Design And Finite Element Based Structural Optimization of An Onion Harvester Chassis Using Solid Works CAD–FEA Integration

Shaikh Kasifsohel Mahebubsab[1], Sachin Mallikarjun Nagure[2], Aatmling Narayanpure[3]
[1]*M. Tech Student, Department of Manufacturing Process Engineering, M.B.E. Societys of Engineering, Ambajogai*
[2]*Department of Manufacturing Process Engineering Asst. Prof. M.B.E. Societys of Engineering, Ambajogai*
[3]*Founder. of CAD Infield Technologies, Pune*

***Abstract*—The chassis is the primary load-carrying structure of a self-propelled onion harvester and must safely support the engine, gearbox, harvesting mechanism, hydraulic system, operator platform and axle assembly under field operating conditions. This paper presents a CAD–FEA integrated design and structural optimization study of an onion harvester chassis carried out using SOLIDWORKS Weldments and SOLIDWORKS Simulation. A parametric 3D model of the welded tubular chassis was developed and a static structural analysis was performed under a total operating load of approximately 5.0 kN, derived from the dead and live loads of the mounted components and amplified by a dynamic factor of 1.25 to represent field operating conditions.**

Fixed geometry boundary conditions were applied at the front axle and rear wheel mounting faces, and a beam-based mixed mesh was used to capture axial, bending, shear and torsional effects in the tubular members. Seven successive design iterations were analyzed, progressively introducing cross-members, truss reinforcement and improved load paths. The maximum equivalent (von Mises) stress reduced from 193.00 MPa in the initial design to 168.24 MPa in the optimized design, while the maximum resultant deformation reduced from 7.49 mm to 3.63 mm, both remaining within the elastic limit of the IS 2062 structural steel used (yield strength 250 MPa). The results demonstrate that iterative CAD–FEA based optimization is an effective, low-cost approach for validating and improving agricultural machinery chassis structures prior to fabrication.

Index Terms*—Onion Harvester Chassis; Finite Element Analysis; SOLIDWORKS Simulation; Structural Optimization; Von Mises Stress; Factor of Safety

I. INTRODUCTION

Mechanization of onion harvesting reduces labour dependency and improves field efficiency, and the chassis of a self-propelled onion harvester forms the structural backbone that carries the engine, transmission, harvesting mechanism, hydraulic units and operator load [1]. The chassis operates under uneven terrain, variable soil resistance and continuous vibration, so an under-designed frame can lead to excessive deformation, misalignment of the harvesting mechanism and fatigue failure, whereas an over-designed frame adds unnecessary weight and cost.

Conventional chassis design has traditionally relied on empirical calculations and physical prototyping, which are time-consuming and do not accurately predict local stress concentrations in complex welded structures.

Computer-Aided Engineering (CAE) tools that integrate CAD modeling with Finite Element Analysis (FEA), such as SOLIDWORKS, now allow stress distribution, deformation and failure-prone regions to be predicted virtually before fabrication, reducing development time, material usage and cost.

The present work develops a parametric CAD model of an onion harvester chassis, defines realistic static load cases from component weights, performs FEA-based static structural analysis, identifies critical stress and deformation regions, and iteratively optimizes the frame through reinforcement and load-path improvement, validated through comparative analysis of successive simulation results.

II. LITERATURE REVIEW

Walunj et al. (2023) studied the design and structural analysis of a self-propelled onion harvester chassis, evaluating tubular frames of AISI 4130, 4140 and 4340 steel under bending, torsional and operational loading, and reported that an optimized tubular chassis provides a favourable strength-to-weight ratio while FEA reliably identifies stress concentration regions.
Machado et al. (2014) analyzed agricultural harvester frames under machine weight, crop load and operational forces, showing that stress concentrations typically occur near component mounting regions and frame intersections, and emphasized computational validation to reduce field failures.

Pawar et al. (2018) optimized a tractor trolley chassis using FEA, finding critical stress zones near axle supports and cross-member connections, and demonstrated that iterative FEA-based optimization improves strength while reducing weight. Patil and Deshmukh (2019) applied FEA to automobile chassis design and confirmed that CAD-integrated simulation improves design accuracy and reduces development time.

For welded tubular structures, beam element modeling is computationally efficient and accurate for global structural response [2], bonded contact conditions adequately represent welded joints [3], and mesh convergence studies are essential for reliable stress prediction [4]. Reinforcement and cross-bracing have also been shown to substantially improve stiffness in agricultural frames without a large weight penalty.
Limited published work specifically addresses FEA-based static optimization of onion harvester chassis structures using SOLIDWORKS Simulation under realistic component loading, which is the gap addressed by the present study.

III. MATERIALS AND METHODOLOGY

3.1 CAD Model and Material

A 3D parametric model of the onion harvester chassis was developed in SOLIDWORKS Weldments using standard circular hollow steel pipe sections (21.3 × 2.3 mm and 26.9 × 3.2 mm), selected for their strength-to-weight ratio, torsional rigidity and weldability. Figure 1 shows the isometric CAD model of the chassis.

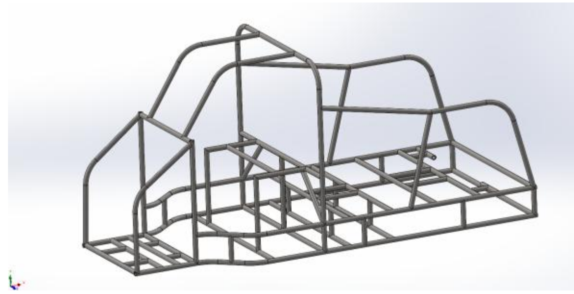

Fig. 1. Isometric CAD model of the onion harvester chassis

The chassis members are fabricated from IS 2062 Grade A structural steel, assumed homogeneous, isotropic and linearly elastic. Material properties used in the simulation are given in Table 1.

Table 1. Material properties of structural steel used

Property	Symbol	Value	Unit
Young's Modulus	E	2.10×10^5	MPa
Poisson's Ratio	ν	0.30	—
Density	ρ	7900	kg/m³
Yield Strength	σy	250	MPa
Ultimate Tensile Strength	σUTS	410	MPa
Shear Modulus	G	80,769	MPa

3.2 Load Case Definition

Static loads were derived from the weights of the major mounted components — engine (50 kg), gearbox (50 kg), harvesting unit (100 kg), hydraulic pump and cylinders (50 kg), front axle (50 kg), operator (100 kg), auxiliary components (30 kg) and chassis self-weight (80 kg) — giving a total static load of 510 kg (5003.1 N ≈ 5.0 kN). To represent field-induced vibration and impact effects within a static framework, a dynamic amplification factor (DAF) of 1.25 was applied:

$$F\_design = 5003.1 \times 1.25 \approx 6.25\ kN$$

Each component load was applied at its respective mounting location on the frame to represent the actual field-operating condition.

3.3 Boundary Conditions and Meshing

Fixed geometry constraints (restraining all six degrees of freedom) were applied at the front axle mounting faces and rear wheel/support mounting faces, representing chassis support through the wheel–

ground contact during field operation — a conservative assumption that allows evaluation of maximum stress and deformation.

A mixed mesh strategy was used: beam elements (with six degrees of freedom per node, based on Euler–Bernoulli/Timoshenko beam theory) for the slender tubular members, and second-order 10-node tetrahedral solid elements for mounting blocks, base plates and connection zones. SOLIDWORKS Simulation generated beam elements automatically along member centrelines, with automatic mesh refinement at load-transfer points, member intersections and support locations. Figure 2 shows the discretized finite element model.

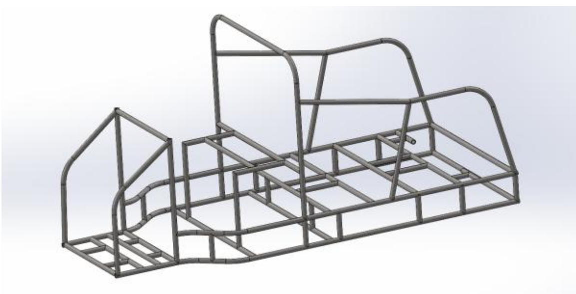

Fig. 2. Mixed mesh finite element model of the chassis

Welded joints were represented using global bonded contact, treating the welded frame as a single continuous structure, with beam joint connections at member intersections transferring axial force, shear, bending and torsion. Component-to-chassis interfaces (engine, gearbox, hydraulic system, harvesting unit and operator platform) were modeled as bonded interactions.

3.4 Analysis Approach

Linear static structural analysis was performed using the FFEPlus solver in SI units (mm, N, MPa, kg), with large-displacement, thermal and friction effects switched off, consistent with the assumption of small elastic deformation. Von Mises stress, resultant displacement, beam axial/shear forces, bending and torsional moments, support reaction forces and factor of safety were evaluated. The chassis was analyzed through seven successive design iterations (Static 1–Static 7), progressively introducing cross-members, truss reinforcement, optimized beam arrangement and local reinforcement at the harvesting-unit support, with results compared at each stage to guide optimization.

IV. RESULTS AND DISCUSSION

Table 2 summarizes the maximum von Mises stress and maximum resultant deformation obtained at each of the seven design iterations.

Table 2. Maximum stress and deformation across design iterations

Study	Max. Stress (MPa)	Max. Deform. (mm)	Remark
Static 1	193.00	7.49	Initial design
Static 2	303.42	6.21	Reinforcement added
Static 3	301.02	4.87	Load path improved
Static 4	163.00	4.11	Cross-members optimized
Static 5	165.60	3.70	Truss members added
Static 6	168.20	3.63	Local reinforcement
Static 7	168.24	3.63	Optimized design

The initial design (Static 1) exhibited a maximum von Mises stress of 193.00 MPa and a maximum deformation of 7.49 mm, concentrated at the front harvesting-unit support and longitudinal member intersections, where the front chassis extension behaves similarly to a cantilever under the harvesting mechanism load.

The intermediate reinforcement stages (Static 2–3) temporarily increased peak stress at newly loaded members before load-path optimization in Static 4 reduced it substantially.

Progressive addition of truss members and local reinforcement in Static 5–7 reduced deformation further while stabilizing peak stress, yielding a final optimized design with a maximum stress of 168.24 MPa and a maximum deformation of 3.63 mm — a 12.8% reduction in peak stress and a 51.5% reduction in maximum deformation relative to the initial design.

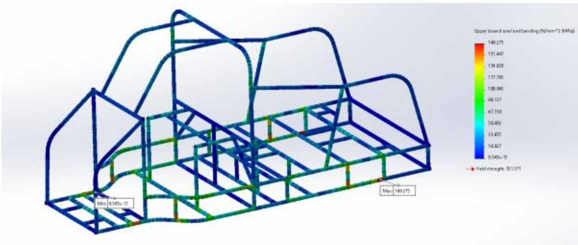

Fig. 3. Von Mises beam stress distribution — optimized design (Static 7)

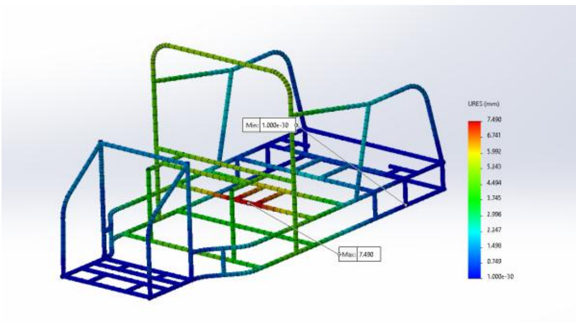

Fig. 4. Resultant displacement contour of the chassis

The stress contour of the optimized design (Fig. 3) shows that the majority of structural members remain in the low-stress (blue/cyan) range, with only localized, moderate stress concentration near the harvesting-unit mounting region and welded joints carrying concentrated component loads — consistent with the load-transfer pattern expected for a welded tubular frame under combined axial, bending and torsional loading. The displacement contour (Fig. 4) confirms that maximum deflection occurs at the unsupported front cantilever section, while the remainder of the frame shows minimal displacement, indicating adequate stiffness to maintain alignment of the harvesting mechanism, engine and transmission system.

With a material yield strength of 250 MPa, the optimized design gives a factor of safety of approximately:

$FOS = \sigma y / \sigma VM = 250 / 168.24 \approx 1.49$

This is close to the conventional recommended minimum of 1.5 for agricultural machinery under static loading, indicating that the optimized frame is structurally adequate but would benefit from further local reinforcement near the harvesting-unit support to raise the safety margin. The longitudinal members were confirmed as the principal load path, carrying the majority of axial and bending load from the engine, gearbox, operator and harvesting mechanism, while cross-members distributed transverse loads and reduced localized stress concentration. Overall, the results confirm that iterative FEA-based design modification — through added cross-bracing, truss members and local reinforcement — is effective in simultaneously improving stiffness and reducing peak stress without a major increase in structural weight.

V. CONCLUSION

A CAD–FEA integrated methodology was used to design and structurally optimize the chassis of a self-propelled onion harvester. A parametric weldment model was developed in SOLIDWORKS, realistic static loads (≈5.0 kN, amplified to ≈6.25 kN with a dynamic factor of 1.25) were applied at component mounting locations, and static structural analysis was performed using a beam-based mixed mesh with fixed supports at the axle and wheel mounting faces.

Across seven design iterations, the maximum von Mises stress was reduced from 193.00 MPa to 168.24 MPa and the maximum deformation was reduced from 7.49 mm to 3.63 mm, both remaining within the elastic range of the IS 2062 structural steel used, giving a factor of safety of approximately 1.49 for the optimized design. The harvesting-unit support and front cantilever region were identified as the critical zones requiring further local reinforcement. The study confirms that CAD-integrated FEA is an effective, low-cost tool for validating and optimizing agricultural machinery chassis structures prior to fabrication, and future work should extend this static evaluation to dynamic/modal analysis, fatigue life assessment, topology optimization and experimental validation of a fabricated prototype.

ACKNOWLEDGMENT

The author gratefully acknowledges the guidance and support of [Guide Name] and the Department of Mechanical Engineering, [Institute Name], during this MTech project work.

REFERENCES

[1] Walunj, S. Nalawade, R. Kadam, S. Thawkar, R. Mane, and P. Wable, “Design of chassis for self-propelled onion harvester,” in *Proc. Conf. on Agricultural Machinery Design*, 2023.

[2] A. Machado et al., “Structural simulation of agricultural harvester frames under operational loading,” Journal of Agricultural Engineering Research, 2014.
[3] S. Pawar et al., “Analysis and optimization of tractor trolley chassis using finite element methods,” International Journal of Mechanical Engineering and Technology, 2018.
[4] R. Patil and S. Deshmukh, “Finite element analysis in automobile chassis design,” International Journal of Automotive Engineering, 2019.
[5] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, Concepts and Applications of Finite Element Analysis, 4th ed. New York, NY, USA: John Wiley & Sons, 2002.
[6] W. Fricke, “Fatigue analysis of welded joints: State of development,” Marine Structures, vol. 16, no. 3, pp. 185–200, 2003.
[7] K.-J. Bathe, Finite Element Procedures. Englewood Cliffs, NJ, USA: Prentice Hall, 2006.
[8] J. S. Arora, Introduction to Optimum Design, 2nd ed. San Diego, CA, USA: Elsevier Academic Press, 2004.
[9] M. P. Bendsøe and O. Sigmund, Topology Optimization: Theory, Methods and Applications, 2nd ed. Berlin, Germany: Springer-Verlag, 2003.
[10] Bureau of Indian Standards, IS 2062: Hot Rolled Medium and High Tensile Structural Steel—Specification, 7th rev. New Delhi, India: BIS, 2011.
[11] Bureau of Indian Standards, IS 800: General Construction in Steel—Code of Practice, 3rd rev. New Delhi, India: BIS, 2007.
[12] Bureau of Indian Standards, IS 1161: Steel Tubes for Structural Purposes—Specification, 4th rev. New Delhi, India: BIS, 1998.
[13] ASTM International, ASTM A36/A36M-19: Standard Specification for Carbon Structural Steel. West Conshohocken, PA, USA: ASTM International, 2019.
[14] Dassault Systèmes, SOLIDWORKS Simulation Help—Static Analysis. Waltham, MA, USA: Dassault Systèmes, 2023.
[15] American Welding Society, ANSI/AWS D1.1/D1.1M: Structural Welding *Code—Steel*, 24th ed. Miami, FL, USA: American Welding Society, 2020.