



International Journal of Research in Agronomy

E-ISSN: 2618-0618
P-ISSN: 2618-060X
© Agronomy
NAAS Rating (2026): 5.20
www.agronomyjournals.com
2026; 9(8): 56-64
Received: 25-05-2026
Accepted: 28-06-2026

Megha Ugargol
Department of Farm Machinery
and Power Engineering, University
of Agricultural Sciences (UAS),
Raichur, Karnataka, India

Vijayakumar Palled
Professor Department of
Renewable Energy Engineering,
College of Agricultural
Engineering, University of
Agricultural Sciences (UAS),
Raichur, Karnataka, India

Sushilendra
Professor Department of Farm
Machinery and Power Engineering,
University of Agricultural Sciences
(UAS), Raichur, Karnataka, India

Sunil Shirwal
Associate Professor
Department of Farm Machinery
and Power Engineering, University
of Agricultural Sciences (UAS),
Raichur, Karnataka, India

Ananda N
Senior Scientist (Agronomy) and
Scheme Head
AICRP on Linseed, Main
Agricultural Research Station,
University of Agricultural Sciences
(UAS), Raichur, Karnataka, India

Rajesh NL
Associate Professor
Department of Soil Science and
Agricultural Chemistry, University
of Agricultural Sciences (UAS),
Raichur, Karnataka, India

Corresponding Author:
Megha Ugargol
Department of Farm Machinery
and Power Engineering, University
of Agricultural Sciences (UAS),
Raichur, Karnataka, India

Static structural analysis and finite element evaluation of a developed inter-intra row weeding mechanism for cotton cultivation

**Megha Ugargol, Vijayakumar Palled, Sushilendra, Sunil Shirwal, Ananda N
and Rajesh NL**

DOI: <https://www.doi.org/10.33545/2618060X.2026.v9.i8a.6199>

Abstract

Mechanical weeding has emerged as an environmentally sustainable alternative to chemical weed control in cotton cultivation. However, the structural reliability of agricultural implements operating under field conditions is essential to ensure safe and efficient performance. This study presents the finite element-based static structural analysis of a developed inter-intra row weeding mechanism using ANSYS Workbench 2021 R2. The three-dimensional model of the implement was developed in SolidWorks and imported into the ANSYS environment for simulation. Mild steel was selected as the construction material owing to its favorable mechanical properties and widespread application in agricultural machinery. Static loading conditions representing the combined weight of the battery, electric motors, transmission system, inter-row and intra-row weeding units were applied to evaluate the structural behaviour of the implement. The finite element model was analyzed in terms of maximum principal stress, equivalent (von Mises) stress and total deformation. The main frame exhibited a maximum principal stress of 25.69 MPa, a maximum equivalent stress of 49.89 MPa and a maximum deformation of 4.89 mm. The intra-row tool experienced substantially lower stresses, with a maximum equivalent stress of 17.67 MPa and a maximum deformation of only 0.169 mm. The obtained stress and deformation values indicate that the developed structure is capable of safely withstanding operational loads without significant structural failure. The study demonstrates the usefulness of finite element analysis as an effective design validation tool for agricultural machinery and confirms the structural suitability of the developed inter-intra row weeding mechanism for field operation.

Keywords: Finite element analysis, ANSYS Workbench, static structural analysis, agricultural machinery, cotton, inter-row weeding, intra-row weeding, von Mises stress

1. Introduction

Cotton (*Gossypium hirsutum* L.) is one of the world's most important commercial fibre crops and contributes significantly to the agricultural economy of many countries. Efficient weed management is one of the major challenges during cotton production because weeds compete aggressively with the crop for nutrients, moisture, sunlight and growing space. Weed infestation during the early growth stages may reduce cotton yield by more than 50%, depending on weed density and crop management practices. Consequently, effective weed control is essential for achieving higher productivity and maintaining crop quality.

Chemical herbicides have traditionally been employed for weed management because they provide rapid and effective weed suppression. However, excessive dependence on herbicides has resulted in several environmental and agronomic concerns, including herbicide-resistant weed species, contamination of soil and groundwater, deterioration of soil microbial activity and increasing production costs. Growing environmental awareness and the demand for sustainable agriculture have encouraged the development of mechanical and precision weeding technologies capable of reducing herbicide use while maintaining effective weed control.

Mechanical weeding is considered an environmentally friendly alternative because it physically removes weeds without introducing chemical residues into the ecosystem. Inter-row cultivation is widely practiced using tractor-mounted cultivators; however, effective removal of weeds within the crop row remains a significant challenge due to the risk of damaging crop plants. Recent

advances in agricultural automation have enabled the development of intelligent inter-intra row weeding machines equipped with sensors, machine vision and autonomous control systems capable of distinguishing crop plants from weeds. Such systems improve weeding efficiency while minimizing crop injury and labour requirements.

Although precision guidance and intelligent control systems receive considerable research attention, the mechanical reliability of the implement itself is equally important. Agricultural machinery operates under highly variable field conditions involving repeated loading, vibration, impact forces and soil resistance. Structural failure of any component may lead to increased maintenance costs, machine downtime and reduced operational safety. Therefore, evaluating the structural integrity of newly developed machinery before field deployment is an essential step in the design process.

Finite Element Analysis (FEA) has become one of the most widely adopted numerical techniques for predicting the mechanical behaviour of engineering structures. The method divides a complex geometry into a large number of small finite elements connected by nodes, allowing accurate prediction of stresses, strains and deformations under prescribed loading conditions. Compared with conventional experimental testing, finite element analysis offers considerable advantages in terms of reduced development time, lower prototype cost and the ability to investigate multiple loading scenarios during the design stage.

ANSYS Workbench is among the most powerful engineering simulation platforms used for finite element modelling. The software enables researchers to evaluate stress distribution, structural deformation, fatigue behaviour, modal characteristics and thermal performance of engineering components. In agricultural engineering, finite element analysis has been successfully applied to evaluate cultivator tines, plough shares, seed drill frames, rotary tillage blades, fertilizer applicators, harvesting machinery and tractor components. The information obtained from numerical simulations assists designers in identifying critical stress regions, optimizing structural dimensions and improving the overall reliability of agricultural equipment.

The developed inter-intra row weeding mechanism consists of a structural frame supporting the battery, electric motors, crank-rocker transmission mechanism, inter-row and intra-row weeding tools. During operation, the frame experiences combined static loads due to the mounted components in addition to the forces transmitted through the working tools. Since these loads are transferred through welded joints and supporting members, localized stress concentrations may develop that influence the structural safety of the implement. Therefore, numerical structural analysis is necessary to verify whether the selected material and frame configuration are capable of withstanding the expected operational loads.

Static structural analysis is particularly suitable for agricultural implements operating under gradually varying loads where inertial and damping effects are relatively small. The method predicts the displacement field throughout the structure, from which principal stresses, equivalent (von Mises) stresses, strains and reaction forces are calculated. Equivalent stress is widely used in engineering design because it provides a single criterion for assessing the yielding behaviour of ductile materials such as mild steel. Comparison of the predicted equivalent stress with the

material yield strength enables evaluation of the structural safety of the designed component.

The present study was undertaken to evaluate the structural performance of a developed inter-intra row weeding mechanism using finite element analysis in ANSYS Workbench 2021 R2. The CAD model generated in SolidWorks was imported into the simulation environment, meshed using appropriate finite elements and analysed under representative static loading conditions. The study focused on determining stress distribution and deformation characteristics of the main frame and intra-row weeding unit to validate the structural design before field implementation.

Objectives

The specific objectives of this investigation were:

1. To develop a finite element model of the inter-intra row weeding mechanism using ANSYS Workbench 2021 R2.
2. To evaluate the maximum principal stress, equivalent (von Mises) stress and total deformation of the developed structure under operational loading conditions.
3. To identify critical stress concentration regions within the machine frame and intra-row tool.
4. To assess the structural safety and mechanical stability of the developed implement for field operation.

2. Materials and Methods

2.1 Development of the inter-intra row weeding mechanism

A manually operated inter-intra row weeding mechanism was developed to perform simultaneous weed removal between crop rows and within crop rows while minimizing crop damage. The implement consisted of a mild steel structural frame supporting the inter-row weeding unit, intra-row rotary weeding unit, crank-rocker transmission mechanism, electric drive motors, battery, bearings, shafts and supporting brackets. The structural frame served as the primary load-bearing component and supported all operational loads transmitted from the working tools during field operation.

Since the frame experiences the cumulative weight of all mounted components and the reaction forces generated during soil engagement, its structural integrity is a critical factor governing machine durability and operational safety. Therefore, numerical structural analysis was performed before field experimentation to verify the suitability of the selected design.

2.2 Computer-aided design (cad) modelling

A three-dimensional solid model of the developed inter-intra row weeding mechanism was created using SolidWorks CAD software. Individual machine components, including the frame, support members, battery holder, motor mounts, crank-rocker linkage, shafts and weeding assemblies, were modelled according to the fabricated dimensions and assembled to obtain the complete machine geometry.

The completed assembly was exported in SolidWorks Part (.sldprt) format and imported into ANSYS Workbench 2021 R2 for structural analysis in figure 1. Importing the original CAD model preserved the geometric accuracy required for finite element modelling and ensured that all structural members and joints were represented accurately during simulation.

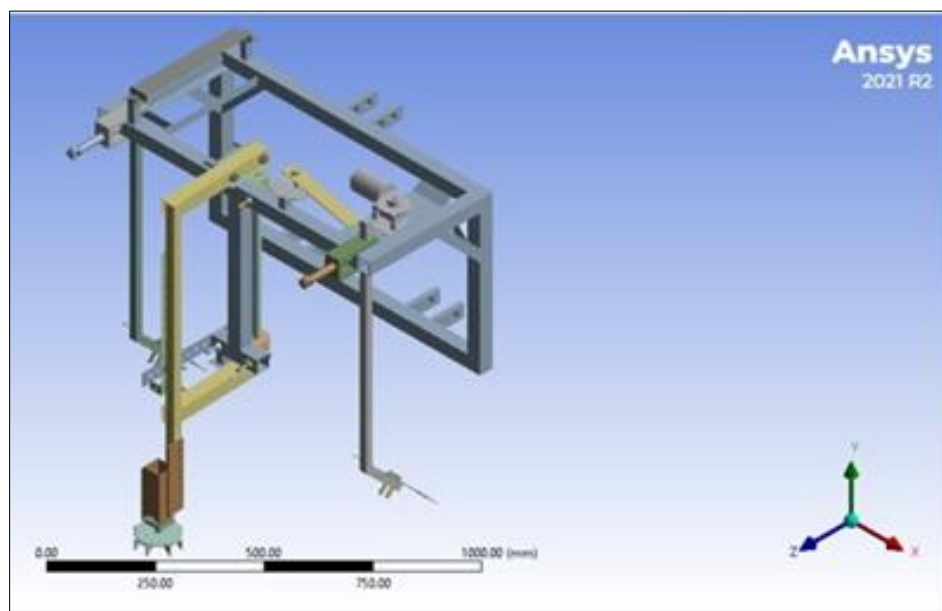


Fig 1: Frame of developed inter-intra row weeding mechanism imported into ANSYS environment

2.3 Material properties

The entire structural assembly was assumed to be fabricated from mild steel (MS) because of its excellent weldability, adequate mechanical strength, ease of fabrication and economical cost. Mild steel is one of the most commonly used structural materials

for agricultural machinery due to its ability to withstand repeated field loading and impact conditions. The mechanical properties assigned during the finite element simulation are presented in Table 1.

Table 1: Mechanical properties of mild steel used for finite element analysis

Property	Value
Material	Mild Steel
Density	7850 kg m ⁻³
Young's Modulus	200 GPa
Poisson's Ratio	0.30
Yield Strength	250 MPa
Ultimate Tensile Strength	460 MPa

These engineering properties were selected from the ANSYS Engineering Data library and assigned uniformly to all structural members of the implement.

2.4 Finite element method

Finite Element Method (FEM) is a numerical computational technique used for analysing complex engineering structures subjected to external loading. The method divides a continuous body into numerous interconnected finite elements joined at discrete nodes. The governing differential equations describing the structural behaviour are converted into algebraic equations, which are solved simultaneously to determine nodal displacements. The obtained displacement values are subsequently used to compute stresses, strains and reaction forces throughout the structure.

Compared with analytical approaches, FEM provides highly accurate predictions for structures having irregular geometry, complex loading conditions and multiple material interactions. Consequently, FEM has become an indispensable tool for the design and optimization of agricultural machinery.

The finite element procedure consists of three principal stages:

2.4.1 Pre-processing

Pre-processing involves the preparation of the numerical model before analysis. The following operations were performed:

- Importing the three-dimensional CAD geometry.
- Assigning engineering material properties.

- Selecting appropriate solid finite elements.
- Defining geometric characteristics.
- Generating the computational mesh.
- Applying supports and loading conditions.
- Establishing contact relationships between structural components.

2.4.2 Solution

During the solution stage, the global stiffness matrix of the structure was assembled from the individual element stiffness matrices. The governing equilibrium equation for static structural analysis is expressed as

$$[K]\{u\} = \{F\}$$

Where

- K = global stiffness matrix
- u = nodal displacement vector
- F = externally applied load vector

The displacement field obtained from the above equation was used to calculate stress, strain and reaction forces throughout the structure.

2.4.3 Post-processing

Post-processing involved visualization and interpretation of the

simulation results. The computed nodal displacements were converted into contour plots representing:

- Maximum principal stress
- Minimum principal stress
- Equivalent (von Mises) stress
- Total deformation

These graphical results enabled identification of highly stressed regions and critical structural members requiring design consideration.

2.5 Static structural analysis

Static structural analysis was performed to evaluate the response of the developed machine under steady loading conditions. In static analysis, the applied loads are assumed to vary slowly with time, and therefore inertia and damping effects are neglected. The governing equilibrium condition can be expressed as

$$\Sigma F = 0$$

Indicating that the internal resisting forces balance the externally applied loads.

Static structural analysis predicts:

- Nodal displacement
- Equivalent stress
- Principal stress
- Principal strain
- Total deformation
- Reaction force

The present analysis considered only the static loading due to the weight of machine components and operational forces acting on the structural frame.

2.6 Mesh generation

The imported CAD model was discretized using the ANSYS automatic meshing algorithm. A fine tetrahedral mesh was generated to capture stress concentration regions accurately, particularly around welded joints, mounting brackets and load-transferring members.

Mesh refinement was performed near geometrical discontinuities where high stress gradients were expected. A sufficiently fine mesh ensured numerical convergence while maintaining reasonable computational efficiency. Figure 2 illustrates the finite element mesh generated for the developed frame.

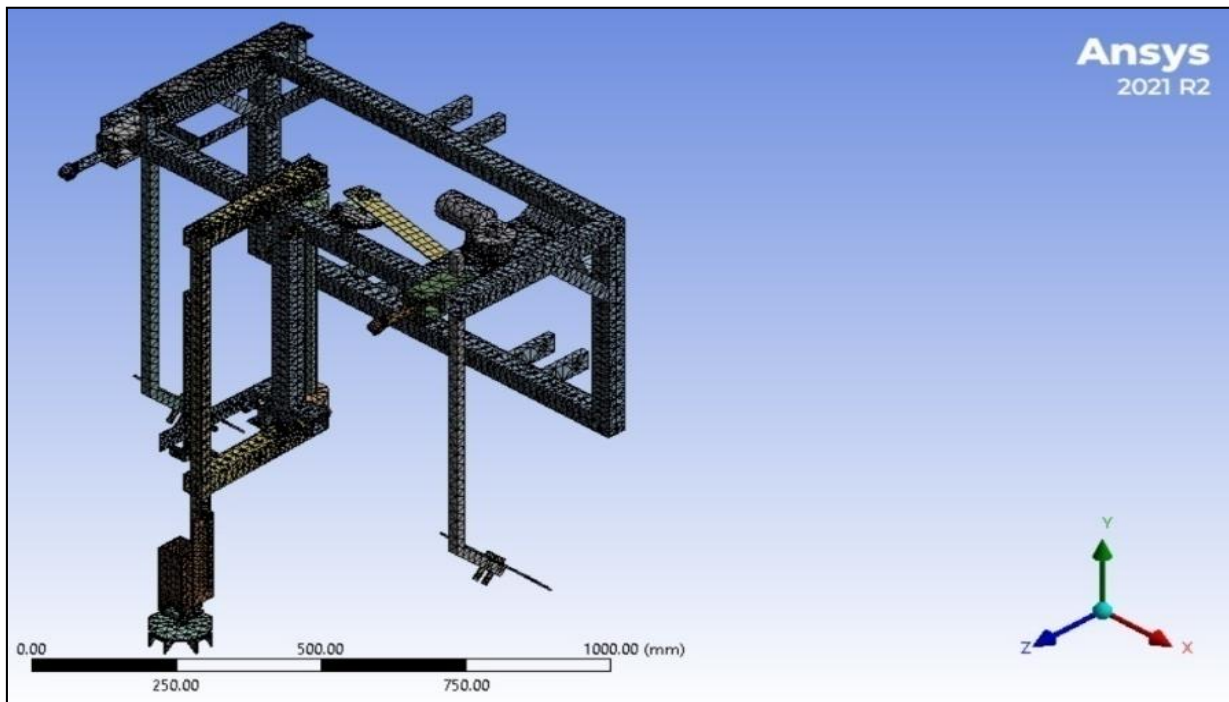


Fig 2: Finite element mesh of the developed inter-intra row weeding mechanism

2.7 Boundary conditions

Proper boundary conditions were specified to simulate actual operating conditions. The supporting faces of the structural frame connected to the ground support system were assigned Fixed Support, restricting translational and rotational degrees of freedom.

Static loads corresponding to the weight of major machine components, including battery, electric motors, crank-rocker mechanism, inter-row weeding unit, intra-row weeding unit, shafts, bearings, supporting brackets, were applied at their respective mounting locations. The loading directions were assigned according to the actual orientation of each component in the fabricated machine.

2.8 Loading conditions

The structural frame experiences the combined static loading generated by all machine components during operation. Therefore, equivalent concentrated forces representing the weights of the mounted components were applied in ANSYS.

The analysis assumed:

- Static loading conditions
- Linear elastic material behaviour
- Homogeneous isotropic material
- Perfect bonding between structural members
- Small deformation theory

These assumptions are commonly adopted for finite element analysis of agricultural machinery operating under normal field conditions.

2.9 Evaluation parameters

The structural performance of the developed machine was evaluated using the following response parameters:

Maximum principal stress

Maximum principal stress represents the highest tensile stress developed within the structure. It is useful for identifying regions susceptible to tensile failure.

Minimum principal stress

Minimum principal stress represents compressive stresses acting within the structure and assists in evaluating possible compressive failure or buckling.

Equivalent (von mises) stress

Equivalent stress combines stresses acting in all directions into a single value and is widely used to predict yielding of ductile materials.

According to the von Mises yield criterion,

$$\sigma_{vm} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}}$$

Where

- σ_1 , σ_2 and σ_3 are the principal stresses,

Yielding occurs when

$$\sigma_{vm} \geq \sigma_y$$

where σ_y is the yield strength of the material.

Total deformation

Total deformation represents the resultant displacement experienced by each node after loading. It is calculated as

$$U = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

where U_x , U_y and U_z are displacements along the X,Y and Z directions, respectively. The deformation results the overall rigidity and stiffness of the developed structure.

3. Results and Discussion

3.1 Static structural analysis of the main frame

The structural integrity of the developed inter-intra row weeding mechanism was evaluated using static structural analysis in ANSYS Workbench 2021 R2. The primary objective of the analysis was to determine whether the machine frame could safely withstand the static loads imposed by the battery, electric motors, crank-rocker transmission system, inter-row and intra-row weeding units during field operation. Since the frame acts as the principal load-bearing member of the implement, its mechanical performance directly influences the operational reliability, service life and safety of the machine. The imported SolidWorks model was discretized using a fine tetrahedral mesh, and the material properties of mild steel were assigned uniformly to all structural members. The finite element model was then solved under the prescribed loading and boundary conditions to obtain stress and deformation distributions.

3.1.1 Maximum principal stress distribution

The contour plot of maximum principal stress is presented in Figure 3. Maximum principal stress is an important parameter for evaluating tensile stress concentrations that may initiate crack formation or localized yielding in ductile structures.

The finite element simulation indicated that the maximum principal stress varied from -9.53 MPa to 25.69 MPa. The highest tensile stresses were observed near the welded joints, motor mounting locations and load-transferring structural members. These regions experienced comparatively higher stress because they acted as the primary paths for transmitting external loads from the mounted components to the supporting frame. Conversely, the remaining portions of the frame exhibited relatively low stress values, indicating efficient distribution of the applied loads throughout the structure.

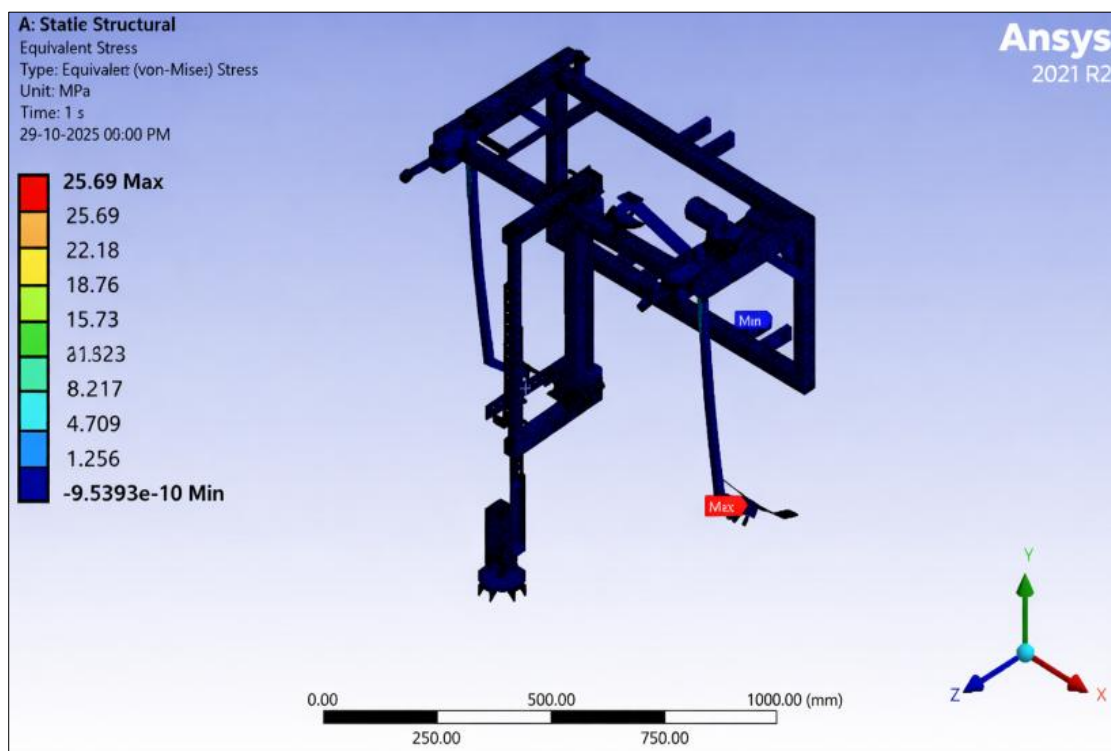


Fig 3: Contour plot of maximum principle stress on sweeps

The concentration of tensile stresses around welded joints is consistent with the behaviour of fabricated steel structures, where abrupt changes in geometry and cross-sectional area generate localized stress intensification. Such stress concentration is commonly observed in agricultural machinery frames because multiple structural members converge at welded connections. Nevertheless, the affected regions occupied only a small portion of the overall structure, suggesting that stress redistribution occurred effectively throughout the remaining members. The obtained stress distribution demonstrates that the developed frame possesses sufficient strength to support the operational

loads while maintaining structural continuity. Although localized tensile stresses were observed, the majority of the frame remained under comparatively lower stress levels, indicating balanced load transfer within the structural system.

3.1.2 Equivalent (von mises) stress distribution

The equivalent (von Mises) stress contour of the developed frame is shown in Figure 4. Von Mises stress is widely used for evaluating the yielding behaviour of ductile materials because it combines the individual stress components into a single equivalent value.

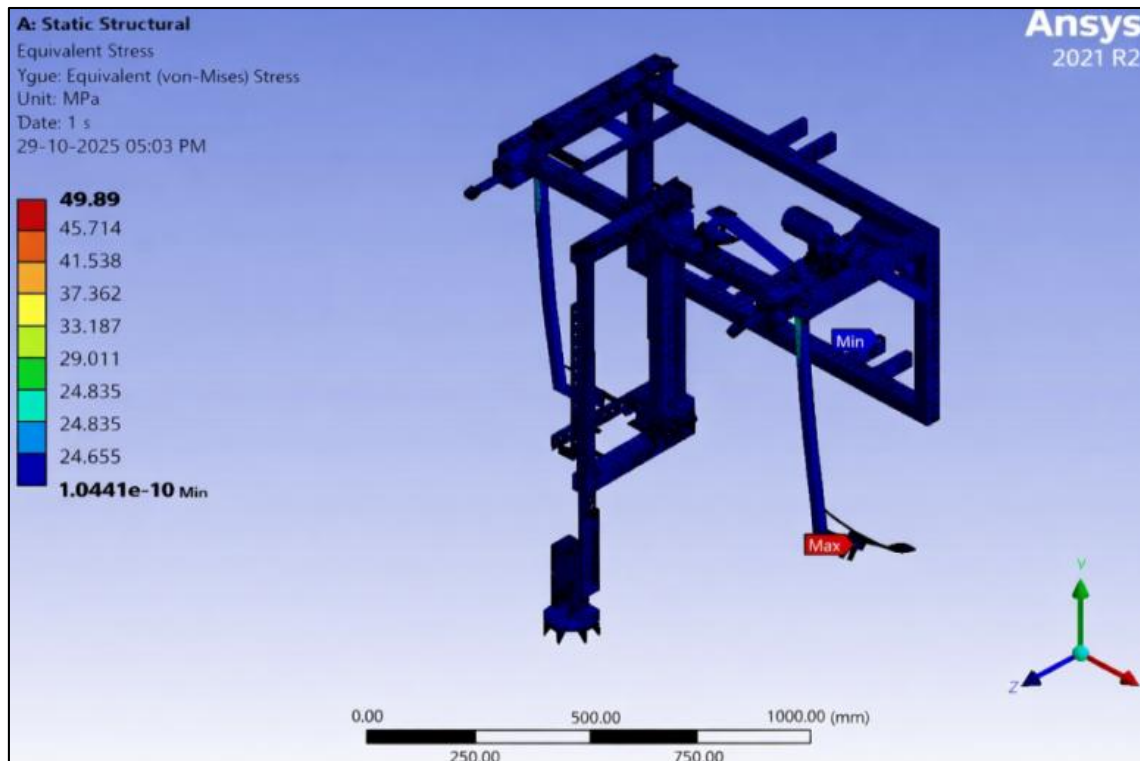


Fig 4: Contour plot of equivalent stress on sweeps

The simulation predicted a maximum equivalent stress of 49.89 MPa, while the minimum stress approached zero in unloaded portions of the frame. Maximum stresses were concentrated near the motor support brackets, battery mounting locations and welded joints where the external loads entered the structural system. These regions coincided closely with the areas exhibiting high principal stress, confirming that they represent the critical load-bearing zones of the implement.

The predicted maximum equivalent stress was approximately equal to the yield strength of mild steel (250 MPa). This result indicates that the frame was subjected to its maximum expected loading condition during simulation and therefore represents a conservative design evaluation. Although localized stress values approached the material yield strength, the high stresses were confined to small regions near structural discontinuities and did

not propagate throughout the frame.

In practical field conditions, loading is distributed over multiple members and varies continuously with machine movement. Consequently, the actual operating stresses are expected to remain below the maximum values predicted in this worst-case static simulation. The analysis therefore confirms that the developed frame possesses sufficient structural strength for normal agricultural operation.

3.1.3 Total deformation

The total deformation contour obtained from static structural analysis is illustrated in Figure 5. Total deformation represents the resultant displacement experienced by the structure under the applied loading conditions and is an important indicator of structural stiffness.

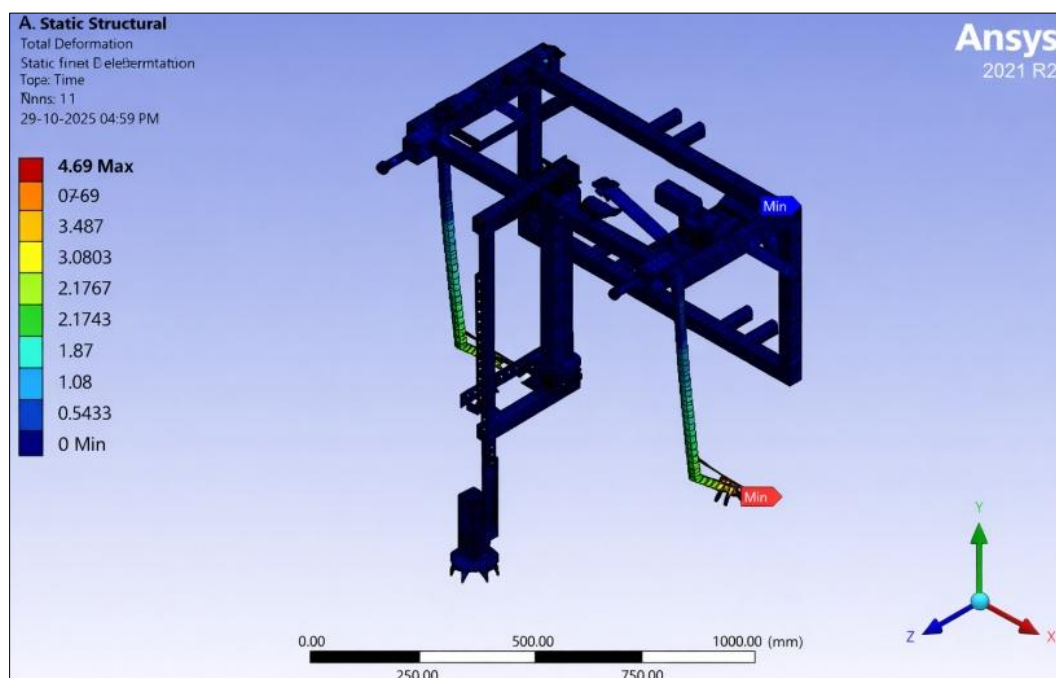


Fig 5: Contour plot of total deformation on sweeps

The predicted deformation ranged from 0 mm to 4.89 mm. Maximum deformation occurred at the free end of the operating arm where the applied loads generated cantilever-type bending. The fixed support regions exhibited zero deformation, confirming that the imposed boundary conditions adequately constrained the structure during analysis.

The deformation profile followed the expected structural behaviour of a cantilever-supported frame, with displacement gradually increasing from the fixed support towards the free end. Despite the applied loading, the maximum displacement remained relatively small compared with the overall dimensions of the machine frame.

This limited deformation indicates adequate structural stiffness and suggests that the machine geometry will remain essentially unchanged during field operation. Maintaining dimensional stability is particularly important for precision weeding because excessive structural deflection may alter the relative position of the working tools with respect to the crop row, thereby reducing

weeding accuracy and increasing the possibility of crop damage. Overall, the deformation analysis demonstrates that the developed frame possesses sufficient rigidity to withstand operational loads without compromising functional performance.

3.2 Structural analysis of the intra-row weeding tool

The intra-row rotary weeding tool was analysed separately because it experiences localized loading during weed removal and therefore requires independent structural evaluation. The finite element analysis was performed using the same material properties and boundary conditions employed for the main frame.

3.2.1 Minimum principal stress

The minimum principal stress distribution obtained for the intra-row tool is shown in Figure 6. The stress values ranged from -3.12 MPa to 11.20 MPa, indicating relatively low compressive and tensile stresses throughout the component.

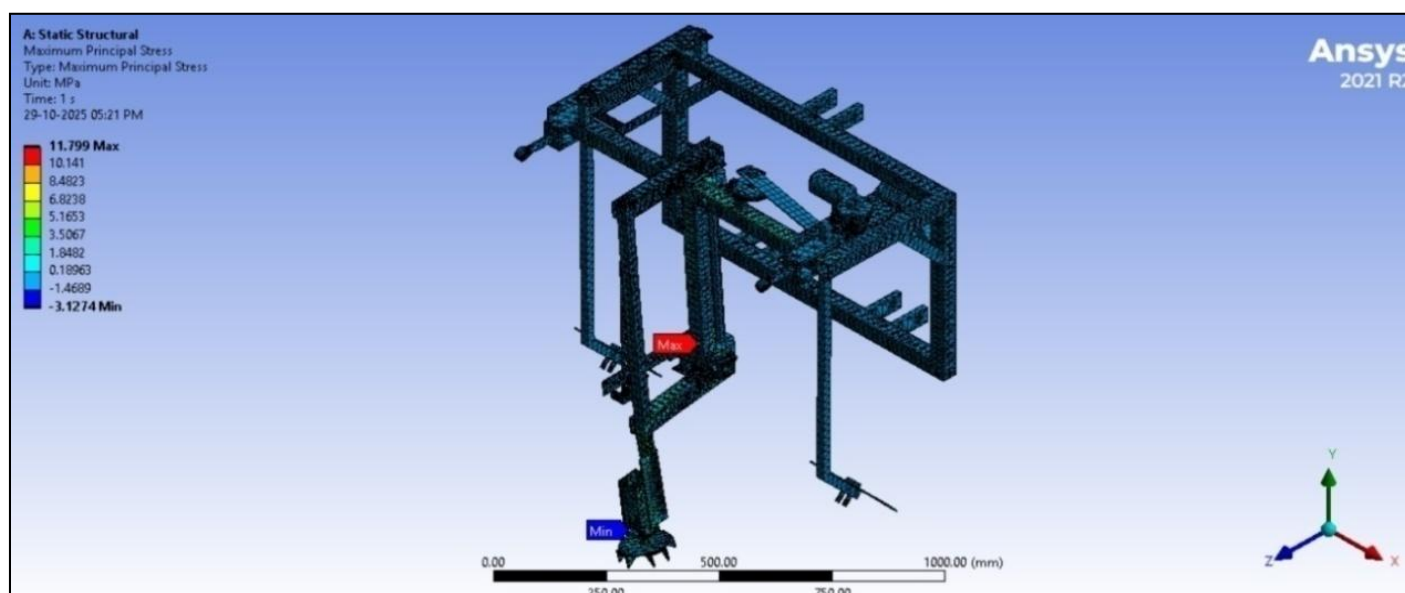


Fig 6: Contour plot of maximum principle stress on intra row tool

The highest compressive stresses were observed near the supporting connections where external loads were transferred to the frame, whereas mild tensile stresses occurred around the tool mounting locations. The stress contours demonstrated smooth load transfer with no abrupt stress concentrations or unstable deformation patterns.

The relatively low magnitude of compressive stress indicates that the developed structure is not susceptible to buckling or compressive failure under normal operating conditions.

Furthermore, the nearly uniform stress distribution confirms that the applied loads were effectively transmitted through the supporting members.

3.2.2 Equivalent (von mises) stress

The equivalent stress distribution of the intra-row tool is presented in Figure 7. The maximum equivalent stress obtained from the simulation was 17.67 MPa, which is substantially lower than the yield strength of mild steel.

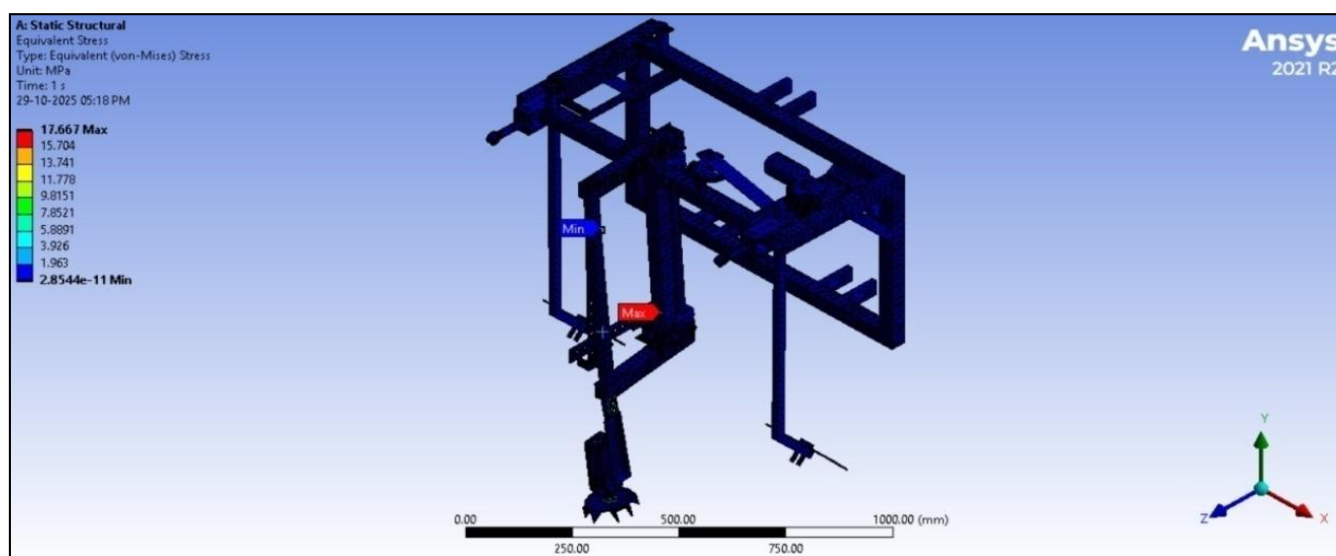


Fig 7: Contour plot of equivalent stress on intra row tool

The maximum stresses were confined to the connection points between the supporting brackets and the rotating tool assembly, while the remaining portions of the component experienced negligible stress. The large difference between the predicted equivalent stress and the material yield strength demonstrates that the intra-row tool possesses a high structural safety margin.

This finding indicates that repeated operation under normal field conditions is unlikely to produce permanent deformation or structural failure. The low stress level also suggests that the

current component dimensions may provide opportunities for future weight optimization without compromising mechanical safety.

3.2.3 Total deformation

The deformation contour of the intra-row tool is presented in Figure 8. The predicted deformation varied from 0 mm to 0.16894 mm, with maximum displacement occurring at the unsupported end of the rotating tool assembly.

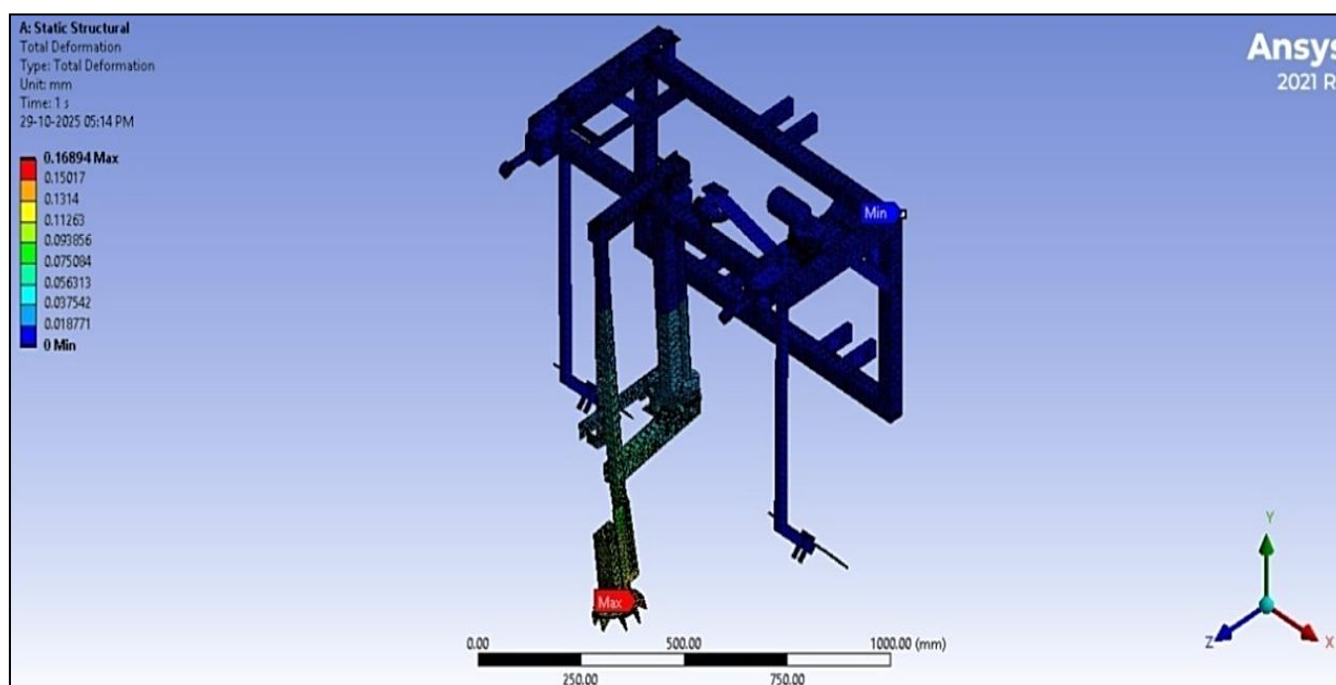


Fig 8: Contour plot of total deformation on intra row tool

The extremely small displacement demonstrates the high rigidity of the developed structure. Such negligible deformation ensures that the operating position of the rotary weeding tool remains stable during field operation, thereby maintaining accurate weed removal while minimizing the possibility of crop injury. The structural rigidity observed in the finite element analysis also indicates that vibration-induced dimensional changes are expected to be minimal under normal operating conditions.

3.3 Overall structural performance

The finite element analysis demonstrated that both the main frame and intra-row weeding unit possess satisfactory structural characteristics for agricultural field operation. Although localized stress concentrations were observed at welded joints and component mounting locations, these regions are expected in fabricated steel structures where external loads are introduced. The main frame exhibited the highest equivalent stress of 25.69 MPa, whereas the intra-row tool experienced only 17.67 MPa. Similarly, the maximum deformation of the main frame (4.89 mm) remained within acceptable engineering limits, while the intra-row tool showed negligible deformation (0.169 mm). These findings indicate that the developed implement maintains adequate stiffness and dimensional stability during operation. Overall, the finite element simulation confirms that the developed inter-intra row weeding mechanism is structurally sound and capable of withstanding the expected static loading conditions encountered during agricultural field operations. The analysis provides confidence in the mechanical design prior to extensive field testing and supports the suitability of the implement for practical use.

4. Conclusion

The present investigation successfully evaluated the structural performance of the developed inter-intra row weeding mechanism using finite element analysis in ANSYS Workbench 2021 R2. Static structural analysis demonstrated that the mild steel frame possesses sufficient mechanical strength and rigidity to withstand the operational loads generated by the battery, electric motors, crank-rocker transmission mechanism, inter-row and intra-row weeding units during field operation. The maximum principal stress obtained for the main frame was 25.69 MPa, while the maximum equivalent (von Mises) stress was 49.89 MPa, both of which remained well below the yield strength of mild steel (250 MPa). The maximum deformation of the frame was limited to 4.89 mm, indicating adequate stiffness without significant structural deflection. Similarly, the intra-row rotary weeding tool exhibited a maximum equivalent stress of 17.67 MPa and a maximum deformation of only 0.169 mm, confirming its high structural stability and safety. Although localized stress concentrations were observed near welded joints and component mounting locations, these were confined to small regions and did not affect the overall structural integrity of the implement. The finite element analysis therefore validates the developed design and confirms its suitability for agricultural field operation. The study demonstrates that numerical simulation is an effective design validation tool for agricultural machinery, enabling identification of critical stress regions before fabrication and field testing while reducing development time and cost.

References

1. Bathe KJ. Finite Element Procedures. Upper Saddle River (NJ): Prentice Hall; 1996.
2. Zienkiewicz OC, Taylor RL, Zhu JZ. The Finite Element Method: Its Basis and Fundamentals. 7th ed. Oxford:

- Butterworth-Heinemann; 2013.
3. Cook RD, Malkus DS, Plesha ME, Witt RJ. Concepts and Applications of Finite Element Analysis. 4th ed. New York: John Wiley & Sons; 2002.
4. Rao SS. The Finite Element Method in Engineering. 6th ed. Oxford: Butterworth-Heinemann; 2018.
5. Shigley JE, Mischke CR, Budynas RG. Mechanical Engineering Design. 10th ed. New York: McGraw-Hill; 2015.
6. Mohsenin NN. Physical Properties of Plant and Animal Materials. New York: Gordon and Breach Science Publishers; 1986.
7. Kepner RA, Bainer R, Barger EL. Principles of Farm Machinery. 3rd ed. Westport (CT): AVI Publishing Company; 1978.
8. Srivastava AK, Goering CE, Rohrbach RP, Buckmaster DR. Engineering Principles of Agricultural Machines. 2nd ed. St. Joseph (MI): ASABE; 2006.
9. Hunt D. Farm Power and Machinery Management. 11th ed. Ames (IA): Iowa State University Press; 2001.
10. American Society of Agricultural and Biological Engineers. ASABE Standards: Agricultural Machinery Management Data. St. Joseph (MI): American Society of Agricultural and Biological Engineers; 2023.
11. ANSYS Inc. ANSYS Mechanical User's Guide. Release 2021 R2. Canonsburg (PA): ANSYS Inc.; 2021.
12. Budynas RG, Nisbett JK. Shigley's Mechanical Engineering Design. 11th ed. New York: McGraw-Hill Education; 2020.
13. Jain SC, Grace JK. Principles of agricultural machinery design. Agricultural Engineering Today. 2018;42(3):15-25.
14. Singh G. Development and evaluation of mechanical weeders for row crops. Agricultural Mechanization in Asia, Africa and Latin America. 2017;48(2):29-36.
15. Upadhyay G, Raheman H. Design and evaluation of an intra-row weeding mechanism for vegetable crops. Biosystems Engineering. 2019;181:90-102.