Biological Forum – An International Journal 15(4): 539-543(2023)

ISSN No. (Print): 0975-1130
ISSN No. (Online): 2249-3239

To use the Finite-element Method for Optimization the Frame of a Straw Collector Cum Seeder Machine

Rahul Gautam[1] and A.K. Shrivastava[2*]

[1]Ph.D. Scholar, Department of Farm Machinery and Power Engineering, College of Agricultural Engineering Jawaharlal Nehru Krishi Vishwa Vidyalaya (JNKVV), Jabalpur *(Madhya Pradesh), India.*
Professor, Department of Farm Machinery and Power Engineering, College of Agricultural Engineering Jawaharlal Nehru Krishi Vishwa Vidyalaya (JNKVV), Jabalpur (Madhya Pradesh), India.

(Corresponding author: A.K. Shrivastava)*
(Received: 14 February 2023; Revised: 14 March 2023; Accepted: 19 March 2023; Published: 20 April 2023)
(Published by Research Trend)

ABSTRACT: An effective numerical method for solving frame design problems is presented. The method has been shown to be applicable to non-swaying structures with a plane frame that can be partially or completely raised from the ground. Finite element analysis (FEA) is a computational technique that breaks down complex structures into small elements and solves them numerically using various partial differential equations. In agriculture, engineers can use FEA-based numerical simulation to study the behavior of various input products to optimize the design of any machine without developing a prototype. The individual beams are solved alternately in two orthogonal directions and a relaxation technique is used in each joint to adjust both solutions and achieve rotational compatibility. The present study focused to design and simulates the frame of tractor operated straw collector cum seeder machine by analysis structure in the FEA technique. 3D isometric view of frame of a tractor-operated straw collector cum seeder use as a material in this analysis was created using CAITA V5 and then structure analysis was performed by ANSYS R18.0.

After 3D modeling by CATIA V5, boundary condition was defined. To analysis the static structure analysis of the frame structure to find equivalent results total deformation, equivalent elastic strain, directional deformation (X-axis), minimum principal elastic strain, maximum principal elastic strain, and maximum principal stress were found to be 5.61 mm, 1.06 mm/mm, 1.80 mm, -5.76 mm/mm, 4.89 mm/mm and 0.00055MPa respectively at 100N scraping forces. The static structure analysis calculates the effect of steady (or static) load condition on a structure, while ignoring inertia and damping effect, such as those caused by time varying loads. The FEA approach was found to be a scientific and highly effective method for designing and simulating the frame of a tractor-operated straw collector cum seeder machine.

Keywords: CATIA, ANSYS, frame, deformation, strain, stress and FEA etc.

INTRODUCTION

India's economy is based on agriculture. In its various agro-ecological regions, much of the land is used for farming, and a wide variety of crops are cultivated. According to the 2014-15 land use statistics, the total area is 328.7 Mha, the planted net and the plowed area are 140.1 Mha and 198.4 Mha respectively, with an investment rate of 142%. The net investment area accounts for 43 percent of the total area. A total of 68.4 Mha of land is irrigated. Total rice production for 2018-19, (MoA, 2022-23) is expected to be a record 116.42 Mt. This data is also an impressive 8.62 Mt than five years ago of 107.80 Mt. Wheat productions is expected to raise the record 102.19 Mt, up 2.32 Mt from last year's value 99.87 Mt. over a five-year average of 94.61 Mt. but both the bulk of the crop residues on and off the farm are bought naturally.

Indian agriculture currently produces 500-550 Mt of crop residues per year, with cereals and fiber crops accounting for 58% and 23% of crop residue, respectively. The remaining 19 percent of crop residue is made up of sugarcane, pulses, oilseeds, and other crops. However, a large portion of these crop residues, to about 90-140 Mt per year, are burned on-farm mainly to clear the fields in order to enable the planting of crop production (NASS, 2012). Mostly the farmers and their users use the crop residues as manure, soil mulching, animal feeding, fuel for domestic and industrial purposes etc.

Rice straw production is estimated to be 21 Mt per year, to up to approximately 5% used as cattle feed, 2% used to develop farm structures, 5% used as a raw material for making paper and cardboard, and 7% used as packaging material for horticultural crops and goods like chinaware (Sehgal *et al.*, 1999). Due to the low temperatures and limited time between rice harvesting and wheat sowing, incorporating rice straw before wheat planting is challenging. Because it is a quick and easy method of disposing of paddy straw, approximately 70-75 percent of straw is burned in the fields to ensure that the wheat crop is sown on time. By burning one tonne of paddy straw, approximately 400 kg organic carbon, 5.5 kg nitrogen, 2.3 kg phosphorus, and 25 kg of potash and 1.2 kg of sulfur were lost (Anon., 2014).

The finite element method (FEM) is one of the best-known simulation methods that estimates and analyzes the behavior of a system and structure under defined conditions. In the last twenty years, research work in the field of numerical modeling has been completed, which today allows engineers to predict simulations very close to the real problem. Thus, nonlinear phenomena exist as a standard modeling task in structural phenomena such as nonlinear material behavior, deformation, and contact problems. The development of software technology results in simulations with millions of degrees of freedom, eliminating time and cost. Finite element solutions can be provided using ANSYS software for most engineering disciplines such as statics, dynamics, heat flow, fluid, electromagnetic, as well as coupled field problems. Additionally, ANSYS users can run either linear and nonlinear problems where structural nonlinearities may occur due to nonlinear material behavior, deformations, or guidance boundary conditions (Erke and Thomas 2002).

Research gape: There are many technology or appropriate machinery available to manage straw in-situ or ex-situ condition but there no machine available for both in-situ and ex-situ management of the paddy straw at same time. It encompasses the strategy to determine the design value for the developed prototype, to decide the optimum operational parameters in order to enhance the performance efficiency in actual field condition and to evaluate the economic feasibility of the developed prototype.

MATERIAL AND METHOD

Frame of straw collector cum seeder machine was design and simulated by employing the CAITA V5, and then structural analysis was performed using ANSYS R18.0 software respectively and fabricated at JNKVV, Jabalpur. These frame were fabricated using the Mild Steel because of its various physical and mechanical properties suited for design, cheap and readily available everywhere.

A. Main Frame

Due to their availability on the metal market and their frequent use in agriculture, standard 50 and 55 mm square profiles were used in the design of the frame. The technical specifications of this channel profile were taken from the German national standardization organization Deutsches Institut für Normung (DIN) and were taken into account during the design (Valinejad, 2005). Ease of assembly and disassembly was an important factor in designing the frame.

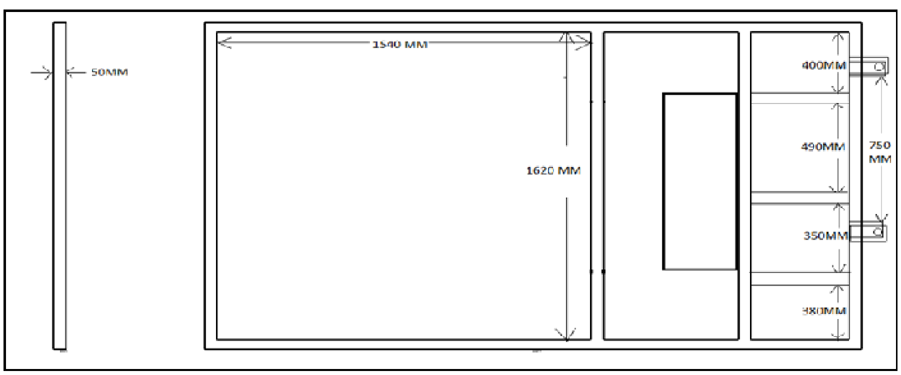

Fig. 1. Main frame of straw collector cum seeder.

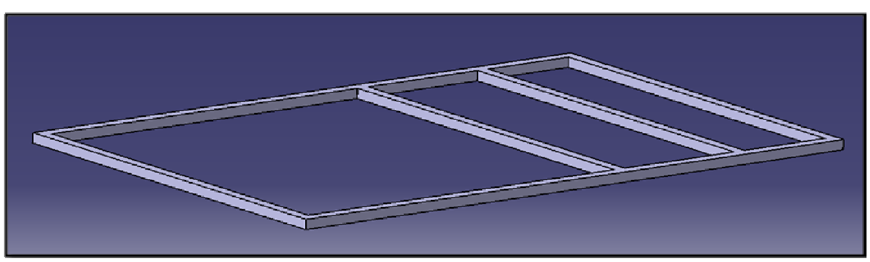

Fig. 2. 3D Isometric view of frame in CATIA.

B. 3-D design stage of the static structure model analysis

• All designs are made using CAITA In version V5 or basically known as the fifth version of CAITA in this it is easy to modify the solid structure easily because the design of blower, cutter and other mechanical parts are designed in CAITA and also alignment in kinematic joint using kinematic tool in analytical machine parts. First, the 2-D structure is basically created using the 2-D tool bar using various tools such as radius flat tools and other basically tools by completing the 2-D structure, then the 2-D the structure is transferred to the 3-D structure by making the actual design, then a different value is given to make the design structure correct as required. Know that the entire design is produced in CAITA. It is then saved to an IGS file by saving as a toolbar. The whole design or now we proceed to testing, so different results are obtained using different tool bars in ANSYS.

- ANSYS is a special software by which we can analyze the actual result of any structure or mechanical structure that basically carries loads, these loads are of different types, directional load or compressive load or expansion load of each type of load can be expressed using this ANSYS software. The tool I used to analyze the actual results of the various spare parts of my machines is a static structure; this static structure is basically put into the field by drop but by dragging to continue the analysis.
- First I entered all the engineering data into the ANSYS file so that ANSYS takes the correct data that I need to design according to the strength of the machine. Engineering data.
- Then I put in the IGS file from CAITA by selecting it and the geometry is imported After importing the geometry from CAITA our average process is to start because first I open the geometry we have to create a mesh which basically means the number of points is generated by the result will be improved. The generated mesh is very fine which will make the result finer basically this mesh is known as diamond mesh. After all the slurry is made, now our main work will begin, this work will put all the forces that are applied by the machine, After applying all the forces, be it fixed forces, rotational forces or bearing joint each joint is verified and all forces are entered in mega Pascal which is a unit of force then time will be entered which basically means how long the result will run the result will run for one hour each result is obtained correctly by running one hour of the relationship.
- All the results that I got in ANSYS are now put into a graph, so a line graph is obtained, which is shown below this line graph, it shows that my material will be the best to use and the machine that I made will work properly in the field without breaking and this machine will be accepted by the farmer so this machine will not produce any kind of waste so the production will be faster in the future, I expect this machine can replace the older machine which can't do maximum work like my machine.
- A 3D isometric view of a tractor-operated straw picker frame with a seed drill used as material in this analysis was created using CAITA V5, and then structural analysis was performed using ANSYS R18.0. After 3D modeling using CATIA V5, the boundary condition was defined. Analyze the static structural analysis of a frame structure to find the equivalent results of total strain, equivalent elastic strain, directional strain (X-axis), minimum principal elastic strain, maximum principal elastic strain, and maximum principal stress. Static analysis of a structure calculates the effect of a steady (or static) load on the structure, ignoring inertia and damping effects such as those caused by time-varying loads.
- **Analyzing the model in ANSYS:** After designing the model in CATIA, the CAT FILE has been converted to IGES format. This format makes the design compatible with ANSYS software. After importing the build into ANSYS, the analysis process begins. Various measures designed for appropriate model analysis. Fig. 3 shown the Flow diagram of a procedure of ANSYS software and Table 1 shown the physical properties of the materials.

C. Methodology of Frame

Table 1: Physical properties of the material.

Mild Steel	
Physical Properties	**Details**
Modulus of elasticity (E), GPa	208
Density, kg/m^3	7800
Tensile Strength, MPa	350
Yield Strength, MPa	175
Poisson's ratio	0.29

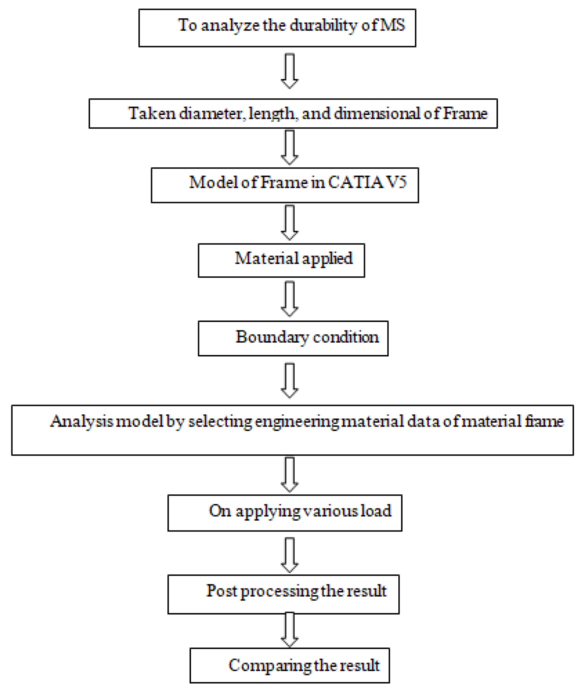

Fig. 3. Flow diagram of a procedure of ANSYS software.

RESULTS AND DISCUSSION

A. Total deformation

The total deformations obtained from the analysis are shown in Fig. 4, respectively. The maximum deformation is observed at the ends of the chassis and decreases as it moves towards the fixed support of the center of the chassis. It also includes the amount of reaction force required to bend the material. It is also used to calculate the displacement of the applied voltage. The maximum deformation observed in the frame is 5.613 mm. The frame structure is then optimized using an optimal space-filling design scheme. An equivalent stress, strain and solid mass were generated for each design point. Red and blue colors represent the maximum (5.613 mm) and minimum (0 mm) values of these output parameters as shown in Fig. 4.

B. Equivalent elastic strain

The team performed FEA of the frame using ANSYS. A load of 100N was applied at the point where the frame should abut on the profile joint. For this analysis, the equivalent elastic strain was calculated, which showed that a maximum value of 1.06 mm/mm was found (Fig. 5).

C. Directional deformation

Directional deformation acts in directions like X, Y & Z. in case of total deformation; it is the square root of the total square of the X, Y and Z direction. For this analysis, the directional deformation strain was calculated; where the X direction axis showed that a maximum value of 1.80 mm was found (Fig. 6).

D. Minimum & maximum principal elastic strain

The distribution of the material is determined by the minimum and maximum principal elastic stress of the product. The color swatch of the model shows the minimum and maximum principal elastic stress. The minimum principal elastic stress was -5.760 mm/mm and the maximum principal elastic stress was 4.89 mm/mm (Fig. 7-8).

E. Maximum principal stress

As shown in Fig. 9, the maximum principal stress is calculated based on the estimated yield failure criteria in ductile materials. The maximum principal stress measured was 0.000556 MPa, while the minimum was -0.0006552 MPa (Jakasania *et al*., 2016). Conducted a similar study and suggested that when designing the model, the working stress should be less than the maximum or ultimate stress at which the material fails".

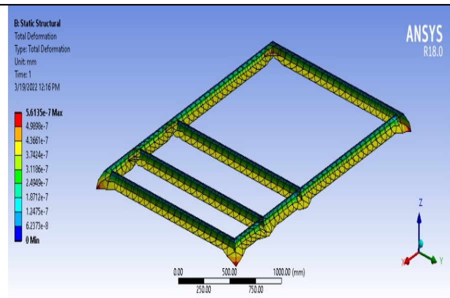

Fig. 4. Total deformation.

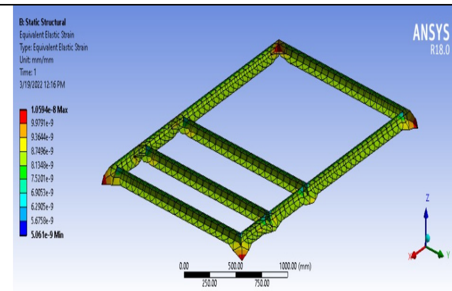

Fig. 5. Equivalent elastic strain.

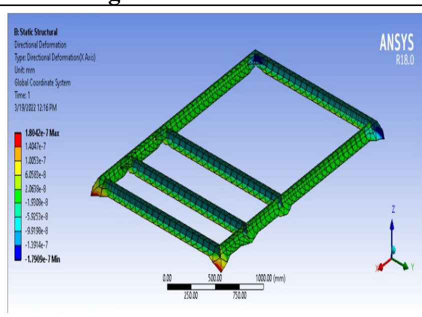

Fig. 6. Directional deformation.

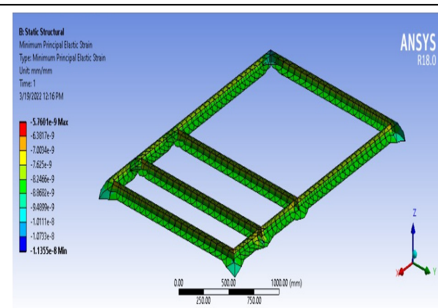

Fig. 7. Minimum principal elastic strain.

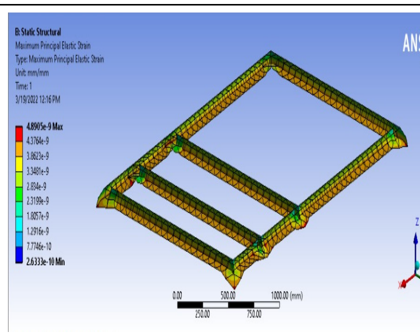

Fig. 8. Maximum principal elastic strain.

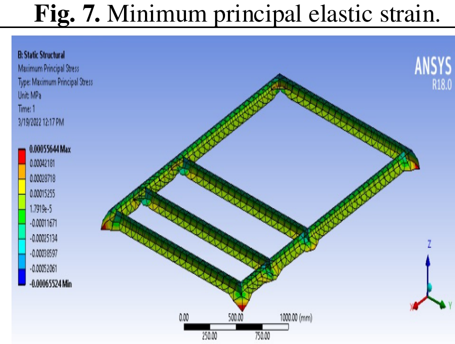

Fig. 9. Maximum principal stress

Table 2: Analysis results of frame.

Name of analysis	Maximum	Minimum
Total deformation	5.613-7mm	0mm
Equivalent elastic strain	1.059-8mm/mm	5.06mm/mm
Directional deformation (X-axis)	1.80-7mm	-1.70-7mm
Minimum principal elastic strain	-5.76-9mm/mm	-1.135-8mm/mm
Maximum principal elastic strain	4.89-9mm/mm	2.63-10mm/mm
Maximum principal stress	0.00055MPa	-0.00065MPa

CONCLUSIONS

A 3D model was created in CATIA V5 design software to design and simulate the frame of the straw picker and seed drill, and the static structure analysis was performed in ANYSIS version R18.0 software. At 100 N scraping force, the simulated results predicted that the total maximum strain was 5.61 mm, the maximum equivalent elastic strain was 1.06 mm/mm, the maximum directional strain (X-axis) was 1.80 mm, the minimum principal elastic strain was 4 .89 mm/mm, the maximum principal elastic stress was 4.89 mm/mm and the maximum principal stress was found to be 0.00055 MPa. It was found that the stress values are within the yield limits of the material. As a result, it was found that the FEA technique is a scientific and very effective approach to the design and frame of the straw picker and planter, and the prototype of the tractor straw picker and planter frame can be developed based on that.

FUTURE SCOPE

The finite element method (FEM) is one of the best-known simulation methods that estimates and analyzes the behavior of a system and structure under defined conditions. In the last twenty years, research work in the field of numerical modeling has been completed, which today allows engineers to predict simulations very close to the real problem.

Acknowledgement. The authors are grateful and thankful to department of Farm Machinery and Power Engineering, College of Agricultural Engineering, JNKVV, JABALPUR for providing financial support and all kind of assistance to AICRP on FIM project at JNKVV, Jabalpur Centre.
Conflict of Interest. None.

REFERENCES

Anonymous (2014). Parali na saado Suchajji varton karo: Bulletin published by Punjab Agricultural University, Ludhiana, India, Pp 2.

Erke, W. and Thomas, N. (2002). Structural Dynamic Capabilities of ANSYS. Germany: CADFEM GmbH.

Jakasania, R. G., Vadher, A. L. and Yadav, R. (2016). Structural analysis of parabolic type subsoiler using CAD software. *Int J of Sci, Environ and Technology, 5*(5), 3415-3422.

MoA (Ministry of Agriculture). (2023). Govt. of India, New Delhi. www.agricoop.nic.in.

NAAS, (2012). Management of crop residues in the context of conservation agriculture. Policy Paper No. 58, National Academy of Agricultural Sciences, New Delhi, (58), 12.

Sehgal, V.K., Arora, M. and Arora, S. (1999). Paddy straw-A valuable waste Proc. Annual Day of ISAE (Punjab Chapter) held at PAU, Ludhiana on Feb 26.

How to cite this article: Rahul Gautam and A.K. Shrivastava (2023). To use the Finite-element Method for Optimization the Frame of a Straw Collector Cum Seeder Machine. *Biological Forum – An International Journal, 15*(4): 539-543.