Advanced International Journal for Research (AIJFR)
E-ISSN: 3048-7641 ● Website: www.aijfr.com ● Email: editor@aijfr.com

Drilling Fixture Analysis by Developing Finite Element Model Using ANSYS

Ms. Apoorva Tiwatane [1], Prof. Amit Desai[2]

[1]M.Tech, Student, Department of Mechanical Engg., Bharati Vidyapeeth's College of Engineering, Kolhapur. M.S., India.
[2] Asst. Professors, Department of Mechanical Engg., Bharati Vidyapeeth's College of Engineering, Kolhapur. M.S., India.

Abstract
The objective of this analysis is to evaluate the stress distribution and structural integrity of the Drilling Fixture under the specified loading conditions. A Linear Static Finite Element Analysis (FEA) has been performed to assess the stress levels and deformation behavior of the fixture. The 3D CAD model used for the analysis was developed based on the geometric details provided in the relevant engineering drawings, which are listed in the following chapter. The finite element model was created and analyzed using the ANSYS software package. The results obtained from the analysis are used to verify the adequacy of the fixture design and ensure that the induced stresses remain within the allowable limits of the selected material.

Keywords: Drilling fixture, vertical fixture, angular operations, stress distribution, structural integrity, cylinder block.

1. Introduction

The vertical operation fixture is developed to be used in cylinder block manufacturing and is a type of fixture which is designed for performing machining operations on the surface or the holes which are either vertical or at a certain angle to the basic axes of the cylinder block. This is used to perform various operations like precise machining on the complex shapes such as oil holes, coolant holes, side mount face, and angle drilling. This fixture is intended to fix the cylinder block at a particular location on the table and orient it at a desired angle. The angular operations using this fixture will be carried out by providing machining programs to the table. This angled fixture is a unique kind of fixture which is made to locate the component in vertical orientation, and table will be oriented at a certain angle for machining purpose. This fixture keeps the cylinder block flat or straight on the table, and the table will orient at an angle to perform machining on straight or angled surfaces. This fixture enables machining of straight and angled holes or surfaces accurately and consistently. The cylinder block fixture is used for manufacture of cylinder blocks since it increases the production rate, provides accuracy and consistency in production process.

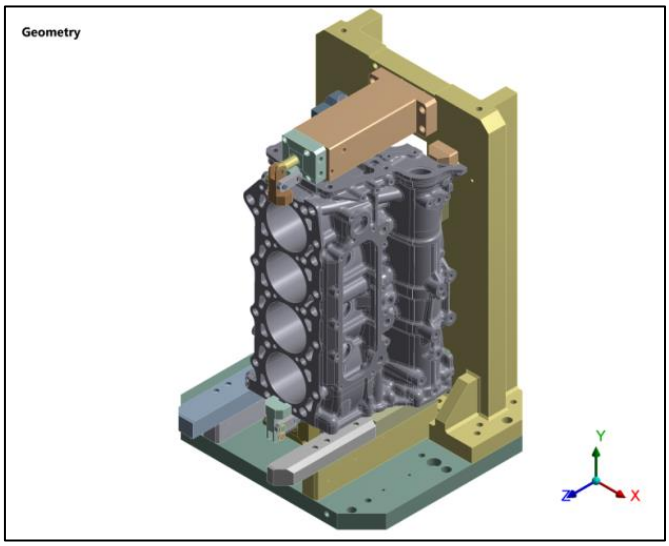


Fig.1. 3-D Model Geometry view

Analysis Using Ansys

The Finite Element Analysis (FEA) is a numerical method used to predict the structural behavior of components under different loading conditions. In this study, a Linear Static Structural Analysis of a drilling fixture was performed using ANSYS Workbench to evaluate stresses and deformations generated during machining operations. The objective of the analysis was to verify that the drilling fixture can safely withstand hydraulic clamping forces, cutting forces, and self-weight without exceeding the allowable stress limits of the material.

- **Geometry Preparation**

The 3D CAD model of the drilling fixture assembly was created based on the available engineering drawings. The model included all major structural components such as base plate, vertical support plate, clamps, rest pads, support pads, work piece (Engine Block Casting). Small geometric features that do not significantly affect the structural behavior were removed to simplify the analysis and reduce computational time.

- **Material Properties**

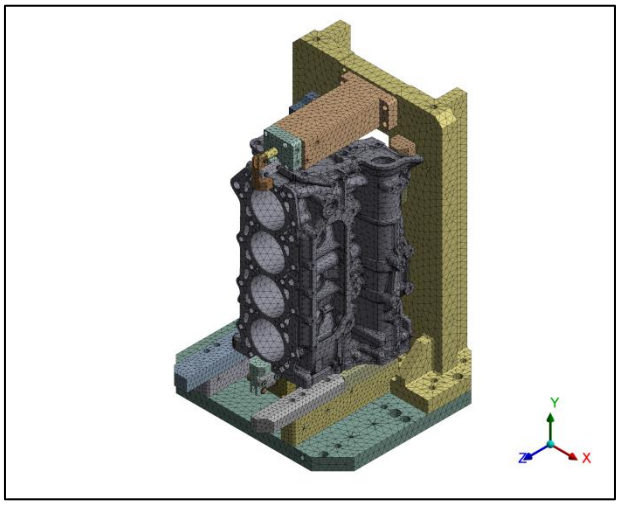
Material properties were assigned to all fixture components before performing the analysis. These properties are required by the software to calculate stresses and deformations.

For Mild Steel (MS):

Property	**Value**
Young's Modulus	205800 MPa
Poisson's Ratio	0.30
Yield Strength	235 MPa
Density	7850 kg/m³

- **Meshing**

The geometry was divided into a large number of small finite elements. This process is called meshing. A finer mesh was provided near clamping locations, rest pads, support pads, cutting force locations. Mesh quality was checked using standard parameters such as aspect ratio, Jacobean Ratio, skewness, element quality. All mesh quality parameters were found within acceptable limits, ensuring reliable analysis results.



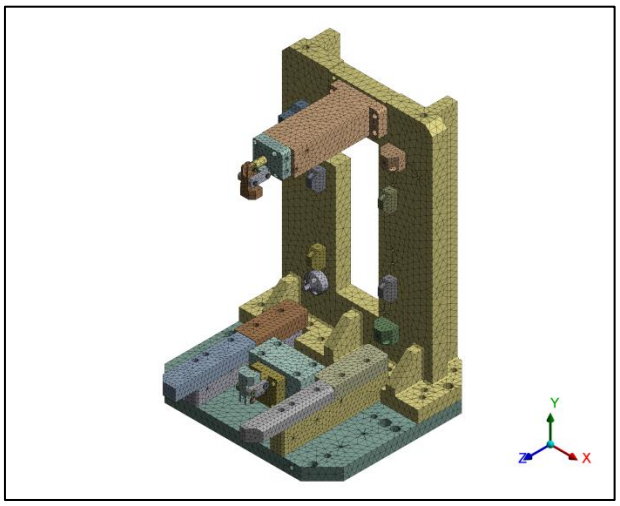


Fig.2. Isometrics view of Mesh

- **Boundary Conditions**
  - **Fixed Support:**

The base plate of the fixture was assumed to be rigidly attached to the machine table. Therefore, a Fixed Support boundary condition was applied at the mounting locations of the base plate. This prevented movement in all directions.

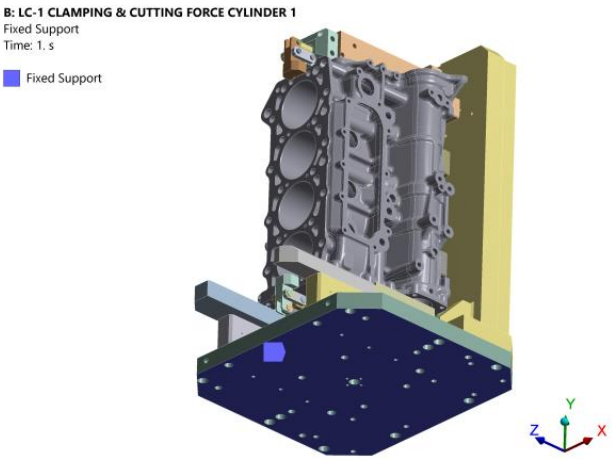


Fig.3. Fixed Support Boundary Condition

- **Self-Weight**

The self-weight of the fixture was considered by applying standard Earth gravity: g = 9.81 m/s². This load accounts for the weight of all fixture components.

- **Hydraulic Clamping Force**

Hydraulic cylinders are used to hold the casting firmly during machining. A hydraulic clamping force of **7853.9 N** was applied through the top and bottom cylinders at the contact locations between the clamp and casting.

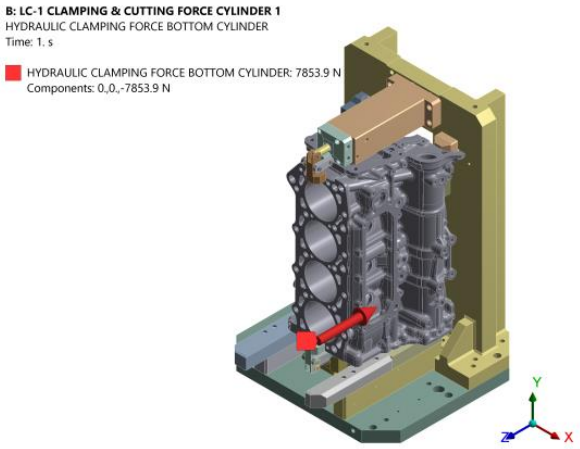


Fig.4. Hydraulic Clamping Force by Bottom Cylinder

- **Cutting Force**

During drilling and boring operations, cutting forces are generated by the cutting tool. To simulate actual machining conditions, a tangential cutting force of **750 N** was applied at the cylinder bore being machined.

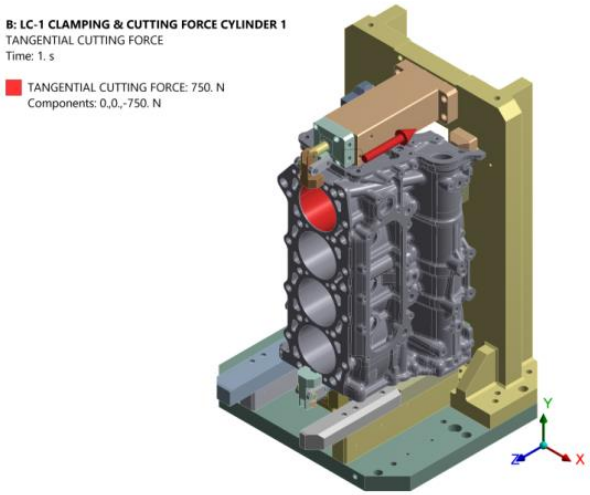


Fig. 5. Tangential Cutting Force in 1st Cylinder Bore

- **Load Cases**

Since the casting contains four-cylinder bores, the cutting force was applied separately to each bore location. For all load cases, gravity, support conditions, and clamping forces remained unchanged. Only the cutting force location was changed. The following load cases were considered:

Load Case	Cutting Force Location
LC-1	Bore 1
LC-2	Bore 2
LC-3	Bore 3
LC-4	Bore 4

- **Result Evaluation**

The maximum stresses obtained from the analysis were compared with the material yield strength. For Mild Steel, the Yield Strength is 235 MPa. The maximum stress observed in the fixture components was significantly lower than the allowable stress limit. Therefore, all major components of the drilling fixture were found to be structurally safe under the applied loading conditions.

Location	Equivalent stress (MPa)	Yield Stress (MPa)	Result
Maximum stress at Clamping Pad	**58.90**	235	**SAFE**
Maximum stress at Vertical Plate	**7.75**	235	**SAFE**
Maximum stress at Base Plate	**2.79**	235	**SAFE**
Maximum stress at Base Support Pad	**17.62**	235	**SAFE**
Maximum stress at Rest Pad	**46.49**	235	**SAFE**

- **Ansys Observations:**

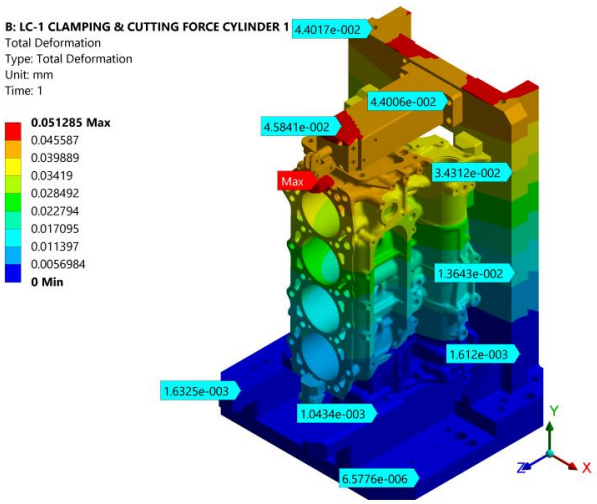


Fig.6. Total Deformation

Conclusion

A Linear Static Finite Element Analysis was successfully performed on the drilling fixture. The analysis included self-weight, hydraulic clamping forces, and machining forces. The obtained stresses were within the allowable limits of the material, and the deformation was very small. Hence, the drilling fixture is considered safe and suitable for the intended machining operation.

References

1. https://www.irjet.net/archives/V3/i8/IRJET-V3I887.pdf
2. Sabah Ibrahim, Abdullah. (2024). Static Structural Analysis on Cylinder Block in Ansys Workbench. 10.13140/RG.2.2.20053.44008.
3. https://www.ijsrd.com/articles/IJSRDV5I110230.pdf