

RESEARCH METHODOLOGY

3.1 Methodology for analysis of dragline boom

The current research work mainly deals with the development of a three-dimensional (3D) Solid model of dragline boom in computer-aided design (CAD) environment followed by the stress analysis and parametric analysis of joint clusters of dragline boom under the different loading conditions. Further, the fatigue analysis has also been performed for the four strategic joint locations on the boom structure. The flow chart presented in figure 3.1 clearly illustrates the research methodology adopted in the present research work.

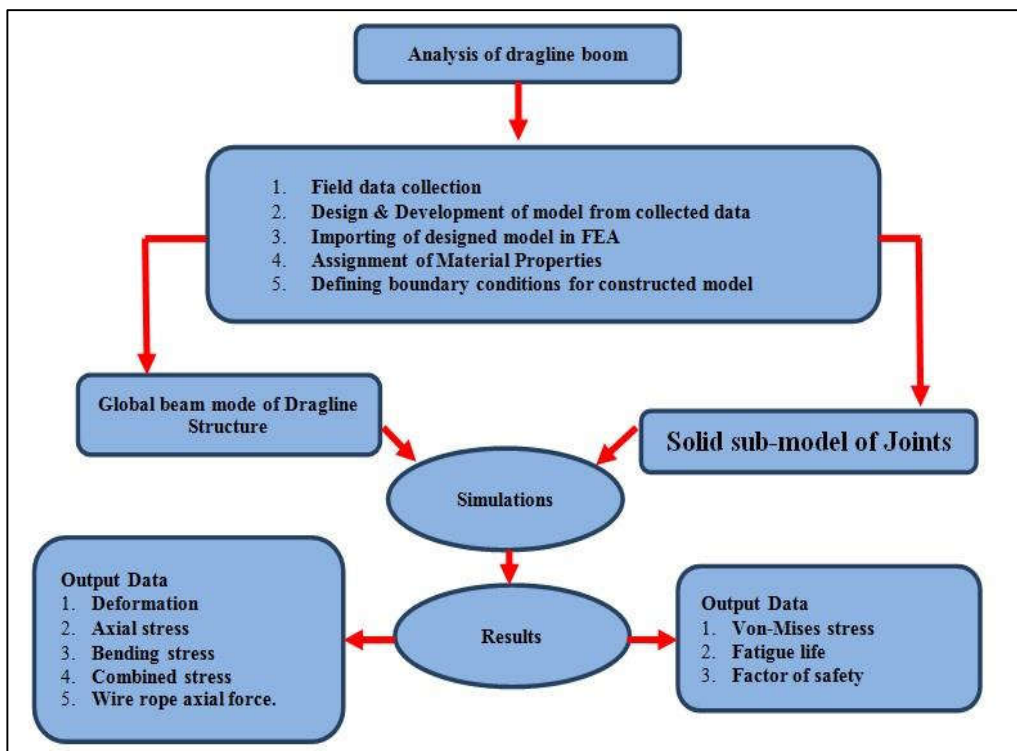


Figure 3.1 Flow chart illustrating the design of research

Chapter 3. Research Methodology

During the field visits, the operations of the dragline were carefully observed and relevant field data, related to the current research work, was obtained.

During the field study on dragline operations, two type of loading conditions, namely static and dynamic loading were observed. The static loading conditions occur during the operation, when the boom and bucket of the dragline become static for a very short period of time. This static condition occurs just before swinging back segment with empty bucket, just before the lowering of empty bucket at the bank and just after filling but before hoisting of loaded bucket. It is noteworthy that this static loading condition exists for a very momentary time period and may remain unnoticed if not viewed carefully. These instants of loading conditions were incorporated in the current analysis.

The dynamic loading conditions occur during the swing-to segment of dragline bucket cycle with filled material for unloading the material from face to disposal site. It also occurs during the swing-back segment of dragline bucket cycle with empty bucket after dumping the material at the disposal site and returning back to the blasted face.

Accordingly, a solid 3D Model of the boom was constructed in SOLIDWORKS software as wireframe design. For performing simulations, the constructed model was imported from SolidWorks to ANSYS software in compatible file format by use of IGES and PARASOLID. Before performing simulation, the wireframe model was converted into 3D solid model by assigning appropriate dimensions and cross sections, which were collected from the field reports, equipment catalogue and personal communications with the field operators as tabulated in table 3.1.

Table 3.1: Specifications of boom and bucket for construction of model

Specifications of dragline boom and bucket	Value
Boom length	96 m
Boom angle	35°
Main chord diameter	406 mm
Main chord thickness	20 mm
Brace chord diameter	207 mm
Brace chord thickness	8 mm
Bucket capacity	24 m ³

Simulations were performed for the following five conditions of the dragline boom (under static and dynamic loading).

1. Self-weight of the boom under static loading.
2. Tare weight of the bucket acting on the boom along with the self-weight of the boom under static loading.,
3. Payload acting on the boom, along with the loading condition as listed in points 1 and 2 above under static loading.
4. Swing to motion of the dragline boom under dynamic loading.
5. Swing back motion of the dragline under dynamic loading.

After successfully performing simulation, various results were evaluated and analyzed. Based on these findings the overall structural behavior of the dragline boom was observed. Furthermore, a number of parametric changes were employed to

understand the impact of these parametric changes on stresses on a single joint location in the boom.

3.2 Data Collection

For arriving at a good level of understanding, the collection of field data is very crucial. For collection of real time data, a number of open cast mines was visited. The field observations were made in various mines of the Northern Coalfields Limited (NCL) and Sasan Coal Mine, India (figure 3.2 (a-d)). As the dragline used in the NCL mines were quite old, the maintenance and breakdown were frequently observed during their normal operational hours. Dragline operations, manoeuvring of boom and its assembly was observed and studied carefully in the field operations. Further, the important specifications and dimensions needed for design of dragline, boom and its related assemblies were collected. Also, the physical properties of overburden material were obtained. Various departments were visited to ensure correct data availability for fulfillment for the current design and analysis work. In the present work the data gathered is mainly related to dragline boom and bucket. The properties of overburden material that was handled, after blasting, by the dragline are tabulated in table 3.2.

Table 3.2: Material properties of overburden

Sr. No.	Parameters	Value
1	Overburden Material	Sandstone
2	Density of overburden (γ)	(1.8 - 2.4) t/m ³
3	Swell factor (S)	0.85
4	Bucket Fill Factor (F)	0.75



Figure 3.2 (a-d) Images from field visits. (a) Dragline model 8200 in SASAN coal mine (b) Ganga dragline model in NCL (c) 24/96 dragline model in NCL and (d) Zoom view of boom foot of the dragline

3.3 Solid 3D Model Generation of Dragline Boom

SolidWorks software has been used for modeling and design of dragline boom. The software is developed by the Dassault systems and was first introduced in year 1995. SolidWorks is mostly preferred for any industry that wants to perform the design and analysis of structures, static structural analysis, dynamic analysis, thermal analysis etc. It is Computer Aided Design (CAD) system, globally utilized by researchers, industry professionals, simulation experts and R&D departments from the numerous industries to simulate and assimilate various CAD operations on critical components/structures. Some of the utilities of this tool are 3D Model Creation,

Chapter 3. Research Methodology

Construction of 2D Drawings, Sheet Metal Modeling and Development, Structural Modeling and Simulation, Welding Simulation, Concept Design and Realization, Tubing, Piping, Cable, PCB, Printed Circuit Boards simulation.

SolidWorks provides the latest solution from Design to manage digital data with a user-friendly interface. It also offers high-end CAD simulation modules to perform design and development. User can simulate a 3D model on the same graphic interface without switching over to other platform and they can also perform virtual model simulation for important properties, such as, stress, fatigue, life-cycle, motion, flow, vibration, frequency, static, linear and non-linear analysis etc.

Advantages of SolidWorks

The salient advantages of SolidWorks as an important design tool are listed:

- An interactive user interface.
- The system provides easy to use drawing tools readily available on the main screen.
- A plenty of online learning & technical support is available on the web and the database itself.
- It proves to be an effective tool for construction of 3 D models for complex structure and any change in design parameters incorporated during any phase of the design gets automatically updated for the entire structure without redrawing from the scratch each and every time.

The points discussed above justify the selection of SolidWorks software system as a tool for the application of solid modeling of dragline boom. It saved a lot of time as

compared to other contemporary CAD systems. It also provided a convenient user interface and working experience.

Figure 3.3 shows the schematic diagram of a dragline (Anonymous, 2004); the basic structure of the front end includes boom, suspension ropes, mast and A- frame. For the design and analysis purpose these components were modeled as wireframe diagram (figure 3.4).

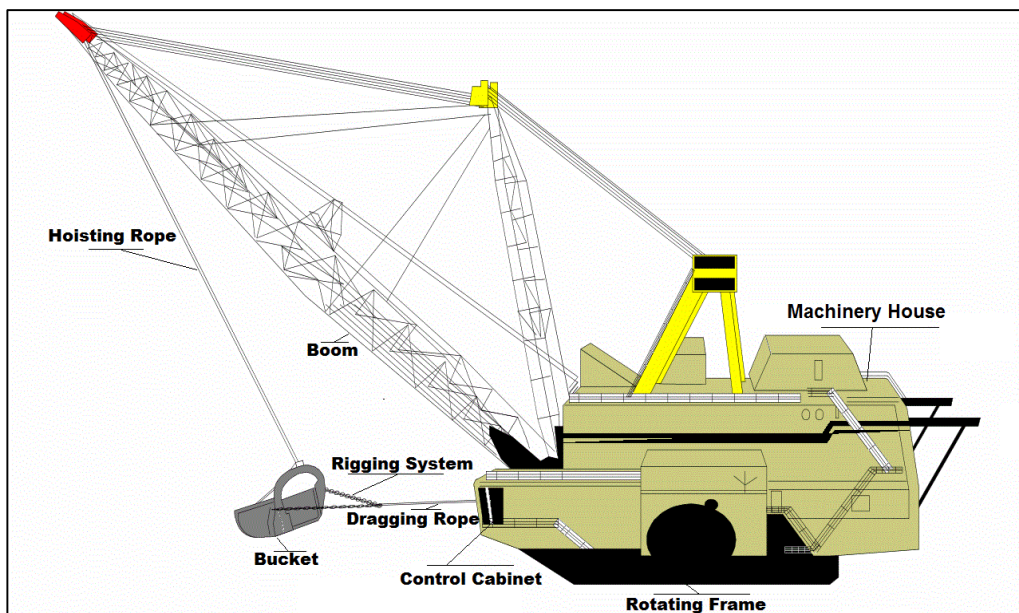


Figure 3.3 Basic Dragline Structure (Anonymous, 2004)

Figure 3.4 illustrate the constructed model of the dragline boom in SOLIDWORKS as wireframe model, which was then imported into ANSYS 18.0 software.

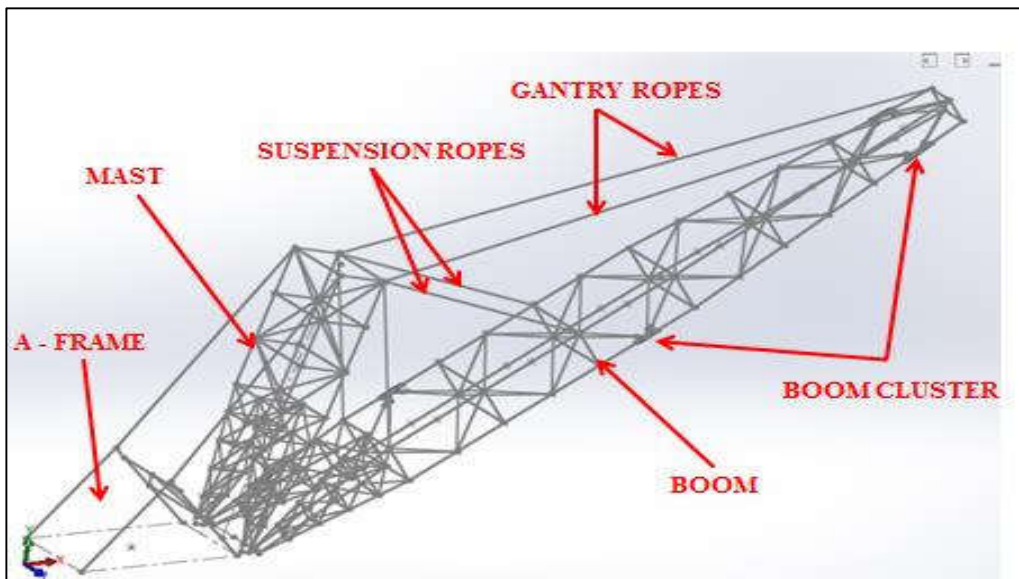


Figure 3.4 Wireframe model of dragline boom

As revealed in figure 3.5 the respective cross sections for tubular members were assigned in the ANSYS workbench after importing the wireframe model through portable PARASOLID format (Shukla. A, and Rai. P, 2019).

After assigning the cross sections the model has been used for the analysis in static structural module.

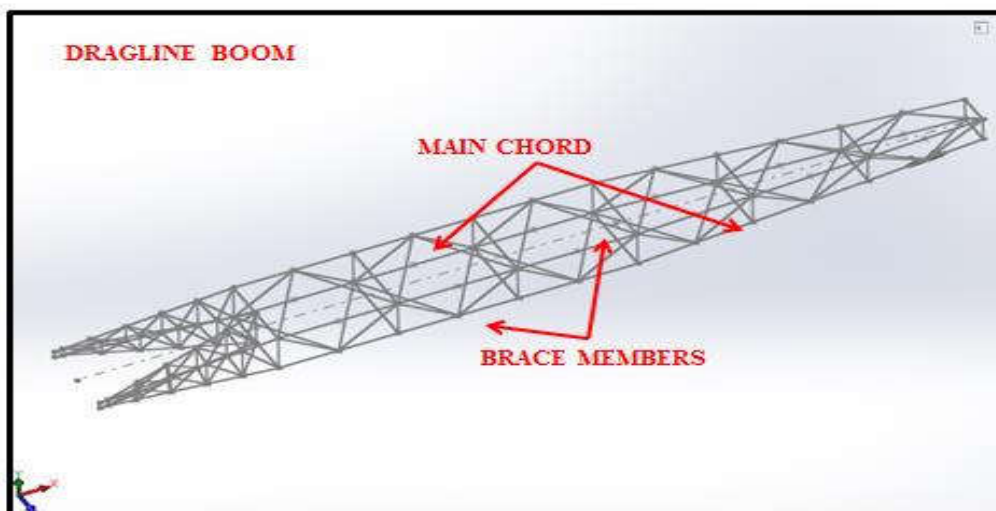


Figure 3.5 Dragline Boom model with cross section assigned in ANSYS workbench design modeler

3.4 Solid 3D Model Generation of Dragline boom Cluster (Joint)

Figure 3.6 illustrate the strategic joint locations for the evaluation of fatigue life and safety factor. Four critical locations of the joints have been selected for in depth analysis of stresses around the interaction point of brace with the main chord. As Joint 1 and 2 are located near the boom point sheave, thus the impact of loading was critical for analysis at these locations. Further, as the Joint 3 was located near the center of gravity of the entire structure, therefore it was also considered suitable for analysis. Joint 4 was near the boom foot (boom root). The reason behind the selection of this joint is its strategic location being near the center of gravity as well as the boom root. These joints have been chosen on the basis of field understanding that the repairing and maintenance activities are carried out frequently at these locations.

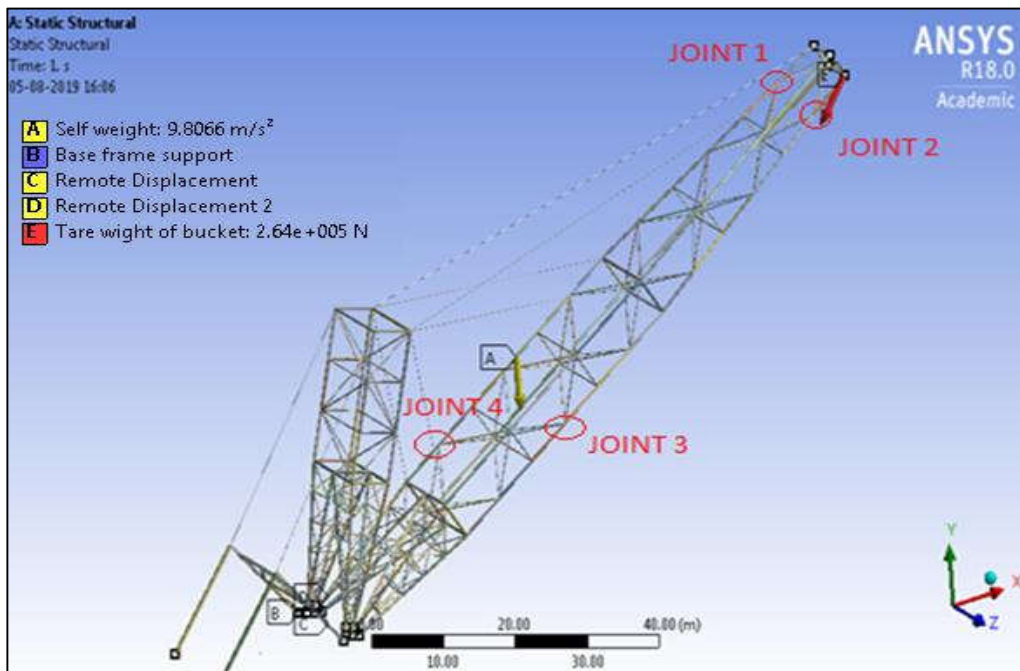


Figure 3.6 Selected joint locations for fatigue analysis

Because the boom design incorporates number of joints, hence any design or analysis of boom structure remains incomplete without critical investigation of joints. Therefore, critical joints observed on the basis of field investigation have been chosen for the analysis of stresses within the joints (Shukla. A, and Rai. P, 2020).

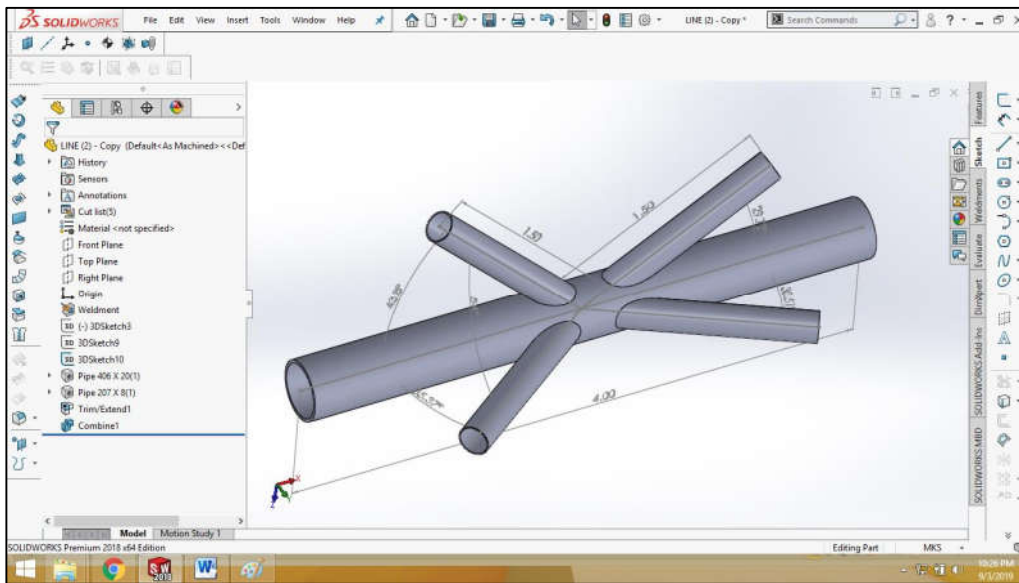


Figure 3.7 Dragline Boom cluster (joint) design model

The basic joint design consists of one main chord and four lacing members connected together by means of welding as shown in figure 3.7.

There were four numbers of key parameters, which are considered for constructing the joint model. These parameters are main chord diameter, main chord thickness, brace diameter and brace thickness.

3.5 Simulation and analysis of dragline boom

The finite element method (FEM) has been used for simulation and analysis of loading and its impact on strategic joints on the dragline booms. The FEM is a powerful technique for determining stresses and deflections in complex shaped

Chapter 3. Research Methodology

structures, which are difficult to analyze by analytical methods. In the finite element method, the structure is divided into a network of small elements connected to each other at a point. In this thesis, dragline boom analysis has been carried out to determine the stresses experienced by boom during static and dynamic loading conditions of the dragline.

A very novel and effective technique of beam to solid sub-modeling has been used in the FEM analysis by ANSYS. As the design of boom structure is very complex and requires very high computing power, so these can be analyzed as beam elements for global model, which requires very less computing power and time by considering the entire global model as beam model. It is a very new functionality which is introduced in ANSYS 18.0. The procedure of the analysis of such type of structure using beam-solid submodelling technique is enumerated:

1. Construction of a global beam model of entire dragline boom structure.
2. To solve the global beam model by applying the real boundary conditions.
3. To feed the obtained results of global beam model solution in terms of applying the obtained boundary condition on the constructed solid joints.
4. To solve the solid sub-model based on the global beam model generated boundary conditions.
5. To compute the problem for obtaining the results with refined mesh values and with local fillets at suitable locations.

The process of getting solution from global beam model to solid submodel for the dragline boom cluster is illustrated in figure 3.8 in order to predict fatigue life and stresses at the local regions. The solution obtained from global beam model A has

been fed to the cluster solid model D, while C represent the CAD models of the four joints.



Figure 3.8 Solution procedures in ANSYS for a critical location in a global beam model

Solid submodel for joint was prepared using SOLIDWORKS for stress analysis as shown in figure 3.9. The modeling of joint in welding regions is very difficult to simulate because of its complexity. Therefore, the weld geometry was replicated by providing the fillet radius of 10 mm for entire model during the simulation process.

Based on the results obtained from the global beam model, fatigue life and factor of safety (FOS) of four selected joints at strategic and critical locations was critically examined.

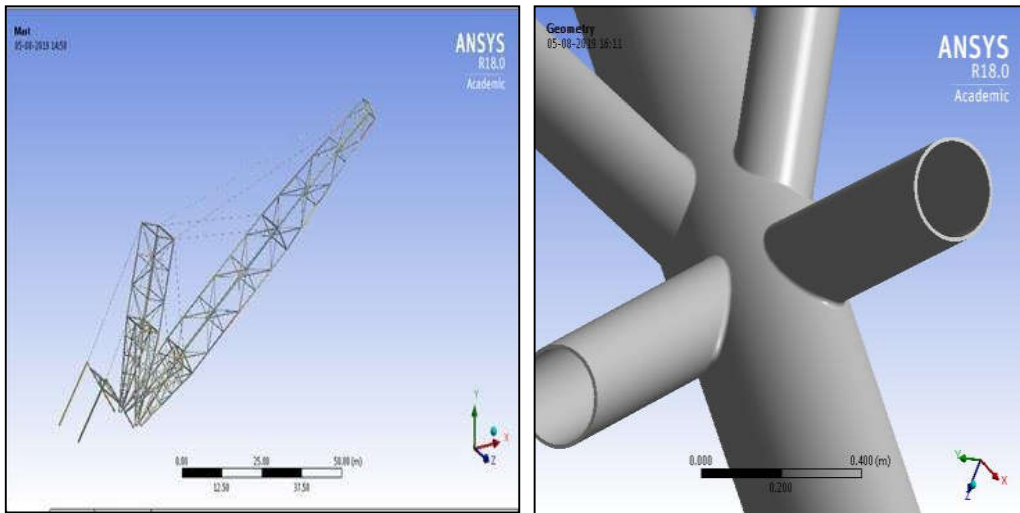


Figure 3.9 Global beam model with solid sub - model of cluster

3.6 Structural material properties of dragline boom

After successfully creating CAD models, the primary step in finite element analysis was to define the material properties of dragline boom's tubular members. The chemical composition and mechanical properties of tubular steel members used in model analysis are given in tables 3.3 and 3.4

Table 3.3: Chemical composition of tubular steel

Constituent Material	Percentage
C	0.14 - 0.2
Si	0.4 – 0.6
Mn	1 -1.4
Ni	Max 0.3
S	Max 0.04
P	Max 0.035
Cr	Max 0.3
N	Max 0.008
Cu	Max 0.3
As	Max 0.08

Table 3. 4 Mechanical properties of tubular steel

Mechanical Property	Value
Density	7850 kg/m ³
Ultimate Tensile Strength	510 MPa
Yield Strength (Tensile)	250 MPa
Poisson's Ratio	0.3
Young's Modulus	2 x 10 ⁵ MPa

3.7 Finite element meshing and boundary conditions for static loading

Meshing is an integral part of the finite element analysis. It also affects the accuracy of the analysis. Various mesh sizes were checked before finally selecting the optimum mesh size. The meshing of the solid bodies was then carried out using the tetrahedron method with element size as 40 mm. Meshing of global beam model is illustrated in figure 3.10. The resultant meshing body thus incorporated 58134 beam elements with 116022 nodes.

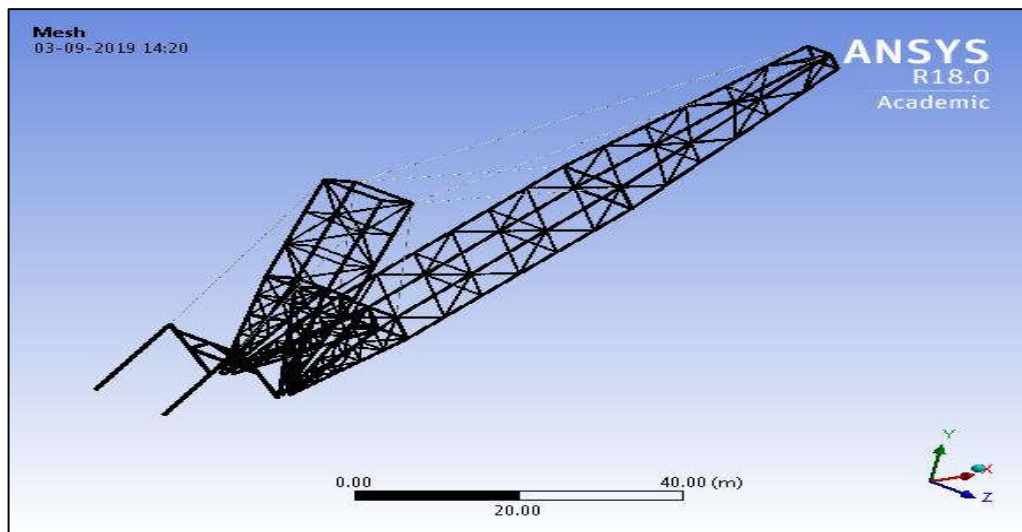


Figure 3.10 Meshed global beam model in Ansys

Figure 3.11 shows the meshing of the solid submodel with refined mesh size at the corners where the bracing members are connected with the main chord. The mesh size was reduced to 10 mm near the fillet region to obtain accurate results while analyzing the sub-model.

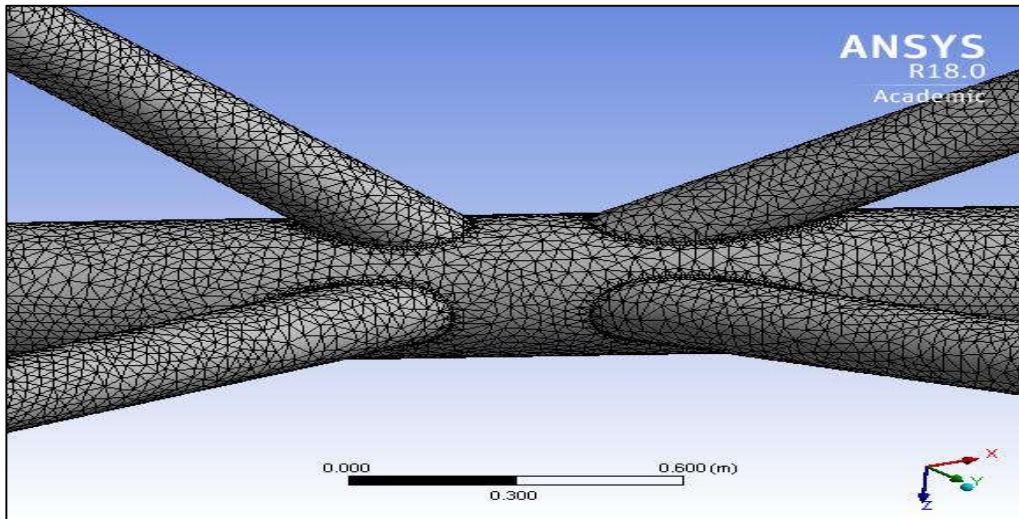


Figure 3.11 Meshed solid sub-model at the local joint regions

3.8 Static loading condition

Static loading conditions occur for a very short time in every cycle of the dragline bucket. In static condition of a dragline, the forces acting on the dragline boom do not change with time and have a constant value. Figure 3.12 (a,b) illustrate the loading and unloading conditions for dragline operation.

Analysis of a static dragline boom is carried out for three different conditions, namely,

- 1- Dead load i.e. dragline boom with self-weight,
- 2- Boom with self-weight and empty bucket weight,

- 3- Boom with self-weight, empty bucket weight and payload of bucket filled with blasted sandstone material.



Figure 3.12 (a) loading and (b) unloading segment of a dragline cycle

Boundary and loading conditions of boom for global model and solid submodel under static loading conditions are illustrated in Figure 3.13 and 3.14 respectively.

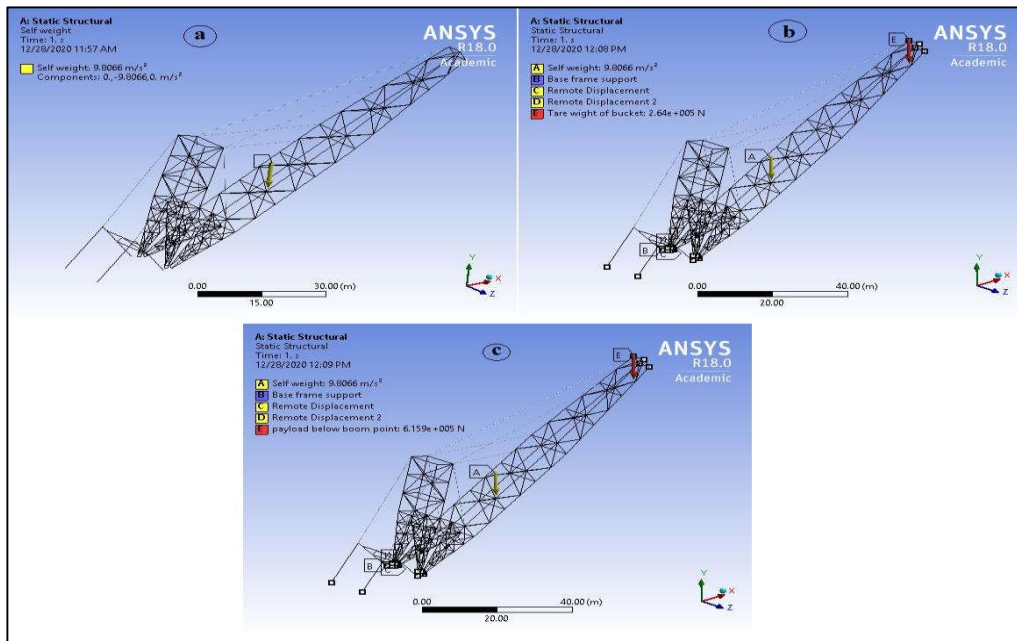


Figure 3.13 Boundary and loading condition of boom model under static conditions (a) boom with self – weight, (b) boom with empty bucket weight, (c) boom with payload in bucket

Chapter 3. Research Methodology

In figure 3.13, point A is the point of application of self-weight, point E shows the load application point of bucket dead load and payload (blast material filled inside the bucket). Points B, C, and D represent the constraints provided for support of A-frame, mast and boom root respectively. These supports were carefully observed during the field visits. A- Frame was fixed at the base of dragline rotating body. The boom root and mast root were hinged about a common point by means of a pin. Point B, C, and D represent the constraints at above discussed locations.

The solid submodel of the joints has their boundary condition derived from the displacements and forces extracted from the global beam model of the structure. While defining boundary condition the remote displacements at the respective nodes was provided to analyze the model. In the figure 3.14 points B,C,D,E,F and G show the application of respective displacements value at the respective nodes.

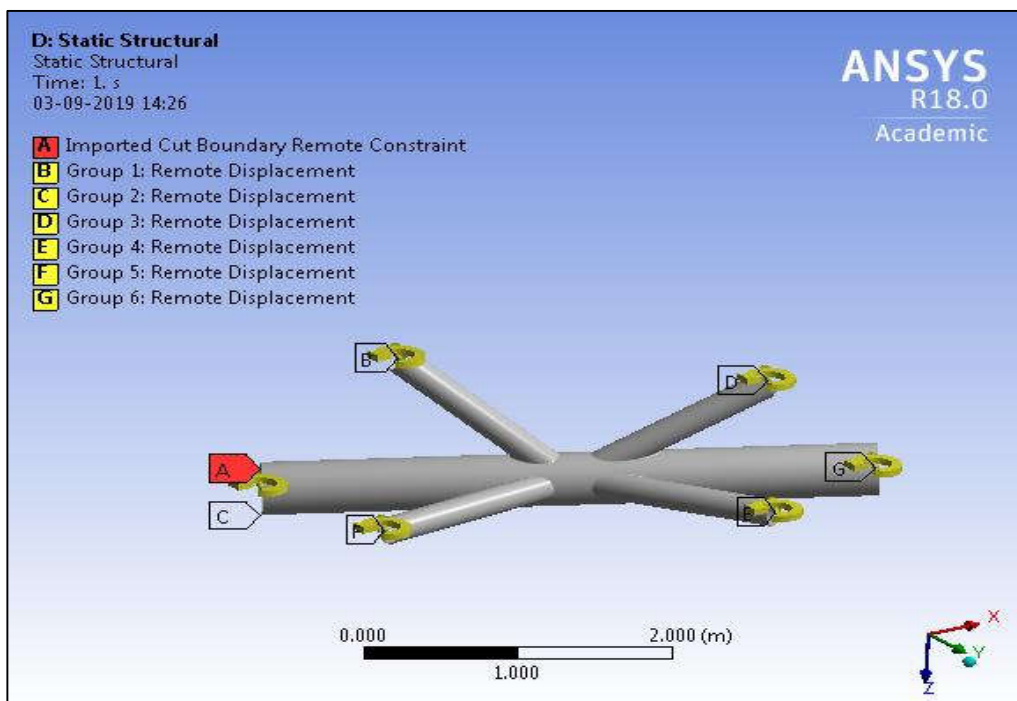


Figure 3.14 Boundary and loading condition of solid submodel under static conditions

3.9 Dynamic Loading Condition

Dragline cycle is composed of the following phases (Rai P., 2000) (i) Filling the bucket by dragging it towards the dragline; (ii) Swinging the loaded bucket along a predetermined arc; (iii) Paying out the dirt onto a spoil pile; (iv) Swinging the empty bucket back to excavation face; and (v) Positioning the bucket to re-load. the cycle has been clearly illustrated in figure 3.15.

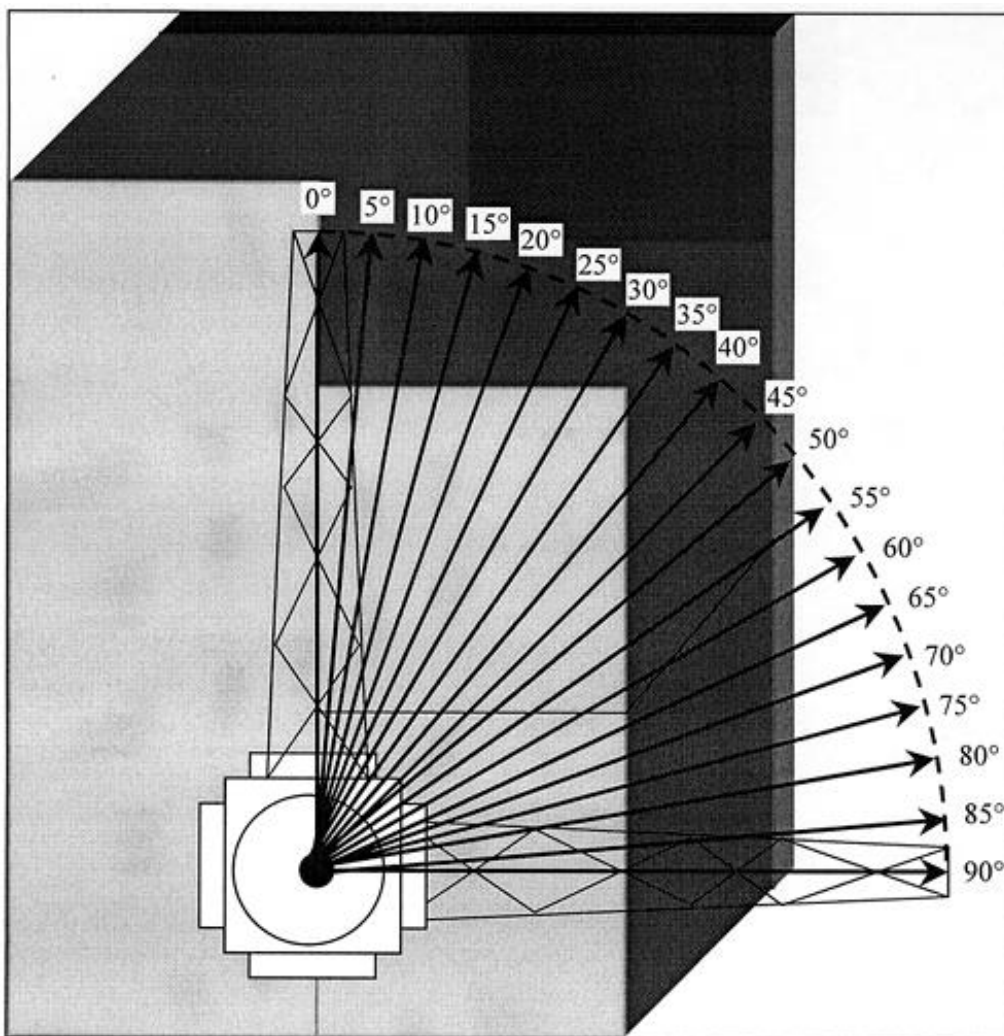


Figure 3.15 Typical dragline swing angles (Rai, P, 2000)

Swing times (both loaded and back) are positively correlated to swing angles. Since swing times may not be reduced, the dragline panel design may be so optimized that dragline swing angles are kept at a minimum. The dragline cycles are mostly swing-dependent. However, in the case of narrow and deep key cuts, cycles tended to be hoist-dependent. Further, when the swing angles were smaller than 50°, cycles became drag-dependent, which took longer than larger-swing-angle cycles due to longer payout processes. Such situation needs to be avoided. Dumping time and repositioning time fluctuate within a narrow time interval. As such they may be considered to be constants for various modes of operation (key cut, main cut and chop cut). Operator's experience plays a significant role in filling, dumping and repositioning phases. Dragline swing angle variation with time has been shown in figure 3.16 as given by (Erdem, B., & DUzgUn, H. (2005)).

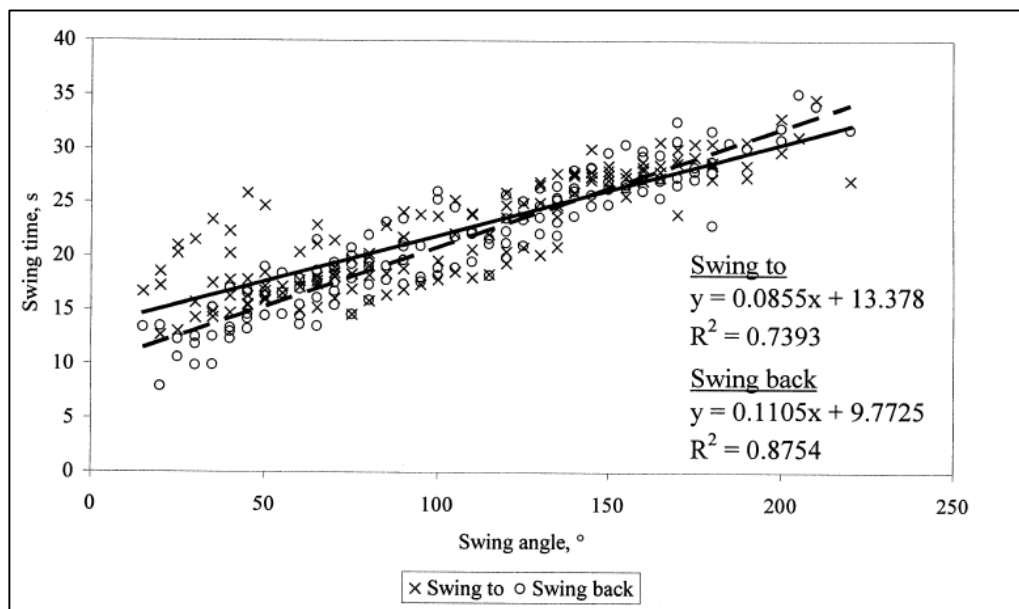


Figure 3.16 Dragline swing angle variation with time (Erdem, B., & DUzgUn, H. (2005))

Considering the swing angle of 90^0 , the average swing to time observed was 21s and average swing back cycle was about 20s. Almost all the dragline swing cycles fall in this range and satisfy the above regression equation. Assuming the average acceleration phase for the dragline to be about 16 s and correspondingly the average constant angular acceleration value of about $0.36 \text{ degree / sec}^2$, the maximum velocity obtained after 16 seconds is 6 degree/ s . The data for swing acceleration and velocity were rigorously gathered during the field visits and observation.

Therefore, in further computations, the maximum angular velocity of dragline of about 6 degree/ s and average acceleration value of about $0.36 \text{ degree/sec}^2$ has been used religiously. The dynamic cases for the dragline boom analysis were considered as follows:

1. Swing to motion of the dragline.
2. Swing back motion of the dragline.

3.10 Boundary conditions for dynamic loading

Dragline boom with swing acceleration is shown in figure 3.17 below. In dragline dynamic swing motion, the boom is rotated along with the tri structure about the revolving frame pin Centre. Hence swing acceleration is required to obtain required swing cycle. In figure 3.17-point F is an additional force provided for the analysis of the model to compute stresses in the dynamic loading condition with filled bucket.

The dragline swing motion is also accompanied by the hoisting simultaneously for dragline operations. However, the current research work is limited to the swing to and swing back motion of dragline.

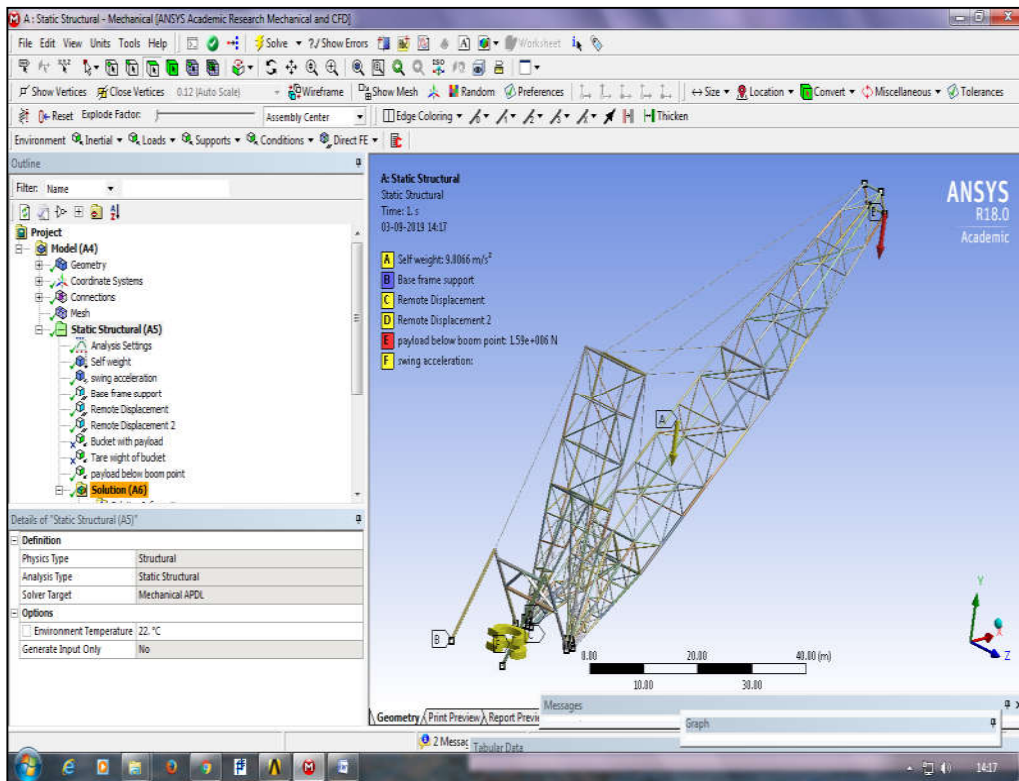


Figure 3.17 Boundary condition for swing motion of boom

3.11 Static load computation

The load computation for static loaded cases has been carried out in this section considered 96 m boom length and 24 m³ bucket capacity, while keeping the boom angle fixed at 39°. The payload was considered as broken sandstone material.

3.11.1 Dead load on boom structure (self-weight of the boom).

The self-weight of the boom for 96 m boom length was computed from the model and was found to be 142-ton approx. Force on the boom can be computed by equation 3.1.

$$F_1 = 142 \times 10^3 \times g \text{ N} \quad (3.1)$$

$$F_1 = 1.42 \times 10^5 \text{ N}$$

Where,

F_1 = load due to self-weight

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

3.11.2 When empty bucket is on ground below dragline boom in vertical position

Empty bucket weight has been computed by assuming density of the bucket as 1100 kg/m^3 of bucket capacity. Taking the bucket capacity as 24 m^3 , empty bucket weight may be computed by the following expression (equation 3.2).

$$m = \rho \times V \quad (3.2)$$

$$m = 1100 \times 24 \text{ Kg}$$

$$m = 26400 \text{ kg}$$

$$\text{Load acting on boom point } F_2 = m \times g \quad (3.3)$$

$$F_2 = 26400 \times 10 \text{ N}$$

$$F_2 = 2.64 \times 10^5 \text{ N}$$

Where,

m = mass of bucket in (kg)

V = bucket capacity in (m^3)

3.11.3 When bucket is completely filled with sandstone and positioned below dragline boom in vertical position

The force calculation can be performed according to equation 3.4.

Total weight acting on boom point (F_T) = empty bucket weight (F_1) + payload in bucket (F_2)

$$\begin{aligned} \text{Payload } F_3 &= \rho \times V \times f \times S && (3.4) \\ &= 2300 \times 24 \times 0.85 \times 0.75 \\ &= 35190.00 \text{ kg} \end{aligned}$$

Total load acting on boom point (F_T) = $2.64 \times 10^5 + 3.5 \times 10^5 \text{ N} = 6.14 \times 10^5 \text{ N}$

Where,

V = volume of empty bucket (m^3)

ρ = density of filled material (kg/m^3)

f = bucket fill factor

S = Swell factor

3.12 Evaluation of axial tension in suspension wire ropes

Usually the boom angle in the field ranges from between 30 -35° degrees from the horizontal. The boom is hinged at the boom root by use of a pin. As the boom is very heavy structure with long length, there must be sufficient support provided along the length of boom axis. For maintaining the constant angle and stability to the boom, support is provided by means of various wire ropes, which prevent the entire

Chapter 3. Research Methodology

structure from excessive stresses around the pin region. Also these suspension wire ropes must be capable of providing support to the boom during the working of draglines by encountering the bending moment generated in the boom about its rotational axis. Therefore, the analysis of these wire ropes becomes essential by computing the axial tensile force in these ropes.

The term "wire rope" refers to a diameter larger than 3/8 inch (9.52 mm). In the past, wrought_iron_wires were used, but now a day's steel is being used as the main material for wire ropes.

The wire ropes are also used for dragging, hoisting and lifting in the cranes, draglines; they work as mechanical power transmission. The wire ropes are subjected to fluctuating forces, by wear, by corrosion and in some cases by extreme forces.

Also however, rope life is finite and the safety is normally ensured by the detection of wire breaks, loss in cross-section, and related failures so that the wire rope can be replaced before any possibility of hazard. Installations should be designed to facilitate the inspection of the wire ropes.

In the current design and analysis wire ropes have been assumed as beam elements for minimizing the complexity. Structural steel has been considered as its construction material. The diameter of the wire has been taken as 111 mm as used in the dragline. The axial forces in the direction of wire connection have been computed by the global beam model analysis after computing the axial forces or tension in the wire ropes, the wire rope stress has been computed by the formula given below (equation 3.5).

$$\text{Wire rope stress} = \frac{\text{Axial force in the direction of its major axis}}{\text{cross section area of the wire}} \quad (3.5)$$

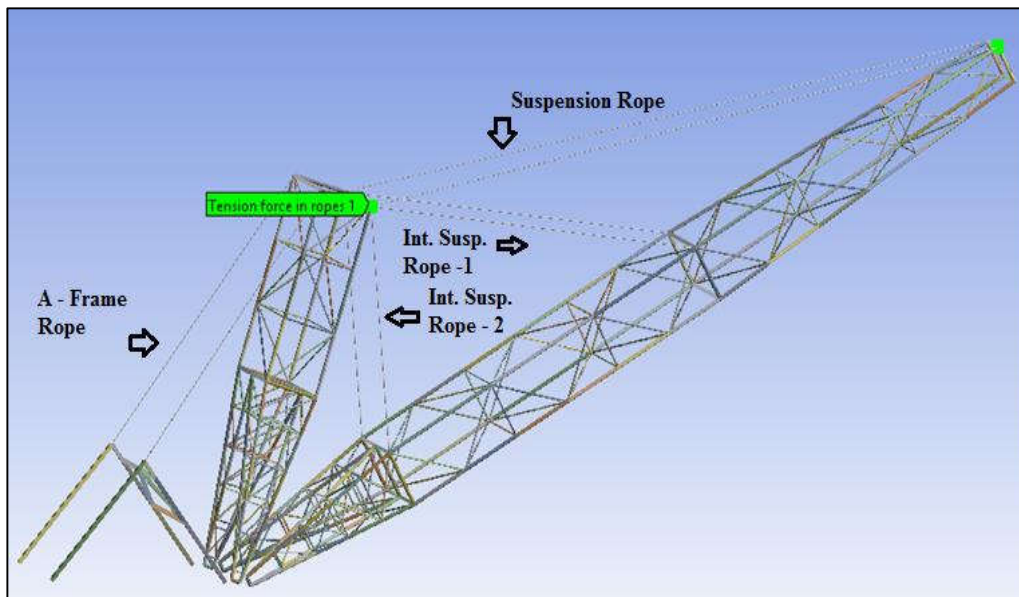


Figure 3.18 Figure showing set up for evaluating axial forces in a dragline suspension ropes

Figure 3.18 shows the connection arrangements for beam connectors. Axial forces are the forces along the direction of the connected wire. These axial forces are evaluated after the analysis and stresses in the ropes can be computed with the help of these forces.

3.13 Parametric analysis of dragline boom cluster

For optimum boom design, the joints of the boom must be properly designed and should have higher fatigue life. As the fatigue life of the joints is computed by the stress generated in the joint, therefore there is a need of parametric analysis in order to obtain an optimum boom design. The basic four parameter of the existing boom namely the main chord diameter, main chord thickness, brace diameter and brace thickness have been assessed. However there are other parameters also that affect the stress value at the joint, viz, angle between boom main chords and brace member,

weld failure ,the joint eccentricity etc. In the current study other parameters have not been considered due to their complexity and very high computational requirements.

The range of the assessment parameters has been considered based on the CIDECT design guide. This design guide gives sufficient information about the ranges of these parameters for circular hollow structural (CHS) members.

The dragline boom has a number of circular hollow steel tubes consisting of various joints known as clusters. These clusters suffer high stresses during the dragline operating cycle and may fail due to fatigue. A number of joint have been deployed in the constructed model of dragline. Mainly CHS T joint and non-overlapping multiplanar CHS KK joints were deployed. In the current work the effect of various non-dimensional parameters associated with the CHS KK joint have been studied and results of the stress values have been determined. As fatigue failure occurs due to high magnitude of stress in the joint cluster, therefore the primary objective of the current parametric analysis was to propose a safe design to reduce the value of fatigue stress in the joint clusters while satisfactorily supporting the load.

CIDECT design guide clearly states the condition for designing of welded hollow steel structures. The ultimate capacity of joint depends on various non-dimensional parameters, which should be followed for safe designs. The parameters discussed below must be satisfied for optimum life of the joint, as per the CIDECT guidelines.

$$30^{\circ} \leq \theta_i \leq 90^{\circ}$$

$$0.2 \leq \beta = \frac{d_i}{d_0} \leq 1$$

$$5 \leq \gamma = \frac{d_0}{2t_0} \leq 25$$

$$g \geq t_1 + t_2$$

$$10 \leq \mu_i = \frac{d_i}{t_i} \leq 50$$

Where,

d_i is the diameter of the brace member,

d_0 is the diameter of the main chord,

t_i is the thickness of the chord,

t_0 is the thickness of the main chord,

g is the gap between the adjacent brace members, and

β, γ, μ_i are the non-dimensional parameters.

These parameters are shown in figure 3.19.

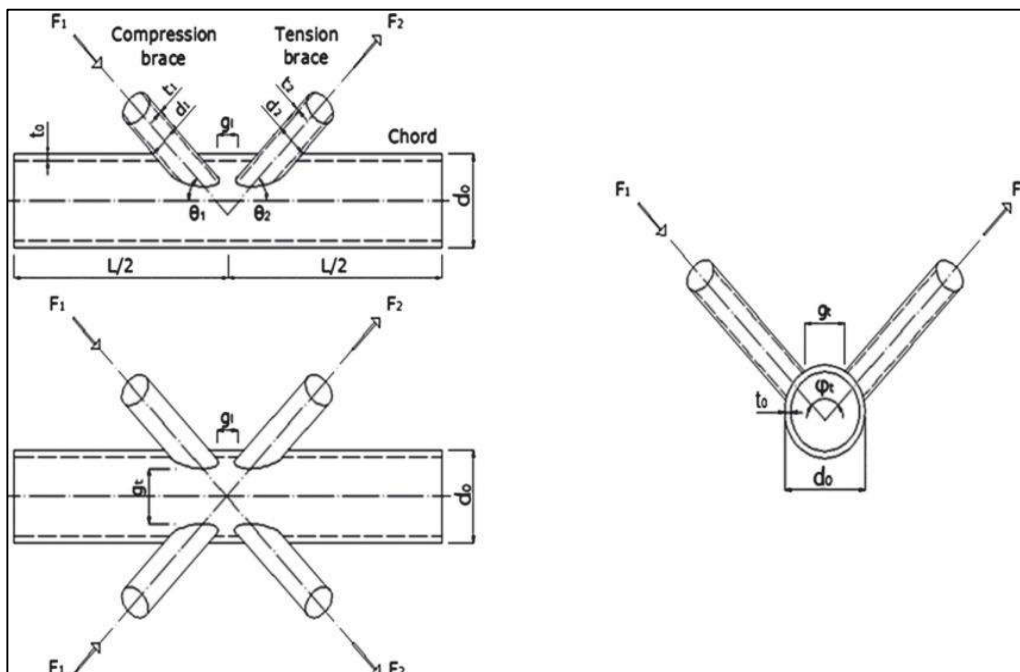


Figure 3.19 Design of CHS Multiplaner KK Joint

Chapter 3. Research Methodology

Thirteen models were subsequently analyzed for obtaining the magnitude of stress in the joint. The critical joint 2 near the boom point was selected for making a comparison between the parametric changes on design of the structure. Von-Mises stress has been evaluated for the selected joints 2. The parametric changes considered for the analysis are tabulated in table 3.5.

Table 3.5: Variation of parameters for the design of Joint 2

Model No.	d_o (mm)	t_o (mm)	d_i (mm)	t_i (mm)
A₁	406	20	207	8
A₂	390	20	207	8
A₃	350	20	207	8
A₄	300	20	207	8
B₁	406	16	207	8
B₂	406	12	207	8
B₃	406	8	207	8
C₁	406	20	190	8
C₂	406	20	100	8
C₃	406	20	80	8
D₁	406	20	207	6
D₂	406	20	207	4
D₃	406	20	207	2