CHAPTER 3

METHODOLOGY

3.1 Research methodology

In the present research work, the stress distribution on a three-dimensional dragline bucket model was determined and critically assessed by using state-of-art tools and techniques, such as, Finite Element Analysis (FEA),Computer-Aided Design (CAD),SOLIDWORKS, ANSYS, MATLAB and analytical methods. Dragline bucket was modeled using AutoCAD and SOLIDWORKS. The simulation and analysis of prepared 3D solid model was carried out on the ANSYS platform, to investigate the stresses, damage, factor of safety, life, and deformation on the bucket. Further the rake angle and resistive force of the dragline bucket was optimized with the help of MATLAB software using state-of-art method and analytical techniques.

A precise overview of the research design is illustrated in Figure 3.20 to give an overall picture of the scope of work covered.

During the field visit, the working and operation of dragline was carefully observed. It was found that in a complete cycle of dragline ,static loading condition occurs when the boom and the bucket get static for a very short period of time. Further, while excavating of the rock by bucket it was observed that bucket teeth were continuously interacting with overburden material. For analysis purpose, the important specifications and dimensions of dragline bucket and its assembly were collected.

Accordingly, a 3D solid model of the bucket and its assembly was developed in AutoCAD. After this, the solid model of bucket was transferred from AutoCAD to SOLIDWORKS environment for compatible file transfer. Simulation of dragline bucket has been done by ANSYS software. Before the simulation, evaluation of the resistive forces on bucket teeth using the analytical method proposed by McKyes (1985) was accomplished.

Simulation of the bucket has been done under two conditions (i) static loading condition and (ii) dynamic loading condition, by applying the loading and boundary conditions. By visualization of simulation results, it has been observed that hitch elements, arc anchors and teeth are critical components of dragline bucket.

When the dragline bucket is excavating the muck, the teeth touches the ground and interacts with the rock. Rake angle plays a critical role in designing of teeth. In this study a relationship between the different angle of bucket teeth and resistive force has been established to optimise the rake angle and minimise the resistive force. In this work to optimised the function of rake angle with the help of MATLAB.

3.2 Field visit

In this research work, the field visit in Northern Coal Field (NCL) and Sasan Coal Mines was undertaken. Dragline operation, manoeuvring of bucket and its assembly was observed and studied carefully in the field operations. Further, the important specifications and dimensions of dragline, its bucket and related assemblies were collected. Also, the physical properties of overburden material were obtained from the research. Figure 3.1 illustrates different views of a typical dragline bucket in the field. The material and rock properties obtained from the field are tabulated in Table 3.1.

Table 3. 1: Material properties of overburden

S.No.	Parameters	Value
1	Density of overburden (γ)	(1.8 - 2.4) t/m^3
2	Cohesion strength of overburden (c)	25 kPa
3	Internal friction angle (ϕ)	40^0
4	External friction angle (δ)	30^0
5	Rake angle of bucket teeth (α)	45^0

(a) (b)

(c) (d)

Figure 3. 1: (a-d) Different views of dragline bucket in mine A

(a) (b)

(c) (d)

Figure 3. 2: (a-d) Different views of dragline bucket teeth in mine A

(a) (b)

Figure 3. 3: (a-b) Dragline model in mine A

3.3 Computerized generation of 3D solid model of dragline bucket

Computer-Aided Design (CAD) offers many proven benefits over hand drawing techniques in mechanical modelling. An element or assembly model can be manipulated without problems and monitored 3-dimensionally on a computer simultaneously. The shape of the models can precisely be changed by altering the geometrical dimensions. CAD allows the designer to illustrate the spatial relationships between the parts and view the model in an under perspective. It prepares a suitable version of the model for simulation in a virtual environment. Mechanical simulations in CAD environment test the model by computing the dynamic relationships among the parts. For instance, one of the common simulations stress test measures the nodal modifications, and distortions on the model under distinct types of stresses. It uses the displacement of meshes or grids to guess the strain and stress components. In design and simulation steps, geometrical relationships on the parts, settlements of the parts in the system, material properties and finite element patterns should be assigned as close as possible to the reality, to get the most efficient and realistic results.

To create and simulate a model related to dragline bucket and its components, it is essentially required to research and criticise the running parts of a dragline and their impact on the productivity (Figure 3.4). A dragline contains swivel control cabinet placed on a fixed base. An outwardly extending boom is fixed axially to the cabinet. Winch assembly on the sides of the cabinet controls the bucket actions by releasing or retrieving ropes. Two pairs of ropes in different duties (hoist and drag ropes) provide the connection between the control cabinet and the rigging mechanism of bucket. Hoist rope lies along the boom as much as the sheave at the farthest point of boom and lies right down to the rigging and bucket assembly. Drag rope is positioned between the drag winch and rigging bucket meeting. In among the dragline operating components, the most stress exposed parts against the resistive forces of the formation are bucket elements. This is because bucket is directly interacting with the broken rock materials and is subjected to maximum abrasion, stress and stress inversion. Determination of the stress, fatigue life, damage factor of safety and deformation on these components are vital to preserving a view about the excavation efficiency of the dragline. These investigations can provide benefit to (i) location of the critical yielding points at the bucket components (ii) optimizing the rake angle of bucket teeth and minimizing the resistive force (iii) implement suitable maintenance plans, (iv) reduce the maintenance price and (v) increase the productivity and availability of the bucket.

The 3D bucket modelling has been carried out in AutoCAD by taking the specifications and dimensions of bucket and its assemblies from the field. The idea of development of 3D solid model of the bucket aims at subjecting the developed model to the working conditions and determine the stresses, damage, fatigue life, factor of safety, fatigue sensitivity and deformation in the bucket and its critical elements.

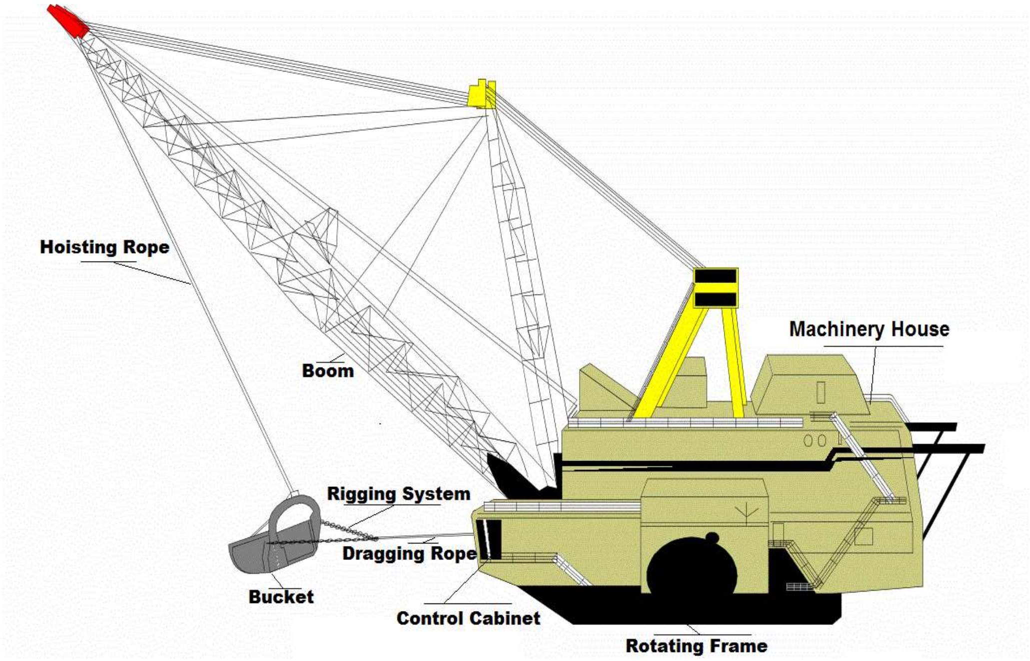


Figure 3. 4: Dragline component (Anonymous, 2004)

Dragline bucket has a massive size and shape as compared to other excavator buckets, being able to remove of 24 m^3 (30 yd^3) to 134 m^3 (175yd^3) overburden (Gilewicz, 1999). Due to its self-weight (70-80 tons approximately for 62 m^3 capacity), the resistance of the formation against its dead load and heavy payload, the varying magnitude of stresses occurs at different portions of the bucket. In this regard, it is important to simulate an accurate description of the bucket version with proper geometry.

Figure 3. 5: Bucket and rigging mechanism in mine A

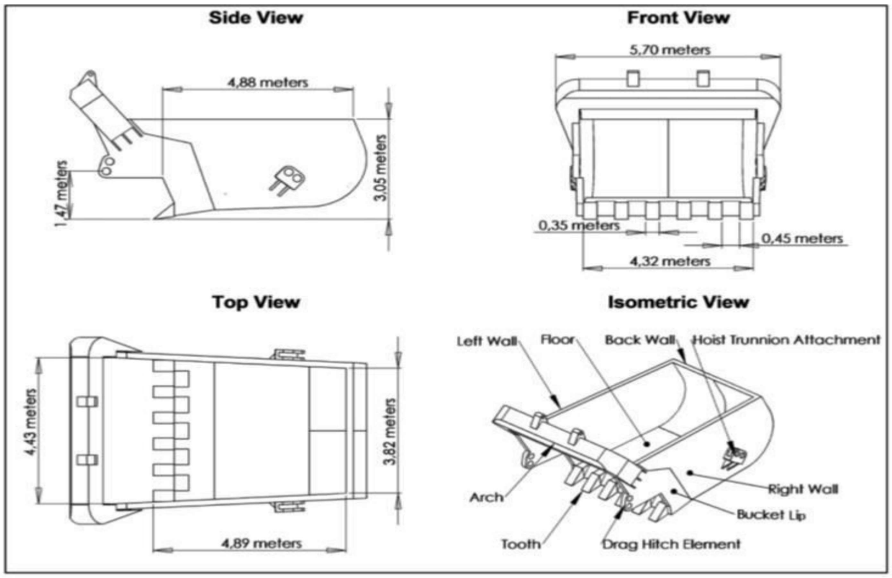


Figure 3. 6: Dragline bucket views with dimensions (Golbasi and Demirel, 2015)

Figure 3.6 presents a general view of the bucket solid model in plan and sections. In the field visit, the important dimensions of operating bucket were collected. The front dimension of the bucket is 4.0 m, and it consists of six digging teeth each one having 0.305 m width. Tooth space between the teeth is 0.435 m, length of bucket is 4.80 m and height of bucket is 3.20 m. According to these dimensions the 3D solid model of dragline bucket in AutoCAD was generated to reveal different views and sections of bucket and its teeth. The dragline bucket solid model generated in AutoCAD was imported to SOLIDWORKS and converted into IGES file because IGES file is compatible with ANSYS. Hence 3D solid model of bucket was imported into ANSYS for subsequent simulation studies.

3.4 ANSYS workbench with FEA

A complete range of finite element simulation solutions is provided by leading computer-added engineering (CAE) organizations. ANSYS workbench is a user-friendly platform. ANSYS is used for advanced engineering simulation technology. It offers a bi-directional connection with a major CAD system. The Workbench environment is geared toward improving productivity and ease of use among engineering teams.

The finite element method (FEM) is a powerful technique for determining stresses and deflections in structures too complex to analyses by strictly analytical methods. With this method, the structure is divided into a network of small elements connected to each other at a point. In this thesis, dragline bucket analysis includes examination and understanding of the stress distribution, life of bucket, safety factor and optimization of rake angle of the bucket body under different loading and working conditions. Numerical modelling technique has been used for examining all the aforestated critical parameters of bucket and its assembly. ANSYS is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. In this thesis, two loading conditions of the bucket were investigated, namely static loading condition and dynamic loading condition.

3.5 Steps in static structural analysis in ANSYS

- Import the solid model
- Input the material properties
- Generate the mesh area
- Application of the loading and boundary conditions
- Model simulation and solving
- Results and analysis

Static Structural analysis system has been considered in which the analysis system appears as several cells arranged in a column and in order to perform the analysis, we proceed from the top of the column to the bottom and complete the tasks required in each cell. A snapshot of the static structural module is given in Figure 3.7.

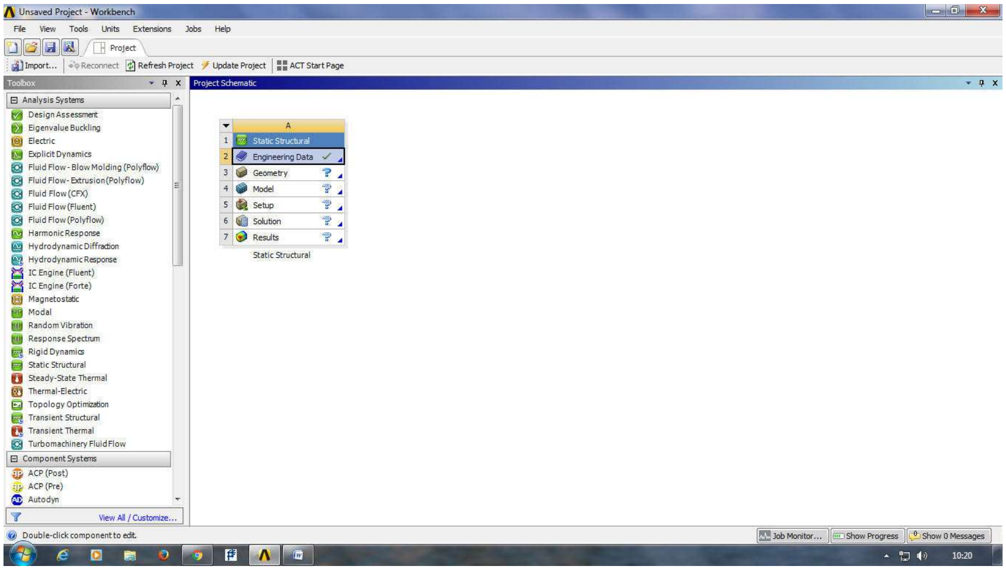


Figure 3. 7: Static structural module

3.6 Static loading condition

The static loading condition occurs when the boom and the bucket get static for a very short period of time. In the static loading condition, the bucket may be fully loaded and just ready to discharge the material (Figure 3.8 a) or, else, immediately after unloading the muck and commencing the swing back operation again the bucket comes to a static condition (Figure 3.8 b). Another instance of static loading condition occurs at the time of placement of the bucket at the bank (toe of the bench) just prior to being dragged against the bank for loading.

(a) (b)

Figure 3. 8: (a, b) Loaded and unloaded bucket in static conditions

3.6.1 Material properties simulated for modelling

Table 3. 2: Material properties of dragline bucket (ANSYS user's manual,2018)

Material	Steel
Density	7850 kg m^-3
Ultimate Tensile Strength	460 MPa
Yield Strength (Tensile)	250 MPa
Poisson's Ratio	0.3
Young's Modulus	2 x 10^5 MPa

The solid model with the capacity of the bucket as 62m^3 and the mass of bucket as 70 ton has been constructed as described in preceding section by taking the real-time dimension of the bucket. When overburden material is filled in the bucket, then the total mass of the bucket gets naturally increased. In the study mines, the overburden material was sandstone, which was being excavated by dragline. The density of this sandstone, being excavated by the dragline varied from 1.8 – 2.4 ton/m^3.

3.6.2 Finite element meshing for static loading

The meshing of the solid bodies has been carried out using the tetrahedron method and element size has been taken as 50 mm. The resultant meshing body incorporated 115982 solid elements and 200292 nodes. Meshing pattern is illustrated in Figure 3.9.

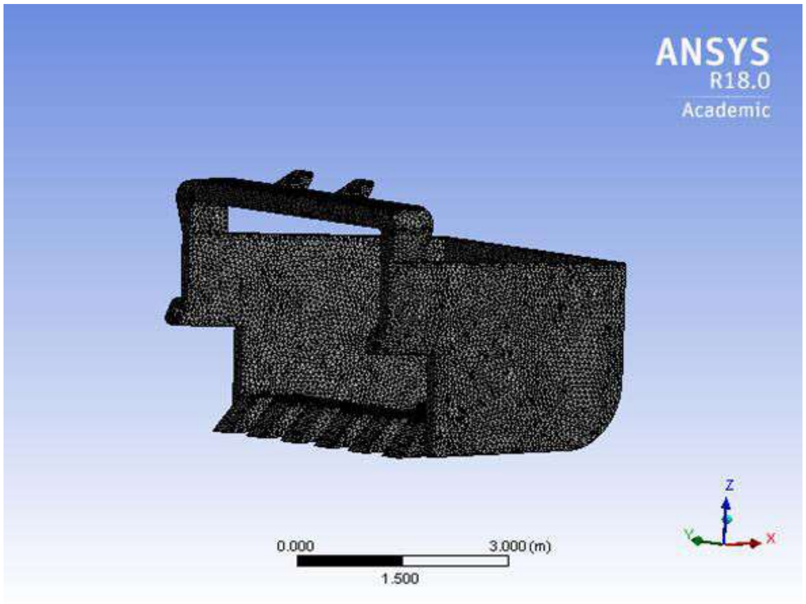


Figure 3. 9: Meshing body of bucket under static condition

3.6.3 Boundary and loading condition for static loading

An analysis has been done under the static condition in which hitch element and arc anchors have been considering to be fixed. The load has been applied at the base of the bucket. Boundary and loading conditions are illustrated in Figure 3.10 for empty bucket. Figure 3.11 shown as boundary and loading condition for a loaded bucket. Two conditions have been simulated, namely, (i) empty bucket (ii) loaded bucket condition. Different loads(as evaluated in the next chapter) have been applied to the model in both cases.

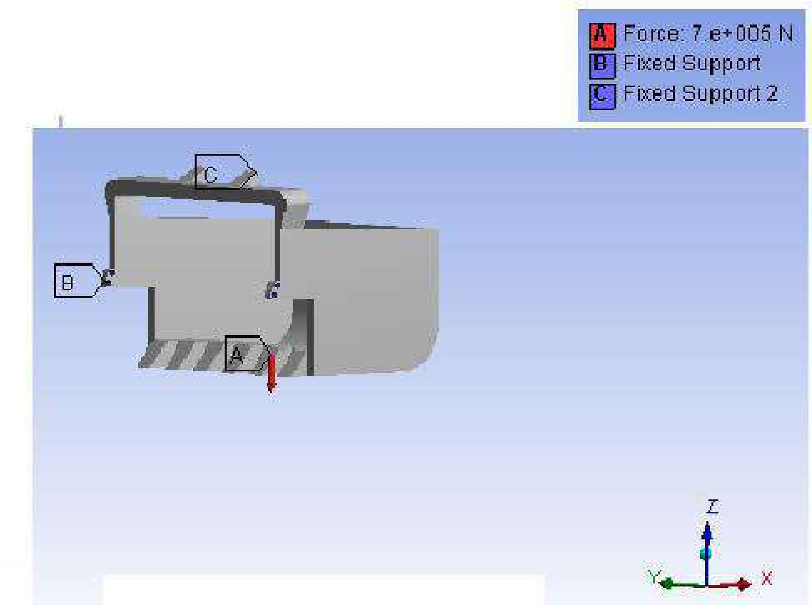


Figure 3. 10: Boundary and loading condition of empty bucket

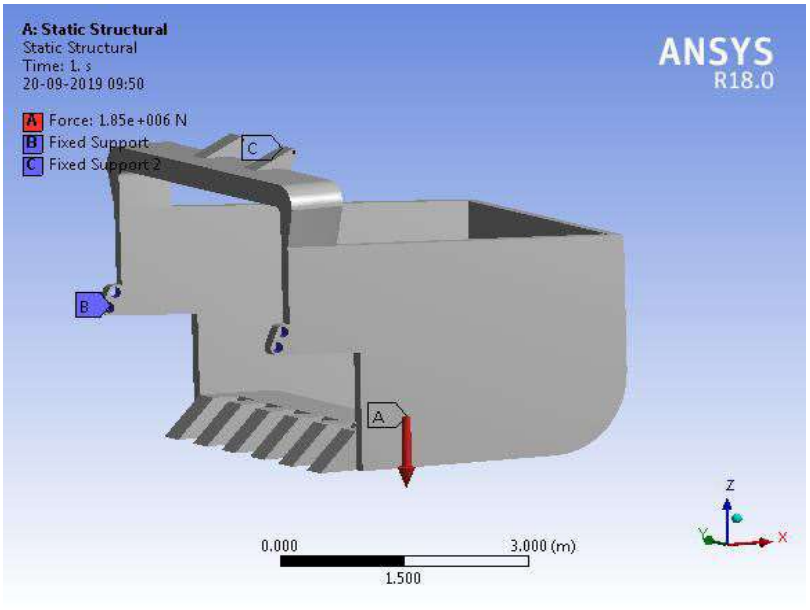


Figure 3. 11: Boundary and loading condition of loaded bucket

3.7 Boundary and loading condition for hitch elements and arc anchors under static loading conditions

The boundary and loading condition that we have used in the loaded bucket is the same boundary and loading condition used in the analysis of hitch elements and arc anchors. Boundary and loading conditions for hitch elements and arc anchors are illustrated in Figure 3.12.

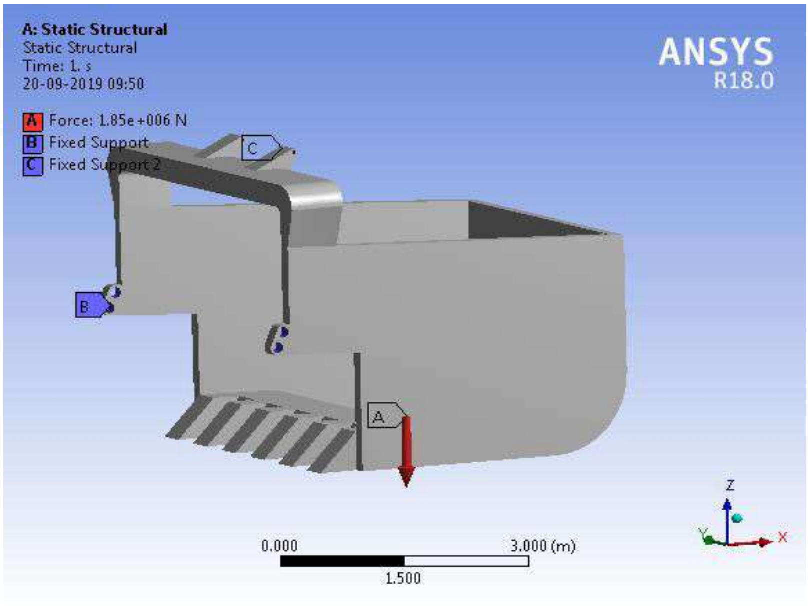


Figure 3. 12: Boundary and loading condition for hitch elements and arc anchors of the bucket

3.8 Steps in explicit dynamic analysis in ANSYS

Explicit dynamic is Ansys tool which is used to the mechanical problems involving short-duration simple loading, material failure and large material deformation.

- Import the solid model
- Input the material properties
- Meshing the bucket
- End time setting
- Application of the loading and boundary conditions

- Model simulation and solving
- Results and analysis

Explicit dynamic analysis has been considered in which the analysis system appears as several sections arranged in a column and in order to perform the analysis. We proceed from the top to the bottom and complete the tasks required in each cell, as shown in Figure 3.13. A snapshot of explicit dynamic module is given in Figure 3.13.

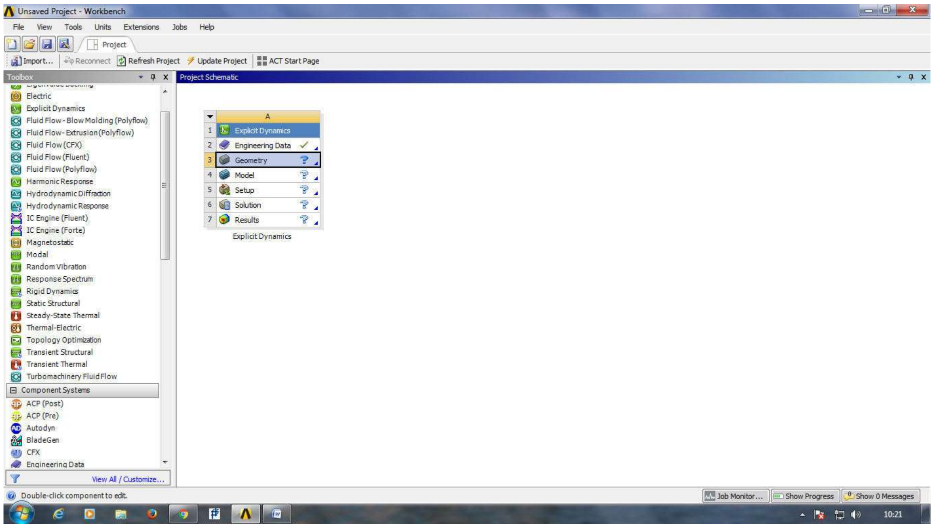


Figure 3. 13: Explicit dynamic module

3.9 Dynamic Loading Condition

In the research work, the dynamic loading condition implies that the bucket moves in a forward direction to fill the broken rock material at a constant velocity 0.5m/s (assume). Direction of movement of bucket illustrated in Figure 3.15.

3.9.1 Material property of the model

Material properties of bucket used in dynamic loading condition be kept same as in the static loading condition.

3.9.2 Finite element meshing for dynamic loading

The material specification corresponds to steel metal with the tensile strength of 460 MPa, showing elastic-perfectly plastic behaviour. The meshing of the solid bodies was carried out using a four-node linear tetrahedron continuum element. The meshing of the solid bodies was carried out using the tetrahedron method and element size was taken as 100 mm. Meshing pattern is illustrated in Figure 3.14. The resultant meshing body incorporates 75542 solid elements and 18315 nodes.

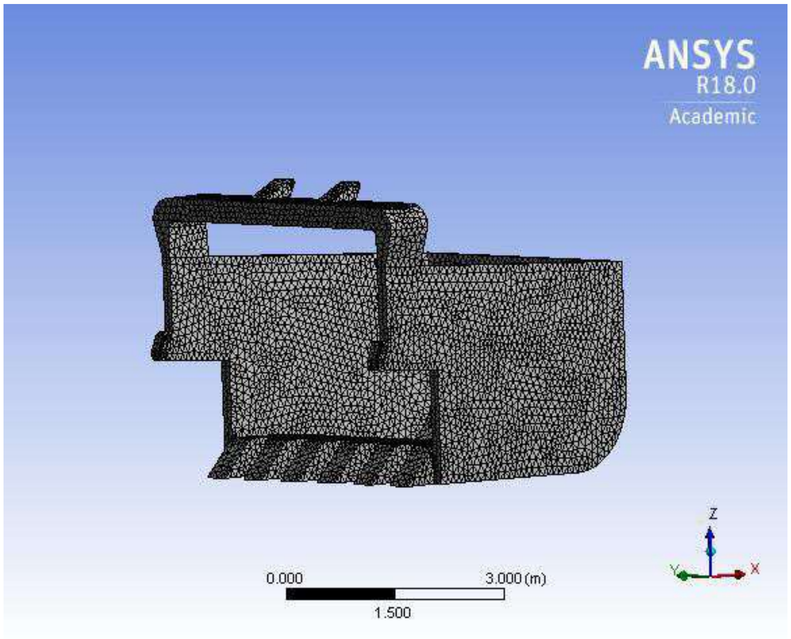


Figure 3. 14: Meshing body under the dynamic condition of bucket

3.9.3 Boundary and loading condition for dynamic loading

In the dynamic analysis, the boundary and loading conditions, external pressure on the teeth, velocity of bucket, and displacement of the bucket teeth have been considered as important parameters for the simulation. The velocity applied at the drag hitch element of bucket is 0.5 m/s(assume), and the external pressure applied on the digging teeth of bucket is 0.78 MPa (the method of computation is given in next chapter). The displacement of bucket teeth is free in the x-direction and zero in y and z-direction. The loading and boundary conditions of the moving bucket model are illustrated in Figure 3.15.

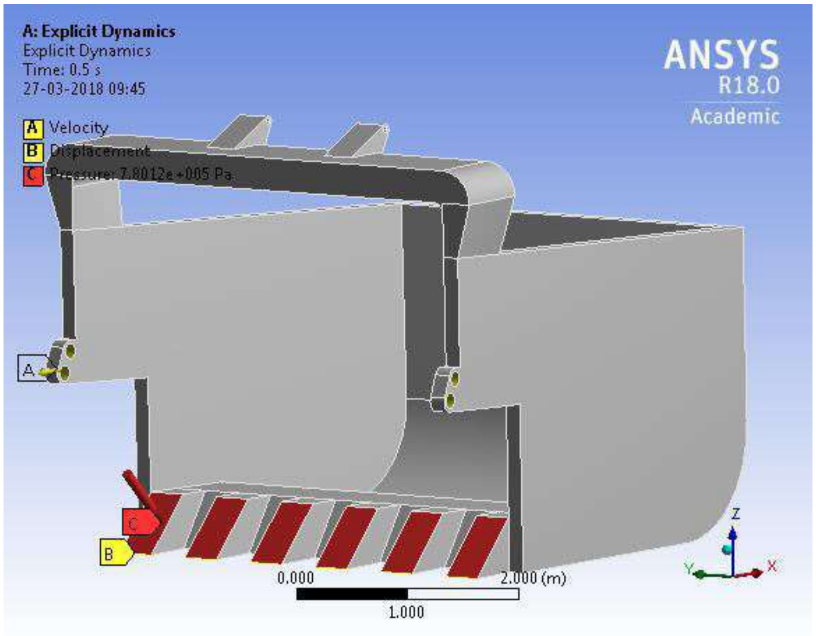


Figure 3. 15: Boundary and loading condition for bucket under the dynamic condition

3.10 Bucket teeth under static loading condition

In this loading condition, the physical properties of the teeth and mesh does not change. Only boundary conditions of the bucket have been changed.

3.10.1 Boundary and loading condition for bucket teeth

In this case, the base of the bucket was fixed in all directions, and pressure force was applied to the teeth to determine the affect of pressure force in empty bucket of teeth. In this analysis, the bucket was assumed to be in an equilibrium state and simulated the model by applying the boundary and loading conditions. Boundary and loading conditions are illustrated in Figure 3.16.

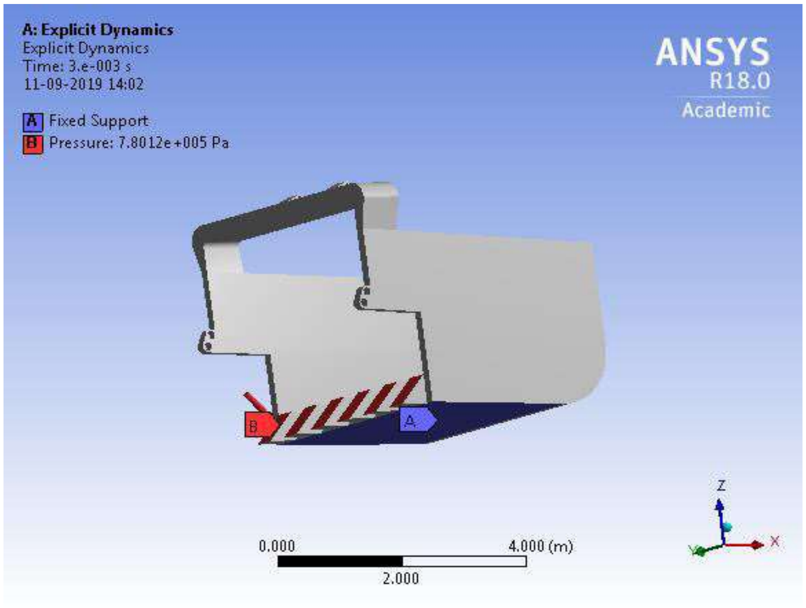


Figure 3. 16: Boundary and loading condition for bucket teeth under static condition

3.11 Optimisation of rake angle of dragline bucket teeth

MATLAB is the high-level language and communicating environment used by millions of engineers and scientists worldwide. The matrix-based language is a natural way to express computational mathematics. Optimization Toolbox provides functions for finding parameters that minimise or maximize objectives while satisfying constraints. The toolbox includes solvers for linear programming (LP), mixed-integer linear programming (MILP), quadratic programming (QP), nonlinear programming (NLP), constrained linear least squares, nonlinear least squares, and nonlinear equations. One can define the optimization problem with functions and matrices or by specifying variable expressions that reflect the underlying mathematics (MATLAB user's manual, 2018).

When the dragline bucket is excavating/digging the muck, the teeth touches the ground and interact with rock. The next step is the process of digging while the bucket teeth cuts the rock that offers resistive force to the bucket teeth.

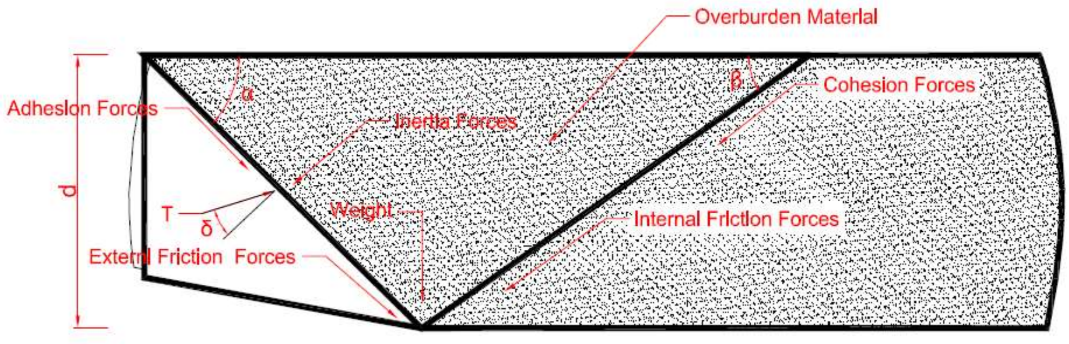


Figure 3. 17: Tooth geometry of bucket

In Figure 3.17, α, δ and β, are rake angle, external friction angle and shear plane angle. Rake angle plays a critical role in designing of teeth. For the optimum value of rake angle, the material flow along the teeth will be very smooth, at the same time specific pressure acting on the teeth will have a lesser value, which increases the tool life. T is cutting force and d is teeth depth.

In this thesis, the active-set method optimization technique has been adopted. The active-set method is mainly significant in optimization theory, as it poses which constraints will affect the result of optimization.

3.11.1 Overburden material properties

In the present study, two types of broken rock material encountered by dragline during excavation have been analysed, namely, sandstone rock and shale rock. The properties of these rock materials are tabulated in Table 3.3.

Table 3. 3: Material properties (Zhu,2012 and Strength,1996)

Material	**Properties**		
Sandstone	Density (kg/m^3)	Cohesion Strength (Pa)	Internal Friction Angle (degree)
	2000	25000	30^0
Shale	2500	30000	27^0

3.11.2 Input parameters of MATLAB

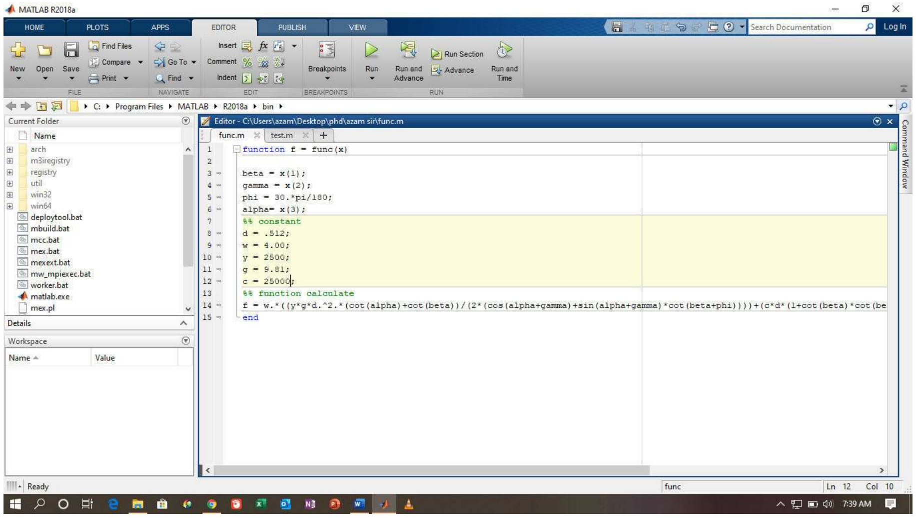
In the present study, certain parameters were assumed to be fixed while some parameters were varied, to study the impact of these variable parameters on the performance of dragline buckets.

Table 3. 4: Input parameter

<table>
<tr><th colspan="3">Fixed Parameter</th><th colspan="2">Variable Parameter</th></tr>
<tr><td>Parameter</td><td colspan="2">Value</td><td>Parameter</td><td>Value</td></tr>
<tr><td>Teeth depth (m)</td><td colspan="2">0.50</td><td>Rake angle</td><td>(15^0 to 90^0)</td></tr>
<tr><td>Bucket width (m)</td><td colspan="2">4.0</td><td colspan="2">Output Parameter</td></tr>
<tr><td>Cohesion strength of sandstone (Pa)</td><td colspan="2">25000</td><td colspan="2">Resistive force (KN)</td></tr>
<tr><td>Gravity (m/s^2)</td><td colspan="2">9.81</td><td colspan="2" rowspan="4"></td></tr>
<tr><td>density (kg/m3)</td><td colspan="2">2000</td></tr>
<tr><td rowspan="2">Internal friction angle (degree)</td><td>sandstone</td><td>30^0</td></tr>
<tr><td>shale</td><td>27^0</td></tr>
</table>

3.11.3 Input parameters function for sandstone

The input functions of sandstone are illustrated as a snapshot in Figure 3.18. Active-set method optimization technique and optimizing the fitness function of sandstone has been used.



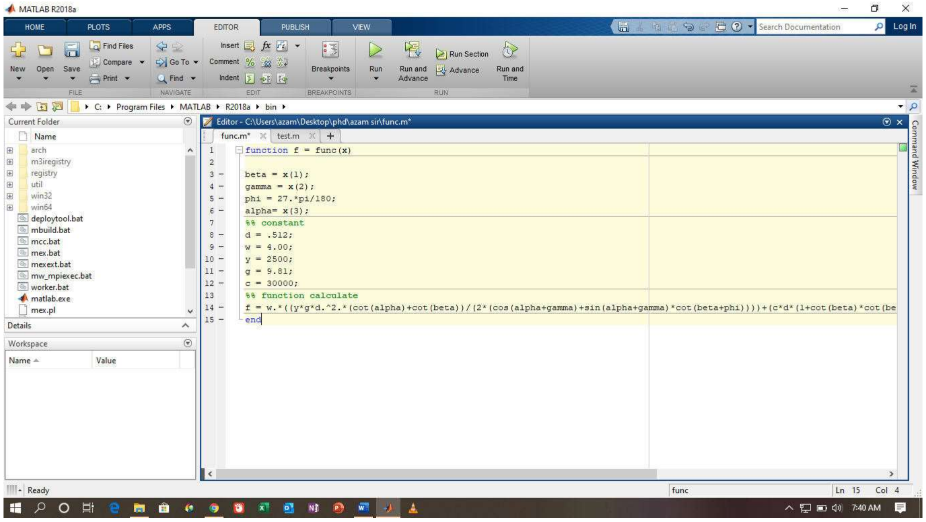
```
function f = func(x)

beta = x(1);
gamma = x(2);
phi = 30.*pi/180;
alpha= x(3);
%% constant
d = .512;
w = 4.00;
y = 2500;
g = 9.81;
c = 25000;
%% function calculate
f = w.*((y*g*d.^2.*(cot(alpha)+cot(beta))/(2*(cos(alpha+gamma)+sin(alpha+gamma)*cot(beta+phi))))+(c*d*(1+cot(beta)*cot(be
end
```

Figure 3. 18: Input function of MATLAB (Sandstone)

3.11.4 Input parameters function for shale rock

The input functions of shale rock are illustrated as a snapshot in Figure 3.19. Active-set method optimization technique and optimizing the fitness function of shale rock has been used.



```
function f = func(x)

beta = x(1);
gamma = x(2);
phi = 27.*pi/180;
alpha= x(3);
%% constant
d = .512;
w = 4.00;
y = 2500;
g = 9.81;
c = 30000;
%% function calculate
f = w.*((y*g*d.^2.*(cot(alpha)+cot(beta))/(2*(cos(alpha+gamma)+sin(alpha+gamma)*cot(beta+phi))))+(c*d*(1+cot(beta)*cot(be
end
```

Figure 3. 19: Input function of MATLAB (Shale rock)

3.12 Simplified block diagram revealing the research design

About the methodology, Figure 3.20 visualizes the flow chart followed in the thesis study.

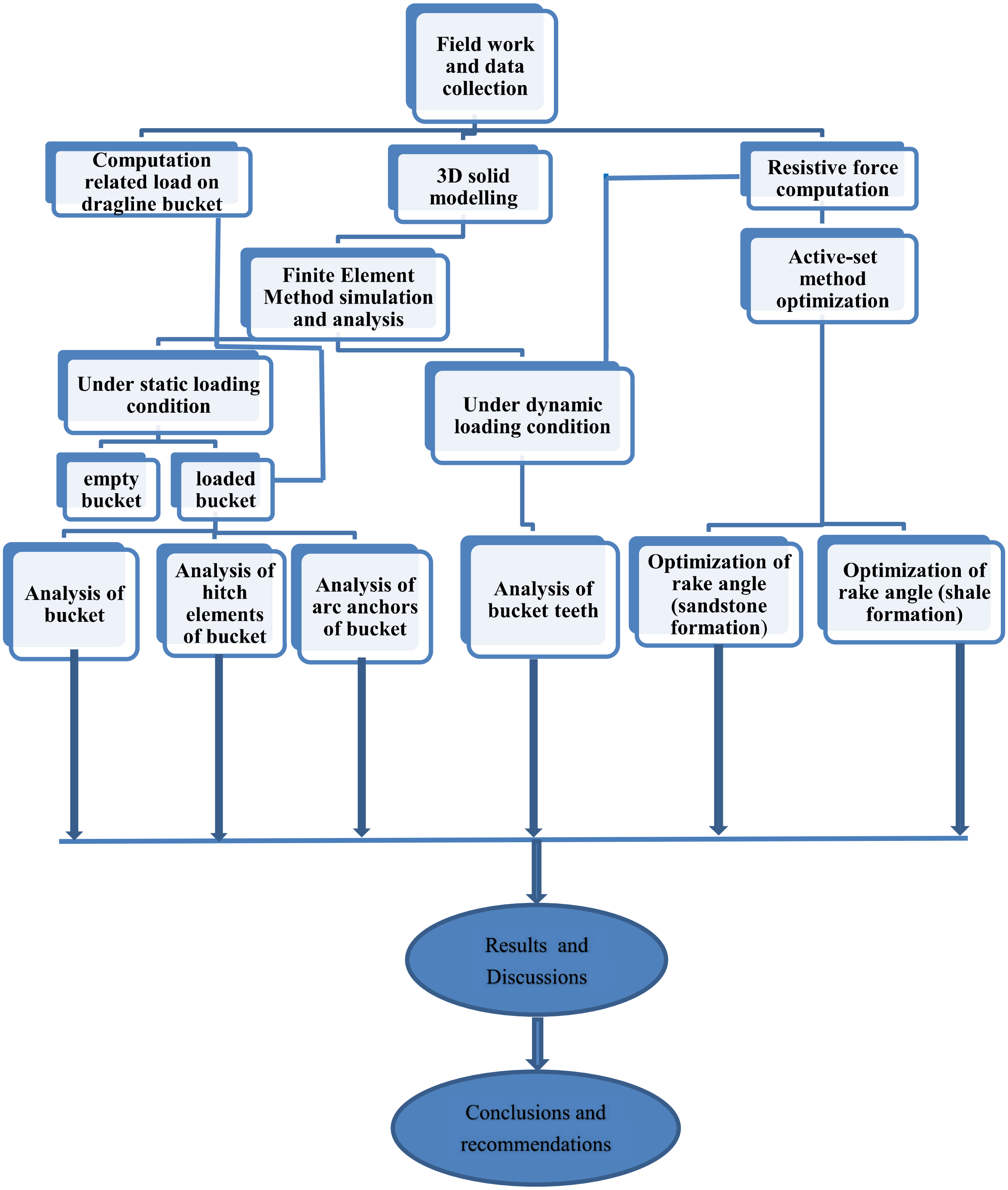


Figure 3. 20: Flow chart of the Thesis