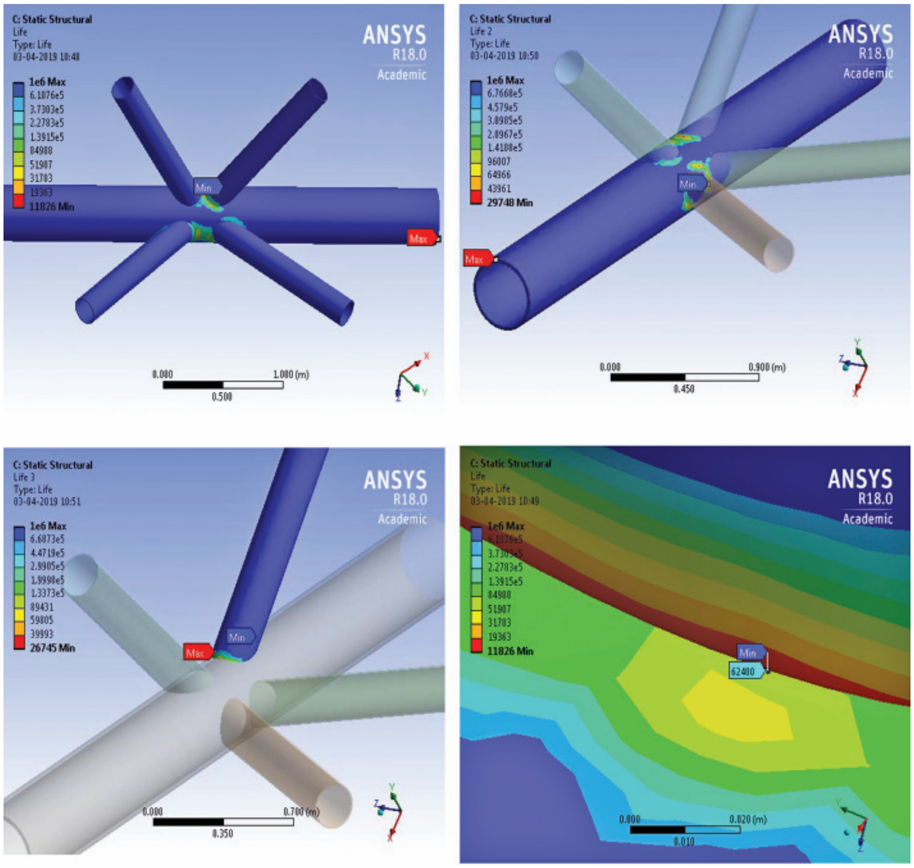


Fig.11 Fatigue life results in solid submodel

3. Stresses are very high near the nodes where bracing members are connected with the main chords. Mainly these points lie inside the weld joint itself and due to the continuous action of cyclic stress cracks may develop at these locations.
4. As can be seen from the Fig.11 submodel, fatigue life is calculated based on the value of von- mises stresses. This tool does not consider a sudden change in cross-section. Any sudden change in cross-section area always gives very high-stress values, so a more detailed model is needed to capture local joint with modelling of weld geometry itself.

References

1. Azam, S. F. and Rai, P., (2018): Modelling of dragline bucket for determination of stress. *ASME J.*, 78, 392–402.
2. Azam, S. F. and Rai, P. (2019): Finite element-based simulation and analysis of dragline bucket in static and dynamic loading condition. 116.
3. Patel, R. K., Singh, A. and Kishore, N. (2017): Assessment of comparative operating cost of new draglinevs. Assessment of comparative operating cost of new dragline vs. Outlive dragline.
4. Circular, F. O. R., Section, H., Joints, W. and Loading, U. F. for Circular and Rectangular Hollow Section Welded Joints.
5. Abo-Elnor, M., Hamilton, R. and Boyle, J. T., (2003): 3D Dynamic analysis of soil-tool interaction using the finite element method. *J. Terramechanics*, 40(1), 51–62.
6. P. Dayawansa, G. Chitty, B. Kerezsi, H. Bartosiewicz, and J. W. H. Price, (2006): "Fracture mechanics of mining dragline booms," *Eng. Fail. Anal.*, vol. 13, no. 4, pp. 716–725.
7. S. Frimpong and N. Demirel, (2009): "Case Study: Planar Kinematics of Dragline for Efficient Machine Control," *J. Aerosp. Eng.*, vol. 22, no. 2, pp. 112–122.
8. Joshi, S., Aloraier, A. S., George, S. and Campus, W. (2011): Measurement of Welding Induced Distortions in Fabrication of a Prototype Dragline Joint/ : a Case Study. Engineering 6, 310–317.
9. Y. Li and W. Liu, "Dynamic dragline modelling for operation performance simulation and fatigue life prediction," *Eng. Fail. Anal.*, vol. 34, pp. 93–101, 2013.
10. F.R. Mashiri, S. Joshi, N.L. Pang, D.P. Dayawansa, X.L. Zhao, G. Chitty, H. Jiao, J.W.H. PriceService loads in dragline tubular structures: a case study of cluster A5Struct. Control. *Health Monit.*, 20 (2) (2013), pp. 210-22.