**DESIGN AND ANALYSIS OF MACHINE TOOL**

 **ELEMENTS USING ANSYS**

**ABSTRACT**

The performance of a machine tool is eventually assessed by its ability to produce a component of the required geometry in minimum time and at small operating cost. It is customary to base the structural design of any machine tool primarily upon the requirements of static rigidity and minimum natural frequency of vibration. The operating properties of machines like cutting speed, feed and depth of cut as well as the size of the work piece also have to be kept in mind by a machine tool structural designer. This paper presents a novel approach to the design of machine tool element for static and dynamic rigidity requirement. Model evaluation is done effectively through use of General Finite Element Analysis software ANSYS. Studies on machine tool column are used to illustrate finite element based concept evaluation technique.

In this project, we will create machine tool element models like spindles, bearing supports, saddles (carriages), work piece tables, and composite housings with columns, arms, beds, etc. using solid works premium 2014 software. And we perform a structural analysis on that created models using finite element method analysis i.e., ansys software. We will find out the stress induced in the machine element particles due to the applied loads and fixtures. This may undergoes Deformations also due to the excess of high loads on the machine tool elements.