FEA-Based Structural Optimization of An Onion Harvester Chassis

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

***Abstract*—The chassis of a self-propelled onion harvester is the primary load-bearing structure that supports the engine, gearbox, harvesting mechanism, hydraulic system, operator platform, and axle assembly, and its structural adequacy directly governs the reliability and safety of the machine under field conditions. This study presents the design, finite element based structural evaluation, and iterative optimization of an onion harvester chassis using a three-dimensional parametric CAD model developed in SOLIDWORKS Weldments and analysed using SOLIDWORKS Simulation. A total static operating load of approximately 5.0 kN, representing the combined weight of the mounted components and self-weight of the structure, was applied, and a dynamic amplification factor of 1.25 was used to account for field-induced operational effects. A beam-based finite element model with fixed boundary conditions at the axle mounting locations was solved through seven successive design iterations (Static 1 to Static 7), progressively introducing cross-members, truss elements, and local reinforcement. The optimized configuration (Static 7) reduced the maximum resultant deformation from 7.49 mm to 3.63 mm, a reduction of about 51.5%, while maintaining the maximum von Mises stress of 168.24 MPa well below the 351.57 MPa yield strength of the structural steel, corresponding to a factor of safety greater than 2.0. The results confirm that integrated CAD-FEA optimization is an effective, low-cost approach for validating and improving welded tubular chassis structures in agricultural machinery prior to fabrication.**

Index Terms*—Onion harvester; Chassis design; Finite element analysis; SOLIDWORKS Simulation; Structural optimization; Von Mises stress; Factor of safety

I. INTRODUCTION

Mechanization in agriculture has become essential to improve productivity, reduce labour dependency, and enhance the efficiency of field operations such as harvesting [1], [2]. Specialized machines such as onion harvesters reduce manual effort while increasing throughput, and the chassis is the fundamental structural framework of such machines, carrying the engine, transmission system, harvesting mechanism, hydraulic units, and operator load. The chassis operates under uneven terrain, variable soil resistance, and continuous vibration, and must sustain combined static and dynamic loading without excessive deformation, misalignment, or fatigue failure.

Conventional chassis design has relied on empirical calculations, which typically lead to over-designed or structurally inefficient frames. The availability of integrated Computer-Aided Design (CAD) and Finite Element Analysis (FEA) platforms, such as SOLIDWORKS, now allows engineers to model, simulate, and validate complex welded structures virtually, predicting stress distribution, deformation, and failure-prone regions before fabrication and thereby reducing development cost, time, and material consumption [3], [4].

The specific objectives of the present work are to: (i) develop a parametric three-dimensional CAD model of the onion harvester chassis; (ii) define realistic load cases from the weights of the major mounted components; (iii) perform static structural FEA to evaluate stress, deformation, and factor of safety; (iv) identify critical stress and deformation zones; (v) implement design improvements through reinforcement and optimized load paths; and (vi) validate the optimized design through comparative FEA results. The scope of the study is limited to static structural analysis of the chassis frame; dynamic, fatigue, and vibration analyses are identified as future scope.

II. LITERATURE REVIEW

Structural design and analysis of agricultural machinery chassis has received considerable research attention because the chassis is the primary load-carrying structure supporting the engine, gearbox, harvesting mechanism, hydraulic system, operator platform, and axle assemblies.

Walunj et al. [5] carried out a detailed structural analysis of a self-propelled onion harvester chassis using ANSYS, evaluating vertical bending, torsional, and operational loading conditions for AISI 4130, 4140, and 4340 steels, and reported that an optimized tubular chassis provided a favourable strength-to-weight ratio while FEA effectively identified critical stress zones. Machado et al. [6] investigated agricultural harvester frames and showed that stress concentrations typically develop near component mounting regions and frame intersections, emphasising the importance of computational validation to reduce field failures.

Pawar et al. [7] optimized a tractor trolley chassis and found that axle mounting regions and cross-member connections are critical stress zones, and that iterative FEA-based optimization can substantially improve strength while reducing material usage. Patil and Deshmukh [8] applied FEA to automobile chassis design and confirmed that CAD-integrated simulation reduces development time while improving the accuracy of failure prediction.

For welded tubular frames, beam element modelling has been shown to provide computationally efficient and sufficiently accurate results [9], [10], while bonded contact conditions adequately represent welded joints for global structural analysis [11]; mesh convergence remains essential to ensure reliable stress predictions [10]. Structural optimization studies further indicate that size optimization of member cross-sections [12], topology optimization of material distribution [13], and reinforcement through cross-bracing and truss members can achieve significant weight and deformation reduction without compromising safety.

Collectively, the reviewed literature confirms that FEA is a widely accepted tool for validating agricultural and vehicular chassis structures, that tubular welded frames analysed with beam elements offer an efficient modelling strategy, and that component mounting locations are consistently identified as critical stress regions. However, limited published work specifically addresses detailed FEA-based static structural optimization of an onion harvester chassis using SOLIDWORKS Simulation under realistic component loading, which forms the motivation for the present study.

III. MATERIALS AND METHODOLOGY

3.1 System Description and CAD Modelling

The onion harvester chassis was modelled as a welded tubular weldment structure in SOLIDWORKS, using circular hollow steel pipe sections selected for their favourable strength-to-weight ratio, torsional rigidity, and weldability (Table 1).

The chassis supports the driver/operator, diesel engine, harvesting mechanism, gearbox/transmission, hydraulic pump and cylinders, front axle assembly, and auxiliary components, with an overall length of 2500 mm, width of 1200 mm, height of 900 mm, and wheelbase of 1800 mm. All members were fabricated from IS 2062 Grade A structural steel, and a complete weldment cut list was generated parametrically for fabrication planning.

Table 1. Standard pipe sections used in the onion harvester chassis

Section Type	Designation	Outside Diameter	Wall Thickness	Application
Circular hollow pipe	21.3 × 2.3	21.3 mm	2.3 mm	Main chassis members
Circular hollow pipe	26.9 × 3.2	26.9 mm	3.2 mm	Load-carrying / reinforcement
Circular hollow pipe	21.3 × 2.3	21.3 mm	2.3 mm	Side supports and bracing
Circular hollow pipe	26.9 × 3.2	26.9 mm	3.2 mm	Critical load-transfer points

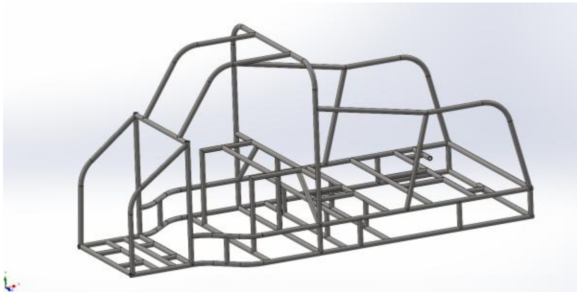

Figure 1. Isometric CAD model of the onion harvester chassis

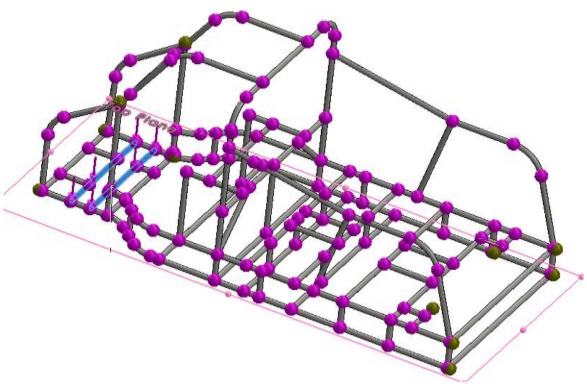

Figure 2. Complete welded structural frame of the chassis (isometric view)

3.2 Material Properties

The chassis material was assigned isotropic, homogeneous structural steel properties in SOLIDWORKS Simulation, as summarized in Table 2. The material was assumed to behave linearly elastically within the operating stress range.

Table 2. Material properties of structural steel used for the chassis

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	351.57	MPa
Ultimate tensile strength	σUTS	410	MPa
Shear modulus	G	80,769	MPa

3.3 Load Case Definition

Dead loads (engine, gearbox, hydraulic system, front axle, auxiliary components, and self-weight) totalled 310 kg (3.04 kN), while live loads (operator and harvesting unit) totalled 200 kg (1.96 kN), giving a total static operating load of 510 kg (5003.1 N). To represent dynamic effects arising from uneven terrain, harvesting resistance, and engine/gearbox vibration within the static analysis framework, a dynamic amplification factor (DAF) of 1.25 was applied, giving a design load of approximately 6.25 kN, consistent with conservative agricultural machinery design practice.

3.4 Boundary Conditions, Contacts, and Meshing

Fixed geometry constraints (all six degrees of freedom restrained) were applied at the front axle and rear wheel/support mounting faces to represent ground support through the wheel axle. A global bonded contact was applied across the weldment frame to represent welded construction, with component-bonded interactions defined at the engine, gearbox, hydraulic pump, harvesting unit, and operator platform mounting regions, and beam joint connections applied automatically at member intersections to transfer axial force, shear, bending, and torsional load between connected members.

The model was discretized using a beam element mesh generated along the centreline of each structural member, which is well suited to slender tubular members whose length substantially exceeds their cross-sectional dimensions, and which significantly reduces computational effort relative to a full solid mesh while retaining accuracy in axial, bending, torsional, and shear response. Mesh density was automatically refined at load-transfer points, member intersections, and support locations. Linear static analysis was performed using the automatic (FFEPlus) solver, with large-displacement, thermal, and friction effects excluded, consistent with the small-deformation, elastic assumptions of the study.

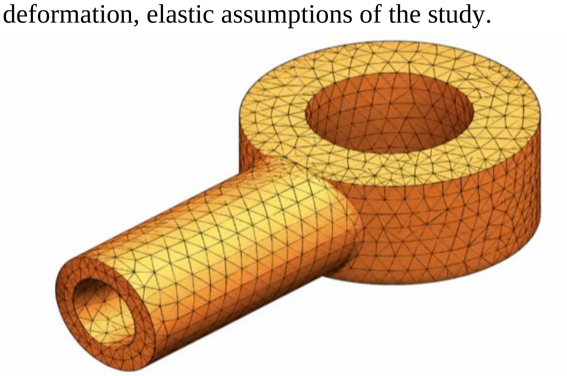

Figure 3. CAD model discretized into finite (beam) elements

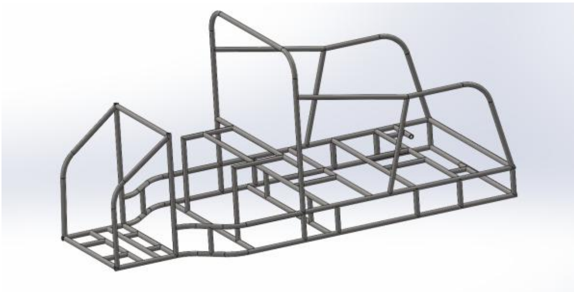

Figure 4. Detailed mixed mesh view of the initial chassis design

IV. RESULTS AND DISCUSSION

4.1 Iterative Structural Optimization

Static structural analysis was carried out over seven progressive design iterations (Static 1 to Static 7), beginning with the initial chassis configuration and successively introducing reinforcement members, improved load paths, optimized cross-members, and truss elements. The maximum von Mises stress and maximum resultant deformation obtained at each stage are summarized in Table 3.

Table 3. Comparison of maximum stress and deformation across design iterations

Study	Maximum stress (MPa)	Maximum deformation (mm)	Design modification
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 results show that the initial design (Static 1) exhibited a maximum deformation of 7.49 mm concentrated in the unsupported front cantilever section carrying the harvesting mechanism. Intermediate reinforcement (Static 2 and Static 3) temporarily increased local stress while reducing deformation, indicating a redistribution of load into the newly added members. Subsequent optimization of cross-members and the addition of truss elements (Static 4 to Static 7) reduced both stress and deformation simultaneously, converging to a stable optimized configuration with a maximum stress of 168.24 MPa and a maximum deformation of 3.63 mm — a 51.5% reduction in deformation relative to the initial design, achieved without a significant increase in structural weight.

4.2 Stress Distribution

In the optimized model, the majority of chassis members remained in the low-stress (blue/cyan) contour range, with localized moderate stress concentrations confined to the front harvesting-unit support and the intersections of longitudinal and transverse members. The maximum stress of 168.24 MPa remains well below the 351.57 MPa yield strength of the structural steel, giving a factor of safety of approximately 2.09, which satisfies the minimum recommended range of 1.5–2.0 for statically loaded agricultural machinery structures.

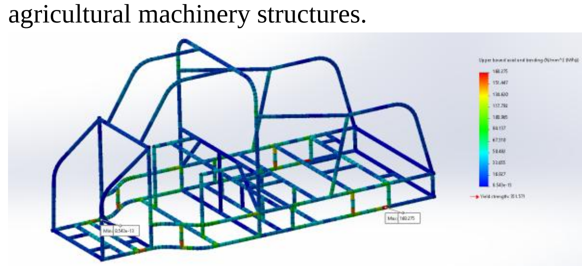

Figure 5. Beam stress (von Mises) distribution of the optimized chassis (Static 7)

4.3 Deformation and Load-Path Behaviour

The maximum resultant displacement occurred near the unsupported front extension of the chassis, which behaves as a cantilever under the concentrated harvesting-unit load. Beam force results indicate that the longitudinal members act as the principal load path, carrying the majority of axial and bending load from the engine, gearbox, operator, and harvesting mechanism, while transverse cross-members distribute load laterally across the frame and reduce localized stress concentration. The deformation of the optimized design remains within limits that are not expected to misalign the harvesting mechanism, engine, or transmission system during field operation.

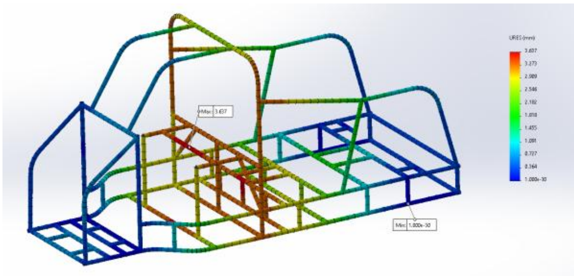

Figure 6. Resultant displacement contour of the optimized onion harvester chassis

4.4 Engineering Discussion

The stress and deformation trends across the seven design iterations demonstrate the effectiveness of iterative, simulation-driven optimization: reinforcement alone (without load-path redesign) can increase local stress even as deformation falls, whereas combined cross-member optimization and truss reinforcement reduce both stress and deformation together. The critical regions identified — the harvesting-mechanism support, the front chassis extension, longitudinal beam intersections, and welded joints under concentrated load — are consistent with the mounting-region stress concentrations reported for other agricultural and vehicular chassis structures [6]– [8], supporting the reliability of the modelling approach adopted in this study.

V. CONCLUSION

This work has demonstrated that a welded tubular chassis for a self-propelled onion harvester can be systematically refined using an integrated CAD–FEA workflow in SOLIDWORKS, without resorting to costly physical prototyping. Starting from a parametric weldment model and a realistic combined loading condition of 5.0–6.25 kN, seven successive design iterations were carried out, each modifying the frame through targeted reinforcement, revised load paths, cross-member redesign, or added truss elements. As the design matured, the peak deformation fell from 7.49 mm in the initial layout to 3.63 mm in the final configuration — a decrease of roughly 51.5% — and the governing von Mises stress settled at 168.24 MPa, comfortably beneath the 351.57 MPa yield limit of the structural steel and yielding a factor of safety near 2.09. These outcomes indicate that the optimized chassis meets accepted structural safety margins for agricultural equipment while avoiding unnecessary material use. Looking ahead, the present static analysis could be extended with dynamic, modal, and fatigue studies, topology-based weight reduction, nonlinear material and large-deflection effects, and physical testing of a fabricated prototype to further corroborate the simulation results.

ACKNOWLEDGMENT

The authors gratefully acknowledge the support of the Department Manufacturing Process Engineering, [M.B.E.Socitey's of Engineering,Ambajogai], for providing the CAD/FEA software resources used in this study.

REFERENCES

[1] S. R. Walunj, S. B. Nalawade, K. R. Kadam, P. R. Thawkar, S. B. Mane, and S. S. Wable, "Design and structural analysis of chassis for a self-propelled onion harvester," *Journal of Agricultural Engineering*, 2023.

[2] K. Dasari, N. Fantuzzi, P. Trovalusci, A. Panei, and M. Pingaro, "Optimal design of a machine structure using parametric structural design and a genetic algorithm," Engineering Structures, vol. 312, Art. no. 118215, 2025.

[3] 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.

[4] K.-J. Bathe, Finite Element Procedures. Englewood Cliffs, NJ, USA: Prentice Hall, 2006.

[5] S. R. Walunj, S. B. Nalawade, K. R. Kadam, P. R. Thawkar, S. B. Mane, and S. S. Wable, "Structural analysis and material evaluation of an onion harvester chassis," 2023.

[6] P. Machado et al., "Structural simulation of agricultural harvester frames under operational loading," 2014.

[7] A. Pawar, "Analysis and optimization of tractor trolley chassis using finite element methods," 2018.

[8] S. Patil and R. Deshmukh, "Finite element analysis in automobile chassis design," 2019.

[9] W. Fricke, "Fatigue analysis of welded joints: State of development," Marine Structures, vol. 16, no. 3, pp. 185–200, 2003.

[10] J. S. Arora, Introduction to Optimum Design, 2nd ed. San Diego, CA, USA: Elsevier Academic Press, 2004.
[11] M. P. Bendsøe and O. Sigmund, Topology Optimization: Theory, Methods and Applications, 2nd ed. Berlin, Germany: Springer-Verlag, 2003.
[12] R. T. Haftka and Z. Gürdal, Elements of Structural Optimization, 3rd ed. Dordrecht, The Netherlands: Kluwer Academic Publishers, 1992.
[13] Bureau of Indian Standards, IS 2062: Hot Rolled Medium and High Tensile Structural Steel Specification, 7th rev. New Delhi, India: Bureau of Indian Standards, 2011.
[14] Bureau of Indian Standards, IS 800: General Construction in Steel—Code of Practice, 3rd rev. New Delhi, India: Bureau of Indian Standards, 2007.
[15] Bureau of Indian Standards, IS 1161: Steel Tubes for Structural Purposes—Specification, 4th rev. New Delhi, India: Bureau of Indian Standards, 1998.
[16] ASTM International, ASTM A36/A36M-19: Standard Specification for Carbon Structural Steel. West Conshohocken, PA, USA: ASTM International, 2019.
[17] Dassault Systèmes, SOLIDWORKS Simulation Help—Static Analysis. Waltham, MA, USA: Dassault Systèmes, 2023.
[18] American Welding Society, ANSI/AWS D1.1/D1.1M: Structural Welding Code—Steel, 24th ed. Miami, FL, USA: American Welding Society, 2020.
[19] J. Moon, J.-W. Yi, B. H. Choi, and H.-E. Lee, “Torsional buckling of I-girder with corrugated webs under uniform bending,” Thin-Walled Structures, vol. 47, no. 1, pp. 21–30, 2009.
[20] H.-X. Wan and M. Mahendran, “Bending and torsion of hollow flange channel beams,” Engineering Structures, vol. 84, pp. 300–312, 2015.