Stress Analysis Using FEM For The Tooth of Earth Mover Shovel to Enhance Structural Strength

#1 Pradip N. Mule, #2 Arun V. Bhosale

1 Mechanical Design Engineering Department, Savitribai Phule Pune University, Pune
2 Mechanical Design Engineering Department, Sinhgad academy of engineering, kondhwa, pune
Savitribai Phule Pune University, Pune
Advent Tool Tech Pvt.Ltd. Pune,India.

ABSTRACT

The shovel of the earth-moving equipment rakes out soil while hitting the ground and nudging the matter into the cavity of the shovel. Teeth are provided along the edge of the bottom face of the shovel to assist penetration in the ground. This helps the digging action of the mechanized shovel that moves the grit and gravel from one location to the other. The fabricated structure of the component is designed to offer robustness while tilling the ground and ferrying the same into a container or dumper. The teeth fastened to the lower edge of the shovel experiences impact action of the shovel while hitting the ground. This dissertation work shall offer research on the topic in terms of investigating the nature and magnitude of stresses induced in the tooth of the shovel that leads to potential failure during operation. Besides the preliminary investigation using empirical formulae, the computational techniques shall be deployed using 'HyperWorks' for Analysis of the behavior of the component while subjected to loading during operation. Variants shall be proposed towards possible solution for enhanced strength. Validation shall be attempted for the basic variant used during the initial phase of the work.

Keywords— Digging action, HyperWorks, Impact action, Potential failure.

I. INTRODUCTION

Earth mover shovel are primarily used to excavate earth and dump the materials. Backhoe digs, elevates, swings and dumps materials by the action of its mechanism, which consists of boom, swing frame, arm, bucket, tooth of bucket and hydraulic cylinders. Due to severe working conditions, backhoe loader parts are subjected to corrosive effects and high loads. The backhoe loader mechanism must work reliably under unpredictable working conditions. Poor strength properties of the backhoe loader parts like boom, arm, tooth of bucket and swing frame limit the life expectancy of the backhoe loader. Therefore, backhoe loader parts must be strong enough to cope up with caustic working conditions of the backhoe loader. It can be concluded that, strength analysis is an important step in the design of backhoe loader parts.

Finite Elemental Analysis (FEA) is the most powerful technique in stress and deformation analysis of the structures working under known load and boundary conditions. One can determine the critical loading conditions of the backhoe loader by performing static force analysis of the backhoe mechanisms. The boundary conditions for stress analysis will be determined according to results of the static force analysis. CATIA model of the parts to be analyzed must be prepared prior to FEA. Preparations of CATIA model and performing FE analysis...
take considerable amount of time and needs experienced user. The shovel of the earth-moving equipment rakes out soil while hitting the ground and nudging the matter into the cavity of the shovel. Teeth are provided along the edge of the bottom face of the shovel to assist penetration in the ground. This helps the digging action of the mechanized shovel that moves the grit and gravel from one location to the other. The fabricated structure of the component is designed to offer robustness while tilling the ground and ferrying the same into a container or dumper. The teeth fastened to the lower edge of the shovel. These teeth experiences impact action of the shovel while hitting the ground. Due to the impact action of the teeth it may lead to potential failure during operation. The shovel of the earth-moving equipment rakes out soil while hitting the ground and nudging the matter into the cavity of the shovel. Teeth are provided along the edge of the bottom face of the shovel to assist penetration in the ground. This helps the digging action of the mechanized shovel that moves the grit and gravel from one location to the other. The fabricated structure of the component is designed to offer robustness while tilling the ground and ferrying the same into a container or dumper. The teeth fastened to the lower edge of the shovel experiences impact action of the shovel while hitting the ground. Besides the preliminary investigation using empirical formulae, the computational techniques shall be deployed using 'HyperWorks' for Analysis of the behavior of the component while subjected to loading during operation. Variants shall be proposed towards possible solution for enhanced strength. Validation shall be attempted for the basic variant used during the initial phase of the work.

2.2 Probable Solution:
The Design of the tooth is a potential area that could be reviewed for finding solution. Alternative material used for manufacturing can also be proposed for longer life. CAE methodology could be utilized for solving the Finite Element Model while determining the variants. The Design parameters like the Geometry and/or the Material and/or Boundary conditions can be varied to find its effect on the failure of the tooth. The key objective for the work being to reduce the incidence of failure during usage on the field. Use of FEM Solvers like HyperWorks, MSC Nastran or ANSYS can address the analytical treatment to the given problem. Mathematical treatment can be offered using Empirical formulae and Literature Review for the proposed work.

III. OBJECTIVES OF PROJECT
1. Study the existing system.
2. Mathematical calculation for forces.
3. Modeling and meshing of the component.
4. Finite element Analysis.
5. Validation of results with Experimental results.

IV. SCOPE OF PROJECT
Preliminary investigation using empirical formulae, the computational techniques shall be deployed using 'hyperworks' for analysis of the behavior of the component while subjected to loading during operation. Variants shall be proposed towards possible solution for enhanced strength.

V. METHODOLOGY

VI. SOFTWARE USED FOR FE ANALYSIS
1. 3D Modeling: The softwares used for 3D modeling are CATIA V5.
2. **Pre-processor**: Hyper Mesh is a software used as a preprocessor.

3. **Solver**: The suitable type of softwares such as NASTRAN, Radioss, ANSYS can be used as a solver.

4. **Post-processor**: Hyper View is used for the presentation of the result.

VII. EXPERIMENTATION AND VALIDATION

**Experimentation**

Charpy impact test shall be proposed for experimentation.

**Charpy impact test:**

It is also known as the charpy v-notch test, is a standardized high strain rate test which determines the amount of energy absorbed by a material during fracture. This absorbed energy is a measure of a given material's notch toughness & acts as a tool to study temperature dependant ductile- brittle transition. It is widely used in industry since it is easy to prepare and conduct and results can be obtained quickly and cheaply.

**Validation**

Validation shall be attempted by comparing the results obtained through comparison of finite element analysis and experimentation.

REFERENCES

[1] Martin Genet, Wenyi Yan, Thanh Tran-Cong “Investigation of a hydraulic impact: a technology in rockBreaking”, Received: 23 August 2007 / Accepted: 5 August 2008 / Published online: 26 August 2008.


