International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-10, Issue-08, Nov 2024

Structural UDL Analysis of Base Structure of Folding Fixture

V. Guru shanker, Mechanical Engineering, Vignan Institute of technology and science, India, gurushanker43@gmail.com

A. Shiva subramanya sai, Mechanical Engineering, Vignan Institute of technology and science, India, saiakella19@gmail.com

M. Ashok, Mechanical Engineering, Vignan Institute of technology and science, India, ashoknewashok@gmail.com

Abstract. This study presents a structural analysis of the base structure of a folding fixture under various loading conditions with a focus on uniformly distributed loads (UDL). The base structure is a critical component designed to withstand operational stresses while maintaining structural integrity and stability. Using finite element analysis (FEA), we modeled the base structure and evaluated its response to different UDL scenarios, simulating real-world operating conditions. The analysis includes the assessment of stress distributions, deflection patterns, and critical load-bearing capacity under each loading condition. Material properties and boundary conditions were defined to closely replicate the actual environment. The results highlight areas of high stress concentration, potential deformation, and the overall safety factor, providing insights for structural optimization. This research aims to ensure that the folding fixture meets performance and safety standards, informing design improvements for durability and efficiency.

Keywords- Folding Fixture, Deflection Analysis, Uniformly Distributed Load (UDL), Safety Factor.

I. INTRODUCTION

A folding fixture is a frame structure made up of beam members connected at joints, either rigidly or with pins, and oriented in various directions. This fixture serves as a specialized tool to hold workpieces or machine components securely in place during manufacturing processes. The structural members of the fixture are designed to withstand both bending and axial loads, ensuring stability and proper positioning of the workpiece.

Deflection, in this context, refers to the bending deformation of the fixture when subjected to loads. Specifically, it describes the change in shape and position of the beam's longitudinal axis, known as the neutral axis, which, under bending, shifts from a straight line to a curved shape—called the deflection curve.

This study aims to analyze the deflection behavior of the fixture using different engineering materials, each affecting the fixture's performance under load. A static structural model in ANSYS is employed to evaluate the deflection across four selected materials. A simply supported beam with a uniformly distributed load across its entire span is used as the model to simulate real-world loading conditions. The results provide insights into material selection and design improvements, contributing to enhanced durability and efficiency of the fixture in engineering applications.

II. LITERATURE REVIEW

The fixture set up for components is done manually which requires more cycle time for the loading and unloading of the material. So, it was required to develop a system by which we can improve productivity and time. The positions of locators and clamps, and the values of clamping force should be properly selected and calculated so that the workpiece deformation due to clamping and cutting force is minimized and uniformed. Weifang Chen & Lijun Ni & Jianbi n Xue (2007) [1] developed a multi-objective model for controlling the deformation of the workpiece. The main objective is to find an optimal layout or positions of the fixture elements around the workpiece and optimal clamping force. The objective is two folded and one of them is to minimize the maximum elastic deformation of the machined surfaces, and another is to maximize the uniformity of deformation. Authors have used the ANSYS software package to calculate the deformation of the workpiece under given clamping force and cutting force. The maximal cutting force is calculated in the cutting model and the force is sent to the finite element analysis (FEA) model along with the other boundary conditions. Then the optimization procedure is followed to get the optimum result. This process repeats itself until we get the optimum result. The model for the FEA analysis is a semi-elastic contact model considering friction effect, where the materials are assumed linearly elastic. In FEA, machining deformation under the cutting force and the clamping force is

 DOI : 10.35291/2454-9150.2024.0407

International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-10, Issue-08, Nov 2024

calculated using a finite element method under a certain fixture layout, and the deformation is then sent to optimization procedure to search for an optimal fixture scheme. The software package of ANSYS is used for FEA calculations in this study.

III. MODELLING & ANALYSIS

The engineering design process is a critical step in product development. Designing a fixture begins with evaluating its functional requirements to achieve a balanced combination of design characteristics at a reasonable cost. At this stage, the tool design process should clearly define the problem to be solved and establish the desired outcomes.

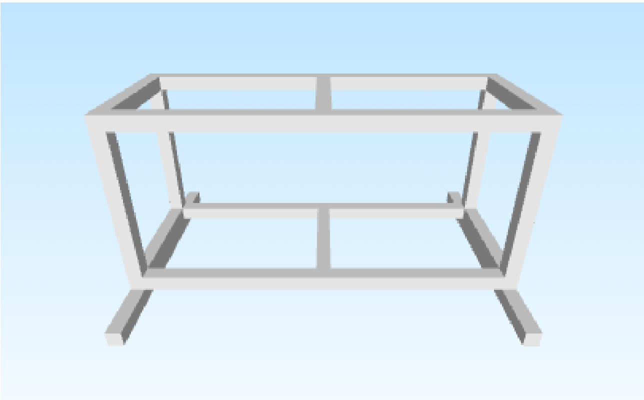

Fig.1 Folding Fixture CAD Model

The design process also includes analyses related to the product, such as determining load distribution, material selection, and other critical factors. This step ensures the design meets both functional and performance requirements.

Table.1 Dimension & Physical Properties Folding Fixture

Geometrical Model Dimensions	
Length in X direction	2.7 m
Length in Y direction	1.8 m
Length in Z direction	1.35 m
Mass	1898.7 kg
Centroid X	-1.2484 m
Centroid Y	5.7375e-017 m
Centroid Z	0.6221 m
Scale factor value	1
Moment of Inertia Ip1	916.86 kg·m²
Moment of Inertia Ip2	2603.1 kg·m²
Moment of Inertia Ip3	2396.6 kg·m²
Material Properties	
Type of Material	Structural Steel
Density	7850 kg /m^3
Thermal Conductivity	55 W /m K
Poisson's Ratio	0.3

Additionally, this process explains how the folding fixture is analyzed. CAD tool is utilized for design modeling and Finite Element Analysis (FEA). The primary goal of the FEA analysis is to determine deformation values under applied loads and estimate the maximum allowable strength based on the material's yield strength. These values are essential to assess whether the part design is suitable and meets the specified requirements.

Table 2. shows the Mesh details of fixture

FEA Statistics	
Nodes	10572
Elements	4732
Mesh Metric	None
Stiffness Behavior	Flexible

IV. RESULTS & DISCUSSION

Static analysis includes deformation analysis; Von-Mises stress analysis, translational displacement analysis, and principal stress evaluation. From these results, several critical factors must be understood to ensure the product remains free from failure. Additionally, key material properties such as Young's modulus, yield strength, and density are identified in the analysis, as they significantly impact the product's performance. This data is crucial for determining potential failure factors before proceeding to manufacturing.

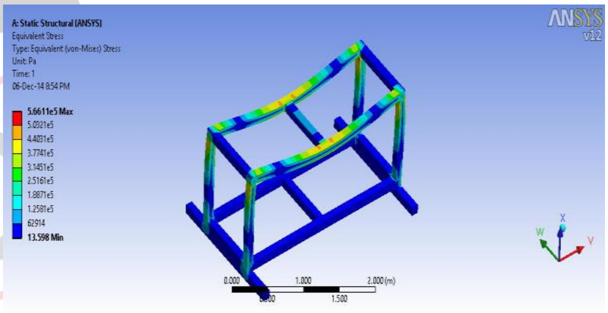

Fig.2 Total deformation of beam

Fig. 2 shows the total deformation refers to the combined displacement of the structure in all three coordinate directions (X, Y, Z). This result provides an overall measure of how much the structure moves from its original position due to applied loads and boundary conditions. Red color areas experiencing the maximum deformation (2.2342×10−5 m or 22.342 µm), Likely occurs at the center of the curved top beam, which is the most flexible region and the Blue color areas with minimum deformation (almost 0 m), Found at fixed or supported areas, such as the base corners or where constraints are applied.

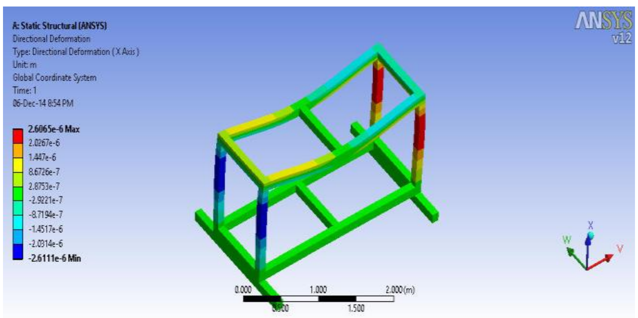

Fig.3 Directional Deformation along X-Axis

 DOI : 10.35291/2454-9150.2024.0407

The fig. 3 shows the directional deformation (X-axis) for a static structural analysis. This indicates how much each point of the structure has shifted along the X-axis due to the applied loads and constraints. Red color areas experiencing the highest positive deformation along the X-axis Likely occur near the top part of the frame, where the structure bends., and the blue color areas with the most negative deformation, Found in the bottom support areas, where constraints or loading cause opposite deformation.

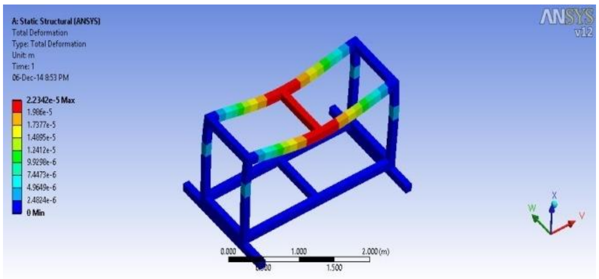

Fig.4 Equivalent (Von-mises) Stresses

Fig. 4 shows the Equivalent Stress (von-Mises Stress) distribution of a structure analyzed in ANSYS, Von-Mises stress is a measure used to determine whether a material will yield or fail under complex loading. It combines stresses from all directions into a single value for comparison against the material's yield strength. The top curved beam and areas near connections (junctions) show maximum stress. These areas are critical for potential failure or yielding. Red color Maximum stress (5.6611×105 Pa) and Blue color Minimum stress (13.598 Pa), typically at fixed or lightly loaded regions.

V. CONCLUSION

This holding fixture has significant changes to job handling during process operations. By utilizing modern tools in system design, significant improvements can be ensured. To meet multifunctional and high-performance fixturing requirements, an optimum design approach can be employed to provide comprehensive analyses and determine the overall optimal design.

The structural analysis highlights that the curved top beam is the most critical region in terms of both deformation and stress. The maximum total deformation is 22.342 µm

22.342µm, occurring at the top beam, while the maximum von-Mises stress is 566.11 kPa,concentrated near the curved beam and connection points. In contrast, the vertical supports and base experience minimal deformation and stress, providing stability to the structure.

Overall, the structure is rigid under the applied loads, with small deformation magnitudes. However, the stress concentration at the curved beam indicates the risk of yielding or failure under increased loading. To address this, design optimizations are recommended, such as increasing the thickness of the curved beam, redistributing stresses with reinforcements, or using materials with higher yield strength. These improvements will ensure the structure remains safe, functional, and capable of withstanding the applied loading conditions.

REFERENCES

[1]. Bi ZM, Zhang WJ. Flexible fixture design and automation: review, issues and future direction. International Journal of Production Research 2001;39(13): 2867–94

[2]. Weifang Chen ,Lijun Ni & Jianbin Xue "Deformation control through fixture layout design and clamping force optimization" Int J Adv Manuf Technol 38:860–867 DOI 10.1007/s00170-007-1153-2,2008.

[3]. Shailesh S Pachbhai and Laukik P Raut (2014), "A Review on Design of Fixtures", International Journal of Engineering Research and General Science, Vol. 2, No. 2, ISSN, pp. 2091-2730.

[4]. J. Cecil "A Clamping Design Approach for Automated Fixture Design" Int J Adv Manuf Technol 18:784– 789,2008

[5]. Nicholas Amaral,Joseph J. Rencis,Yiming (Kevin) Rong "Development of a finite element analysis tool for fixture design integrity verification and optimization" Int J Adv Manuf Technol 25: 409–419, 2005

[6]. Y. Wang, X. Chen. N, Gindy "Surface error decomposition for fixture development" Int J Adv Manuf Technol DOI 10.1007/s00170-005-0270-z, 2007

[7]. R. Huntera, J. Riosb, J.M. Pereza, A. Vizan, "A functional approach for the formalization of the fixture design process", International Journal of Machine Tools & Manufacture 46 (2006) 683–697

[8]. A. Senthil Kumar, V. Subramaniam, T.B. Teck, "Conceptual design of fixtures using machine learning techniques", International Journal of Advanced Manufacturing Technology 16 (2000) 176–181

 DOI : 10.35291/2454-9150.2024.0407